

ATHENA₂₀₁₄

The AutoCAD application for design in metal construction and facade engineering

CAD-PLAN GmbH
Frankfurter Straße 59-61
63067 Offenbach, Germany
Tel: +49-69-800818-0
Fax: +49-69-800818-18
info@cad-plan.com
www.cad-plan.com

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A Installation and Authorization

This capital contains instructions for the installation and authorization of ATHENA 2014.

1 System requirements

The recommended system requirements for ATHENA 2014 depend on the version of AutoCAD that is used. You can obtain this information from your AutoCAD documentation.

The requirements for the current AutoCAD versions can be found in the Internet under www.autodesk.com. After selecting the product, select the menu point Functions and Specifications. Here, you will also find information on graphics cards which are suitable or certified for AutoCAD.

ATHENA 2014 can be installed under the following AutoCAD versions and vertical products:

Prefix	Label	Release
7xxx	2009	17.2
8xxx	2010	18.0
9xxx	2011	18.1
Axxx	2012	18.2
Bxxx	2013	19.0
Dxxx	2014	19.1

Suffix	Label
x001	AutoCAD
x00A	AutoCAD OEM
x003	Inventor Series
x004	AutoCAD Architecture
x005	AutoCAD Mechanical
x006	AutoCAD MEP
x007	AutoCAD Electrical
x012	ADT for Raster Design
x013	Inventor Professional
x014	Inventor Professional for Routed Systems
x015	Inventor Professional for Simulation

You will find the prefix in the registry under:

HKEY_CURRENT_USER\Software\Autodesk\AutoCAD\R19.0\ACAD-D004:407

The highlighted number D7004 would therefore be the version 2014 (B) of AutoCAD Architecture (004).



When using vertical products (e.g. AutoCAD Architecture) we recommend a pure AutoCAD profile as the basis for the installation.

To do this, install the appropriate product with the pure AutoCAD profile. It is essential to follow the information in your software package and in case of doubt to consult your ATHENA Support Team.

2 Installation of ATHENA 2014

You can operate ATHENA 2014 as a single license or as a network license.

If you want to operate ATHENA as a single license, you need a hardware protection plug (hardware lock) for each workstation.

If you wish to operate ATHENA 2014 as a network license (floating license), you need a hardware protection plug for the server (here you must also install a license manager).

Hardware protection plugs are currently only available for USB slots.

ATHENA 2014 is available in 32 and 64 bit versions. The installation program automatically detects your environment and installs the appropriate version.

2.1 Installation variants

Irrespective of whether you wish to operate ATHENA as a single license or a network license, you can install the program files on the local hard disk or in a server path.

Advantages of the local installation: You can use ATHENA even when the server fails (assuming AutoCAD is also used as a single license).

Advantages of the server installation: This type of installation simplifies the administration required, because the installation of updates must only be implemented once. Networks are primarily recommended where there are many CAD workstations.



Both installation variants can take place in combination with an AutoCAD single license (SLM) or a network license (NLM).

If you are operating 32-bit and also 64-bit systems in one network, a separate server installation is required for each version.

Installing a single license locally

You install ATHENA on each workstation as described in section *Install ATHENA* on page 12. You can specify the installation paths for the program folders in the appropriate dialog boxes of the setup. Use local folders for each workstation (we recommend the default folder for this).

Installing a single license on the server

You install ATHENA on the first workstation as described in section *Install ATHENA* on page 12. You can specify the installation paths for the program folders in the appropriate dialog boxes of the setup.

Then set up an ATHENA profile for AutoCAD on all further workstations. The procedure is described in the section *Create ATHENA profile* on page 17.



For each workstation use the same server paths.

Installing an ATHENA network license locally

Install the drivers for the hardware protection plug and the license manager on the server as described in section *Hardware protection - installing the drivers/ license manager* on page 18.

You then install ATHENA on each workstation as described in section *Install ATHENA* on page 12. You can specify the installation paths for the program folders in the appropriate dialog boxes of the setup. Use local folders for each workstation (we recommend the default folder for this).

Installing an ATHENA network license on the server

Install the drivers for the hardware protection plug and the license manager on the server as described in section *Hardware protection - installing the drivers/ license manager* on page 18.

You install ATHENA on the first workstation as described in section *Install ATHENA* on page 12. You can specify the installation paths for the program folders in the appropriate dialog boxes of the setup.

Then set up an ATHENA profile for AutoCAD on all further workstations. The procedure is described in the section *Create ATHENA profile* on page 17.



For each workstation use the same server paths.

2.2 Planning the installation

In particular when installing on a number of workstations in a company network, you should plan which data are to be installed on a local drive and which are to be installed on a server drive. Below you will find an overview of the folders with default paths and a brief description of the folder contents.

The standard program folder (depending on the operating system used, e.g. C:\PROGRAM FILES) is given in the following in each case with PROGRAMS\.

The folder for user-specific application data (depending on the operating system used, e.g. C:\USERS \USERNAME \APDATA \ROAMING) is given in the following in each case with ...USERDATA\.

ATHENA folder:

ATHENA:	Folder for ATHENA 2014 program files ...PROGRAM FILES\CAD-PLAN\2014\ATHENA
Data local:	Folder for local settings, e.g. drawing frames, text boxes, plot script files. ...USERDATA\CAD-PLAN\2014\ATHENA\DATALOCAL
Data group:	Folder for group settings, e.g. standard part definitions. ...USERDATA\CAD-PLAN\2014\ATHENA\DATAGROUP
Systems:	Folder for files from the profile manufacturers. Here, further subdirectories for various manufacturers are set up in which you can copy profile files and system sections (*.dwg) from the manufacturers. ...USERDATA\CAD-PLAN\2014\ATHENA\SYSTEMS
User data:	Folder for user blocks and temporary drawings. ...USERDATA\CAD-PLAN\2012\ATHENA\SYSTEMS\USER
Project folder:	Folder for project drawings C:\DWG

CPL (CAD-PLAN core functions) folder:

CPL:	Folder for general program files for CAD-PLAN applications. ...PROGRAM FILES\CAD-PLAN\2014\CPL
Data local:	Folder for local settings for CAD-PLAN applications. ...USERDATA\CAD-PLAN\2014\CPL\DATALOCAL
Data group:	Folder for group settings for CAD-PLAN applications. ...USERDATA\CAD-PLAN\2014\CPL\DATAGROUP

Local settings and group settings

When ATHENA is installed in a network, you can move files from the folders DATALOCAL to the folder DATAGROUP and vice versa. Thus, you can control which settings are to be user-specific and which settings are to be the same for all users.



If the same files are present in both folders, the files from the folder DATALOCAL are given priority.

The default settings after installation appear as follows:

Content of the folder:

...USERDATA\CAD-PLAN\2014\ATHENA\DATAGROUP

ath_htr.dex

In this file ATHENA saves user-defined materials for the computation of the thermal resistance - see also Chapter *User catalog* on page 130.

Content of the folder:

...USERDATA\CAD-PLAN\2014\ATHENA\DATALOCAL

ath_dim.dex

Dimension settings for the corresponding dimension styles are defined in these files - see also Chapters *Dimension styles* on page 58 and *Manage dimension styles* on page 454.

ath_nm_prop.dex

ATHENA saves material assignments for standard parts in this file - see also Chapter *Standard Part*, Section *Materials* on page 255.

ath_obj_order.dex

ATHENA saves item and process numbers in this file - see also Chapter *Item* on page 125.

ath_obj_prop.dex

ATHENA saves various object properties in this file - see also Chapters *Layer assignment* on page 108, *Hatch pattern assignment* on page 110 and *Label* on page 117.

ath_txt.dex

Text styles are defined in this file - see also Chapters *Text styles* on page 59 and *Manage text styles* on page 526.

ath_var.dex

System settings are defined in this file - see also Chapters *AutoCAD system variables* on page 57 and *Manage system variables* on page 609.

blocklib.dat

ATHENA saves block management data in this file - see also Chapter *Block manager* on page 579.

plot*.scr

The settings for the plot routines are defined in these files - see also Chapters *Plotter configuration* on page 61 and *Plotting* on page 620.

Content of the folder:

...USERDATA\CAD-PLAN\2014\CPL\DATAGROUP

Contains no data

Content of the folder:

...USERDATA\CAD-PLAN\2014\CPL\DATALOCAL

cpl_layer.dex

ATHENA saves layer settings in this file - see also Chapter *Layer* on page 85.

cpl_mat.dex

ATHENA saves material settings in this file - see also Chapter *Material* on page 81.

cpl_base.cfg

ATHENA-specific settings are saved in this file. These are settings which are defined in the Dialog box ATHENA options.

2.3 Installation requirements

Please check the requirements quoted below in good time before the installation so that where required you can carry out corrections.

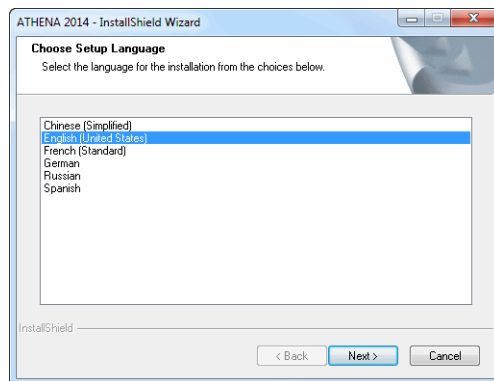
- Make sure that the operating system and AutoCAD are properly installed. We recommend a full AutoCAD installation.
- Currently, ATHENA 2014 is supplied with Sentinel HASP HL Pro or a Sentinel HASP HL Net hardware protection. Older Sentinel/SafeNet (Rainbow) SuperPro or SuperProNet hardware protection plugs are **not** supported and must be replaced.
- If you want to use ATHENA with a vertical Autodesk product (e.g. AutoCAD Architecture), we recommend that an AutoCAD profile is set up when installing it.
- To ensure that the ATHENA installation routine can set up an AutoCAD profile, AutoCAD must have been started once before the installation is started.
- Make sure that the appropriate user rights are available on the workstation. Since drivers for the hardware protection may have to be installed, we recommend local administrator rights.

2.4 Install ATHENA

Installing ATHENA 2014 on a single-station computer

1. Insert the ATHENA 2014 CD-ROM into the CD-ROM drive on your computer. The installation process starts automatically. If you have deactivated the autorun function, you must start the installation routine manually. To do this, click Run in the Start menu. Enter the drive letter for the CD-ROM drive and then setup.exe (e.g. d:\setup.exe).

Dialog box Selected a setup language.



2. Select the language of the installation routine and click the button Next >.

Dialog box Welcome screen



On the welcome screen you have access to information, drivers and to the actual ATHENA installation routine.

Installation information

Opens the help file with information about the installation.

Manual

Opens the help file with the command reference.

Drivers

Opens an Explorer window with the folder containing drivers and diagnostic programs for the hardware protection plug.

Useful information

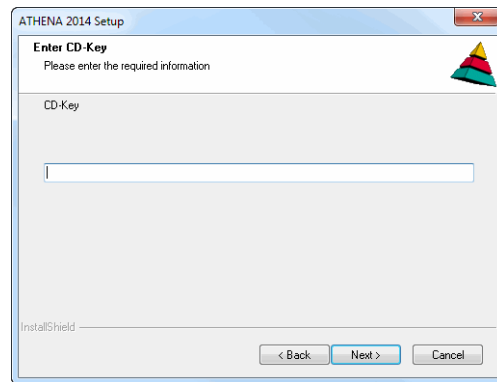
Opens an Explorer window with the Sample folder. Here you will find example drawings, plotting styles, script files and other useful items.

Install ATHENA

Starts the ATHENA installation program

3. Press Install ATHENA to start the program installation.
4. A welcome dialog box for the installation routine now appears. Here, click the button Next >.

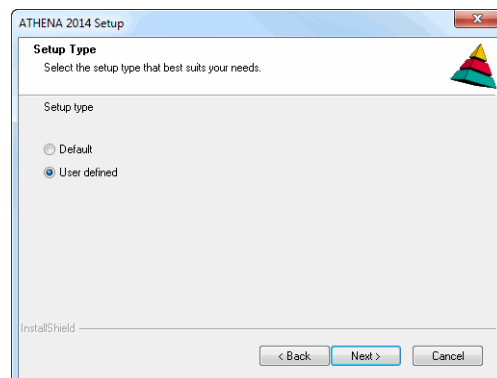
Dialog box Enter the CD key



5. Enter the CD key in the input line and click the button Next >.
You will find the CD key in the software packaging.

6. Carefully read the software license agreement. To continue the installation you must accept the license agreement. Activate the option I accept the conditions of the license agreement and click the button Next >.

Dialog box Setup type



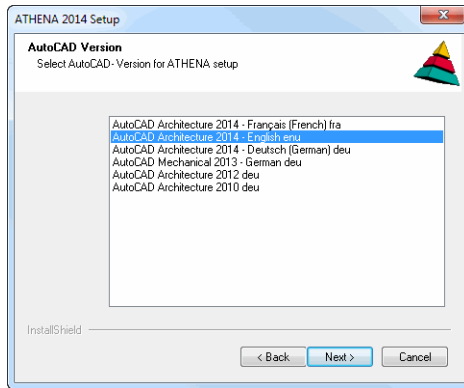
7. Decide on a setup type and click the button Next >.
Choose Standard to install ATHENA 2014 for the latest AutoCAD version.
You can only specify the most important installation paths.
If you have installed a number of AutoCAD versions, you should select the

option User defined. You then have the possibility of selecting the AutoCAD version for which ATHENA 2014 is to be installed. Furthermore, with this variant you can specify all the paths.



This installation variant is particularly recommended for advanced users. In the following steps only the user-defined installation is described.

Dialog box AutoCAD version

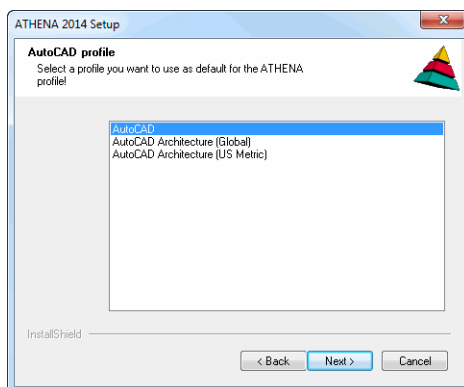


8. Here, the AutoCAD versions installed on the computer are displayed. Select the AutoCAD version from the list for which ATHENA 2014 is to be installed and click the button Next >.



To install ATHENA 2014 for a number of AutoCAD versions, the setup routine must be repeated appropriately. For the following installations use the option Only configure (refer also to Section *Create ATHENA profile* on page 17).

Dialog box AutoCAD profile

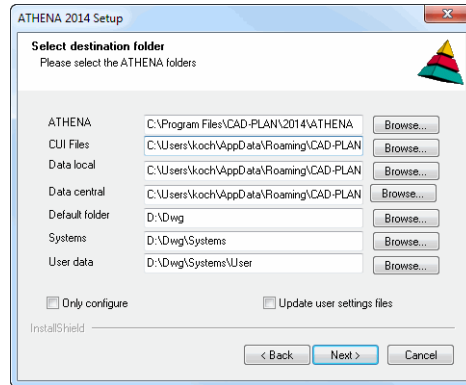


9. Here, the existing AutoCAD profiles are displayed. Select a profile from the list to use it as a template for the ATHENA 2014 profile and click the button Next >.



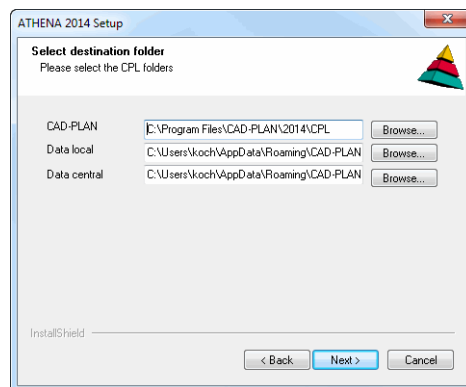
It is recommended that the AutoCAD profile is used.

Dialog box Select the ATHENA folder

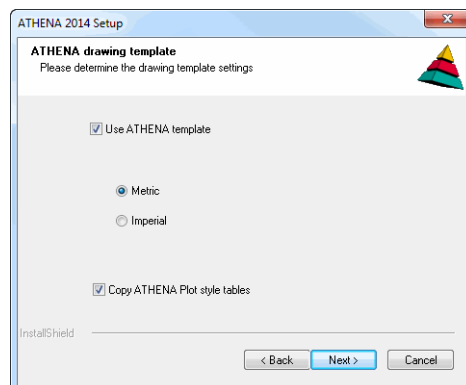


10. Specify here the destination folders for ATHENA 2014. Click the Browse button to specify new destination folders. In the dialog box Select folders there are all the folders linked to your computer. Alternatively, you can also write the destination folders directly into the appropriate input fields. Click the button Next > to proceed with the installation.

Dialog box Select CAD-PLAN folder



11. Specify here the destination folders for the CAD-PLAN program modules. Click the button Next > to proceed with the installation.



12. Here, you select whether you want to use a template and, where applicable, change its units.

Activate the tick box Copy ATHENA plot style tables if you want to use plot style tables which have been matched to the ATHENA layer. They are automatically copied from the setup into the AutoCAD plot styles folder. Click the button Next > to proceed with the installation.

13. In the last dialog box the setup shows the current settings. Check these settings again and click Next > to start the installation.
The installation starts and the progress of the installation is indicated.
14. After the installation the dialog box InstallShield Wizard concluded appears.
Click the button Finish installation to conclude the installation.
15. Plug the hardware protection (dongle) into a USB interface on your computer.
The installation is now concluded.

ATHENA authorization

When you run ATHENA 2014 for the first time, you have to carry out the Authorization. For this, you have to enter your company name and two authorization codes to release the ATHENA 2014.



Enter the company name and codes exactly as stated on the code form. Pay attention to the capitalization and the spaces.

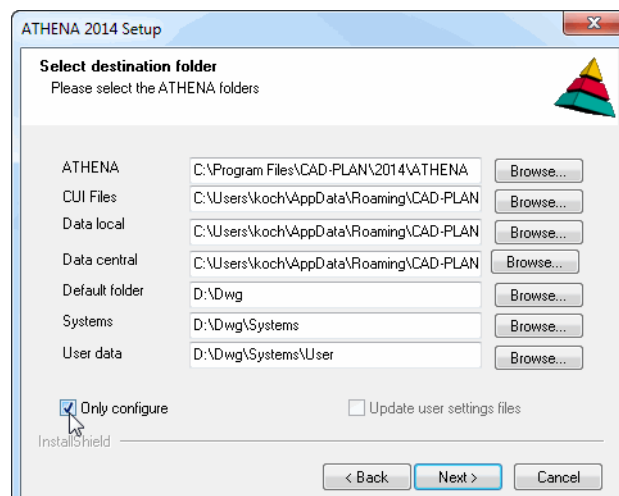
2.5 Create ATHENA profile

Creating ATHENA profiles is necessary:

- For an installation on the server. After Setup has copied the required files into the network folder, you must create an ATHENA profile at each workstation on which the ATHENA 2014 is to be used.
- For an installation for another AutoCAD version. If you have installed many AutoCAD versions on the computer, you must create an ATHENA profile for each further version which is to be used.

To do this, insert the installation CD-ROM into the CD-ROM drive and start the installation program. Proceed as described in the section *Installing ATHENA 2014 on a single-station computer* on page 12. In the dialog box folder specify the installation paths and activate the option Only configure.

Dialog box Choose folder (Only configure)



All other steps are identical to the steps for a standard installation.

2.6 Hardware protection - installing the drivers/license manager

Manual installation of the system drivers for the hardware protection plug is necessary:

- If you want to operate ATHENA as a network license and the hardware protection plug is connected to a server.
- If the driver must be reinstalled for unexpected reasons.
- If the driver has to be updated.

The installation of the license manager is required:

- If you want to operate ATHENA as a network license and the hardware protection plug is connected to a server.

Old Sentinel/SafeNet (Rainbow) SuperPro or SuperProNet hardware protection plugs are not supported and must be replaced. Before installing the drivers, check which hardware protection plug you have. You will find the name of the manufacturer on the hardware protection plug.



On the installation CD in the folder DRIVER you will find a document in which all available hardware protection plugs are illustrated. If you are not certain which hardware protection plug you have, you can find out with the aid of this document.

Installing the HASP driver

Once you have made sure that you have connected a Sentinel (Aladdin) HASP HL Pro or HASP HL Net hardware protection plug, please carry out the following steps:

1. Place the ATHENA installation CD into your CD/DVD drive and click the Driver button on the start screen. An Explorer window with the folder DRIVER opens.
2. Change to the folder HASP HL DRIVER SETUP.
3. Execute the file HASPUserSetup.exe to install the driver.
4. To do this follow the instructions of the installation assistant.
5. Then return to the Explorer window and change to the folder DRIVER.

Installing the HASP license manager

This step is only required when you have connected a hardware protection plug to a server.

1. Change to the folder HASP LICENSE MANAGER.
2. Execute the file lmsetup.exe to install the license manager.
3. Here too, follow the instructions of the installation assistant.

2.7 Checking the hardware protection plug

Localizing the hardware protection plug

If ATHENA does not find the hardware protection plug or you are using several hardware protection plugs in the network, there are two methods of specifically addressing the hardware protection plug.

1. You can specify the name of the server (or computer to which the hardware protection plug is connected).
This method is recommended when you are using a hardware protection plug in the network or when you have connected several hardware protection plugs to different servers.
2. You can specify the key ID of the hardware protection plug.
This step is only recommended when you have connected several hardware protection plugs to a server.

For both cases of application you have to adapt the `ath_hasp_srm.xml` file.

Application Case 1

In the folder `DRIVER\HASP KEY ACCESS\XML_HOSTNAME\` you will find the file `ath_hasp_srm.xml`. This file is used to localize a network dongle on a server.

Copy this file to the folder `USERDATA\CAD-PLAN\2014\ATHENA\DATAGROUP` and change the following lines:

```
<license_manager hostname="SERVERNAME" />
```

Replace `SERVERNAME` by the name of your license server or by the local host if the dongle is connected to the local computer.

Application Case 2

In the folder `DRIVER\HASP KEY ACCESS\XML_KEY-ID\` you will find the file `ath_hasp_srm.xml`. This file is used to localize a network dongle by using its key ID.

Copy this file to the folder `USERDATA\CAD-PLAN\2014\ATHENA\DATAGROUP` and change the following lines:

```
<hasp id="11111111" />
```

Replace `11111111` by the key ID of your dongle.



The key ID is displayed in the browser under the address `http://localhost:1947`. If several HASP hardware protection plugs are connected to the computer, you can recognize the one used by ATHENA by the **Vendor ID 78720**.

3 Migration of user files

If you have installed an earlier version of ATHENA on your computer, you can accept the user-defined settings for ATHENA 2014. To do this some files from folders in the earlier ATHENA version have to be copied into folders of the new version.

Note: You should copy the user-defined files immediately after the installation to prevent newly adapted data from being overwritten.

In the following you will find a list of adaptable user files with a short description.

In the folder C:\USERS\USERNAME\APPDATA\ROAMING\CAD-PLAN\20XX\ATHENA\DATAGROUP\ you will find the following files:

- ath_htr.dex: User library for thermal resistance

In the folder C:\USERS\USERNAME\APPDATA\ROAMING\CAD-PLAN\20XX\ATHENA\DATALOCAL\ you will find the following files:

- ath_nm_prop.dex: Material assignments for standard parts
- ath_order.dex: Item and process numbers. This file is used from version ATHENA 2008. In earlier versions the file ath_obj_order.dex (folder: ...\\CAD-PLAN\20XX\ATHENA\DATALOCAL) was used.
- ath_obj_prop.dex: Object properties
- ath_dim.dex: Dimension styles
- ath_txt.dex: Text styles
- ath_var.dex: System variables
- blocklib.dat: Library for the ATHENA block manager
- plot*.scr: Plot script files for the fast print function
- ath_frame*.dwg: Drawing frame
- ath_caption*.dwg: Text boxes
- ath_level_symbol_0xx.dwg: Levels

In the folder C:\USERS\USERNAME\APPDATA\ROAMING\CAD-PLAN\20XX\CPL\DATALOCAL\ you will find the following files:

- cpl_layer.dex: Layer settings
- cpl_mat.dex: Material definitions

Please note: Some of these files are not present during the installation. They are only created as required. For example, the file ath_nm_prop.dex is only created when you assign an additional material to a standard part.

Note on the path specifications: Here the standard Windows Vista user paths are specified. If you are using a different operating system or are not using the standard paths for the installation, they will be different. The USER NAME specified in the path corresponds to your Windows user name (log-in name).

4 Uninstalling ATHENA 2014

Backup the user settings

Please note that during the uninstall of ATHENA 2014 important user adaptations may be lost which are to be possibly used for other users or later ATHENA versions. For this reason we urgently recommend that the following files are backed up:

- **ath_htr.dex**
In this file ATHENA saves user-defined materials for the computation of the thermal resistance - see also Chapter *User catalog* on page 130.
- **ath_nm_prop.dex**
ATHENA saves material assignments in this file - see also Chapter *Standard Part*, section *Materials* on page 255.
- **ath_obj_order.dex**
ATHENA saves item and process numbers in this file - see also Chapter *Item* on page 125.
- **ath_obj_prop.dex**
ATHENA saves various object properties in this file - see also Chapters *Layer assignment* on page 108, *Hatch pattern assignment* on page 110 and *Label* on page 117.
- **ath_stil.dex**
Text styles are defined in this file - see also Chapter *Text styles* on page 59.
- **ath_user.rsx**
This file contains predefined, multilingual standard texts which can be used for label texts. For editing the program **TextManager** is used. You will find further information in the Chapter *Multilingual standard texts* on page 60.
- **blocklib.dat**
ATHENA saves block management data in this file - see also Chapter *Block manager* on page 579.
- **cpl_layer.dex**
ATHENA saves layer settings in this file - see also Chapter *Layer* on page 85.
- **cpl_mat.dex**
ATHENA saves material settings in this file - see also Chapter *Material* on page 81.
- **plot*.scr**
The settings for the plot routines are defined in these files - see also Chapters *Plotter configuration* on page 61 and *Plotting* on page 620.
- **ath_sysvar.dex**
System settings are defined in this file - see also Chapter *AutoCAD system variables* on page 57.
- **ath_dim.dex**
Dimension settings for the corresponding dimension styles are defined in these files - see also Chapters *Dimension styles* on page 58 and *Manage dimension styles* on page 454.

Uninstalling ATHENA 2014

You can uninstall ATHENA via the dialog box Software in the Windows Control Panel. For this, proceed as follows:

1. Click in the menu Start > Control panel.

2. Double click the icon Programs and Functions (Software) in the Control Panel.
3. Select CAD-PLAN ATHENA 2014 from the list of programs currently installed.
4. Click on uninstall to remove the program.
5. Follow the further instructions of the setup routine.

When you click the Yes button in the information dialog box, ATHENA 2014 is removed from your computer without further warning.

If you click No, the uninstall process is canceled without removing ATHENA 2014.

Uninstalling CAD-PLAN core functions

For the ATHENA core functions there is a dedicated entry with the name CAD-PLAN core in the Windows Control Panel. Uninstalling the core functions is identical to uninstalling ATHENA.

Note: Please ensure that before uninstalling you are not using any further applications requiring these functions.

5 Updates from the Internet

Updates for current ATHENA versions as well as drivers and tools are available in the Internet under the following address: <http://www.cad-plan.com>.

The updates include error rectification, updated standard parts data and small improvements, but no new functions.

You can download the updates as a ZIP file free of charge. For security reasons you need a password for decompressing the ZIP file which you can obtain by telephone or e-mail from your ATHENA Support.

B Introduction and General Remarks

ATHENA is an AutoCAD application which has been specially developed for the sector of metal construction / facade engineering. This application was created in 1990 in the normal planning routine of an engineering consulting office for facade engineering.

ATHENA integrates the design and drafting side of the CAD application significantly more intensively than programs from the manufacturers of profile systems. Planning with all profile systems is supported by the program, but is not dependent on it.

The name of the program was chosen from Greek mythology. ATHENA was the goddess of arts and crafts. With her powers of inspiration we hope she will always be at your side to ensure the success of your project.

1 Help When Working with ATHENA

Manual and Training

This manual serves as a reference book and is intended to help to find answers to questions when working with ATHENA 2014. You can obtain further support directly from CAD-PLAN GmbH or an authorized ATHENA dealer. From these sources it is also possible to learn how to operate ATHENA in training sessions or, say, to find out about new features after an upgrade.

Help

ATHENA provides help which represents as far as possible the electronic version of the ATHENA manual.

You can access context-sensitive help topics by pressing the button **?** or by pressing **Help** in the dialog boxes or by entering **?** at input requests.

2 Typographical Conventions

The following typographical conventions are used in the ATHENA 2014 User Manual to highlight text elements:

Dialog box title	Dialog box Options
Dialog box components (e.g. dialog field sections)	Dialog box section Construction
File names	athena.cui, *.dex, *.dim
Folder names	ATHENA\DATALOCAL
Text in ASCII files	**sheet metal
References to other chapters	See also the chapter " <i>Complete Caption</i> ".
Input requests	Command prompt <i>Select objects:</i>
Explanations of input requests	<i>Select the objects to be changed.</i>

3 New in this version

In this chapter you will find a list of the new features, improvements and changes in ATHENA 2014.

3.1 ATHENA 2014 (AutoCAD 2009-2014) January 2014

3.1.1 General Adaptations

3.1.1.1 Ribbon

The ribbon has been restructured and new commands have been added.

3.1.1.2 Hardware protection

Older Sentinel/Rainbow hardware protection plugs are no longer supported and will be replaced.

Network protection plugs must be updated and can be more conveniently managed.

3.1.1.3 Libraries, Library Objects

The folder structure is now also illustrated in the drawing. In this connection *.olbx has been introduced for libraries of the new file type. Drawings with the old structure are opened in the compatibility mode.

3.1.1.4 Help

The ATHENA Help is now available on-line (it can be switched off in the ATHENA options).

3.1.1.5 Temporary display for ATHENA objects

ATHENA line objects (sheet metal, membrane, welded seam, etc.) are now generated directly for drawings. Consequently, there is no more temporary display which disappears on zooming and panning.

3.1.1.6 Add selected

This AutoCAD command can also be used for ATHENA objects.

3.1.1.7 Performance

The performance on changing from layouts has been improved (performance switch in the ATHENA options).

3.1.1.8 Design environment

If you have loaded external layers and materials via the design environment, layer and material changes can be saved alternatively in the loaded fields or your own files.

3.1.1.9 Optimized leaders

ATHENA leaders have been optimized. With scalable leaders with several scales the position of the leader can be changed separately for each scale.

3.1.1.10 New dialog boxes

Dialog boxes for commands of the LogiKal interface and Welded seam have been reviewed.

3.1.2 New commands

3.1.2.1 ATH_LEADERTYPE

Controls the use of optimized labels. These system variables influence both the generation of new leaders and also the conversion of existing leaders from earlier ATHENA versions.

3.1.2.2 Sheet metal dimension

Generates automatic dimensions

3.1.2.3 Label background color ON and Label text background color OFF

Activates or deactivates the **background** infill color for text for all labels in the current drawing.

3.1.2.4 Add leader line and Remove leader line

Two new commands to supplement or remove leader lines on leader labels.

3.1.2.5 Add leader line segment and Remove leader line segment

Two new commands to supplement or remove leader line segments on leader labels.

3.1.2.6 Modify object label

Changes the labeling texts of several objects of the same groups (e.g. label properties of all standard profiles, standard screws, membranes...).

3.1.2.7 Align leaders

Aligns leaders in the current view or in the current coordinate system.

3.1.2.8 Wall manager and Apply wall

Two new commands for managing and inserting wall cross sections.

3.1.2.9 Model Inspection

Used for checking constructions, e.g. before generating production documentation.

3.1.2.10 Visualize axis model

Draws regions in the sub-areas or an analyzed axis model (3D position) in order to visualize it.

3.1.2.11 Copy analyzed axis model

Copies an analyzed axis model (3D position).

3.1.2.12 Recalculate axis model

Leads to a recalculation of an analyzed axis model (3D position) after geometrical changes.

3.1.2.13 Detach position

Removes the job assignments and position numbers assigned with the command from bars, infills and frame elements.

3.1.2.14 Export NC-X

Exports bars of a project in the NC-X format. Here a *.ncw file is written.

3.1.2.15 Export IFC

Exports a 3D model in the IFC file format.

3.1.2.16 List of facade elevations

This command creates a parts list for the facade elevations, and writes it to the Windows clipboard.

3.1.3 Changes and expansions of commands

3.1.3.1 ATHENA options

New dialog box with sub-dialog boxes.

3.1.3.2 Spacer

Chamfers on the four sides of the block can now be switched separately.

3.1.3.3 Standard Part

With manufactured parts there is now a direct link to the manufacturer's web site.

If you insert standard profiles as cross sections, you can change the insertion point by pressing the CTRL key.

3.1.3.4 Semi-finished product

If you insert a semi-finished product as cross section, you can change the insertion point by pressing the CTRL key.

3.1.3.5 Generating a section from 2D

Sections through the facade elevation+ are now associative.

3.1.3.6 Section symbol

Better standard-compatible display of the section symbol.

3.1.3.7 Levels and dimensions

The dialog fields for editing levels and dimensions have been expanded with a new menu field for standard texts.

3.1.3.8 Load layer

In addition to the drawing layers, material layers are now also loaded.

3.1.3.9 Display modes

Display simply now shows up to 16 edges. For contours with more than 16 edges the enclosing rectangle is shown as before.

3.1.3.10 Bar list, Infill list, List of frame elements

Optionally a list of the positions can be generated according to the tag.

3.1.3.11 Bar diagram

Optionally the same parts can be brought together (bar diagram according to the tag). Processes can be dimensioned.

3.1.3.12 Infill diagram

Optionally the same parts can be brought together (infill diagram according to the tag).

C Program operation

In this section you will find information about the operation of ATHENA 2014 with a mouse or a tablet magnifier. The key functions of the pointing devices are assigned in the ATHENA adaptation file. Here, you can retrospectively modify the assignments.

1 **Mouse Operation**

You can call ATHENA commands with the mouse by clicking the icon assigned to the command.

All commands can be found in well laid-out groups on the ATHENA ribbon tab. Toolbars and menus are also supported.

If you move the mouse over an icon, the command name is displayed on a small flag. In the status line of the AutoCAD program window you can see a comprehensive description of the relevant command.

The mouse keys are assigned current commands (e.g. Zoom) to speed up working with ATHENA. You will find the precise key assignment in Chapter *Mouse key functions*.

1.1 Mouse key functions

Apart from the normal functions (Pick and Enter), you can call other important functions with the mouse keys.

Left mouse key: Selects commands on the monitor (menu or toolbars) and objects on the graphics screen.

Right mouse key: *Return* or *Enter*.

If provided:

Center mouse key (OSnap): Object snap function. The object snap functions, Intersection, End point, Center, Point and Basis are used. The priority is defined by their sequence.

Mouse wheel: You can zoom in the drawing by turning the mouse wheel.
If you double click with the wheel, zooming takes place to the drawing extents.
If you move the mouse with the mouse wheel pressed, you can move in the drawing display (Pan function).

Shift + right mouse key: If the key combination Shift + right mouse key is pressed, the Zoom context menu opens next to the cross hairs and the transparent Zoom commands can be selected.

Ctrl + right mouse key: If the key combination Ctrl + right mouse key is pressed, the Object snap context menu opens next to the cross hairs and one of the object snap methods can be selected.

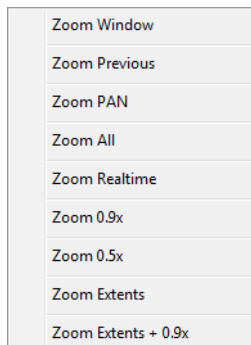


Fig. 1.1: Zoom context menu

You will find further information about zoom functions in the AutoCAD documentation.

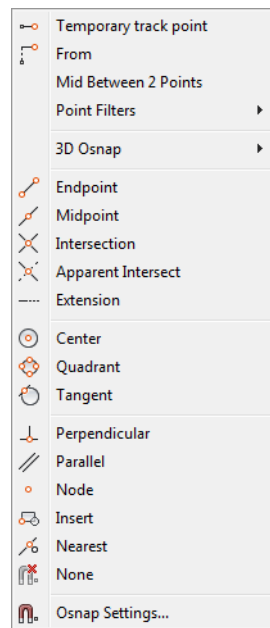


Fig. 1.2: Object snap context menu

You will find further information about object snap functions in the AutoCAD documentation.

You can adapt the mouse key assignment to your own requirements. You will find further information on this subject in the AutoCAD documentation.

1.2 Ribbon

The AutoCAD ribbon is expanded by additional tabs containing ATHENA commands.

ATHENA tab

This tab contains the majority of the ATHENA commands and is divided into the following groups:

- Draw
- Standard parts
- Modify
- Label
- Drawing aids
- Hatch
- Layer
- Extras
- Modeling

ATH tab

This tab contains all ATHENA commands. This tab is deactivated as standard and must be switched on as required. The group division is similar to that of the ATHENA pulldown menu:

- ATH Draw
- ATH Sheet Metal
- ATH Engineering
- ATH Dimension
- ATH Modify
- ATH Drawing Aids
- ATH Text
- ATH Block
- ATH Viewport
- ATH Layer
- ATH Service

1.3 Menus

In the menu bar there are in addition to the AutoCAD pulldown menus, four ATHENA pulldown menus:

Menu ATHENA

Here you find the command [ATHENA options](#) for defining the default settings. Furthermore, you will find all the ATHENA commands in submenus. In addition here you will find the submenu with the ATHENA Help, the remote maintenance program, various links to the CAD-PLAN web site and the [Tablet configuration](#).

Menu Modeling

This pulldown menu contains the commands which are mainly used for designing in the third dimension.

1.4 Toolbars

You can call all ATHENA commands directly via the basic toolbars. The basic toolbars contain all ATHENA commands. The grouping is similar to the pull-down menu ATHENA. An ATH is placed in front of the toolbar name.

Further more, to enable practicable working, so-called working toolbars with frequently used commands are available. This toolbar is prefixed by the name ATHENA.

Main toolbars

- ATHENA Flyout
(The ATHENA flyout toolbar includes the other basic toolbars as flyouts with varying icons. The icon of the last selected command is always used as the flyout icon.)
- ATHENA Flyout Fix
(The ATHENA Flyout Fix toolbar includes the basic toolbars listed below as flyouts with fixed icons. The first command is used as the flyout icon.)
- ATH Draw
- (ATH Hatching)
- (ATH Standard Parts)
- ATH Sheet Metal
- ATH Engineering
- ATH Dimension
- ATH Modify
- ATH Drawing Aids
- ATH Text
- ATH Block
- (ATH Profile Manufacturers)
- ATH Viewport
- ATH Layer
- ATH Service
- ATH Numerical Block
- ATH Protractor Card
- ATH Manager
- ATH Apply
- ATH Analyze

Working toolbars

- ATHENA Modify
- ATHENA Dimension
- ATHENA Block
- ATHENA Functions
- ATHENA Auxiliary Lines
- ATHENA Layer
- (ATHENA Plot)
- ATHENA Standard
- ATHENA Text
- ATHENA Draw

1.5 Elements of the user interface

Ribbons, toolbars and menus are saved in the adaptation file `athena.cui(x)` and should not be changed. The adaptation file `athena.cui(x)` is adapted and expanded for each new ATHENA version. The user's own adaptations are therefore overwritten and must be carried out again.

If you wish to adapt your working environment, you should do this in a dedicated company/user adaptation file. You will find further information on this in the Chapter *Adapt user interface* on page 63.

2 Tablet Operation

ATHENA contains tablet organization with the tablet overlay which is required. The user-friendly tablet arrangement is subdivided into various functional areas, which enable fast and objective working with the commands of the CAD program.

All the required commands are accommodated graphically on the tablet and are consequently quickly available.

The keys on the 4-key magnifier are assigned ergonomically with commands. You will find a detailed description of the key assignment in the chapter *Key functions on the tablet magnifier*.

The tablet also includes a user-friendly layer organization; all layers are classified according to line widths and line types.

2.1 Tablet configuration



Make sure that a suitable tablet driver is available for your operating system and install it beforehand. If necessary, ask your tablet manufacturer for compatible driver software.

To configure the tablet carry out the following steps.

In the AutoCAD options set the **Wintab Compatible Digitizer** as the current pointing device. You will find further information about system pointing devices in the AutoCAD documentation.

Then select the command Tablet config new in the menu ATHENA > Help > Tablet configuration.

Now you must define the menu sections. To do this, click the corner points of the individual menu sections with the Pick key on the tablet magnifier. The numbers of columns and rows are automatically accepted.

Command prompt

*Enter _tablet option [On/Off/CAL/CFG]: _cfg
(The option is automatically selected.)*

*Number of required tablet menus (0-4) <0>: 4
(The number of tablet menu sections is automatically specified.)*

*Digitize left upper corner of menu section 1:
(Click the left upper corner of menu section 1 (P1 in the figure Menu sections on the tablet overlay) with the Pick key of the tablet magnifier.)*

*Digitize left lower corner of menu section 1:
(Click the left lower corner of menu section 1 (P2 in the figure Menu sections on the tablet overlay) with the Pick key of the tablet magnifier.)*

*Digitize right lower corner of menu section 1:
(Click the right lower corner of menu section 1 (P3 in the figure Menu sections on the tablet overlay) with the Pick key of the tablet magnifier.)*

*Enter the number of columns for menu section 1: (1-8574) <44>: 44
(The number of columns is automatically entered.)*

*Enter the number of rows for menu section 1: (1-974) <5>: 5
(The number of rows is automatically entered.)*

*Digitize left upper corner of menu section 2:
(Click the left upper corner of menu section 2.)*

*left lower corner of the menu section . . .
(With the following queries proceed as described above.)*

*Respecify defined pointing area on the monitor? [Yes/No] <N>: _Y
(Yes it is automatically selected.)*

*Digitize left lower corner of defined pointing area:
(Click the left lower corner of the pointing area (P13 in the figure Menu sections on the tablet overlay) with the Pick key of the tablet magnifier.)*

*Digitize right upper corner of defined pointing area:
(Click the right upper corner of the pointing area (P14 in the figure Menu sections on the tablet overlay) with the Pick key of the tablet magnifier.)*

*Specify free pointing area on monitor? [Yes/No] <N>: _Y
(Yes it is automatically selected.)*

*Is the free pointing area on the monitor equal in size to the defined pointing area? [Yes/No] <Y>:
(Confirm this query with **ENTER**.)*

Switch the free pointing area on and off with F12. Do you also wish to define a key for switching the free pointing area? [Yes/No] <N>:
(Confirm this query with **ENTER**.)

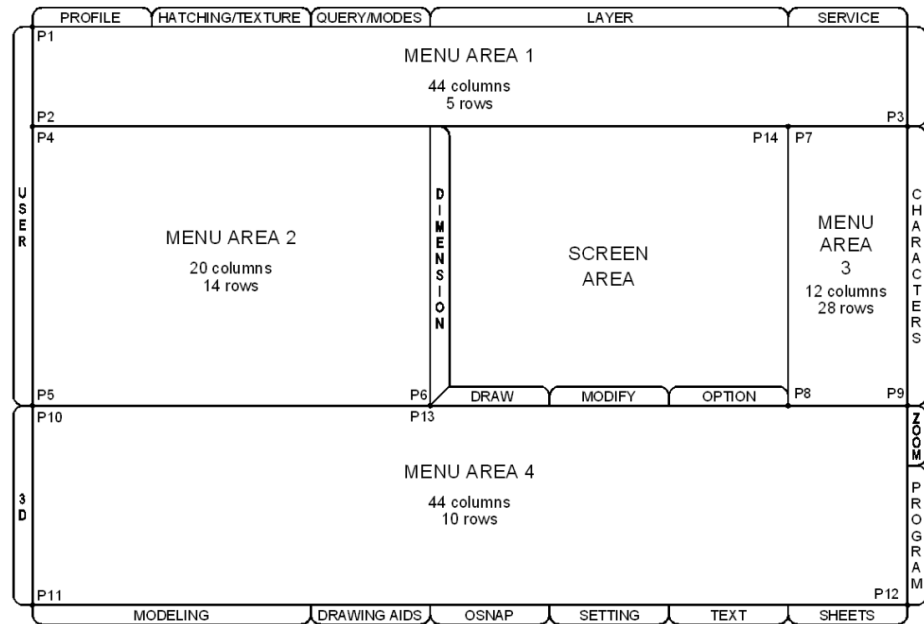


Fig. 2.1: Menu sections on the tablet overlay

If you wish to use your own tablet overlay with modified columns and rows, you can modify the tablet configuration - number of columns and rows - with the command Tablet config change in the menu ATHENA.

2.2 Key functions on the tablet magnifier

When you operate ATHENA with the tablet and 4-key magnifier, the following key assignment is set as standard:

Key 1 (Pick):	Selects commands on the tablet and objects on the graphics screen.
Key 2 (Enter):	Enter key.
Key 3 (OSnap):	Object snap function. The object snap functions, Intersection, End point, Center, Point and Basis are used. The priority is defined by their sequence.
Key 4 (Zoom):	If this key is pressed, the Zoom context menu opens next to the cross-hairs and the various transparent Zoom commands can be selected:

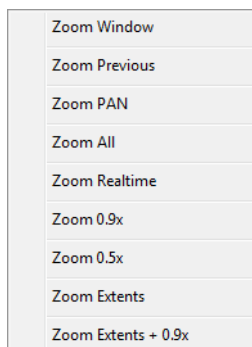


Fig. 2.2: Zoom context menu

You will find further information about zoom functions in the AutoCAD documentation.

The sequence of key assignment is also affected by the tablet driver (Wintab). This must have previously been correctly installed and configured. You will find further information in the documentation for your digitizing tablet.

It is also possible to operate ATHENA with a 16-key magnifier. In this respect you can supplement the key assignment for further keys in the menu file. You will find further information about menu adaptation in the AutoCAD documentation.

D Configuration

This section describes how you can adapt the ATHENA environment to suit your requirements. Change the language setting of ATHENA 2014. Create your own layers and materials or adapt them to those available. Match the plot function to your output devices.

You can adapt the default settings for the following subsections:

- System Configuration
 - ATHENA system variables
 - AutoCAD system variables
 - Dimension styles
 - Text styles
 - Adapt user interface
 - Plotter configuration
 - Slide display
- Options
 - ATHENA options
 - Paths
 - Layer
 - Material

1 System Configuration

You can set up the basic system settings of the ATHENA environment. You can adapt the drawing environment to your own requirements and define your own text styles and dimension styles and use special ATHENA plot commands.

Chapters in this section:

- ATHENA system variables
 - ATH_LEADERTYPE
- AutoCAD system variables
- Dimension styles
- Text styles
- Multilingual standard texts
- Plotter configuration
- Adapt user interface
- Slide display

1.1 ATHENA system variables

Analogous to AutoCAD system variables, ATHENA system variables control the function of certain ATHENA commands.

The input also occurs as with the AutoCAD system variables: First, enter the name of the variable, confirm it by pressing the Enter key and then change the value of the variable. The values available for the relevant variable are documented further below in this section.

You will find further information about system variables in the AutoCAD documentation.

1.1.1 ATH_LEADERTYPE

Controls the use of optimized labels. These system variables influence both the generation of new leaders and also the conversion of existing leaders from earlier ATHENA versions.

Labels in the new format can be positioned independently of the set label scale. For example, the label can be arranged closer to the component when the scale is set at 1:1 and with the scale 1:2 further away from the component.

On deactivation the position of the new labels no longer changes.



Labels in the new format cannot be edited in earlier versions of ATHENA!

The following values can be entered for *ATH_LEADERTYPE*:

- | | |
|---|---|
| 0 | Leaders from earlier ATHENA versions are not converted on opening the drawing.
New leaders are produced in the old format . |
| 1 | Default setting!
Leaders from earlier ATHENA versions are not converted on opening the drawing.
New leaders are produced in the new format . |
| 2 | Leaders from earlier ATHENA versions are converted on opening the drawing.
New leaders are produced in the new format . |

ATH_LEADERTYPE controls leaders which have been generated with the following commands:

- Leader
- Parts labeling
- Label tags
- Coordinate label
- Position symbols
- Edge symbol
- Surface symbol

1.2 AutoCAD system variables

When you activate the option Default settings in the Dialog box ATHENA options, ATHENA loads the AutoCAD system settings in new drawings from the file ATHENA\DATA\Local\ATH_VAR.DEX.

In the ath_var.dex the most important system settings are defined with the optimum values for ATHENA.

The ASCII file system.var from earlier ATHENA versions is no longer used. For reasons of compatibility though it is converted during the program start into an ath_var.dex if the file ath_sysvar.dex has not been found.

Notes

- If ATHENA is installed in a network, you can move the file ath_var.dex into the DATAGROUP folder to ensure that all ATHENA users in the network use the same system settings.
- There are a number of ways of adapting the AutoCAD system settings. You will find further information on this in the AutoCAD documentation (key word System variables).
- Use the command *Manage system variables* to save or to load the system settings of the drawing.

1.3 Dimension styles

When you activate the option Default settings in the Dialog box ATHENA options, ATHENA loads the dimension styles in new drawings from the file ATHENA\DATALOCAL\ath_dim.dex. This is the library file containing the dimension styles.



As standard, metric (mm) and British (inch) dimension styles are provided. Which is displayed depends on the current setting for the drawing units.

The ASCII files with the file extension *.dim from earlier ATHENA versions are no longer used. For reasons of compatibility though they are converted during the program start into an ath_dim.dex if the file ath_dim.dex has not been found.

Notes

- If ATHENA is installed in a network, you can move the library file ath_dim.dex into the DATAGROUP folder to ensure that all ATHENA users in the network use the same dimension styles.
- Use the AutoCAD command **_dimstyle** to create and change dimension styles. You will find further information in the AutoCAD documentation.
- Use the command **Manage dimension styles** to manage dimension styles of the drawing or of the library.

1.4 Text styles

When you activate the option Default settings in the Dialog box ATHENA options, ATHENA loads the text styles into new drawings from the file ATHENA\DATALOCAL\ath_txt.dex. This is the library file containing the text styles.

The ASCII file ath_stil.def from earlier ATHENA versions is no longer used. For reasons of compatibility though it is converted during the program start into an ath_txt.dex if the file ath_txt.dex has not been found.

Notes

- If ATHENA is installed in a network, you can move the library file ath_txt.dex into the DATAGROUP folder to ensure that all ATHENA users in the network use the same text styles.
- Use the AutoCAD command **_style** to create and change text styles. You will find further information in the AutoCAD documentation.
- Use the command **Manage text styles** to manage text styles of the drawing or of the library.

1.5 Multilingual standard texts

Frequently required, multilingual standard texts, which are primarily used for label texts, can be gathered in the text database ATHENA\DATALOCAL\ath_user.rsx.

You can edit this text database to supplement or modify texts. To do this, double click the file ath_user.rsx.

For editing the program **TextManager** is started. You can find further information about the TextManager functions in the program help.

1.6 Plotter configuration

ATHENA supports different output media (e.g. plotters, printers ...) through ATHENA's own plot programs which you can address via the corresponding tablet fields or icons.

- In order to be able to use the plotting capabilities of ATHENA, you must carry out some settings.
- Configure the required plotter in AutoCAD. You will find further information on this subject in your AutoCAD documentation.

Tip: For reasons of performance, printers and plotters should be installed only under Windows (not under AutoCAD).

The directory \ATHENA\DATALOCAL contains the following script files:

- plot-01l.scr: for Output device 01, Layout space
- plot-01m.scr: for Output device 01, Model space
- plot-02l.scr: for Output device 02, Layout space
- plot-02m.scr: for Output device 02, Model space
- Etc.

There is a script file each for plotting in the model section and for plotting in the layout section. You will find further information about model and layout sections in your AutoCAD documentation.

In the script files the relevant sequence of commands for plotting must be defined.

You can use the following example for a laser printer (HP LaserJet 4V on Server CAD1).

;Plot...	You can specify a printer name after the semicolon.
__plot	Plotting without dialog box (Do not change line)
__Y	Detailed plot configuration (Do not change line)
" "	Enter layout name (accept default, do not change line)
\\SERVER\LASERJET4V	Enter name of output device
A3	Specify paper format
M	Specify paper units (inch / millimeter, do not change line)
!ATH_PL_ROT	Drawing orientation (this is computed, do not change line)
__N	Plot upside down?
__W	Specify plotting range (window, do not change line)
!ATH_PL_P1	Lower left corner of window (do not change line)
!ATH_PL_P2	Upper right corner of window (do not change line)
!ATH_PL_SCL	Specify plotting scale (A=adapt, 1=M1:1, 2=M1:2 etc., do not change line)

0.0,0.0	Specify plot offset
_Y	Plot with plotting styles?
monochrome.ctb	Specify plotting style table name
_Y	Plot with lineweights?
_N	Scale lineweights with plotting scale? (Line missing with PLOT0xM.SCR)
_N	Plot last paper space? (Line missing with PLOT0xM.SCR)
!ATH_PL_HID	Enter setting for shaded plot...? (Do not change line)
_N	Output plot to a file?
_Y	Save changes in register model?
_Y	Continue plot?
(graphscr)	Switch to graphics mode

The corresponding script files plot-0xm.scr and plot-0xl.scr are almost identical. The difference consists of two lines: Scale lineweights with plotting scale? and Plot paper space last? which are not present when plotting in the model space (files plot-0xm.scr).

Notes

- Use the command **Adjust plot script files** to define the most important settings in these script files in a dialog box. You will find further information in the Chapter *Adjust plot script files* on page 621.
- Six different plot script files are present as default in ATHENA. If these are not sufficient, you can create (copy) further plot script files according to the scheme plot??l.scr or plot??m.scr.

1.7 Adapt user interface

Adaptations to the user interface can be made in an adaptation file, *.cui(x). Adaptation files are based on the XML format and can be adapted with the AutoCAD command [_CUI](#) in a clearly laid-out dialog box.

Adaptation files mainly contain the following components:

- Toolbars
- Menus
- Tabs and groups in the ribbon
- Context menus
- Keyboard short-cut commands
- Double-click actions
- Mouse keys

Nowadays lower-level components are likely to be listed in the legacy section:

- Tablet menus
- Tablet buttons
- Screen menus.
- Image tile menus

In addition to the main adaptation file for the relevant AutoCAD version, for example acad.cui(x), during the program start ATHENA loads the ATHENA adaptation file (athena.cui(x)) and a user adaptation file (ath_user.cui(x)) as partial adaptation files.

You will find detailed information about user adaptations and adaptation files in your AutoCAD documentation.



We recommend that for your own adaptations you use a user or company adaptation file (e.g. ath_user.cui). Since the acad.cui(x) or athena.cui(x) can be overwritten by updates or service packs, you may possibly lose valuable settings.

1.8 Slide display

Enlarging the slide displays (pictures)

If the slide displays (pictures) are too small, you can enlarge them in the file `base.dcl` which can be found in the AutoCAD support folder.

Extract from the file, `base.dcl`:

```
icon_image : image_button {  
    color                = 0;  
    width                = 12;  
    aspect_ratio         = 0.66;  
    allow_accept         = true;  
    fixed_height         = true;  
    fixed_width          = true;  
}
```

For enlargement, you must change the value `width` (default setting: `width=12`).
With a screen resolution of 1280x1024 pixels, we recommend a value of 18.

2 Options

You can set default settings for the dialog boxes present in ATHENA to your requirements. These default settings affect the objects which you create with the various ATHENA programs.

Chapters in this section:

- ATHENA options
- Start drawing
- Design environment
- Display
- Paths
- Authorization
- Product Information
- Label
- Dimensioning
- Material
- Layer
- Cutting
- Presettings

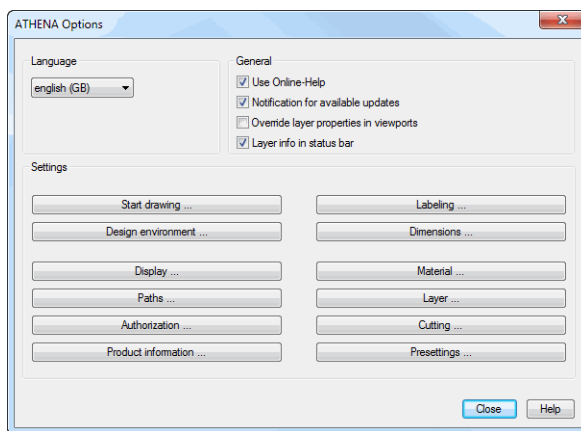
2.1 ATHENA options



Ribbon:	Tab ATHENA > Group Extras > ATHENA options
Menu:	ATHENA > ATHENA options
Toolbar:	Not present
Command input:	ath_optionen

Here you can adapt basic settings, which affect objects and the working environment, to your own requirements. Apart from the general settings, you have access to other sub-dialog boxes, where you can adapt the settings for certain sections (e.g. Label, Layer, etc.).

Dialog box ATHENA options



Dialog box section Language

Here the language of the dialog boxes and input requests controlled by ATHENA changes.



This setting has no influence on the language of the user interface (ribbon, toolbar and menus)!

Dialog box section General

Use online help

Controls whether the local help or the online help is shown. The online help is more up to date, but you need an Internet connection. If no Internet connection is available, the local help is always called.

Notification of available updates

This shows a notice if an update (service pack) is available for your ATHENA version. The notice is displayed in a speech balloon in the program window at the lower left and contains a link for downloading the service pack.

Overwrite layer properties in viewports

Supports the replacement of layer properties in layout viewports (Color, Line type, Lineweight) for ATHENA objects.



This tick box affects the performance during layout changes. Under some circumstances the change of layout is slower if you activate the tick box!

Layer information in status bar

ATHENA displays the name of the current layer and the number of hidden objects (commands Objects invisible and Objects visible) in the status bar.

Dialog box section Settings

The buttons in this section lead to sub-dialog boxes where you can view and manage the various object settings and data.

Start drawing

Opens the Dialog box Start drawing. You will find further information on this in the Chapter *Start drawing* on page 68.

Design environment

Opens the Dialog box Design environment. You will find further information on this in the Chapter *Design environment* on page 70.

Display

Opens the Dialog box Display options. You will find further information on this in the Chapter *Display* on page 73.

Paths

Opens the Dialog box Directories. You will find further information on this in the Chapter *Paths* on page 74.

Authorization

Opens the Dialog box Authorization. You will find further information on this in the Chapter *Authorization* on page 76.

Product Information

Opens the Dialog box Product Information. You will find further information on this in the Chapter *Product Information* on page 77.

Label

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 78.

Dimensioning

Opens the Dialog box Dimensions. You will find further information on this in the Chapter *Dimensioning* on page 80.

Material

Opens the Dialog box Material properties. You will find further information on this in the Chapter *Material* on page 81.

Layer

Opens the Dialog box System layer. You will find further information on this in the Chapter *Layer* on page 85.

Cutting

Opens the Dialog box Manage cutting classes. You will find further information on this in the Chapter *Cutting* on page 89.

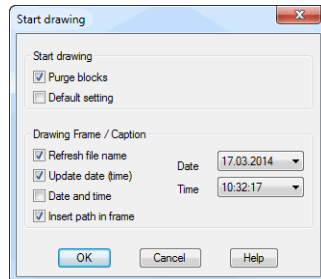
Presettings

Opens the Dialog box Presettings. You will find further information on this in the Chapter *Presettings* on page 91.

2.2 Start drawing

Controls options for the drawing start and the updating of the caption.

Dialog box Start drawing



Dialog box section Start drawing

Purge blocks

If you activate the tick box, ATHENA clears all unreferenced blocks from the drawing on opening it.

Default setting

If you activate the tick box, ATHENA loads the following default settings when opening drawings:

- The drawing limits are set to the format DIN A0 (1189 x 841mm).
- The ATHENA dimension settings are loaded from the file `ath_dim.dex`. Depending on the drawing units the dimension style ATHENA metric or Imperial is loaded.
You will find further information on this in the Chapter *Dimension styles* on page 58.
- Layers from the file `cp1_layer.dex` are loaded. You will find further information in the Chapter *Layer* on page 85.
- The system variables from the file `ath_sysvar.dex` are loaded. You will find further information in the Chapter *AutoCAD system variables* on page 57.



You should activate Default setting when you are not using your own template files (*.dwt). We recommend template files for more flexible working.

Dialog box section Drawing frame/caption

Refresh file name

When you activate the tick box, ATHENA updates the file name in the text box on opening drawings. For this, the text box must satisfy certain conditions. You will find further information on this in the Chapter *Complete caption* on page 555.

Update date (time)

If you activate the tick box, ATHENA updates the date and time in the text box when drawings are terminated. For this, the text box must satisfy certain conditions. You will find further information on this in the Chapter *Complete caption* on page 555.

Date, time

In the pick lists you can select in which format the date and time are displayed in the caption.

Date and time

If you activate the tick box, the current time of day is displayed in the text box next to the current date. For this, the text box must satisfy certain conditions. You will find further information on this in the Chapter *Complete caption* on page 555.

Insert path in frame

When you activate the tick box, ATHENA displays the path and the file name of the current drawing at the bottom right in the drawing frame. These details are updated when you Complete caption or when you save the drawing.

2.3 Design environment

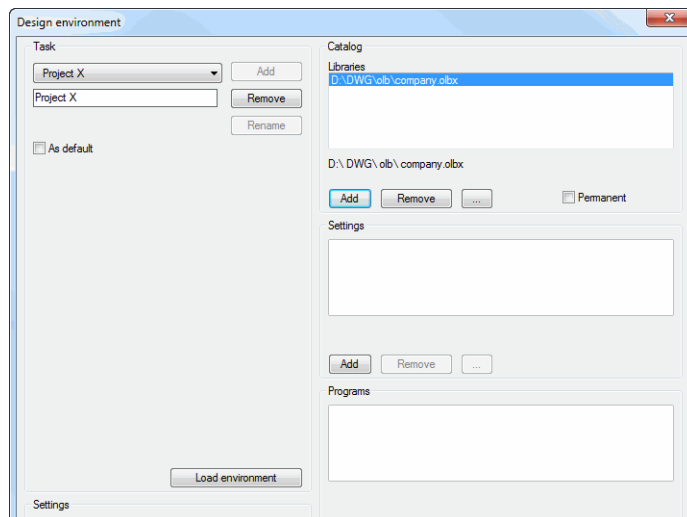


Ribbon:	Tab ATHENA > Group Extras > Design environment
Menu:	ATHENA > Service
Toolbar:	Not present
Command input:	ath_task

Manages design environments.

When working on a project, access to certain libraries and programs is required. These accesses may differ depending on the project. ATHENA offers the possibility of saving and recalling the configuration of libraries and programs specific to the task in hand. This type of task-specific configuration is known as the design environment.

Dialog box Design environment



Dialog box section Task

In this dialog box section you can save, rename and remove tasks. To do this, enter a name in the input field and confirm it with the Enter key.

Add

Adds the new task to the list.

If a new entry is created while settings in the catalog, settings and program sections exist, a confirmation query appears for acceptance of the data.

Remove

Deletes the selected task from the list.

Rename

Saves the selected task under a new name.

As default

Automatically loads a saved environment on starting a drawing. Only one entry can be the default in the list. It is identified with * (asterisk).

Create environment

Loads the environment (the libraries, settings and programs assigned to the task) in the current drawing.

Dialog box section Catalog

In this dialog box section, libraries, which are to be made available when the environment is established, are assigned to the catalog. Usable ATHENA libraries have the extension *.olb. You will find more on the subject of libraries in the Chapters *Assembly library* on page 157 and *Assembly catalog* on page 159.

Add

Adds a new library to the task. For this the standard dialog box is opened for the file selection.

Remove

Deletes the selected library.

[...]

Changes the selected library. For this the standard dialog box is opened for the file selection.

Permanent

Permanently loads individual libraries. They are retained even after the environment is changed. This setting of this option must be carried out separately for each library.

Dialog box section Settings

In this dialog box section, settings, which are to be made available when the environment is established, are assigned to the catalog. Usable setting files are:

- ath_dim.dex (dimension settings)
- ath_nm_prop.dex (material assignments for standard parts)
- ath_obj_order.dex (item and process numbers)
- ath_obj_prop.dex (object properties)
- ath_txt.dex (text styles)
- ath_var.dex (system settings)
- cpl_mat.dex (material definitions)
- cpl_layer.dex (system layer)

Information on these *.dex files can be found in the section *Local settings and group settings* on page 8.

Add

Adds a new settings file to the task. For this the standard dialog box is opened for the file selection.

Remove

Deletes the selected settings file.

[...]

Changes the selected settings file. For this the standard dialog box is opened for the file selection.

Dialog box section Programs

In this dialog box section you can specify the necessary programs which are to be made available when the environment is established. Programs can be Lisp routines, scripts and independent Windows programs.

Add

Adds a new program to the task. For this the standard dialog box is opened for the file selection.

Remove

Deletes the selected program.

[...]

Changes the selected program. For this the standard dialog box is opened for the file selection.

Dialog box section Settings

Export

Exports all saved tasks with their settings to a *.dex file.

Import

Imports all saved tasks with their settings from a *.dex file.



With the import/export functions you can transfer existing tasks from one PC to another PC. Note that during the import any existing tasks will be removed.

Notes

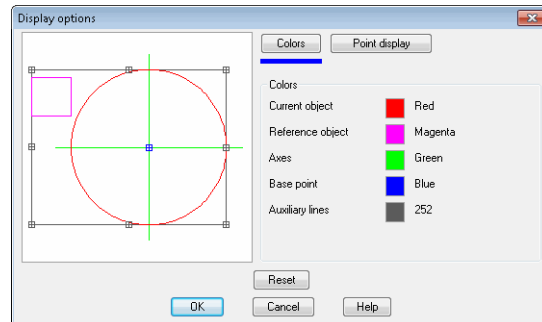
Once an environment has been established, the objects in the included libraries can be accessed via the catalog.

2.4 Display

Controls the display settings of the preview in dialog boxes.

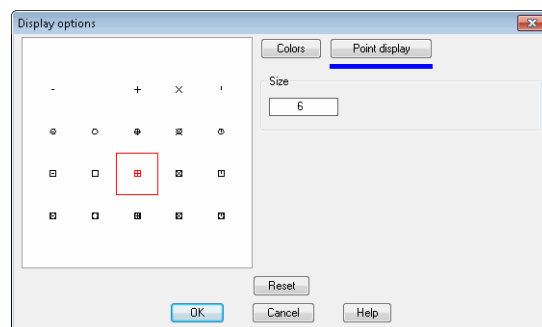
Dialog box Display options

RegisterschaltflächeColors



To change the color for an object type, click in the color field of the object to be changed. Then the AutoCAD dialog box Color is opened (refer to the AutoCAD documentation). The change is displayed in the graphics window.

RegisterschaltflächePoint display



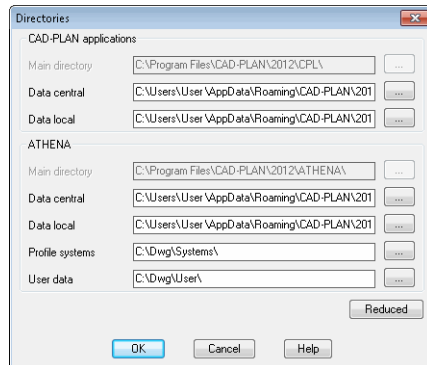
In this section you select the symbol which is to be used for the display of points. Enter a numerical value in the input field to change its size. The entered value corresponds to the pixel size of the point.

The changes made are saved overarching sessions with OK. The basic setting can be restored using Reset.

2.5 Paths

Manages ATHENA data paths.

Dialog box Directories



Dialog box section CAD-PLAN applications

This sub-dialog box is displayed when you have clicked the button Extended. CAD-PLAN applications are modules which overarch programs and which are needed by ATHENA and other CAD-PLAN applications. Here, you can define in which folders ATHENA searches for central data (e.g. materials and layers) and local data (e.g. basic settings).

Dialog box section ATHENA

Here, you can define in which folders ATHENA searches for central data (e.g. standard parts) and local data (e.g. drawing frames) as well as profile systems and user blocks.

To specify the position of a folder, you can write the path directly into the appropriate input field. To search the folder position, click the button [...]. ATHENA starts the dialog box Browse for folder and you can select a new position.

Folder structure for ATHENA and CAD-PLAN applications:

- **Main directory:**
Folder in which the ATHENA program files are saved. You define this folder during the installation and cannot change its position retrospectively.
- **Data central:**
In this folder ATHENA saves all data files which cannot be modified or which can only be modified within ATHENA (e.g. standard part definitions, layer definitions). Companies with many users should divert this folder to a server drive to produce a company standard.
- **Data local:**
Here, ATHENA saves files which can be modified by the user (e.g. drawing frames, dimension styles). Companies with many users should divert this folder to a local drive to enable the user to carry out the user-specific or project-specific settings.
- **Profile systems:**
In this folder ATHENA searches for files of profile manufacturers. You will find information on this in the chapter *Insert profiles* on page 560.
- **User data:**
In this folder ATHENA saves the user blocks. You will find information on user blocks in the Chapters *Save user block* on page 574 and *Insert user block* on page 575.

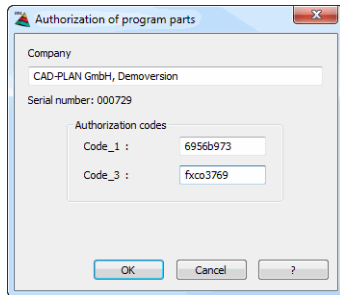


The files of the saving locations "Data central" and Data local" can be interchanged as required. Thus it is ensured that each company with two or more ATHENA workstations manages the settings centrally or locally. Files at the saving location Data local are given priority if they duplicated.

2.6 Authorization

Here you can enter and change your authorization codes. This is necessary after the installation of ATHENA and when you have received an authorization code to convert a temporary license into a permanent license.

Dialog box Authorization



Company

Enter the company name here. Please ensure the correct spelling.

Authorization codes

Here enter the codes 1 and 3. Here too, ensure the correct notation.



Once you have installed ATHENA, the Dialog box Authorization is automatically displayed during the first start.

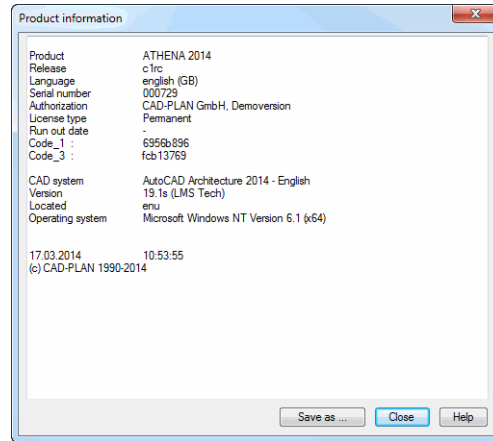
If you have received the authorization codes by e-mail, we recommend that they are inserted into the input fields via the Windows clipboard.

If you have a temporary license, the date of expiry is shown at the lower left. If you enter the company name or a code incorrectly, the notice appears "Incorrect input".

2.7 Product Information

Shows information about the installed ATHENA, AutoCAD and Windows versions. This information is particularly helpful for support queries.

Dialog box Product Information



The dialog box displays important information about your installed ATHENA version. Furthermore, information about the AutoCAD version and the operating system is displayed.

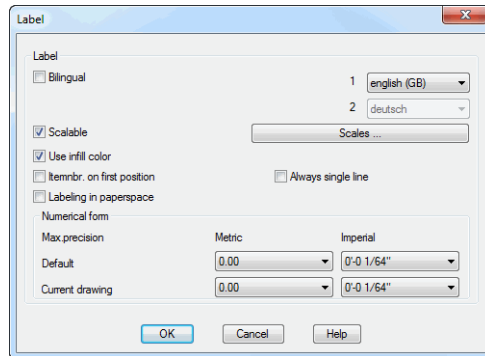
Save as...

Saves the product information in a text file. To do this, the Save dialog box is opened where you can specify the file name and the place of storage.

2.8 Label

Manages settings for labels.

Dialog box Label



Dialog box section Labeling

Bilingual

Activates the bilingual labeling of objects.

You can change the label languages in the selection menus, whereby the second language selection only becomes effective when the tick box Bilingual is turned on.

Scalable

Activates the automatic scaling feature of the labels. The button Scales opens the Dialog box Label scales where you can assign scales to the labels from a list.

You will find further information about scalability in the section *Scalability (label objects)* on page 105.

Use infill color

Activates the infill color for the text background of labels.



If no infill color is used, labels do not cover other objects.

Item in first place

When labeling blocks, this causes the item number in the first position of the label to be shown.



The label texts of the block must first have been defined with the command Assign block label.

Always single line

Generates single line block labels.



The label text of the block must first have been defined with the command Assign block label.

Labeling in paper space

Inserts labels in the layout in the paper space. A viewport must be active for the object selection.

Dialog box section Numerical form

Defines the accuracy for numbers (magnitude figures) in the Position model. Here, a magnitude-dependent accuracy is involved. The default setting 0.00 gives for example:

0.5647 => 0.565

5.768 => 5.77

45.45 => 45.5

556.5 => 557

A composite pane with the dimensions (width x height x thickness) of 1256.4x678.5x32.7 is labeled as follows: 1256x679x32.7.

Maximum precision metric/imperial

Default

Defines the maximum accuracy used in new drawings.

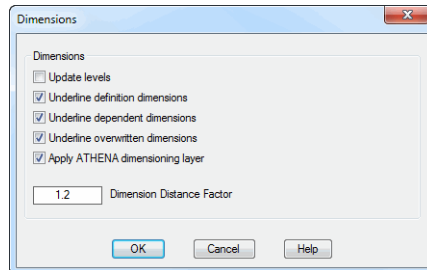
Current drawing

Defines the maximum accuracy used in the current drawing.

2.9 Dimensioning

Manages settings for dimensions.

Dialog box Dimensions



Dialog box section Dimensions

Update levels

If you activate the tick box, ATHENA updates the levels on opening a drawing. We recommend this option when you have modified the levels file (ath_hkote.dwg) and this modification is to take effect in the existing drawings.



This tick box does not function if the Scalability (label objects) uses levels. If scalable levels are to be able to be updated, the levels must be defined as label objects in the definition drawings. With such levels the scalability can then no longer be inhibited!

Underline definition dimensions

If you deactivate the tick box, dimension figures of dimensions, which have been created with the command Defining interrupted dimensions, are no longer underlined.

Underline dependent dimensions

If you deactivate the tick box, dimension figures of dimensions, which have been created with the command Set interrupted dimension, are no longer underlined.

Underline overwritten dimensions

If you deactivate the tick box, dimension figures which have been overwritten are no longer underlined.

Apply ATHENA dimensioning layer

If you deactivate the tick box, the current layer is used when creating dimensions.



If you deactivate the three tick boxes for underlining dimensions, differentiation between virtual and true dimensions is no longer possible. We recommend that these tick boxes are not deactivated!

Dimension distance factor

Here you specify the base line distance between automatically generated dimensions.



The distance is a factor taken into account in the dimension figure height and the distance of the dimension figure to the base line.

$$\text{Dimension distance} = \text{Factor} * \text{dimscale} * (\text{dimgap} * 2 + \text{dimtext})$$

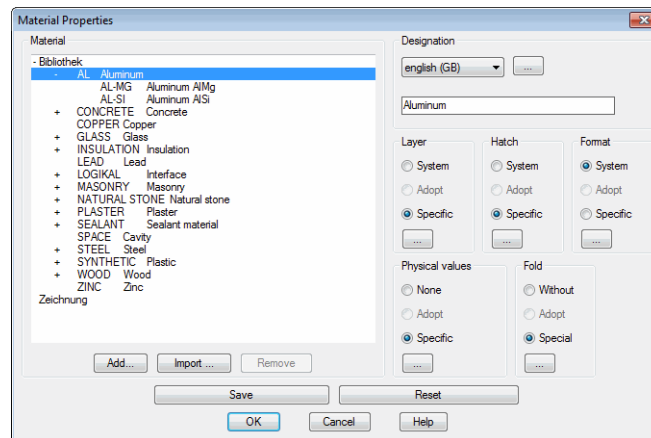
2.10 Material



Ribbon: Tab ATHENA > Group Extras > Material properties
Menu: ATHENA > Service
Toolbar: Not present
Command input: ath_mat_sys

Here, you manage the materials and their properties. ATHENA uses materials in many program modules. Different program modules use different properties of the materials. For example, the routine Sheet metal section for the material aluminum uses the bending allowances whereas the program lx Required/Deflection/Collapsing stress uses the physical material properties.

Dialog box Material properties



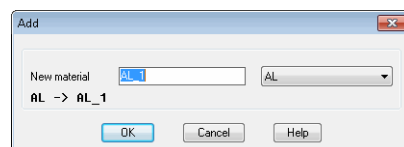
Dialog box section Material

Here all materials are shown in a tree structure. First, the name appears, then the designation of the material in the set language. A + in front of the material indicates that derived materials are available. To open a material branch you must click it with the mouse. The various properties of the selected material can be modified in the part of the dialog box on the right and in sub-dialog boxes.

You can create new materials, import them and delete any that are not required.

Add

Click the button Add to create a new material.



In the dialog box you can select from the list on the right the material whose properties you would like to use as default. You enter the name of the new material in the input field New material. ATHENA creates a new type of material with the properties of the selected material.

Import

Imports materials as well as hatching, layers, blanks and bending tables assigned to them from other ATHENA settings files (cpl_mat.dex and cpl_layer.dex).

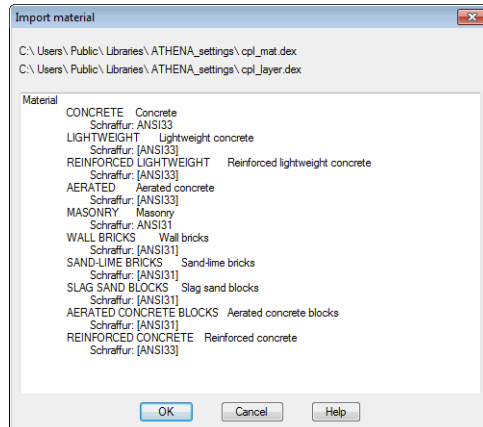
A standard dialog box for material selection appears. Here, you navigate to the folder with the appropriate settings files and select a `cpl_layer.dex` or `cpl_mat.dex`.



It is essential that both files are present, because links exist between layers and materials.

Once you have selected one of the two files, the following dialog box appears:

Dialog box Import materials



The dialog box displays all materials which are not present and can be imported.

Click OK to import the listed materials. Click cancel if you do not want to import the materials.

Remove

When you click the Remove button, you delete the selected material from the list.

Notes

- The materials AL, STEEL and SPACE are required by the system. You cannot delete these materials, but you can change their properties if required.
- The material SPACE has a special status. It is used, for example, for cavities (interpane space) in glazing.

Dialog box section Label

You can save the material designations in various languages. To specify foreign language material designations, select the language from the list. You then enter the name in the relevant language in the input field.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Dialog box section Layer

Here you define the layer settings for the selected material.

System

When you activate the option, ATHENA uses the system layer as the selected material (see also Dialog box System layer).

Adopt

The option is only active with derived materials. Activate this option to use the layer settings of the higher level material.

Specific

With this option ATHENA uses the material-dependent layer. Click the button [...] to change the material layer. You will find further information on this in the Chapter *Layer assignment* on page 108.

Dialog box section Hatch

Here you define the hatch settings for the selected material.

System

When you activate this option, ATHENA uses the system hatch as the selected material (see also Dialog box System layer).

Adopt

This option is only active with derived materials. Activate this option to use the hatch settings of the higher level material.

Specific

With this option ATHENA uses the material-dependent hatching. Click the button [...] to change the material hatching. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Dialog box section Physical values

Here you define the physical values for the selected material.

None

If you activate this option, ATHENA does not make any physical values available for the material. You cannot then use it for structural analysis and building calculations.

Adopt

The option is only active with derived materials. Activate this option to use the physical values of the higher level material.

Specific

When you activate this option, ATHENA uses the physical values defined for the material. Click the button [...] to change the physical values. You will find further information on this in the Chapter *Physical values* on page 112.

Dialog box section Fold

Here you define the settings of bending allowances for sheet metal sections for the selected material.

None

If you activate this option, ATHENA does not make the material available for sheets (command: Sheet metal section).

Adopt

The option is only active with derived materials. Activate this option to use the fold settings of the higher level material.

Specific

If you activate this option, ATHENA uses the allowances defined in the relevant bending tables for the selected material. Click the button [...] to change the bending tables and bending allowances. You will find further information on this in the Chapter *Sheet metal computation values* on page 114.

Dialog box section Format

Here, you define for the selected material the number format which ATHENA uses for evaluating bars and infills in the 3D section.

System

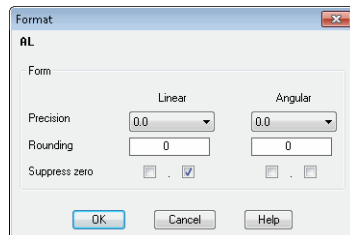
If you activate this option, ATHENA uses the setting of the AutoCAD variables *dimdec* and *dimrnd*.

Adopt

The option is only active with derived materials. Activate this option to use the format settings of the higher level material.

Specific

When you activate this option, the format settings are used which you have defined in the dialog box Format.



Precision

Defines the accuracy for linear figures and angles. You can select the desired precision in the pick list.

Rounding

Defines the rounding rules for linear figures and angles. If you enter the value 0.25 here, all distances are rounded to the nearest quarter level. If you enter the value 1.0, all distances are rounded to the nearest integer number.

Suppress zero

Activation of the corresponding tick box causes preceding or following zeros with linear figures or angles to be suppressed.

Save

When you click the button, the current settings are saved and loaded as default into new drawings.



ATHENA saves these settings in the files *cpl_mat.dex* and *cpl_layer.dex*.

If you have loaded materials via the Design environment, when saving you can select whether the standard settings or the settings of the design environment are to be saved.

Reset

When you click the button, the previously saved settings are restored.

[>]

When you click this button, the Dialog box Material properties is expanded. The Dialog box Material properties is reduced with the button [<].

Click OK to close the dialog box. If you have not clicked the Save button, ATHENA saves the settings for the duration of the drawing session and returns to the starting dialog box (Dialog box ATHENA options).

2.11 Layer



Ribbon: Tab ATHENA > Group Layer > System layer
Menu: ATHENA > Service
Toolbar: Not present
Command input: ath_layer_sys

Here, you manage the system layer used by ATHENA as well as the layer and hatch assignments to the various objects and object constituent parts.

Dialog box System layer

Dialog box section Object

From the pick lists of the various groups you can select the object type whose layer or hatch properties you would like to change. The current layer properties of the selected object are displayed in the Dialog box section Details. The current hatch properties are displayed in the Dialog box section Hatch. Hatch properties are only displayed once you have selected the infill of the respective object.

Note: ATHENA uses the assigned layer and hatching as the system properties for the relevant objects. When you assign a material to an object in its generating dialog field, the material properties are used instead of the system properties.

System layers or system hatching and their application

- System
 - Viewport: Layer for viewport
 - Viewport marking: Layer for viewport marking in the model space
 - Auxiliary Lines: Layer for auxiliary lines
 - Data: Layer is used by the system
 - Text, tables: Layer for results tables (e.g. results from the computation of the Thermal resistance)
- Dimensioning
 - General dimensioning: Layer for linear and angular dimensioning

- Dependent on interrupted dimensions: Layer for dependent (computed) interrupted dimensions
- Definition of interrupted dimensions: Layer for defined interrupted dimensions
- Levels: Layer for horizontal and vertical levels
- Center lines, axes: Layer for axes
- Labeling: Layer for leaders and parts labels.
- Draw
 - 0-0 to 7-1: General drawing layer
- Standard parts/semi-finished products
 - Section outlines: Layer for semi-finished products and standard part profiles
 - Miscellaneous parts: Layer for other standard parts (screws, plugs, etc.)
 - Hidden lines: Layer for hidden lines of the standard parts (e.g. holes in panels)
 - Axes: Axes layer
 - Thread lines: Layer of thread lines for screws
 - Infill/hatching: Layer and hatching for standard parts
 - Contour edge: Layer for tangential transitions for standard parts
 - Solids: Layer for standard solid parts
- Sheet metal
 - Sheet outline: Layer for the sheet outline
 - Sheet infill: Layer and hatching for the sheet
 - Infill for core layer: Layer and hatching for core layer with composite panels
 - Solids: Layer for sheet solids
 - Fold downwards: Layer for fold lines with sheet developments
 - Fold upwards: Layer for fold lines with sheet developments
 - Rolled edge: Layer for rolled edges with sheet developments
 - Coating: Layer for coating lines with sheet metal sections
- Membrane
 - Membrane outline: Layer for membrane outline
 - Membrane infill: Layer and hatching for the membrane infill
- Welded seam
 - Welded seam: Layer for the welded seam
- Spacer
 - Spacer outline: Layer for the external outline of the spacer
 - Spacer infill: Layer and hatching for the spacer
- Insulation
 - Insulation outline: Layer for the insulation boundary
 - Insulation infill: Layer and hatching for insulation
- Gasket
 - Gasket outline: Layer for the external outline of the gasket
 - Gasket infill: Layer and hatching for the gasket
- Seal
 - Seal outline: Layer for the external outline of the seal
 - Seal infill: Layer and hatching for the seal
 - Infill sealing cord: Layer and hatching for sealing cord
- Section symbol
 - Axes: Layer of the lines of the section symbol
 - Symbol: Section symbol layer
- Axis symbol
 - Outline: Layer for the external outline of the axis symbol
 - Glazing axis: Layer for glazing axis
 - Infill/hatching: Layer and hatching for the symbol
- Wall layer
 - Outline: Layer for the external outline of the wall layer

- Infill/hatching: Layer and hatching for the wall layer
- Grid division
 - Full contour: Layer for the complete (unprocessed) contours
 - Trimmed contour: Layer for the processed contours
 - Infill/hatching: Layer and hatching for the grid elements
- Element elevation
 - Frame: Layer for the frame
 - Sash: Layer for the sash
 - Field symbol: Layer for the field symbol (tilt and turn symbol)
- Facade elevation
 - Profile: Layer for the profiles
 - Hidden: Layer for the hidden profiles
 - Center lines: Layer for center lines

Dialog box section Details

The active object type is shown above the section Details.

Here ATHENA displays the layer properties for the selected object type. You can adapt the layer properties to your own requirements.



It is not possible to change the layer properties if the corresponding layer is already present in the drawing. In this case use the AutoCAD layer properties manager.

Name

To use a new layer name, write it into the Name input field. You can also assign an existing layer to an object type. To do this, select a layer from the pick list.

Description

Defines a layer description. The layer description is additional information which is also displayed in the AutoCAD layer properties manager.

Color

To change the layer color, click the color area and select a new color.

Lineweight

To change the lineweight of the layer, choose a lineweight from the drop-down menu. We recommend that you use the default lineweight.

Line type

To change the line type, select a new line type from the drop-down menu.

Plot style

To change the plot style, select a new plot style from the list. This drop-down menu is deactivated when you are using color-dependent plot styles (of color).

Off for display

Switches the layer off or on.

Lock for editing

Locks or unlocks the layer.

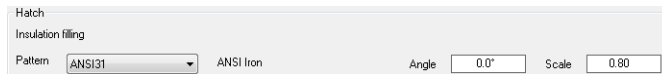
Do not plot

Defines whether the layer is plotted or not

Freeze

Freezes or thaws the layer.

Dialog box section Hatch



The active object type is displayed above.

Here ATHENA displays the hatching properties for the selected object type. You can adapt the hatching properties to your own requirements.

Pattern

To change the pattern of the system hatching, select a new pattern from the list.

Angle

Enter a new angle in the input field Angle to alter the hatching angle.

Scaling

Enter a new scale factor in the input field Scale to alter the hatching scale.

When you click the Save button, the current settings are saved and loaded as default into new drawings.



ATHENA saves these settings in the file `cpl_layer.dex`.

When you click the Reset button, the previously saved settings are restored.

If you have loaded the layer via the Design environment, when saving you can select whether the standard settings or the settings of the design environment are to be saved.

Click OK to close the dialog box. If you have not clicked the Save button, ATHENA saves the settings for the duration of the drawing session and returns to the Dialog box ATHENA options.

Notes

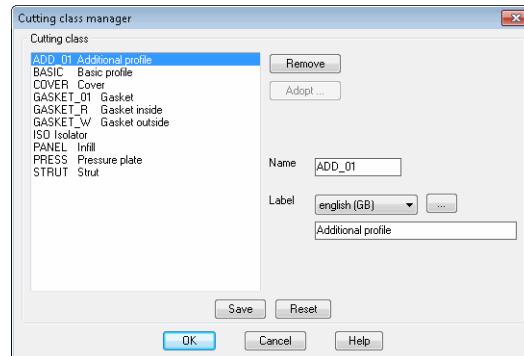
- After termination of the dialog box, use the command **Load layer** to activate the new layer properties in the drawing.
- You will find further information about layers in your AutoCAD documentation.

2.12 Cutting

When you click the button Product information ... in the Dialog box ATHENA options, ATHENA starts the Dialog box Manage cutting classes.

This function is used for creating and managing cutting classes. Components which meet at a junction and are to be cut together must have identical cutting classes. These components are processed during the Cutting according to the type of joint.

Dialog box Manage cutting classes



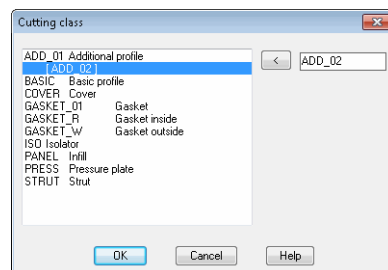
All defined cutting classes are displayed in the list. They can be selected with the mouse for processing.

Remove

Deletes the cutting class from the list.

Adopt

Opens the dialog box for the cutting class.



Here the cutting classes are displayed which are assigned to the assemblies imported from libraries into the drawing. These cutting classes are temporary and are displayed in square brackets for better differentiation.

[<]

Adopts the selected cutting class. Due to the adoption the square brackets are removed and the cutting class can be saved in the Dialog box Manage cutting classes.

Name

To create a new class of cutting, you enter the name here. This is accepted into the list when you quit the box or press the Enter key.

Label

Here, you can enter or change the designation of the current cutting class depending on the language. To do this, select the desired language from the list and enter the designation.

[...]

This button opens the Dialog box Designation, where you can more conveniently manage the multilingual text. You will find further information in the section *Designation* on page 122.

Save

The current list is saved in the file cpl_mat.dex using Save. A query is issued which you must confirm.

Reset

Using Reset the last saved settings are restored.

The dialog box is closed with OK. If you click OK without previously saving, the changes made are retained only for the current drawing session. Cancel closes the dialog box and changes which have not been saved are discarded.



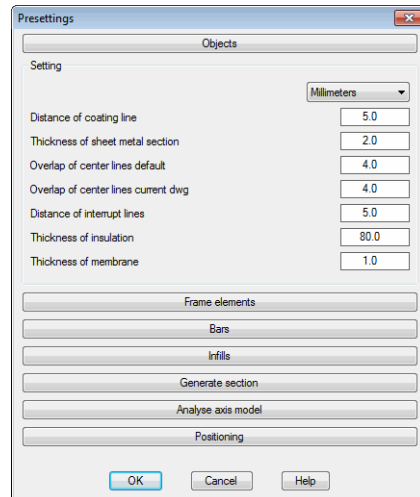
The cutting class is important information for the assembly, which with multi-part assemblies (profile combinations) facilitates the correct cutting of the single components. This means that components with the same cutting class are cut or processed according to the set cutting.

Refer also to Chapter *Cutting* on page 772.

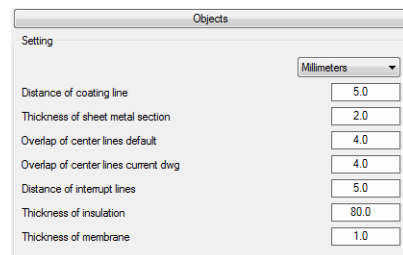
2.13 Presettings

When you click on the Presettings ... button in the Dialog box ATHENA options, ATHENA starts a dialog box where you can carry out the presettings for various functions.

Dialog box Presettings



Drop-down menu Objects



In the upper pick list you can define in which units the values of this dialog box section are displayed. You have a choice between millimeters and inches.

Distance of coating line

Here you define the default distance between the sheet metal section and the coating line.

Thickness of sheet metal section

Here you define the default sheet thickness for the command Sheet metal section.

Overlap of center lines default

Here you define the default center line overlap of center lines, which are created with the commands Standard Part, Semi-finished product, Center lines and Axis, for new drawings.

Overlap of center lines current dwg.

Here you define the default center line overlap of center lines, which are created with the commands Standard Part, Semi-finished product, Center lines and Axis, for current drawings.

Distance of interrupt lines

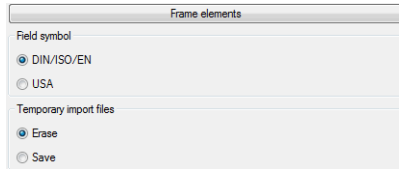
Here you define the default gap between the interrupt lines which are created by the command Interruption.

Thickness of insulation

Here you define the default thickness of the thermal insulation which is created with the command Insulation.

Membrane thickness

Here you define the default thickness of the membrane which is created with the command Membrane.

Drop-down menu Frame elements**Dialog box section Field symbol****DIN/ISO/EN**

Opening symbols for frame elements are generated such that the opening points to the hinged side and the tip points to the handle side.

USA

Opening symbols for frame elements are generated such that the opening points to the handle side and the tip points to the hinged side.

Dialog box section Temporary import files**Erase**

Temporary import files are deleted. This is the normal case.

Save

Temporary import files are deleted.

If there are problems during import, you can activate this setting and send the import file to Technical Support for checking.

Drop-down menu Bars**Dialog box section Marking of bars without solid**

Controls whether with null bars a cone is displayed showing the bar direction.



The size of the cone for marking the direction depends on the current screen section. After zooming and regeneration the size of the new screen section is adapted.

On

Activates the direction marking for null bars.

Off

Deactivates the direction marking for null bars.

Dialog box section Machine support

Machining center

As standard, uses the machine support, which was set for the CNC machining center (CMC), for calculating the cutting angle.

Saw

As standard, uses the machine support, which was set for the saw, for calculating the cutting angle.

Development segmentation

These settings affect the calculation of the development of the round tubes. The default values are suitable for tubes up to a diameter of approx. 200 mm. With larger diameters you should enlarge the arc length.

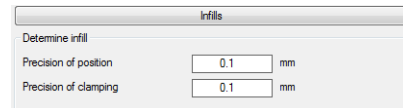
Arc length

Specifies the arc length for which a segment is inserted for segmented development.

Minimum number of segments

Specifies the minimum number of segments.

Drop-down menu Infills



Dialog box section Determine infill

These settings act on the automatic infill determination. You will find further information in the section *Determine infill* on page 787.

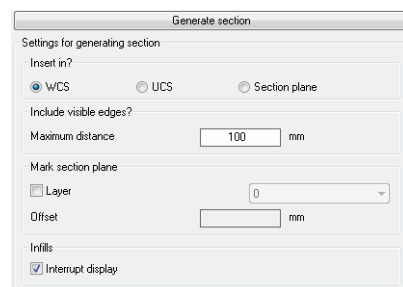
Precision of position

Inaccuracies in the offset of infill positions are ignored up to the stated value. Larger inaccuracies are displayed by the message: **Conflict - infill positions of boundary objects**.

Precision of clamping

Inaccuracies in the clamping thicknesses of the infill positions are ignored up to the stated value. Larger inaccuracies are displayed by the message: **Conflict - clamping of boundary objects**.

Drop-down menu Section generation



These settings act on the section generation. You will find further information on this in the section *Generating a section from 3D* on page 836.

Dialog box section Insert in

WCS

Inserts the generated section in the XY plane of the World Coordinate System.

UCS

Inserts the generated section in the XY plane of the current User Coordinate System.

Section plane

Inserts the generated section in the specified sectional plane.

Dialog box section Include visible edges**Maximum distance**

Specifies the maximum distance to the sectional plane in which visible edges of the solid of unsectioned bars and infills are displayed in section.

E.g.: You are generating a horizontal section through two mullions. If a transom is located less than 100 mm below the sectional plane, its visible edges are shown in section.

Dialog box section Mark section plane**Layer**

The tick box activates the sectional plane marking. In the pull-down menu you can select a layer for marking.

Enlarge by

Specifies the enlargement of the sectional plane marking.



The sectional plane is identified by a region. The size of the region corresponds to the enclosing rectangle of the sectioned objects plus the enlargement.

Dialog box section Infills**Interrupt display**

Generates infills with interrupted display (right and left section instead of complete infill).

Drop-down menu Analysis of axis model**Length mark weather side**

Specifies the length of the red lines which mark the weather side during the analysis of axis models.

Drop-down menu Positioning**Dialog box section Format**

Defines default values for job data and parts for the Positions.

E **Command reference**

The command reference of this documentation consists of three parts.

In the general part you will find information about functions which arise at a number of different places of the program. This mainly involves subdialog fields which can be opened by other commands, that is generally not directly.

Then follows the reference for the commands needed for two-dimensional working.

The last and also the most complex section contains functions which are mainly needed for working in the third dimension.

1 General Functions and Information

This section describes general functions, dialog boxes, dialog box sections and controls. They are used at many different places in the program and are always the same or similar.

Commands in this section:

- Compatibility of ATHENA objects
- Libraries, Library Objects
- Grips, grip editing
- General functions of the dialog boxes
- Scalability (label objects)
- Format settings for labels
- Layer assignment
- Hatch pattern assignment
- Physical values
- Sheet metal computation values
- Label
- Label Scale
- Scale
- Designation
- Material selection
- Cutting class
- Item
- Change dimension text
- Dimensioning options
- User catalog
- Double click
- Add selected

1.1 Compatibility of ATHENA objects

Many ATHENA objects (standard parts, sheet metal sections, membranes, etc.), which have been produced with earlier ATHENA versions, are updated on opening the drawing or when editing the object so that they are compatible with the latest ATHENA techniques. For this reason these objects cannot then be processed with older ATHENA versions.



For the reasons stated above the processing of drawings with different ATHENA versions is not recommended.

1.2 Libraries, Library Objects

You can save objects or also properties of objects in so-called libraries. Libraries are not drawings. They contain objects however, which can be used in drawings. You can transfer objects from drawings into libraries and vice versa.

Objects are presented in libraries with a folder structure which you yourself can determine. Since ATHENA 2014 this folder structure can be of any depth and is also represented in the drawing.

In earlier ATHENA versions the folder structure was limited and was not represented in the drawing. When you edit a drawing containing library objects from an earlier ATHENA version, the compatibility mode notice is displayed. In the compatibility mode no folder structure is replicated; see the following graphics:

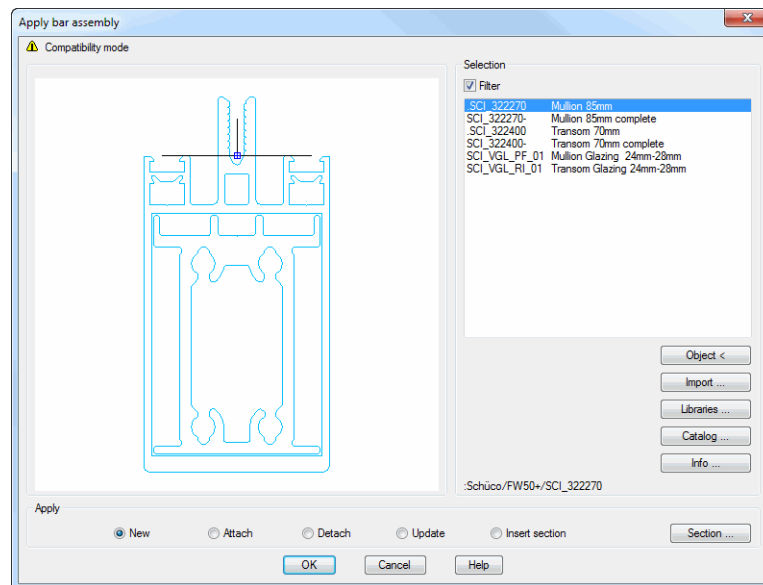


Fig. 1.1: Dialog box Apply bar assembly in compatibility mode

Before you can transfer an object into a library, it must have been saved in the drawing.



Objects which have been saved in libraries are principally not linked to objects in the drawing.

File format of libraries

Libraries of the current ATHENA version are saved with the file extension *.olbx. The folder structure within the library can be of any depth.

Libraries from earlier ATHENA versions have the file extension *.olb. The folder structure within the earlier library was limited to two levels.

You can open an earlier *.olb library to import objects which it contains. You can however only save this library in the new *.olbx format.

The *.olb file is not overwritten; *.olbx files are not downwards compatible! This means that they cannot be opened with earlier ATHENA versions to import objects that they contain.

Associated commands:

- Management of objects
- Saving objects

- Object selection
- Assembly library
- Assembly catalog
- Assemblies folder

1.3 Grips, grip editing

Apart from the well-known grips many ATHENA objects can be processed with additional grips.

In contrast to the familiar square grips the additional grips have a different shape:

- Round grip: Grip for moving objects. Available for standard profiles and semi-finished products in the base point.
- Triangular grip: Grip for stretching line objects without changing their direction. Available, for example, with sheet metal and leaders.
- Arrow-shaped grip: Grip for mirroring bars.

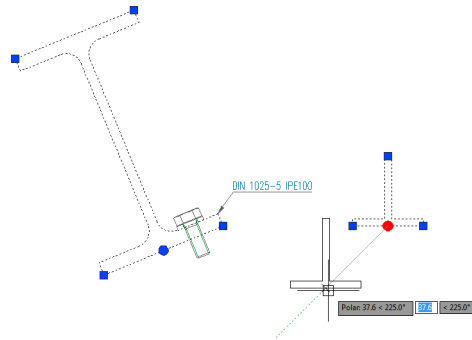


Fig. 1.2: Round grip for moving objects.

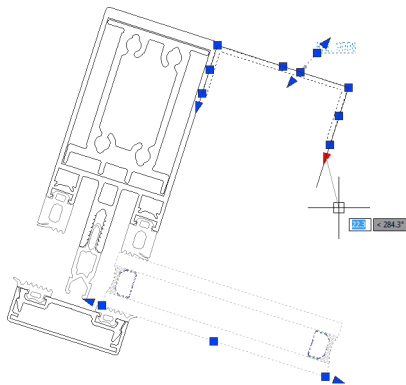


Fig. 1.3: Triangular grip for the linear stretching of objects

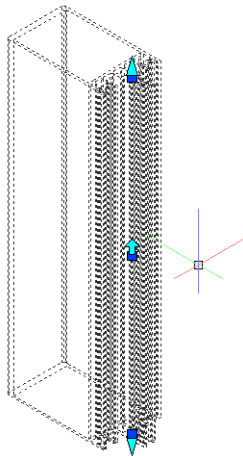


Fig. 1.4: Arrow-shaped grip for mirroring bars

1.4 General functions of the dialog boxes

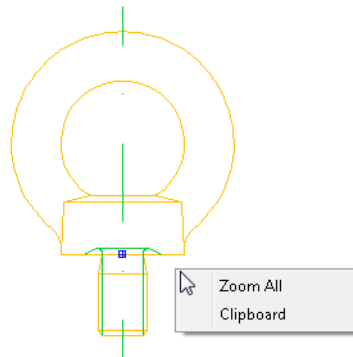
Here, dialog box sections are described which appear in various dialog boxes, but which are the same or similar in their functionality.

Normally, on the left side of the dialog boxes there is the display section with the preview and below the preview and to the right there is the operating section. The operating section on the right next to the preview usually contains a number of tabs.

1.4.1 Object preview

The display section contains a schematic preview of the current object. This preview is used for a visual check and in some cases for the selection of elements for further processing.

Dialog box section Preview



Zooming and panning in the preview

Clicking the preview with the mouse wheel activates the zoom and pan functions. You can zoom in the preview by turning the mouse wheel and also pan with the pressed mouse wheel (as in a drawing). With a right mouse click you can zoom the preview to a full display (**Zoom all**).

Clipboard

Click with the right mouse key to copy a graphical display from the preview to the clipboard.

Special functions

In some dialog boxes you can select objects in the preview and edit them. Where this is possible, it is described in the relevant command section.

1.4.2 Object views

In some dialog boxes to the left next to the preview various buttons are provided with which you can control both the preview itself and also the insertion of the relevant object into the drawing.



2D plan view

Sets the 2D view from above in the preview or inserts the part into the drawing as a 2D plan view.



This is the standard display (plan view) of the standard parts as you are familiar from earlier versions of ATHENA!



2D bottom view

Sets the 2D view from below in the preview or inserts the part into the drawing as a 2D bottom view.



2D side elevation from the left

Sets the 2D side elevation from the left in the preview or inserts the part into the drawing as a 2D side elevation from the left.



This is the standard display (side elevation) of the standard parts as you are familiar from earlier versions of ATHENA!



2D side elevation from the right

Sets the 2D side elevation from the right in the preview or inserts the part into the drawing as a 2D side elevation from the right.



2D front elevation

Sets the 2D elevation from the front in the preview or inserts the part into the drawing as a 2D front elevation.



2D rear elevation

Sets the 2D elevation from the rear in the preview or inserts the part into the drawing as a 2D rear elevation.



ISO view SW (3D solid)

Sets an ISO southwest view in the preview or inserts the part into the drawing as a projection of the ISO view SW.

In some dialog boxes the part is also inserted into the drawing as a solid.



ISO view SE

Sets an ISO southeast view in the preview or inserts the part into the drawing as a projection of the ISO view SE.



ISO view NE

Sets an ISO northeast view in the preview or inserts the part into the drawing as a projection of the ISO view NE.



ISO view NW

Sets an ISO northwest view in the preview or inserts the part into the drawing as a projection of the ISO view NW.



Section (only for stairway)

Sets the stairway section as the preview.

1.4.3 Display options



The Layer button starts the Dialog box Layer assignment, where you can adapt the layer settings of the current object. You will find further information on this in the Chapter *Layer assignment* on page 108.



The Hatch button starts the Dialog box Hatch pattern assignment, where you can adapt the hatch settings of the current object. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.



The Label button starts the Dialog box Label, where you can adapt the label settings of the current object. You will find further information on this in the Chapter *Label* on page 117.

1.5 Scalability (label objects)

Scalable objects are comparable with AutoCAD label objects. If the scalability is activated, these objects are displayed in the viewports of the layouts and in the model section in the size which is determined by the label scale specified for these sections.

You will also find further information about label objects in the AutoCAD documentation.

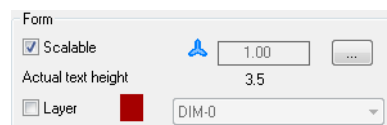
Label objects in ATHENA are:

- Leader
- Coordinate label
- Horizontal levels
- Vertical levels
- Welded seam symbol
- Edge symbol
- Surface symbol

You can activate the scalability of the ATHENA label objects as a basic setup in the ATHENA options or for the individual object in the display section of the relevant dialog box.

1.5.1 Display options for labels

Dialog box section Display



Scalable

Switches the scalability of the object on. With scalability switched on the label scale of the current section (model or layout viewport) is used to control the size of the label.

With scalability deactivated, you can specify the current scale factor in the input field.

[...]

Opens the Dialog box Label scales, where you can assign scales to the leader. You will find further information in the Chapter *Label Scale* on page 120.

Actual text height

Specifies the actual text height. This is calculated from the current text height and the scale factor.

Layer

Activates the layer replacement. If you activate the tick box, you can change the layer set as standard.

1.6 Format settings for labels

In this dialog box section you define the format settings for various label objects.



For leader texts and leader lines the settings of the current dimension style are used as standard. Here, you can overwrite single style settings.

Dialog box section Text format

Dimension style

Switches in replacement of the dimension style. If you activate this tick box, you can select a dimension style from the list.

Text style

Switches in replacement of the text style. If you activate this tick box, you can select a text style from the list.

Text height

Switches in replacement of the text height. When you activate this tick box, you can define a new text height in the input field.



The text height defined here is multiplied by the dimension scale factor.

Distance from base line

Switches in replacement of the distance between the text and the base line. When you activate this tick box, you can define a new distance in the input field.



The distance defined here is multiplied by the dimension scale factor.

Text color

Switches in replacement of the text color. When you activate this tick box, you can define a new text color by clicking the color button.

Infill color

Switches in replacement of the infill color of the leader text. When you activate this tick box, you can define a new infill color by clicking the color button.

Dialog box section Leader lines

Dimension style

Displays the dimension style used for information.

Arrow type

Switches in replacement of the arrow tip. If you activate this tick box, you can select a new arrow type from the list.

Arrow size

Switches on replacement of the arrow size. When you activate this tick box, you can define an arrow size in the input field.

Color

Switches on replacement of the arrow color. When you activate this tick box, you can define a new arrow color by clicking the color button.



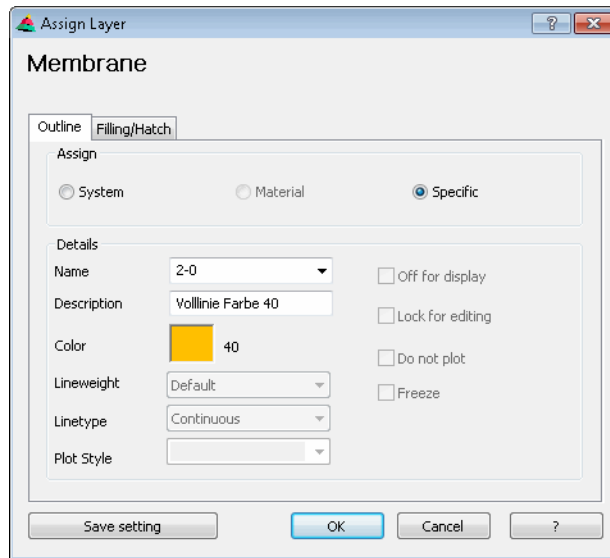
For color changes the dialog box Color is started. You will find further information on this subject in the AutoCAD documentation.

1.7 Layer assignment

Here, you can define layers and layer properties which are to be used for an object.

The Dialog box Layer assignment is used both for drawing objects (e.g. outline and membrane infill) as well as for materials (e.g. AL).

Dialog box Layer assignment



In the upper section of the dialog box the name of the object is displayed for which you define the layer assignment. Most objects consist of several constituent parts. Using tabs you can change between the available constituent parts and separately define the layer assignment. The number and labeling of the tabs depend on the respective object.

Dialog box section Assign

System

Assigns a material-independent system layer to the object. See Layer on page 85.

Material

Assigns a material layer to the object. See Material on page 81.

Specific

Assigns a specific layer to the object. When you choose this option, you can individually define the layer setting.

Dialog box section Details

Here the layer properties are displayed if you have chosen the options System or Material, but the dialog box section is blocked for editing. You can only modify the layer properties when you have selected the Specific option above.

Name

Select the required layer from the list or enter a layer name to create a new layer.

Description

Defines the layer description.

Color

To change the layer color, click the color area and select a new color.

Lineweight

To change the lineweight of the layer, choose a lineweight from the drop-down menu. We recommend that you use the default lineweight.

Line type

To change the line type, select a new line type from the drop-down menu.

Plot style

To change the plot style, select a new plot style from the list. We recommend the use of color-dependent plot styles (From color).

Off for display

Switches the layer off or on.

Lock for editing

Locks or unlocks the layer.

Do not plot

Defines whether the layer is plotted or not

Freeze

Freezes or thaws the layer.

Save settings

Saves the changes as default for the selected object. Otherwise the changes are only valid for the selected object for the duration of the drawing session. This button is not available for materials, because the settings can be saved in the higher level dialog box.

Click OK to close the dialog box and to save the settings. Click Cancel to discard the changes. In both cases ATHENA returns to the starting dialog box.



ATHENA saves the settings for the drawing objects in the file `ath_obj_prop.dex` and for the materials in the file `cpl_mat_dex`.

Notes

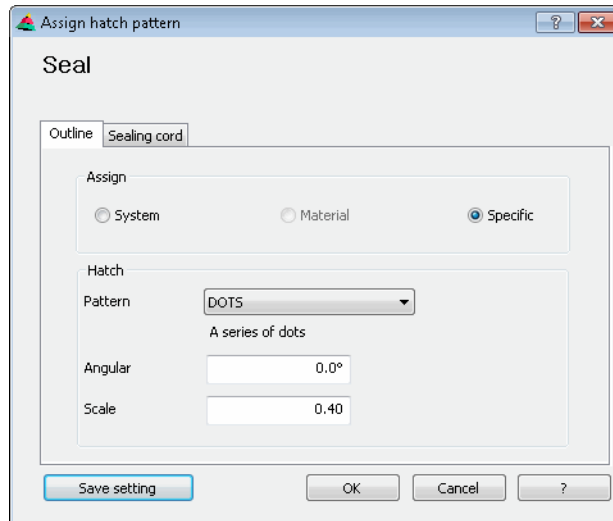
- You will find further information about system layers in the Chapter *Layer* on page 85.
- You will find further information about material layers in the Chapter *Layer assignment* on page 108.
- You will find general information about layers and layer properties in your AutoCAD documentation.

1.8 Hatch pattern assignment

Here, you can define the hatchings which are to be used for an object.

The Dialog box Hatch pattern assignment is used both for drawing objects (e.g. outline and sealing cord of a seal) as well as for materials (e.g. AL).

Dialog box Hatch pattern assignment



In the upper section of the dialog box the name of the object is displayed for which you define the hatch assignment. With some objects several constituent parts can be hatched with different patterns. Using tabs you can change between the available constituent parts and separately define the hatch assignment. The number and labeling of the tabs depend on the respective object.

Dialog box section Assign

System

Assigns a material-independent system hatch to the object. See Layer on page 85.

Material

Assigns a material-dependent hatch to the object. See Material on page 81.

Specific

Assigns a specific hatch to the object. When you choose this option, you can individually define the hatch setting.

Dialog box section Hatch

Here, the hatch properties are displayed without changes being possible. You can only modify the hatch properties when you have selected the Specific option button in the Assign section.

Pattern

To assign a new hatch pattern to the object, select a new pattern from the list.

Angle

Enter a new angle in the input field Angle to alter the hatching angle.

Scaling

Enter a new scale factor in the input field Scale to alter the hatching scale.

Save setting

Saves the changes as default for the selected object. Otherwise the changes are only valid for the selected object for the duration of the drawing session. This

button is not available for materials, because the settings can be saved in the higher level dialog box.

Click OK to close the dialog box and to save the settings. Click Cancel to discard the changes. In both cases ATHENA returns to the starting dialog box.



ATHENA saves the settings for the drawing objects in the file `ath_obj_prop.dex` and for the materials in the file `cpl_mat.dex`.

Notes

- You will find further information about hatching in the Chapter *Layer* on page 85.
- You will find further information about material-dependent hatching in the Chapter *Hatch pattern assignment* on page 110.
- You will find information about hatching, hatch patterns and hatch properties in your AutoCAD documentation.

1.9 Physical values

When you click the button Physical values ... in the Dialog box Material properties, ATHENA starts the Dialog box Physical Material Properties.

Here you enter the physical material parameters which ATHENA uses for the computations in structural analysis and building physics for the selected material.

Dialog box Physical Material Properties

Physical Material Properties

AL Aluminum

Unit
☒ Metric ☐ British - imperial ☐ US - imperial

Coefficient

Density	2710	kg/cbm
Modulus of elasticity	70	GN/sqm
Poisson's ratio	0.33	
Yield strength	95	MN/sqm
Fracture toughness	70	MN/sqm
Thermal conductivity	160	W/(m*K)
Linear expansion coefficient	22	10 ⁻⁶ /K
Specific heat capacity	0.896	kJ/(kg*K)
Thermal transmission coefficient	1	W/(sqm*K)

Omega ☒ Table

OK Cancel ?

Dialog box section Unit

Metric

Activates metric units for material parameters.

British - imperial

Activates British units for material parameters.

US - imperial

Activates American units for material parameters.

Dialog box section Coefficients

Enter the physical material parameters in the appropriate input fields. In doing this, take note of the units located to the right of the input fields. It is not mandatory to enter all values. In the computational dialog fields ATHENA only lists the materials for which the required parameters are present.

Omega ☒ Table

No.	Value
1	1.00
2	1.00
3	1.00

Omega

Activates the table for (omega values) buckling values which are needed for the calculation of supports.

Table

Opens up and closes the table of omega values. Click in a field of the table to activate it and to change the corresponding value.

Click OK to close the dialog box and to save the material values. Click Cancel to discard the settings. In both cases ATHENA returns to the Dialog box Material properties.

1.10 Sheet metal computation values

Here, you determine the computation values which ATHENA uses for developments of sheet metal sections in the selected material.

The specified values are each subtracted from the outer edges of the sheet limb.

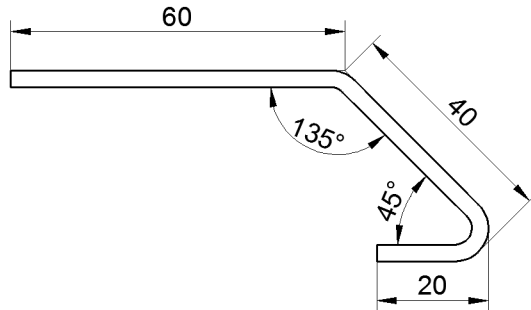


Fig. 1.5: Dimensions for calculating bending allowances



With an obtuse angle the intersection point of the sheet outer edges is measured. With an acute angle the outer point is used for the measurement.

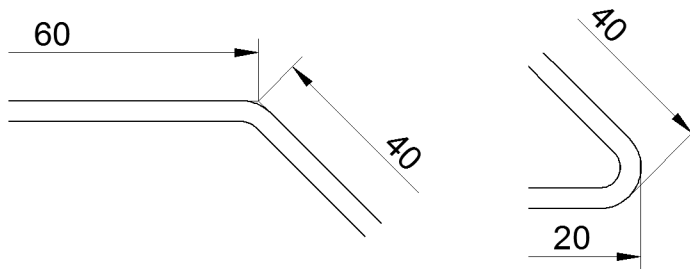


Fig. 1.6: Measurement points: left obtuse angle, right acute angle

Dialog box Sheet metal computation values

Bending allowances for sheet metal

AL Aluminum

Folding tables

Name: AL Tables: AL

Label: english (GB)

Inner bending radius: 1.00

Distance of crimped edge: 1.00

Mode: ☒ Bending angle ☐ Opening angle

Unit: ☒ Millimeter ☐ Zoll

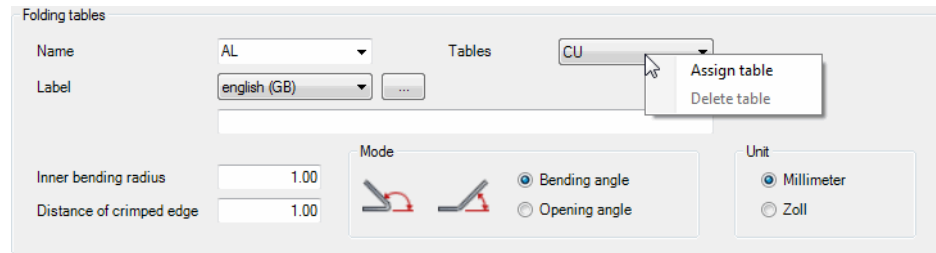
Compensation values

Thickness	15°	30°	45°	60°	75°	90°	105°	120°	135°	150°	165°	180°
1.00	0.27	0.55	0.89	1.31	1.85	2.00	2.00	2.00	2.00	1.67	1.33	1.00
1.50	0.40	0.82	1.31	1.89	2.62	3.00	2.50	2.00	1.50	1.50	1.50	1.50
2.00	0.36	0.76	1.22	1.96	2.89	3.50	3.00	2.50	2.00	2.00	2.00	2.00
2.50	0.49	1.02	1.63	2.37	3.32	4.00	3.33	2.67	2.00	2.00	2.00	2.00
3.00	0.63	1.29	2.05	2.95	4.09	5.00	4.33	3.67	3.00			
4.00	0.89	1.83	2.88	4.11	5.62	7.00	6.67	6.33	6.00			
5.00	0.88	2.02	2.71	4.58	6.22	8.00						

Save setting OK Cancel ?

The active material (e.g. AL - aluminum) is shown above the section Folding tables.

Dialog box section Folding tables



Here you manage the folding tables and their assignment to the current material.

Name

The list contains the tables which are assigned to the current material. You can select an existing table to modify it. To create a new table, enter a new name. With a right click a context menu with further editing options appears.

Release table

Removes the assignment to the current material.

Tables

Contains all available tables. You can select a table. With a right click you obtain further editing options.

Assign table

Assigns the marked table to the active material.

Delete table

Deletes the marked table. Tables assigned to a material cannot be deleted.

Label

Defines the table label in various languages. You can choose a language and enter the text in the input line.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Inner bending radius

Defines to which inner bending radius the allowances apply.

Distance of crimped edge

Defines the default value for the distance of the crimped edge to the sheet.

Mode

Changes the angle display in the values table.

You can choose between display of the bending angle (angle increasing from left to right) and included angle (angle decreasing from left to right).

Unit

Defines the unit in the bending table. The allowance values are converted when you change the unit.

Dialog box section Deductions

Thickness	15°	30°	45°	60°	75°	90°	105°	120°	135°	150°	165°	180°
1.00	0.27	0.55	0.89	1.31	1.85	2.00	2.00	2.00	2.00	1.67	1.33	1.00
1.50	0.40	0.82	1.31	1.89	2.62	3.00	2.50	2.00	1.50	1.50	1.50	1.50
2.00	0.36	0.76	1.22	1.96	2.89	3.50	3.00	2.50	2.00	2.00	2.00	2.00
2.50	0.49	1.02	1.63	2.37	3.32	4.00	3.33	2.50	2.00	2.00	2.00	2.00
3.00	0.63	1.29	2.05	2.95	4.09	5.00	4.33	2.50	2.00	2.00	2.00	2.00
4.00	0.89	1.83	2.88	4.11	5.62	7.00	6.67	2.50	2.00	2.00	2.00	2.00
5.00	0.99	2.02	3.11	4.59	6.22	8.00	7.67	2.50	2.00	2.00	2.00	2.00

Here you define the deductions for the current fold table.

Mark the row you want to edit with the mouse. When you click a marked cell, it is released for editing. You can then change the allowance value directly in the cell. With a right click a context menu with further editing options appears.

Append row

Creates a copy of the marked row above the marked row and releases the cell Thickness for editing.

Insert row

Creates a copy of the marked row at the end of the list and releases the cell Thickness for editing.

Remove row

Removes the marked row from the table.

Save settings

Saves the changes to the bending tables. If you do not save the changes and quit the dialog box with OK, the changes only apply for the drawing session.



The changes are saved in the file cpl_mat.dex.

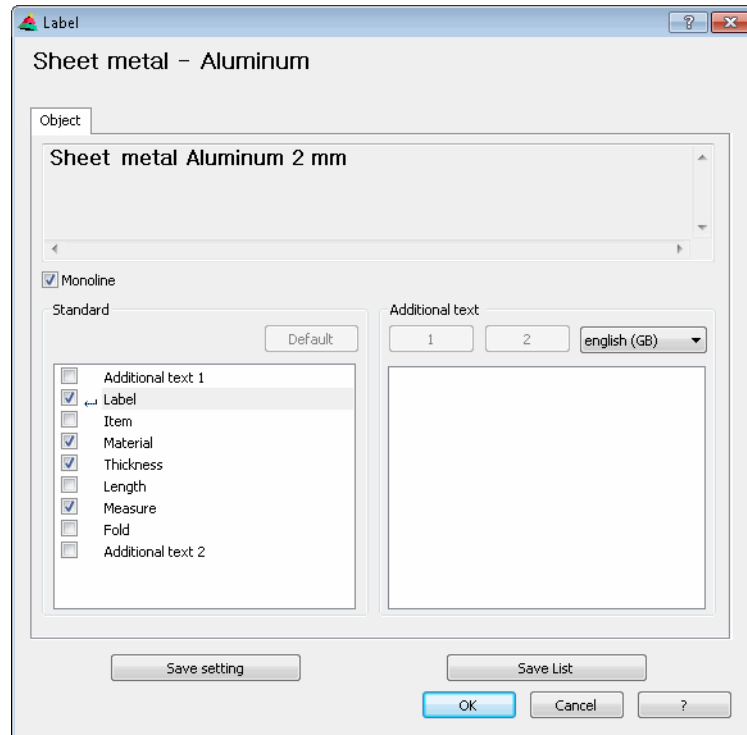
Click OK to close the dialog box and to save the changes. Click Cancel to discard the changes. In both cases ATHENA returns to the starting dialog box.

1.11 Label

In this dialog box you can change the labeling of a part.

You start the dialog box by clicking the button Label ... in the dialog box for the relevant part.

Dialog box Label



The active part is displayed in the upper section.

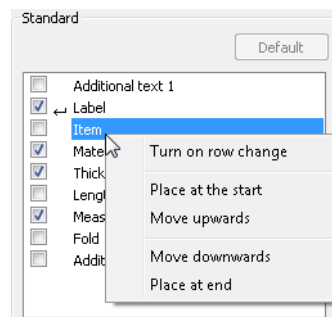
Object

Shows the preview of the label text.

Monoline

Switches the single-line label text on or off.

Dialog box section Standard



Default

Restores the default settings for the label.

List

The list shows the parts of the label available for the active object. With tick boxes you can turn the individual label parts on or off to adapt the label to your

own requirements.

A right click on a label part opens a context menu where you can carry out further label settings.

Turn row change on/off

Turns row change on or off from the selected label part. A row change is identified with a line-feed symbol before the label part.

Place at the start

Sorts the selected label part to the first position.

Move upwards

Moves the selected label part one row upwards (to the front in the label).

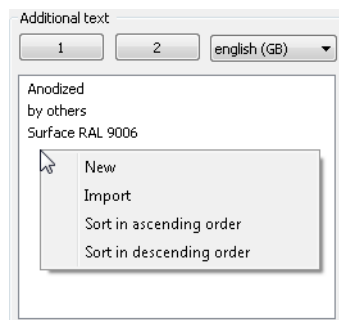
Move downwards

Moves the selected label part one row downwards (to the back in the label).

Place at end

Sorts the selected label part to the last position.

Dialog box section Additional text



In this dialog box section you can define additional texts for the label.

[1]

Defines the marked text as the first additional label part (Additional text 1).

[2]

Defines the marked text as the second additional label part (Additional text 2).

Language

Changes the language of the additional texts. If a text has not yet been translated, it is displayed in the original language and placed in brackets.

List

Shows all available additional texts. You can mark a text to edit it. If you click a marked text again, it is released for editing and can be directly modified.

A right click on a marked text opens a context menu where further editing options are made available.

New

Creates a new text.

Delete

Removes the marked text from the list.

Import

Imports texts from the ATHENA text database `ath_user.rsx`. For this, ATHENA starts the program **TextManager**. Mark a line here, change to ATHENA and click again on the button Import. The marked text is inserted into the list and is available in the appropriate languages. You will find further information on this in the Chapter *Multilingual standard texts* on page 60.

Sort in ascending order

Sorts the list in ascending order; first special characters, then numbers (0-9), then letters (a-Z).

Sort in descending order

Sorts the list in descending order; first letters (Z-a), then numbers (9-0), then special characters.

Save settings

Saves the label settings for the current part.

Save List

Saves the texts for the current part.



ATHENA saves these settings in the file ath_obj_prop.dex.

Notes

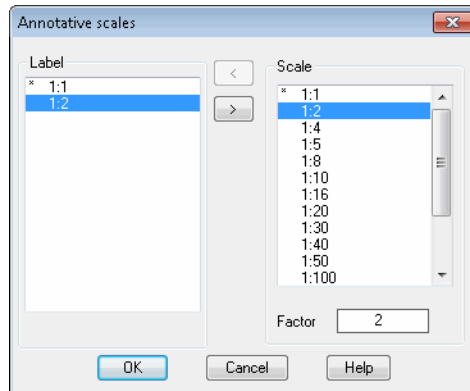
- You will find further information about labeling objects in the Chapter *Parts labeling* on page 425.
- You can change the labeling language by selecting another language in Dialog box ATHENA options. You will find further information in the Chapter *ATHENA options* on page 66.
- You can find further information about the program **TextManager** in the program help.

1.12 Label Scale

In this dialog box you can assign scales to the labels (e.g. leaders).

Depending from which position the dialog box is started, the settings apply to the active label object or as default setting for the new label objects.

Dialog box Label scales



Labeling

Lists the scales assigned to the label.

[<]

Assigns the scale selected in the scale list to the label.

[>]

Removes the scale selected in the scale list from the scale list.

Scales

Lists all available scales.

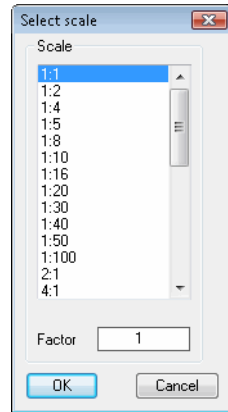
Factor

Defines a new scale factor.

1.13 Scale

Defines the scale of objects. The Dialog box Select scale is always displayed when you have to specify a scale.

Dialog box Select scale



Scale

Defines the scale for the object.

Factor

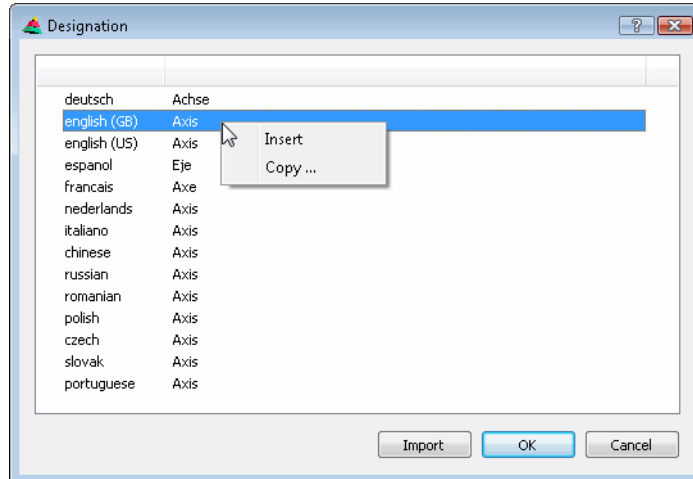
Defines the scale factor. Specify a value here when you want to define a scale not offered in the list.

When you terminate the dialog box with OK, the selected scale is used. With Cancel the previous or predefined scale (1:1) is used.

1.14 Designation

In this dialog box you can conveniently edit multi-language designations.

Dialog box Designation



List

Shows the selected text in various languages. Mark a line with the mouse to edit the text in the respective language.

Import

Imports text from a data base. ATHENA starts the program **TextManager**. Mark a line here, change to ATHENA and click again on the button Import. The marked text is inserted into the list and is available in the appropriate languages.

Text input

The text input is realized by direct cell editing. Click in the line of the relevant language to enter a text or to edit one.

Context menu functions

Copy

Copies the designation with all available translations into the clipboard (not the Windows clipboard). Copied texts can, for example, be inserted for other leaders.

Paste

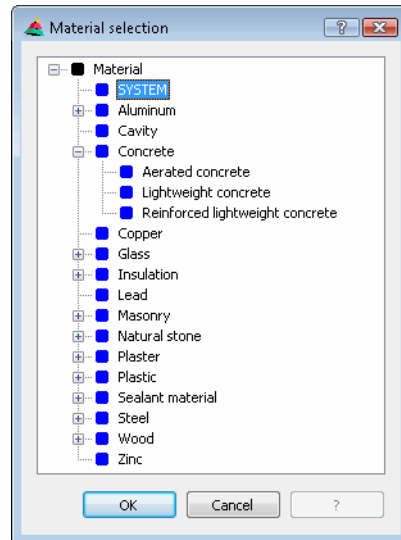
Inserts the designation in all available languages from the clipboard.

You save the changes with OK and discard them with Cancel. In both cases you arrive back at the initial dialog box.

1.15 Material selection

In this dialog box you can select a material to assign it to a part. The Dialog box Material selection is started when you click the button Material... in the dialog box for the relevant part.

Dialog box Material selection



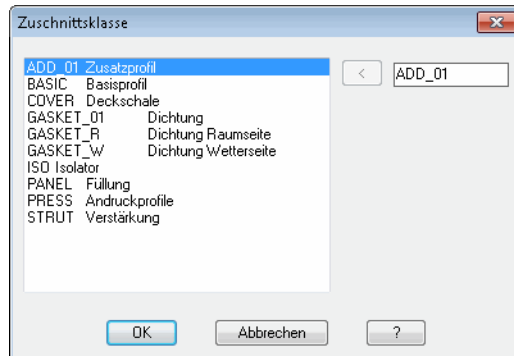
In the dialog box the materials are displayed in a tree structure. Click the + to open a branch. Click the - to close a branch. You can select a material with the mouse.

Click OK to access the initial dialog box. If you click Cancel, you return to the initial dialog box without any change of material.

1.16 Cutting class

Assigns a cutting class to the current component. Components which meet at a junction and are to be cut together must have identical cutting classes. These components are processed during the **Cutting** according to the type of joint.

Dialog box Cutting class



The dialog box is always opened when you want to assign a cutting class to a component.

The list shows all available cutting classes. Select a cutting class here to assign it to the current component and click OK. To create a new cutting class you must write a cutting name in the input field and then click the button [<] to insert the new cutting class into the list.

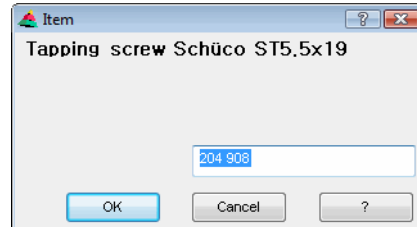
You will find further information in Chapter *Cutting* on page 89.

1.17 Item

In this dialog box an item, process or file number or designation (e.g. for PPC systems) can be assigned to the current object.

You start the dialog box by clicking the button Item ... in the dialog box for the relevant object.

Dialog box Item



Write the item number in the input field provided. When you click OK, the setting for the current object is saved and ATHENA returns to the initial dialog box.



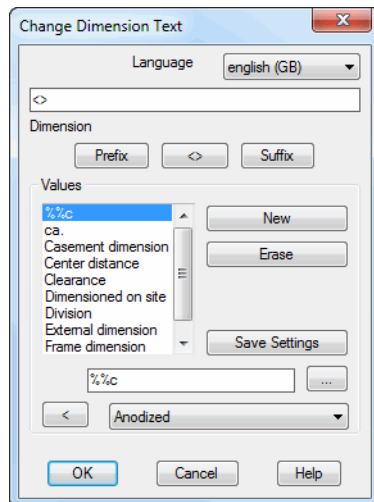
Item numbers are automatically saved in the file `ath_obj_order.dex`.

If, for example you assign an item number to a membrane, it is saved for the thickness and the material.

1.18 Change dimension text

In the dialog box Dimension text you can give dimensions additional text and manage these texts.

Dialog box Dimension text



Language

Defines the language for the dimension texts.

Input line

Shows the dimension text of the current object. You can change dimension text directly in the input line or use the methods described further below.

The object type, e.g. definition dimension, is shown below the input line.

Prefix

Positions the text selected in the value list before the current dimension text.

[<>]

Restores the original dimension text.

Suffix

Positions the text selected in the value list behind the current dimension text.

Dialog box section Values

Value list

Shows the dimension texts available for the selected object. Here, select the text for further use.

If you select a text, it is displayed in the input line. Here, you can modify the text and adopt it by pressing the Enter key.

New

Enables the input line for the definition of a new text. You write a text in the input line and press the Enter key to insert the text into the list.

Erase

Removes the selected text from the list of values.

Save List

Saves the texts of the list.

If you do not save the list, the texts are only available for the duration of the drawing session.

Input line

Defines new texts or displays the text selected in the value list.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.



If you edit texts in the Designation dialog box, you must then save them with the Change button.

The values list is saved in the file ath_obj_prop.dex.

[<] and pick list

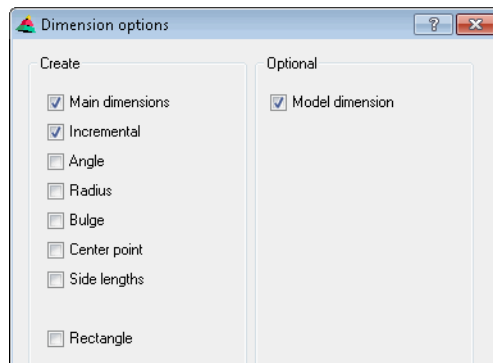
The pick list displays multilingual standard texts from the text database ath_user.rsx. Select a text and click the button [<] to insert the text into the list.

You will find further information about labeling objects in the Chapter *Multilingual standard texts* on page 60.

1.19 Dimensioning options

Here you can define the dimension settings for the automatic dimensioning functions.

Dialog box Dimensioning options



Here you can activate the desired dimensioning option using tick boxes. You can combine the dimensioning options as required. However, it is possible (in dependence of the outline) for dimensions to be produced twice.

Dialog box section Dimension

Maximum dimension

Dimensions the largest linear frame of the enclosing outline rectangle.

Connected dimension

Generates linear dimension chains for all dimension points of the outline.

Angle

Dimensions all angles of the outline which are not 90°.

Radius

Dimensions all radii of the outline.

Receptacle

Dimensions the receptacles for arc-shaped segments.

Center point

Dimensions the centers for arc-shaped segments.

Edge length

Dimensions all outline edges with aligned dimensions.

Rectangle

Also dimensions rectangular outlines (otherwise only model slices).

Dialog box section Optional

Model dimension

Uses special dimension methods for model outlines.



The outlines and dimensioning rules are defined in the file `cp1_model.dex` which can be adapted by the ATHENA Support. Please contact ATHENA Support if outlines are not dimensioned according to your requirements

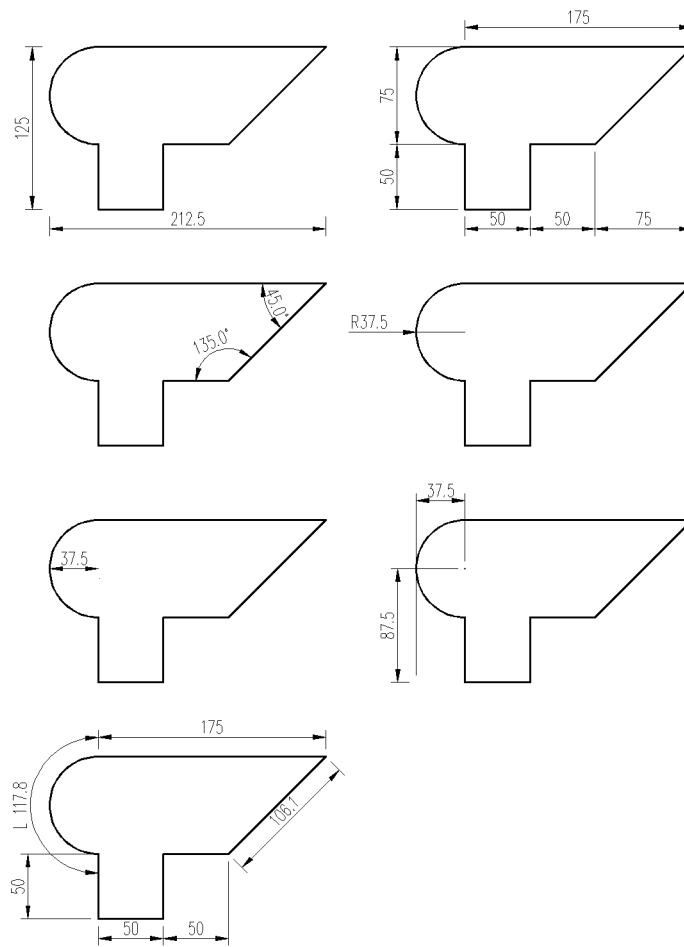


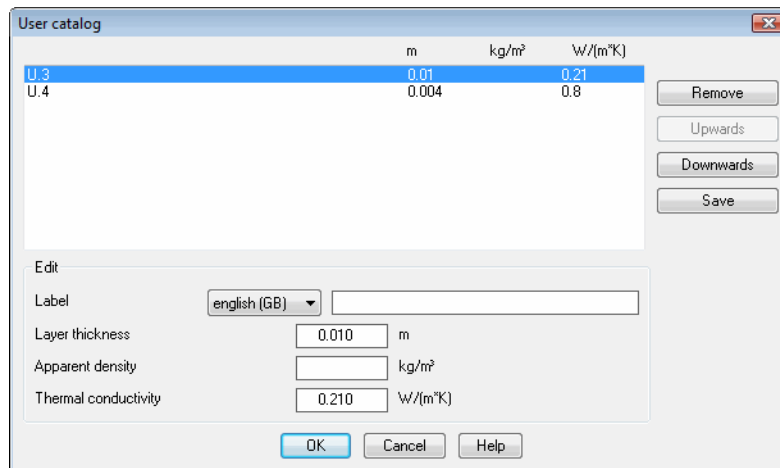
Fig. 1.7: Dimensioning options

1.20 User catalog

With this command you can edit the entries in the user-specific part in the catalog of thermal, moisture and sound-protection parameters.

The command starts when you click the button Edit ... in the Dialog box Thermal resistance or in the Dialog box Roughly determined sound insulation factor for a construction..

Dialog box User catalog



Here you can make changes to the user entries.

With the button Remove you delete the marked entry from the user catalog.

With the button Upwards you move the marked entry upwards.

With the button Downwards you move the marked entry downwards.

When you click the button Save, the changes are saved in the user catalog without the dialog box being terminated.

Dialog box section Edit

Here you can change the building physics values of the selected material. You can adapt the label for various languages by selecting the desired language and writing the label into the input field. Furthermore, you can define the values for layer thickness, apparent density and thermal conductivity in the corresponding fields.

By clicking OK the command is terminated and all changes are saved.



ATHENA saves the user-defined materials in the file ath_htr.dex.

With Cancel the command is terminated without the changes being saved (unless you have previously clicked Save). After closing the dialog box User catalog, the previous dialog box is displayed again.

1.21 Double click

You can edit most AutoCAD and ATHENA objects using a double click. When you double click an object, the associated dialog box with the default settings of the selected object is started. You can then modify the desired parameters.

You can find further information on modifying ATHENA objects by double click in the Chapter *Modify ATHENA* on page 478.

You can find further information on the modification of AutoCAD objects by double click in your AutoCAD documentation.

1.22 Add selected

With this command you generate a new object with the same object type and the same properties as the original object, but you are requested to specify the size, position and other properties.

This AutoCAD command can also be used for ATHENA objects. If, for example, you select a Sheet metal section, the Dialog box Sheet metal section is displayed with all the properties of the selected sheet. You only have to click OK and you can immediately produce a sheet with identical properties.

2 Usage and management of objects

In this chapter you will find information on object management which includes drawings. Example: Many objects can be saved in libraries which can in turn be combined in catalogs. Once defined, these objects can be quickly used in different drawings.

Commands in this section:

- Management of objects
- Saving objects
- Object selection
- Parameters for bar assembly
- Bar properties
- Assembly parameters
- Component variants
- Infill
- Infill parameters
- Apply infill
- Bar
- Bar cross section
- Wall
- Wall parameters
- Frame element
- Assembly library
- Assembly catalog
- Assemblies folder
- Folder properties
- Info
- References
- Projects

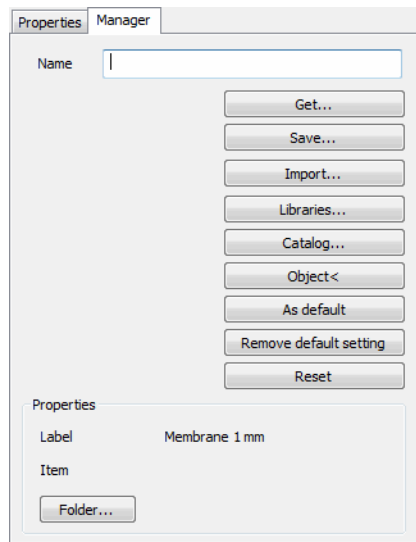
2.1 Management of objects

The creating dialog boxes for many objects have a Manager tab (or register button). This includes an object manager which will be described in this section.



Depending on the object, it is possible that not all of the management functions described here are available.

Tab Manager



Name

Defines the name of the object to be saved.



The name is the unique object designation and is equivalent to a data base code. The name should in no way be confused with the designation. This does not need to be unique and can be saved in different languages.

Get ...

Opens the Dialog box For object selection, where you can load a saved object. See Object selection on page 138.

Save ...

Opens the Dialog box Save, where you can save the current object. See Saving objects on page 136.

Import ...

Opens a standard dialog box for file selection where you can select a drawing (*.dwg, *.dwt or *.dxf) to import the library parts it contains into the current drawing.

Once you have selected a drawing, the Dialog box Assembly library is displayed. Here you can select the required parts and import them into the current drawing. You will find further information on this in the Chapter *Assembly library* on page 157.

Libraries ...

Opens the Dialog box Assembly library, where you can manage objects in libraries. See Assembly library on page 157.

Catalog ...

Opens the Dialog box Assembly catalog, where you can load objects from catalogs. See Assembly catalog on page 159.

Object <

Selects an object in the drawing. For this the dialog box is temporarily closed. The selected object is then marked or displayed in the dialog box.

As default

Saves the current settings as user defaults.

Remove default setting

Removes the user defaults and restores the system defaults. The button is inactive when no user defaults have been saved.

Reset

Restores the default settings (system defaults or user defaults) in the dialog box.



If a dialog box is started for the first time in a drawing session, either the system defaults or the user defaults are displayed, provided they have been saved. If a dialog box is started again during the drawing session, the last status is in each case displayed.

Dialog box section Properties

Label

Defines a label for the current object.



With most objects the current label text is used as the designation. With some objects the designation must be defined by the user.

Item

Defines the item number of the current object.

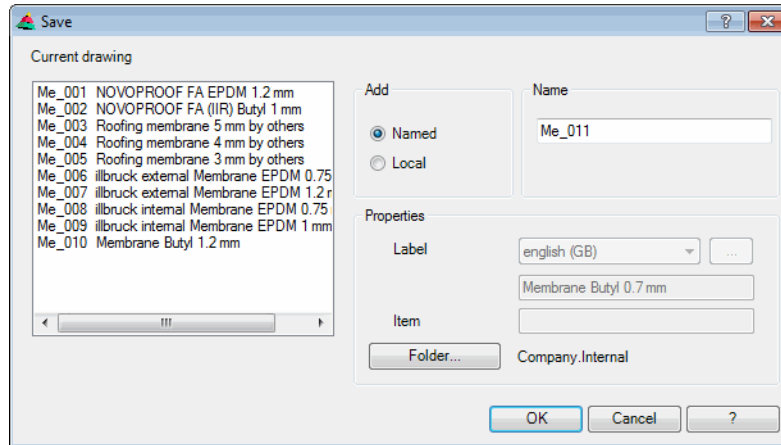
Folder

Opens the Dialog box Assemblies folder, where you can manage the folder structure for the object within the library. See Folder properties on page 164.

2.2 Saving objects

With this function you can save a described object in the drawing. If the drawing is saved, the objects saved in it are retained for future sessions.

Dialog box Save



Dialog box section Current drawing

Shows the objects which are present in the current drawing. This list is informative.

Dialog box section Add

Here you can select the method of saving the object. The issuing of an appropriate name depends on this selection.

Named

Saves the object under a freely selectable name. Here, conformance to the Naming convention should be followed. The issuing of a dedicated name enables better association to the saved objects. The assignment of an object to a name enables the interchange of objects beyond their own medium.

Local

Saves the object with an automatically issued name (e.g. TMP0001) which is automatically incremented.



In earlier ATHENA versions temporarily saved objects were saved using the symbol * (asterisk) and any designation that had been issued and they could not be interchanged beyond their own document.

When you save and interchange objects at a level spanning documents, you should use the named method.

Dialog box section Name

Here, you can issue a name for the object.

Dialog box section Properties

Here you can specify language-dependent labels for the described object. You select a language from the list and add the appropriate text in the line below. You proceed like this for each language. Furthermore, you can issue an item number.

Save With the call of the dialog box from the selection dialog boxes with the button Label [...] changing the existing name is not possible. Only the language-dependent label and the item number can be entered later or changed.

Folder

Opens the Dialog box Assemblies folder. You will find further information on this in the section *Assemblies folder* on page 162.

Naming convention

Conformance to the naming convention is important when issuing dedicated object names. Upper and lower case are taken into account when displaying the object names, but are not differentiated during identification. In the following you will find a list of legal and illegal characters.

Legal characters:

- Letters [a b c ... x y z] and [A B C ... XYZ]
- Numbers [0 1 2 3 4 5 6 7 8 9]
- - (Hyphen)
- _ (Underscore)
- . (Point)

Impermissible characters:

- / (Forward slash)
- \ (Backslash)
- : (Colon)
- * (Asterisk)
- ? (Question mark)
- " (Quotation mark)
- < (Left pointed bracket)
- > (Right pointed bracket)
- | (Vertical line)
- @ (At symbol)

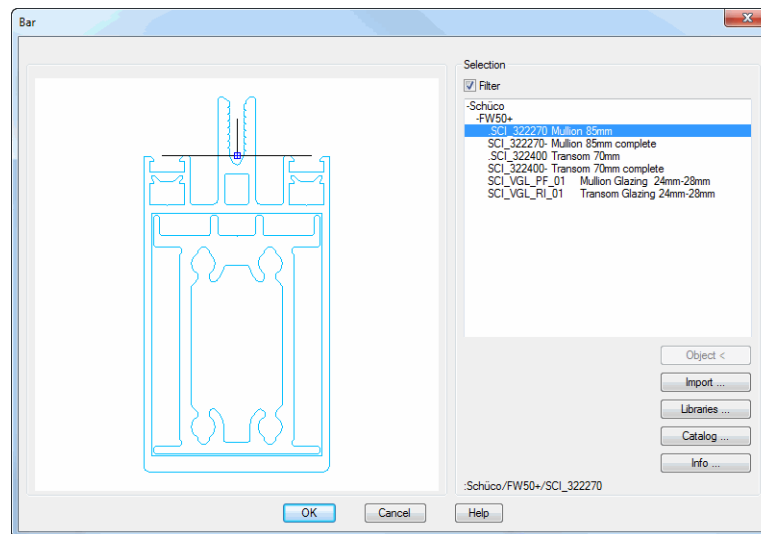
With the entry of illegal characters they are automatically converted into space characters by the program.

2.3 Object selection

With this function you can load or select an object. This is necessary to transfer the object definition into a dialog box or to assign objects to or interchange them with a component definition.

The call for this dialog box may differ and in each case determines the viewed object type. For each call only one object type is listed and the basic functional features are retained. Examples of calling dialog boxes are Bar Assembly Manager, Assembly Manager and Infill manager.

Dialog box For object selection



In the list all objects of the type under consideration existing in the document are listed. With the selection of an object it is displayed in the display section.

Filter

Controls the display of objects to provide a more clear representation.

With bar assemblies reference objects are masked out when you activate the filter. With the filter switched off reference objects are identified with a point before the name.



Exception: Reference objects are not masked if they are used in a drawing as a section or bar!

With semi-finished products, including Membrane, Insulation, etc., objects are masked out which cannot be used in the superordinate dialog box. Example: A membrane cannot be used in the Dialog box Insulation, so with an active filter no membranes are shown.

Object <

Selects an object in the drawing. For this the dialog box is temporarily closed. The selected object is then marked in the dialog box.

Import ...

Opens a standard dialog box for file selection where you can select a drawing (*.dwg, *.dwt or *.dxf) to import the library parts it contains into the current drawing.

Once you have selected a drawing, the Dialog box Assembly library is displayed. Here you can select the required parts and import them into the current drawing. You will find further information on this in the Chapter *Assembly library* on page 157.

Libraries ...

Opens the Dialog box Assembly library. You will find further information in the Chapter *Assembly library* on page 157.

Catalog ...

Opens the Dialog box Assembly catalog. You will find further information in the Chapter *Assembly catalog* on page 159.

Designation ...

Changes the object designation. To do this, the Dialog box Save is opened. You will find further information in the Chapter *Saving objects* on page 136.

Info ...

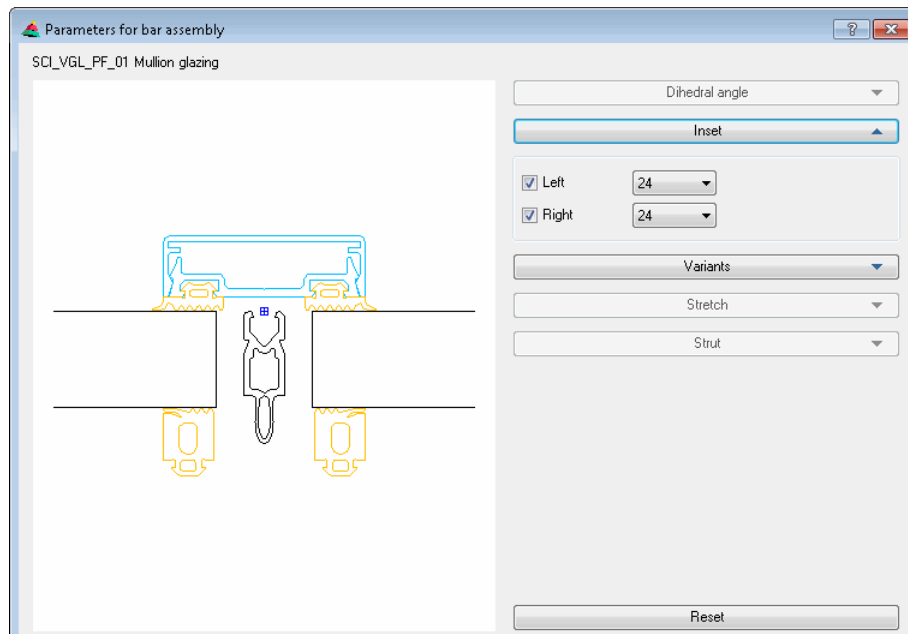
Lists the components of the active assembly. To do this, the Dialog box Component parts is opened. You will find further information in the Chapter *Info* on page 165.

2.4 Parameters for bar assembly

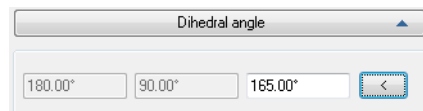
Here you can check the parameters of variable bar assemblies or set them before insertion.

Dialog box Parameters for bar assembly

On the left side the dialog box contains a preview of the current part. On the right side there is the operating section with the various drop-down menus:



Drop-down menu Dihedral angle



Defines the dihedral angle. The first two fields show the defined angular range. In the third field you define the required angle. This must be located within the defined range.

[<]

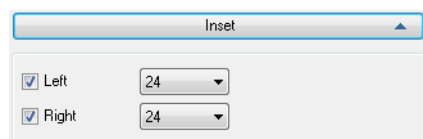
Adopts the angle from an existing bar. For this, the dialog box is temporarily closed and the input request appears:

Command prompt

Select bar or [?]:

Select the bar whose angle you would like to adopt.

Drop-down menu Clamping



Left

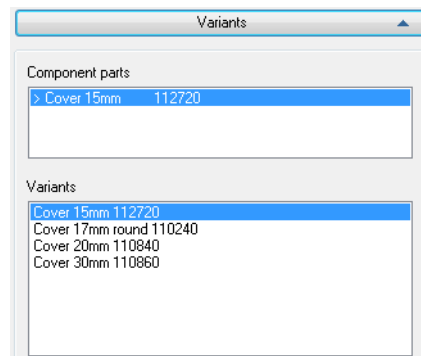
Activates or deactivates the left clamping. When the clamping is activated, all defined clamping thicknesses are made available in the pull-down list.

Right

Activates or deactivates the right clamping. When the clamping is activated, all defined clamping thicknesses are made available in the pull-down list.



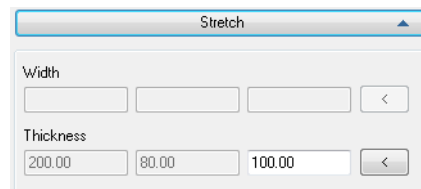
When you make your selection, the clamping thickness is changed and, depending on the definition, an object is displaced or changed. For example, with mullion/transom assemblies the rubber is changed in dependence of the clamping thickness.

Drop-down menu Variants**Component parts**

Displays the components for which the variants have been defined. To change a component variant mark the component here and select a variant in the variant list.

Variants

Here the variants for the selected component are displayed. Here, you select the component variant which you want to use.

Drop-down menu Stretch**Dialog box section Width**

Defines the width of the variable component. The first two fields show the defined range. In the third field you define the required width. This must be located within the defined range.

[<]

Accesses the width in the drawing. For this the dialog box is temporarily closed.

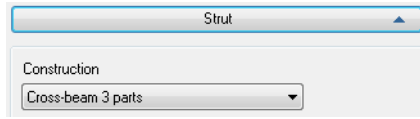
Dialog box section Thickness

Defines the thickness of the variable component. The first two fields show the defined range. In the third field you define the required thickness. This must be located within the defined range.

[<]

Accesses the thickness in the drawing. For this the dialog box is temporarily closed.

Drop-down menu Strut



This selection is active when a function with fixed design rules is selected. This was previously the case with the function Strut. An entered design type can be selected here.

Reset

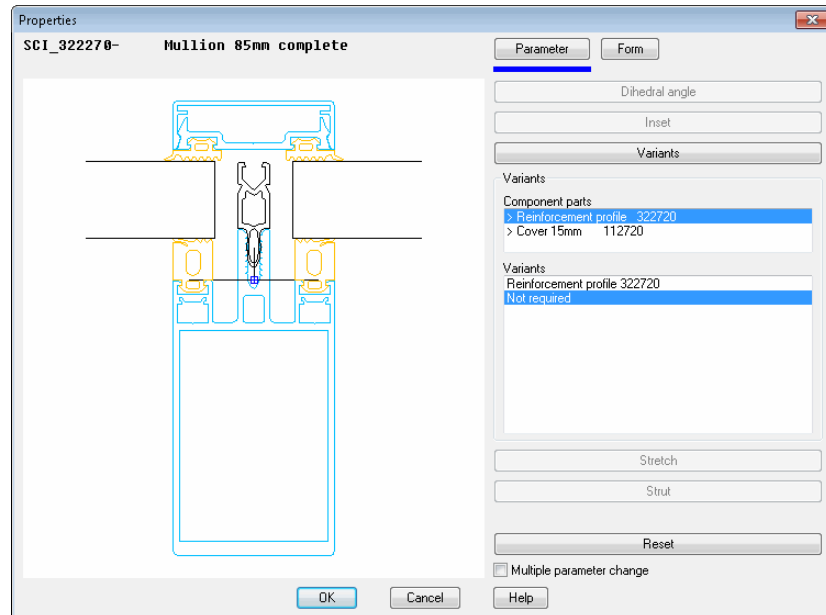
Restores the basic settings of the assembly definition.

2.5 Bar properties

Modifies the properties of one or more bars.

The dialog box is started when you double click a bar and select the option Properties.

Dialog box Properties



On the left side the dialog box contains a preview of the trimmed bar. When you modify the bar properties, they are immediately displayed.

On the right side there is the operating section with the tabs:

- Parameter
- Display

Register button Parameter

A description of the options contained in the various drop-down menus can be found in Chapter *Parameters for bar assembly* on page 140.

Multiple parameter change

Changes the properties for other objects which have to be selected. Then use the following procedure:

Command prompt

Select objects:

Select the bars of which the properties are to be changed. Only the same bars are taken into account.

Register button Display

You will find a detailed description of the display options in the Chapter *Display modes* on page 722.

Dialog box section Visibility

You will find a detailed description of the visibility options in the Chapter *Visibility of bar components* on page 726.

Dialog box section Process steps

Create separately

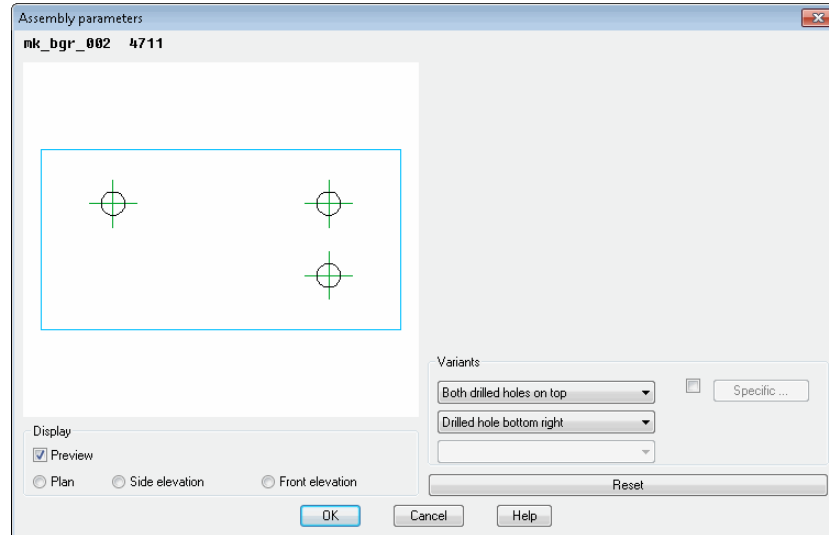
Causes the processes (e.g. drilled holes and notches) to be produced as separate parts.

Sometimes processes on the bar are not shown, because they have been positioned unintentionally outside of the outline. With this option you can check the position of the processes and change the position parameters appropriately.

2.6 Assembly parameters

If you want to use assemblies with variants in the drawing, the Dialog box Assembly parameters is displayed. Here, you can set the variant of the assembly which you want to use.

Dialog box Assembly parameters



On the left side the dialog box contains a preview of the current part. If you change the variant, it is immediately displayed.

Dialog box section Display

Preview

Determines whether a preview of the assembly is displayed in the dialog box or not.

Plan

Shows the plan view of the assembly in the preview.

Side elevation

Shows the side elevation of the assembly in the preview.

Front elevation

Shows the front elevation of the assembly in the preview.



The settings of the preview have no effects on the insertion of the assembly into the drawing!

Dialog box section Variants

With the help of a maximum of three selection menus you can select the variants with which you want to insert the assembly into the drawing.



The number of active selection menus depends on the number of defined variants of the assembly.

Specific

Determines the assembly variant by setting switches. This occurs in the Dialog box Button assignment.

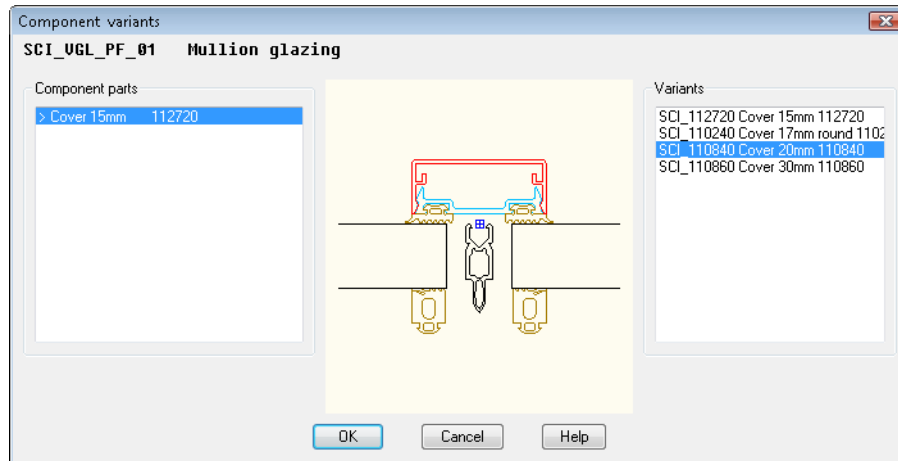
Reset

Activates the definition status of the assembly.

2.7 Component variants

Here you can interchange variable components.

Dialog box Component variants



In this dialog box the components for which variants are assigned are listed on the left side. The assigned variants can be selected on the right side. The current selection is shown in the preview for a visual control.

2.8 Infill

In this dialog box you can modify various properties of an inserted infill section. This dialog box is activated when you double click an infill in the drawing.

Dialog box Infill



Parameter

Changes the infill parameters. For this, the Dialog box Infill parameters is opened, where you can change various parameter settings. You will find further information in the Chapter *Infill parameters* on page 148.

Shear

Displays the infill cut at specified saw and oblique angles in the drawing. The angular settings are carried out in the Dialog box Shear. You will find further information on this in the Chapter *Shear object* on page 838.

Edit

Opens the Dialog box Infill manager, where you can edit the infill. You will find further information in the Chapter *Infill manager* on page 664.



If you want to edit an infill with the infill manager, you must first release it. Changes only affect the selected infill.

If you would like to apply modifications permanently to other infills, you must save them under a new name.

Edge spacer

Changes the viewing properties of the edge spacer. To do this, the Dialog box Bar cross section is opened. You will find further information in the Chapter *Bar cross section* on page 153.

Info

Lists the layers of the active infill. To do this, the Dialog box Component parts is opened. You will find further information in the Chapter *Info* on page 165.

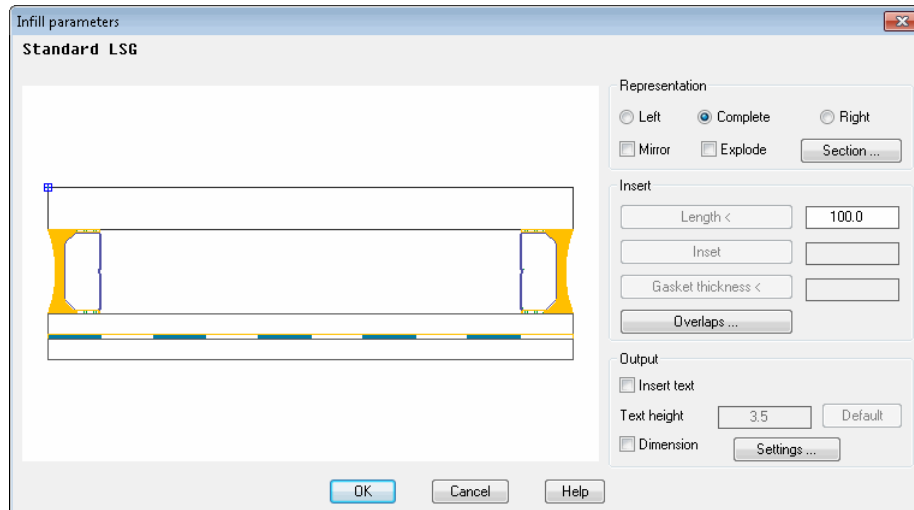
Detach

Detaches the object assignment for the infill definition. When you release the attachment, the switch is deactivated.

2.9 Infill parameters

Here, you can define the section display and the properties of infills.

Dialog box Infill parameters



Dialog box section Representation

Left

Creates the left side of an infill, the right side is interrupted.

Complete

Creates a complete infill without interruption.

Right

Creates the right side of an infill, the left side is interrupted.

Mirror

Mirrors the infill about its own axis.

Explode

Releases the infill into its constituent parts. With a double click the exploded infill can no longer be edited as a whole, but instead only its layers.

Dialog box section Paste

Length, Inset, Gasket thickness

With the buttons Length <, Inset < and Gasket thickness < you can access the appropriate values from the drawing. When you click one of the buttons, the dialog box is temporarily closed and you can select two points. The distance between the points is transferred into the corresponding input field. Alternatively, you can also write the value directly into the corresponding input field.



The insertion point of the infill is changed according to the entries made here, so that it can be correctly positioned.

Overlaps

Opens the Dialog box Overlaps, where you can define overlaps for stepped-edge glazing.

Dialog box section Output

Insert text

Activate this tick box to label a layer.

Text height

Defines the text height of the label text.

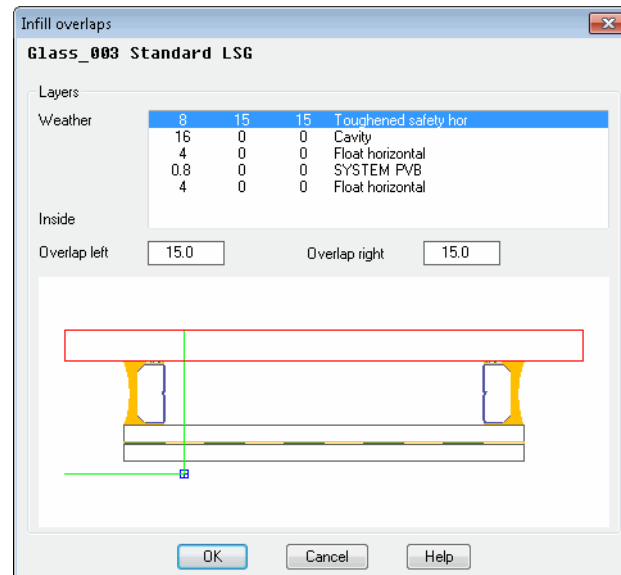
Default

Restores the default text height (current figure height dimension).

Dimensioning

If you activate the tick box, the infill is dimensioned.

Dialog box Overlaps



List

Shows the available infill layers. Select a layer here to change its overlap.

Overlap left

Defines the left overlap of the marked layer. The value entered here is adopted as default for the right side.

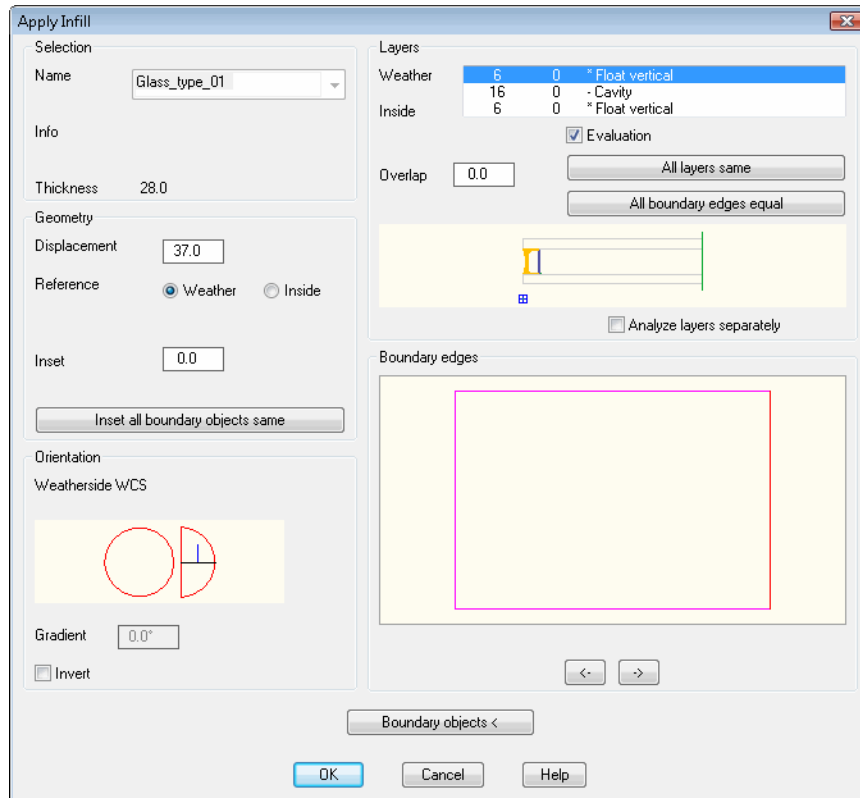
Overlap right

Defines the right overlap of the marked layer.

2.10 Apply infill

The dialog box Apply Infill appears when you insert an infill into the drawing (refer to command *Apply infill* on page 748) or when you modify existing infills, for example with a double click.

Dialog box Apply infill



Dialog box section Selection

Here the current infill is displayed with its properties.

Dialog box section Geometry

Displacement

Specifies the distance of the reference side of the infill to the insertion point of the boundary object (e.g. bars). With the option buttons Weather and Inside you define the reference side of the infill.

Inset

Specifies the inset from the outermost edge of the boundary object (enclosing rectangle) to the infill. If the button Inset all boundary objects same is clicked, then the entered inset is adopted for all boundary objects.

Dialog box section Orientation

Weather side WCS

Displays the reference side of the infill with respect to the World Coordinate System (WCS). The left symbol (circle) shows the weather side in plan view in the WCS. The right hand symbol (semi-circle) shows the weather side as a view in the X/Y plane.

Gradient

Indicates the infill gradient in degrees (°).

Invert

Changes the weather side. The effects are apparent from the symbols.

Dialog box section Layers

Here, all layers of the infill are shown. Click a layer to mark it.

Evaluation

Marks layers for evaluation. The marked layer is identified with *. The commands *Infill list* and *Infill diagram* only take into account layers marked in this way.

Overlap

Defines a value for the overlap in order to create stepped-edge glazing. The overlap initially only applies to the marked layer of the active side (marked red in the preview).

All layers same

Adopts the specified overlap for all layers of the infill.

All boundary edges same

Adopts the specified overlap for all boundary edges which surround the infill.

Analyze layers separately

Analyzes the layers individually. This means that each layer appears separately in the parts list.

Dialog box section Boundary edges

Here you see a preview of the current infill. The active edge is marked in red and its inset or overlap can be changed.

[<-]

Marks the next edge in the counterclockwise direction.

[->]

Marks the next edge in the clockwise direction.

Boundary objects <

Assigns new boundary objects to the current infill. Then a query appears.

If you respond to this confirmation query with No, then the boundary objects will not be changed. The following input request appears if you select Yes.

Command prompt

Select boundary object or [?]:

Use the mouse to choose the first boundary object.

Select boundary object or [Undo/?]:

Use the mouse to choose the next boundary object. This input request is repeated. After selecting the last boundary object, press the Enter key to assign the new boundary objects. The Apply infill dialog box appears once again.

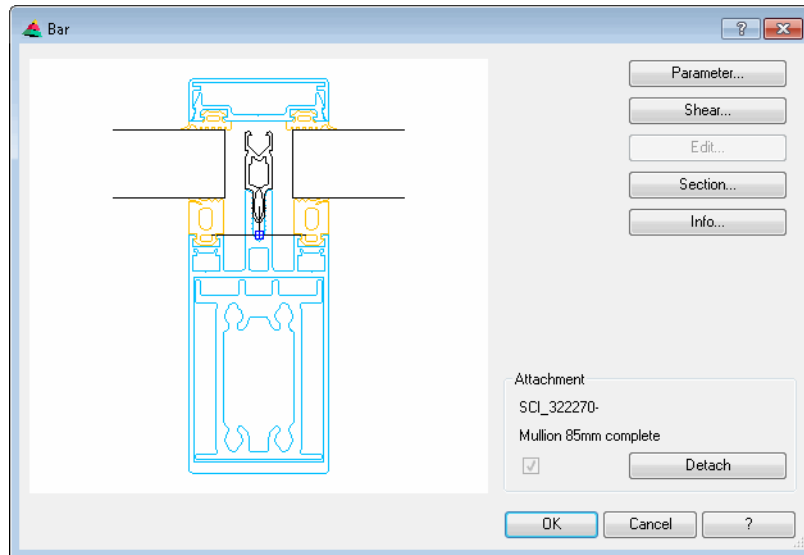
Click OK to quit the dialog box and to accept the changes. With Cancel, the changes are discarded.

2.11 Bar

In this dialog box you can modify various properties of an inserted assembly section.

This dialog box is activated when you double click an infill in the drawing.

Dialog box Bar



Parameter

Changes the parameters of variable bar assemblies. For this, the Dialog box Parameters for bar assembly is opened, where you can change various parameter settings. You will find further information in the Chapter *Parameters for bar assembly* on page 140.

Shear

Opens the Dialog box Shear. You will find further information in the Chapter *Shear object* on page 838.

Edit

Opens the Dialog box Bar Assembly Manager, where you can edit the bar assembly. You will find further information in the Chapter *Bar Assembly Manager* on page 632.



If you want to edit an assembly with the bar assembly manager, you must first release it. Changes only affect the selected bar assembly.

If you would like to apply modifications permanently to other bar assemblies, you must save the bar assemblies under a new name.

Section

Changes the viewing properties of the section display. To do this, the Dialog box Bar cross section is opened. You will find further information in the Chapter *Bar cross section* on page 153.

Info

Lists the components of the active assembly. To do this, the Dialog box Component parts is opened. You will find further information in the Chapter *Info* on page 165.

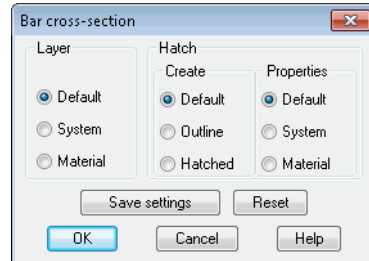
Detach

Detaches the object assignment for the assembly definition. When you release the attachment, the switch is deactivated.

2.12 Bar cross section

In this dialog box you can change the display properties of assembly sections.

Dialog box Bar cross section



This dialog box is opened from other dialog boxes.

Dialog box section Layer

Default

Uses the layer which has been defined in the object properties.

System

Uses the system layer which has been defined for section outlines of the standard parts. See Layer on page 85.

Material

Uses the layer which has been defined for the material assigned to the object. See Material on page 81.

Dialog box section Create

Default

Uses the hatch setting which has been defined in the object properties.

Outline

Does not use any hatching.

Hatched

Uses the hatching corresponding to the properties.

Dialog box section Properties

Default

Uses the default properties of the object definition.

System

Uses the system hatching. See Layer on page 85.

Material

Uses the material hatching. See Material on page 81.

Save settings

Saves the settings.

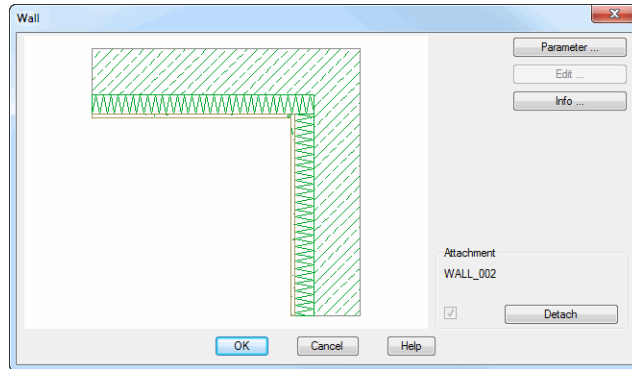
Reset

Restores the settings last saved.

2.13 Wall

You can edit walls with a double-click. To do this, the Dialog box Frame element is shown.

Dialog box Wall



Parameter

Opens the Dialog box Wall. Here, you can modify the wall properties. You will find further information on this in the Chapter *Wall parameters* on page 155.

Edit

Opens the Dialog box Wall manager. Here, you can modify the wall properties. You will find further information on this in the Chapter *Wall manager* on page 671.



Walls, which have been inserted with the Apply wall command, are linked to the saved library object. Before you can modify them, they must be released from the library object.

Info

Opens the Dialog box Component parts. You will find further information on this in the Chapter *Info* on page 165.

Dialog box section Assign

When a link to a saved object exists, its name is shown here.

Detach

Releases the wall from the saved library object. You can then edit its properties.



Detachment occurs immediately and cannot be discarded by closing the dialog box with Cancel.

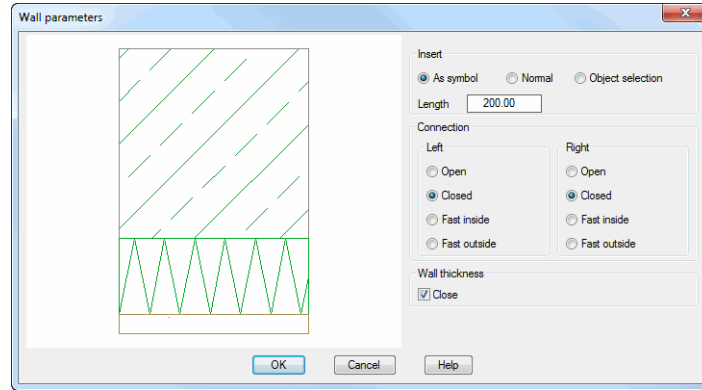
End of program

OK closes the dialog box and the changes are implemented. Cancel closes the dialog box and the changes are discarded (with the exception of Detach).

2.14 Wall parameters

Changes the wall parameters.

Dialog box Wall parameters



Dialog box section Paste

These options are not available when editing walls.

Dialog box section Left/Right

Determines the type of connection to the right or left side of the wall.

Open

Opens the wall on the corresponding side, for example for an interrupted representation.

Closed

Closes the wall on the corresponding side.

Abutment inside

Shows a mounting on the inside of the wall with the entered thickness and width.

Abutment outside

Shows a mounting on the outside of the wall with the entered thickness and width.

Dialog box section Wall thickness

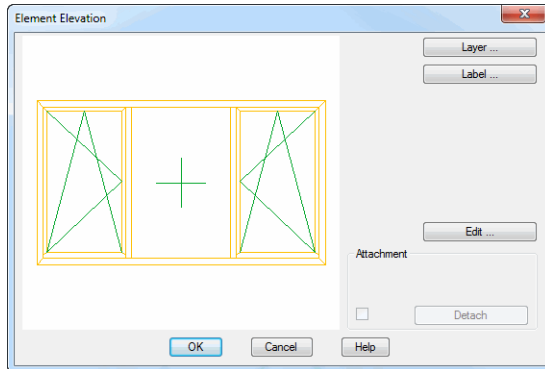
Close

Closes the lowermost layer of the wall structure.

2.15 Frame element

You can edit frame elements with a double-click. To do this, the Dialog box Frame element is displayed.

Dialog box Frame element



Layer

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Edit

Opens the Dialog box Frame Element Manager. Here, you can modify the frame element properties. You will find further information on this in the Chapter *Frame Element Manager* on page 676.



Frame elements, which have been inserted with the Apply frame element command, are linked to the saved library object. Before you can modify them, they must be released from the library object.

Dialog box section Assign

When a link to a saved object exists, its name is shown here.

Detach

Releases the frame element from the saved library object. You can then edit its properties.



Detachment occurs immediately and cannot be discarded by closing the dialog box with Cancel.

End of program

OK closes the dialog box and the changes are implemented. Cancel closes the dialog box and changes are discarded (with the exception of Detach).

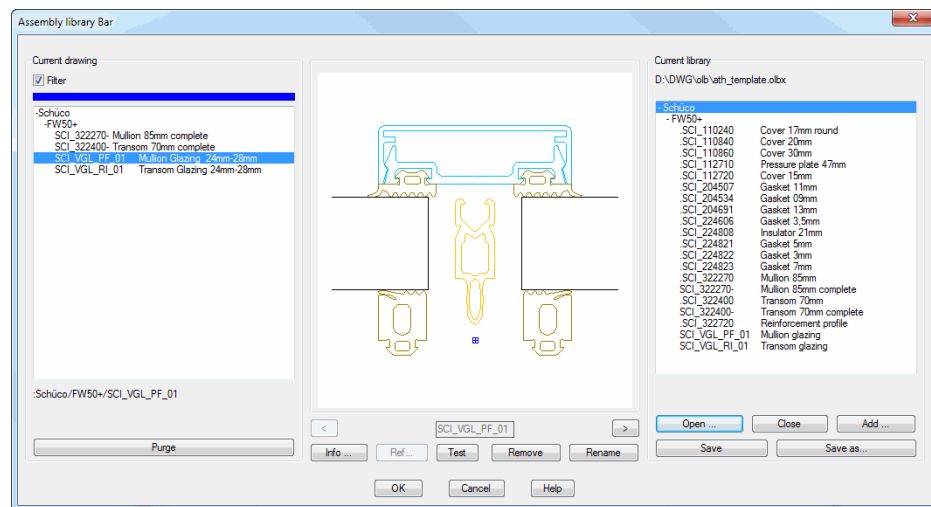
2.16 Assembly library

Libraries are files in which all defined object types are archived independent of the document and can be called. Saved objects can be reciprocally interchanged between libraries and documents.

This dialog box is used for the management of objects in the drawing and in libraries. It enables the copying and removal of objects with additional optional functions for viewing, parameters and object content.

The Dialog box Assembly library appears when you click the button Library ... in the initial dialog box.

Dialog box Assembly library



In the dialog box only the objects are displayed which can be used in the current command. For example, if you start the dialog box from the command Infill, only infills and no screwed joints or bars are displayed.

Dialog box section Current drawing

In the dialog box section Current drawing all objects in the current document are listed in a tree structure. Folder names are identified with +/- and can be expanded or reduced by clicking on them.

Filter

Controls the display of objects to provide a more clear representation.

With bar assemblies reference objects are masked out when you activate the filter. With the filter switched off reference objects are identified with a point before the name.



Exception: Reference objects are not masked if they are used in a drawing as a section or bar!

With semi-finished products, including Membrane, Insulation, etc., objects are masked out which cannot be used in the superordinate dialog box. Example: A membrane cannot be used in the Dialog box Insulation, so with an active filter no insulation is shown.

Purge

Removes these objects completely from the document.

Dialog box section Display

The middle dialog box section provides the preview of selected objects.

The other display options offered depend on the object type and offer various views of or onto the object.

[<]

Copies the selected object from the library into the drawing.

Copies the selected object from the drawing into the library.



Select the relevant superordinate folder to copy its complete content.

If objects with the same name already exist in the target structure, a query appears. Here, you can decide whether the object(s) are to be overwritten or not.



Info

Opens the Dialog box Component parts, where you can change the folder properties. See the Chapter *Info* on page 165.

Ref

Opens the Dialog box References, where you can check in which component the selected one is present as reference.

Test

Opens a separate dialog box for testing available parameters. This option is only available for objects with extended settings.

Remove

Deletes the current assembly from the list.

Rename

Releases the input field where you can change the assembly name.

Dialog box section Current library

In the dialog box section Current library all objects contained in the library are listed in a tree structure. Folder names are identified with +/- and can be expanded or reduced by clicking on them.

Open

Loads a new library. The current library is then removed completely from the list.

Close

Removes the library from the list.

Add

Inserts objects from another library into the current one.

Save

Saves the current library in the list under the specified file name.

Save as

Saves the current library under a new name.



Please follow the information in Chapter *Libraries, Library Objects* on page 99 regarding compatibility and the file format of libraries.

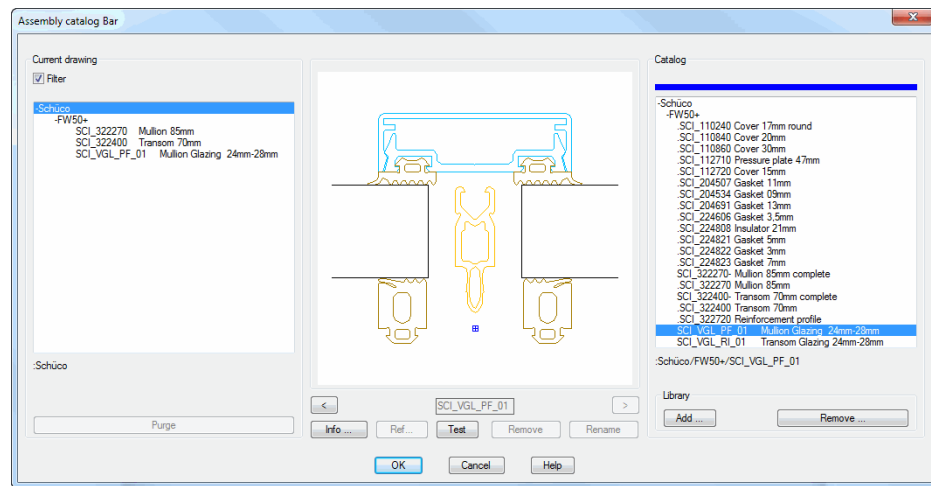
You close the dialog box with OK.

2.17 Assembly catalog

The catalog is a collection of libraries which is intended to simplify access to project-relevant objects. The catalog is part of a definable Design environment which, in a project-related manner, can be set up, saved and recalled at any time. If you have not set up any Design environment, a template library (ath_template.olb) is loaded into the catalog.

This dialog box is used to gain access to the objects brought together in the catalog. It facilitates the import of objects into the document with additional options for viewing, parameters and object content.

Dialog box Assembly catalog



Dialog box section Current drawing

In the dialog box section Current drawing all objects in the current document are listed in a tree structure. Folder names are identified with +/- and can be expanded or reduced by clicking on them.

Filter

Controls the display of objects to provide a more clear representation.

With bar assemblies reference objects are masked out when you activate the filter. With the filter switched off reference objects are identified with a point before the name.



Exception: Reference objects are not masked if they are used in a drawing as a section or bar!

With semi-finished products, including Membrane, Insulation, etc., objects are masked out which cannot be used in the superordinate dialog box. Example: A membrane cannot be used in the Dialog box Insulation, so with an active filter no insulation is shown.

Purge

Removes these objects completely from the document.

Dialog box section Display

The middle dialog box section provides the preview of selected objects.

The other display options offered depend on the object type and offer various views of or onto the object.

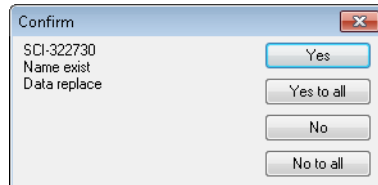
[<]

Copies the selected object from the library into the drawing.



Select the relevant superordinate folder to copy its complete content.

If objects with the same name already exist in the target structure, a query appears. Here, you can decide whether the object(s) are to be overwritten or not.



Info

Opens the Dialog box Component parts, where you can change the folder properties. See the Chapter *Info* on page 165.

Ref

Opens the Dialog box References, where you can check in which component the selected one is present as reference.

Test

Opens a separate dialog box for testing available parameters. This option is only available for objects with extended settings.

Remove

Deletes the current assembly from the list.

Rename

Releases the input field where you can change the assembly name.

Dialog box section Catalog

In the dialog box section Current library all objects contained in the library are listed in a tree structure. Folder names are identified with +/- and can be expanded or reduced by clicking on them.

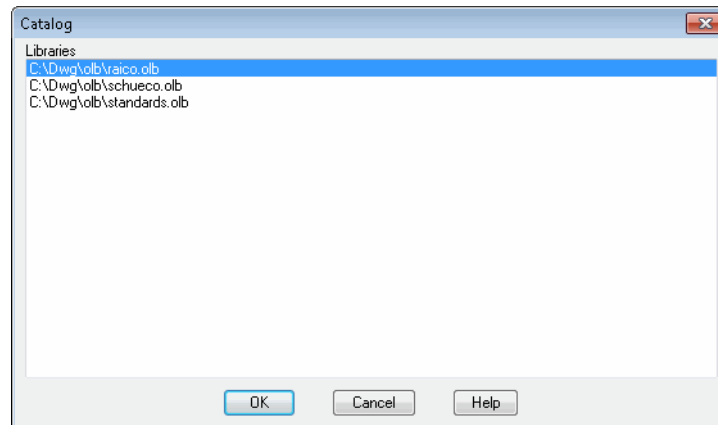
Dialog box section Library

Add

Adds a library to the catalog. To do this, the standard selection dialog box is opened.

Remove

Removes libraries from the catalog. To do this, the Dialog box Catalog is opened.

Dialog box Catalog

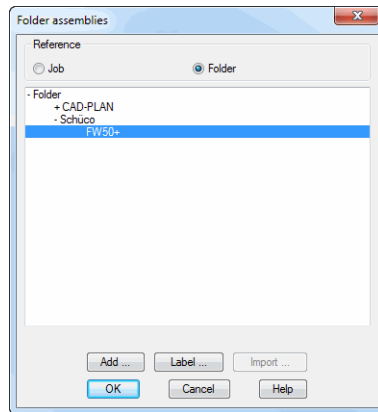
Libraries currently present in the catalog are listed in the dialog box. You remove a library from the catalog in that you mark an entry in the list and terminate the dialog box with OK.

2.18 Assemblies folder

You can create folders and jobs with this function. Both folders and jobs are displayed in libraries as levels.

Folders are used for grouping objects in libraries. Jobs are necessary when objects are to be analyzed, e.g. in lists.

Dialog box Assemblies folder



Dialog box section Reference

Job

Sets the reference to Job. If jobs have already been created in the drawing with the Project Manager, they are displayed.



Jobs are essential if analyzes are to be carried out.

Folder

Sets the reference to Folder. The dialog box changes slightly depending on the selection.

Add ...

Complements the job part in the selected job structure or a subfolder in the selected folder structure.

When the job option is activated, the Dialog box Add is opened, where you can specify the job data. You will find further information about jobs in Section *Project Manager* on page 790.

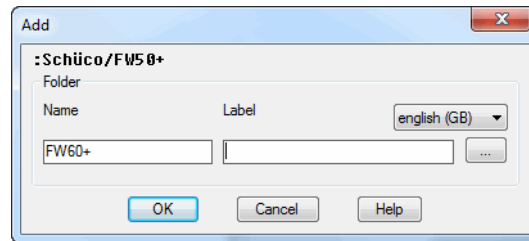
When the profile system option is activated, the Dialog box Add folder is opened, where you can specify the manufacturer and system data.

Designation ...

Opens the Designation dialog box. Here, you can change the designation of a folder or subfolder.

Import ...

Imports a complete project with all jobs and job parts from another drawing. You will find further information on this in the section *Import job* on page 794.

Dialog box Add folder**Dialog box section Profile system****Name**

Defines the name and the language-independent designation of the folder.

Label

Defines the name and the language-dependent designation.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.



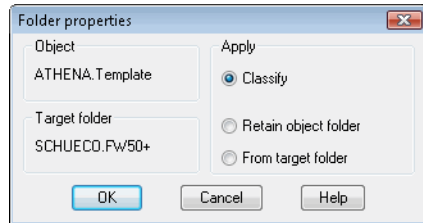
If folders are created in the course of a session and are not filled with objects, they are not retained when the library is restarted. The saving of empty folders is not supported.

2.19 Folder properties

With this function you can decide how object associations are to be dealt with during copying.

Referenced objects must be assigned an association when archived in different folders. You can do this manually during the definition via the function Folder properties or automatically by copying into a folder.

Dialog box Folder properties



Dialog box section Object

Here the existing association of the source object is shown.

Dialog box section Assign to

Here the set destination folder of the library under consideration is displayed.

Dialog box section Apply

Here the options for adoption of the object are given.

Classify

Classifies the object in the folder according to its association. If this is not present, it is automatically created.

Retain object folder

Assigns the object to the target folder and retains its own association.

From target folder

Assigns the object to the target folder and adopts its association.

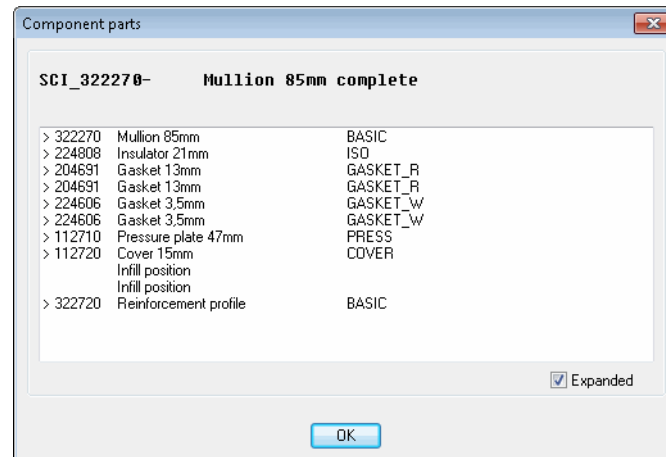
If a number of objects are copied, the option is applied to each object. The selected option becomes effective with the confirmation of the dialog box.

2.20 Info

With this function you can display the content of the marked object or folders in a pick list. The call can take place from various dialog boxes. Examples here are: Assembly library, Assembly catalog and Object selection.

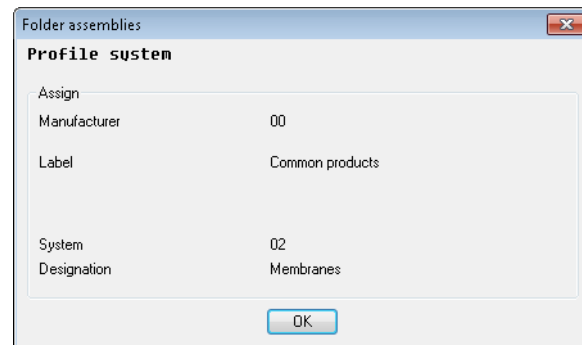
Depending on the selected object, one of the following dialog boxes appears:

Dialog box Component parts



The dialog box lists the component parts included in the selected component with the item number, label and material. Contained references are identified by the symbol > at the start.

Dialog box Assemblies folder

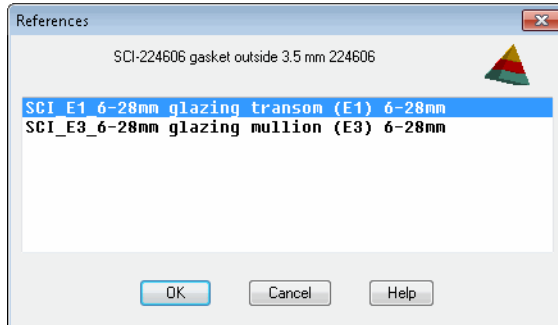


This dialog box shows the structure of the selected folder and informs you whether the folders are assigned to a job or a profile system.

2.21 References

The dialog box shows in which assemblies the current component is present as reference. The call can take place from various dialog boxes. Examples here are: Assembly library.

Dialog box References

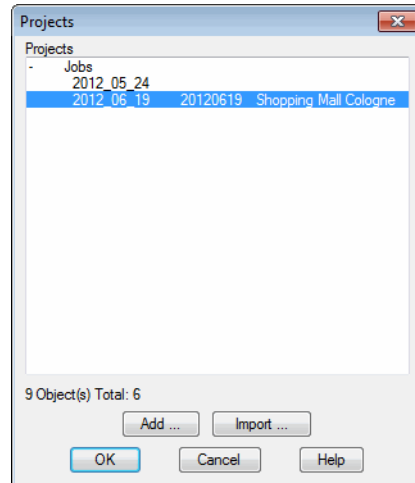


In the upper section the name of the selected component is displayed. The list contains the assemblies in which it is present as reference.

2.22 Projects

This is used for selecting a job or job part. Further actions follow after selecting the job.

Dialog box Projects



In the top section of the dialog box you can see the jobs defined in the drawing. Here, you can select a job.

Below the job list you can see the number of objects assigned to the job.

Add ...

Adds a job part to the selected job structure. For this, the Dialog box Add folder is opened, where you can specify the job data.



A job name must be issued. The name of a job/job part within a project must be unique. For example, if the job name is 001, no job part with the name 001 is admissible.

You will find further information on this in the section *Add job* on page 793.

Import ...

Imports a complete project with all jobs and job parts from another drawing. You will find further information on this in the section *Import job* on page 794.

Cancel

This closes the dialog box without any further action.

OK

Closes the dialog box and uses the data of the selected project for further actions.

F ATHENA Command Reference

This section explains the functions of ATHENA 2014 for general design, mainly in the two-dimensional field.

The chapters are subdivided into functional sections. The functional sections correspond to the submenus in the ATHENA pull-down menu.

1 Drawing

Commands in this section:

- Membrane
- Insulation
- Infill
- Wall
- Facade elevation
- Facade elevation sub-dialog box
- Facade elevation+
- Frame element
- Generating a section from 2D
- Outline
- Grid division
- Infills
- More-centered arch
- Spacer
- Gasket
- Seal
- Hatch
- Standard Part
- Standard parts sub-dialog box
- Edit standard parts
- Screwed joint
- Edit screwed joints
- Semi-finished product
- Drilled hole
- Pipe
- Projection Outline
- Projection objects
- Space projection
- Center lines
- Axis
- Section symbol
- Welded seam
- Welded seam symbol
- Edge symbol
- Surface symbol

1.1 Membrane

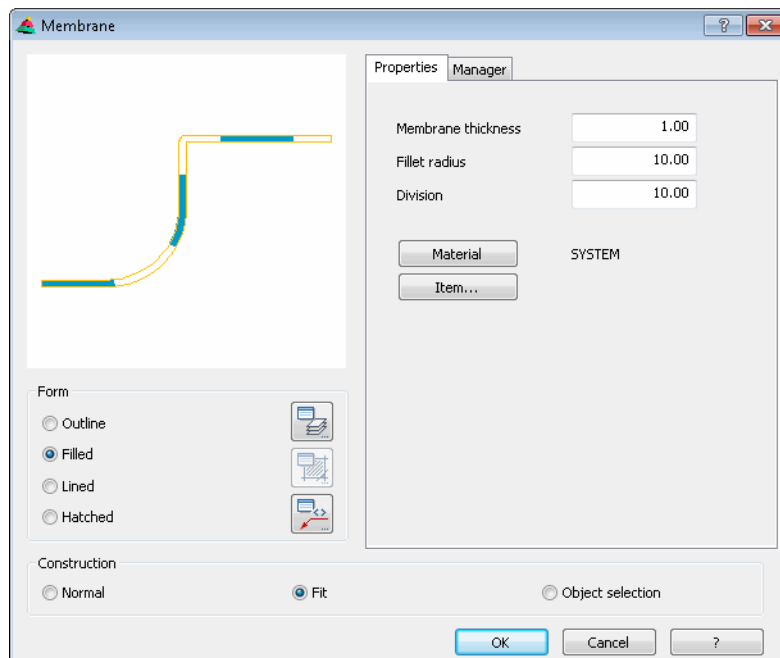


Ribbon: Tab ATHENA > Group Draw > Membrane
Menu: ATHENA > Draw > Membrane
Toolbar: ATH Draw and ATHENA Draw
Command input: ath_folie

With this function you create sealing membranes which can be modified similar to polylines with grips or by stretching. You can create membranes with various possible constructions.

Dialog box Membrane

The dialog box contains the tabs Properties and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Tab Properties

Membrane thickness

Defines the thickness of the membrane.

Fillet radius

Defines the radius to be used by ATHENA for the inside corners of membranes.

Division

Defines the distance between the filled and blank sections of the membrane. This option works in conjunction with the option Filled.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Item

Opens the Dialog box *Item*, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Dialog box section Display**Outline**

Creates a membrane without infill.

Filled

Creates a filled membrane (here the option *Division* is taken into account).

Lined

Produces a lined membrane.

Hatched

Creates a membrane with hatched infill. When you select a material, the material-dependent hatching is used.



Opens the Dialog box *Layer assignment*. You will find further information on this in the Chapter *Layer assignment* on page 108.



Opens the Dialog box *Hatch pattern assignment*. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.



Opens the Dialog box *Label*. You will find further information on this in the Chapter *Label* on page 117.

Dialog box section Construction**Normal**

With the option *Normal* you create a membrane close to reality.

Fit

With the option *Fit* you create a membrane by clicking on snap points.

Object selection

With the option *Object selection* you can create a membrane by selecting a polyline object.



A polyline object may be a polyline or also a Sheet metal section, a Welded seam or a Membrane.

End of program

When you click *OK*, the dialog box is terminated. An input request follows depending on the type of construction selected.



You can terminate the Dialog box *Membrane* directly after calling the command by pressing the *Enter* key to access the input request. In this way you have the possibility of very quickly creating several membranes with identical properties.

E.g.: You have just created a film with certain properties. Now press the *Enter* key twice (once to repeat the *Membrane* command and a second time to terminate the dialog box immediately) and draw a further membrane with the same properties.

Command prompt

Type of construction option *Normal*

Specify starting point of membrane or [?]:

Specify the starting point of the membrane.

You call the help with the option *?*.

Specify direction or [Undo/?]:

Specify the direction of the membrane.

Use the option **Undo** to repeat the previous input request.

Specify next point or [Undo/?]:

Specify the second point of the membrane. If you deviate from the previously specified direction, the membrane is created with a tangential arc to the direction. The membrane runs straight in the specified direction.

Specify side for wall thickness or [Central/Undo/?] <Central>:

Specify on which side the membrane is to be drawn.

Use the option **Central** to draw the membrane centrally to the specified points.

Specify next point or [Undo/?]:

Specify the next membrane point. The last membrane point is always used for the default direction. If you deviate from this direction, the membrane is created with a tangential arc.

This input request is repeated. Press the Enter key to terminate the command.

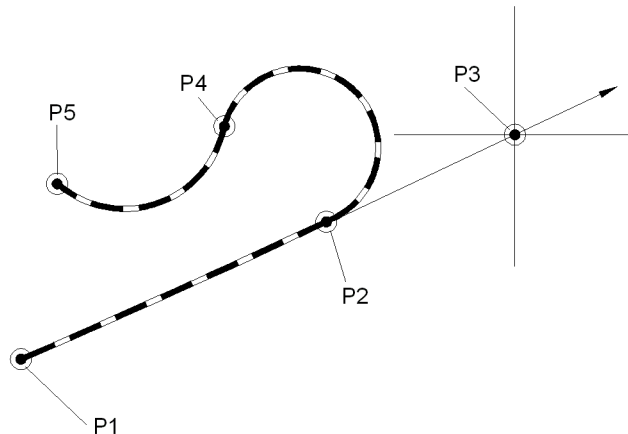


Fig. 1.1: Membrane option Normal

Command prompt

Type of construction option *Fit*

Specify starting point of membrane or [?]:

Specify the starting point of the membrane.

You call the help with the option *?*.

Specify next point or [Arc point/Undo/Direction/Length/?]:

Specify the second membrane point.

Use the option **Undo** to repeat the previous input request.

Specify side for wall thickness or [Central/Undo/?] <Central>:

Specify on which side the membrane is to be drawn.

The wall thickness is produced centrally to the membrane with the option **Central**.

Specify next point or [Arc point/Undo/Direction/ANgle/Length/?]:

Specify the second membrane point.

With the **Option Arc point** you can specify an arc point. Further input requests follow.

With **Option Direction** you can change the direction for the limb. Further input requests follow.

With the **Option Angle** you can specify an angle. Further input requests follow.

With **Option Length** you can specify the length for the limb. Further input requests follow.

This input request is repeated. Press the Enter key to terminate the command.

Specify side for wall thickness or [Central/Same/Undo/?] <Same>:

Specify on which side the membrane is to be drawn.

Press the Enter key to draw the membrane on the side previously specified.

Option Arc point

Specify arc point or [Line/Undo/?]:

Specify an arc point.

With the option **Line** you can draw straight membrane segments.

Specify endpoint of the arc or [Undo/Line/Arc point/?]:

Enter the end point of the arc or select an option.

Option Direction

Specify direction or [Length/Angle/Point/Undo/?]:

Specify the direction of the membrane segment or select an option.

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the membrane segment by pointing with the mouse or entering the length.

Option Angle

Specify angle or [Length/Direction/Point/Undo/?]:

Define the angle relative to the last membrane section by pointing with the mouse or entering the angle or selecting an option.

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the membrane segment by pointing with the mouse or entering the length or selecting an option.

Option Length

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the membrane segment by pointing with the mouse or entering the length or selecting an option.

Specify angle or [Length/Direction/Point/Undo/?]:

Define the angle relative to the last membrane section by pointing with the mouse or entering the angle or selecting an option.

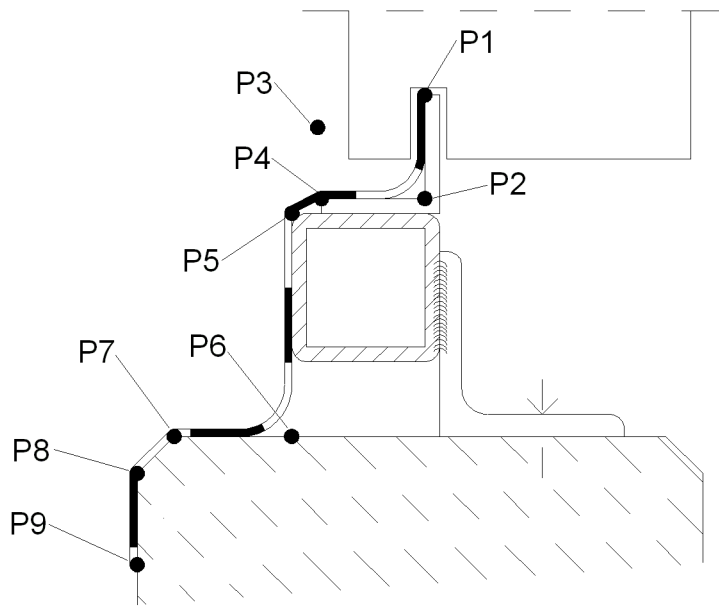


Fig. 1.2: Membrane option Fit

Command prompt

Type of construction option Object selection

Select Polyline or [?]:

Select the polyline object which is to be converted to a membrane.

You call the help with the option ?.

Specify side for wall thickness or [Central/?] <Central>:

Specify on which side the membrane is to be drawn.

The wall thickness is produced centrally to the membrane with the option **Central**.

Delete polyline [Yes/No/?] <Yes>:

With the option **Yes** you delete the existing polyline.

The polyline is retained with the option **No**.

Notes

- You can change a membrane by stretching or with grips. When stretched, the membrane behaves like a polyline.
- You can change membrane properties with the command **Modify ATHENA** or by a double click.
- You can automatically label the membrane with the command **Parts labeling**.
- You can change the layer default settings for the membrane in Dialog box Layer assignment.

1.2 Insulation

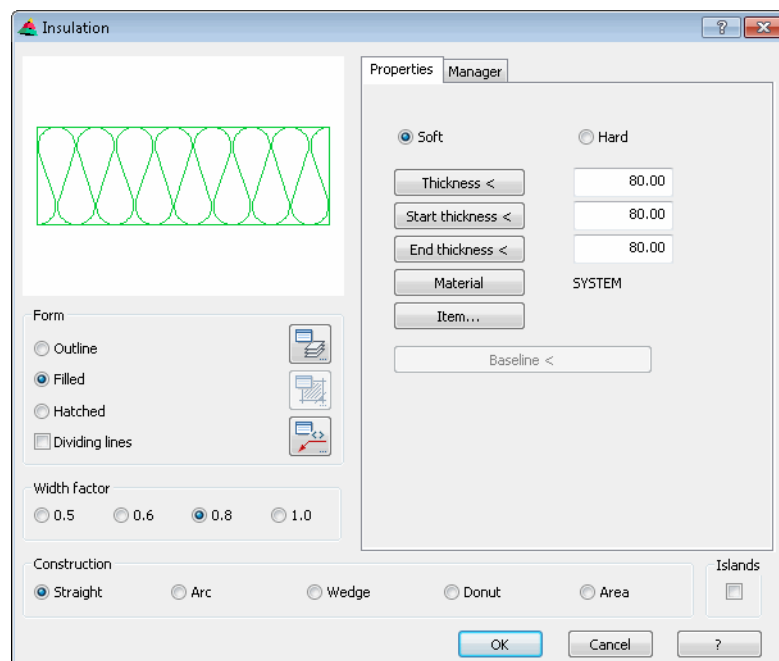


Ribbon: Tab ATHENA > Group Draw > Insulation
Menu: ATHENA > Draw > Insulation
Toolbar: ATH Draw and ATHENA Draw
Command input: ath_dmt0

With this command you create thermal insulation with various shapes and properties.

Dialog box Insulation

The dialog box contains the tabs Properties and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Tab Properties

Soft

Produces soft insulation (round shape).

Hard

Creates hard insulation (polygonal shape).

Thickness

Defines the thickness of the insulating layers. When you click the button Thickness <, you can measure it off in the drawing. The dialog box is temporarily closed and you can define the thickness by clicking two points.

Start thickness

Defines the start thickness of the insulation. When you click the button Start thickness <, you can measure it off in the drawing. The dialog box is temporarily closed and you can define the start thickness by clicking two points.

End thickness

Defines the end thickness of the insulation. When you click the button End

thickness <, you can measure it off in the drawing. The dialog box is temporarily closed and you can define the end thickness by clicking two points.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Baseline

You can change the alignment of flat-shaped thermal insulation with the button Baseline. The button is only active once you have called the dialog box for modifying an insulation.

Dialog box section Display

Outline

Creates an insulation without infill.

Filled

Creates a filled insulation. The infill depends on the options Soft and Hard.

Hatched

Creates insulation with hatched infill. When you select a material, the material-dependent hatching is used.

Dividing lines

Activates dividing lines between the insulation layers.



Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.



Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110. This button is activated when you select the option Hatched.



Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Dialog box section Width factor

Here, you change the displayed width of the insulation from very narrow (0.5) to wide (1.0).

Dialog box section Construction

Straight

Creates straight insulation.

Arc

Creates arc-shaped insulation.

Wedge

Creates wedge-shaped insulation.

Donut

Creates donut-shaped insulation.

Area

Fills a freely shaped area with insulating layers.

Islands

Switches the island detection on or off.

End of program

When you click OK, the dialog box is terminated and an input request follows dependent on the selected type of construction.



You can terminate the Dialog box Insulation directly after calling the command by pressing the Enter key to access the input request. In this way you have the possibility of very quickly creating several insulations with identical properties.

E.g.: You have just created an insulation with certain properties. Now press the Enter key twice (once to repeat the Insulation command and a second time to terminate the dialog box immediately) and draw a further insulation with the same properties.

Command prompt

Straight and wedge types of construction

Specify start point or [Settings/?]:

Specify the starting point of the insulation.

*Use the option **Settings** to run the Dialog box Insulation and to change the insulation properties.*

You call the help with the option ?.

Specify the end point or [Undo/?]:

Specify the end point of the insulation.

*Use the option **Undo** to repeat the previous input request.*

Specify the side on which the insulation is to be drawn [Undo/?]:

Define the side on which the insulation is to be drawn.

Select islands

Select objects:

Select the objects within the insulation to leave the insulation open at these points. This input request only appears when the Islands option has been activated.

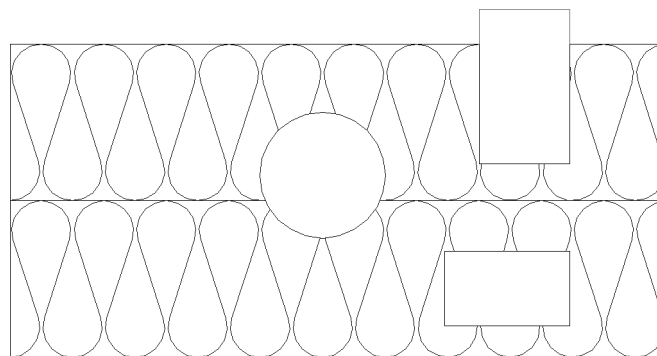


Fig. 1.3: Thermal insulation option Straight (with islands)

Command prompt

Arc type of construction

Specify start point or [Settings/?]:

Specify the starting point of the insulation.

Use the option **Settings** to run the Dialog box *Insulation* and to change the insulation properties.

You call the help with the option **?**.

Specify arc point or [Undo/?]:

Specify an arc point.

Use the option **Undo** to repeat the previous input request.

Specify the end point or [Undo/?]:

Specify the end point of the insulation.

Specify the side on which the insulation is to be drawn [Undo/?]:

Define the side on which the insulation is to be drawn.

Select islands

Select objects:

Select the objects within the insulation to leave the insulation open at these points. This input request only appears when the *Islands* option has been activated.

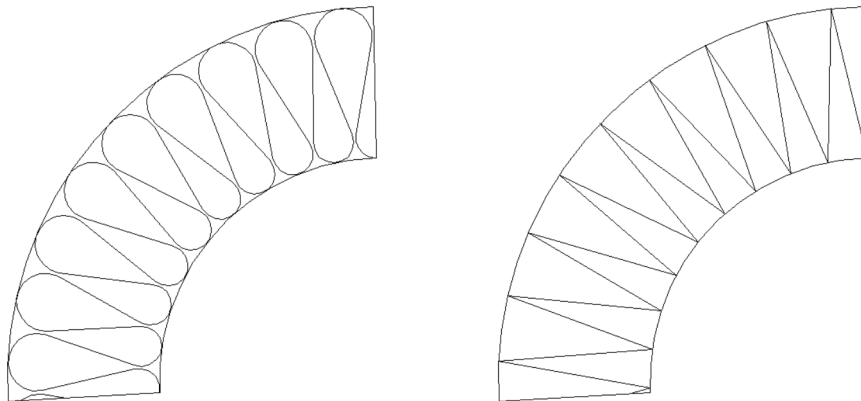


Fig. 1.4: Thermal insulation option Arc

Command prompt

Donut type of construction

Specify center point or [Settings/?]:

Specify the center point of the insulation.

Use the option **Settings** to run the Dialog box *Insulation* and to change the insulation properties.

You call the help with the option **?**.

Specify internal radius or [Undo/?]:

Specify the inner radius of the insulation.

Use the option **Undo** to repeat the previous input request.

Select islands

Select objects:

Select the objects within the insulation to leave the insulation open at these points. This input request only appears when the *Islands* option has been activated.

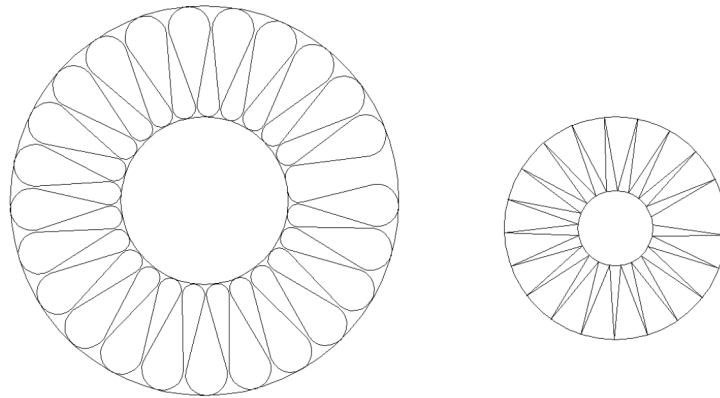


Fig. 1.5: Thermal insulation option Donut

Command prompt

Area type of construction

Select a point in the area or [Settings/Object selection/?]: <Object selection>:

Enter a point within the area to be insulated.

Use the option **Settings** to run the Dialog box Insulation and to change the insulation properties.

Use the **Option Object selection** to select objects.

You call the help with the option ?.

Option Object selection

Select objects:

x object(s) found

Select the objects to be insulated.

Select object as baseline or [Undo/?]:

Click in the vicinity of a line to align the insulation to it.

Use the option **Undo** to repeat the previous input request.

Select islands

Select objects:

Select the objects within the insulation to leave the insulation open at these points. This input request only appears when the Islands option has been activated.

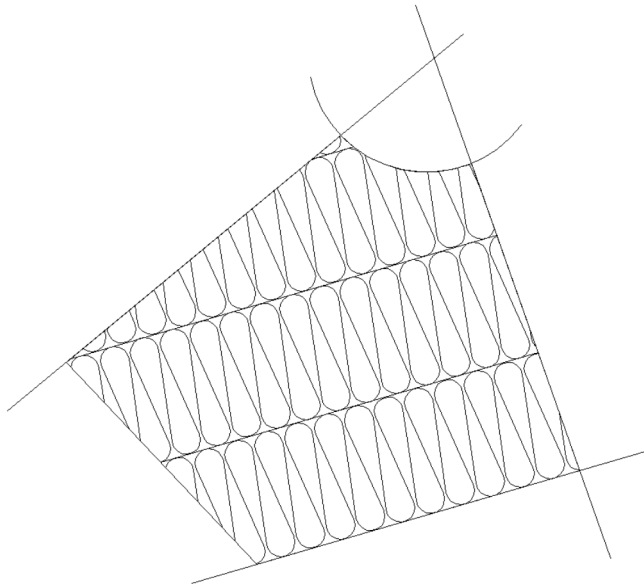


Fig. 1.6: Thermal insulation option Area

Notes

- You can fit insulation to various shapes by stretching or with grips.
- You can change insulation properties with the command **Modify ATHENA** or by a double click.
- You can automatically label the insulation with the command **Parts labeling**.
- You can change the layer default settings for the insulation in Dialog box System layer.

1.3 Infill



Ribbon:	Tab ATHENA > Group Draw > Infill
Menu:	ATHENA > Draw > Infill
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_glpk

With this routine you can create the cross section of an infill. An infill may be glass as well as a panel.

The infill structure is defined in a dialog box. Infills can be saved as library objects.

You will find a detailed description of the dialog box in the chapter *Infill manager* on page 664.

When you click the OK button, the settings in the dialog box are saved and the input request follows:

Command prompt

Option Point

Specify insertion point or [Object/?]:

Use the mouse or enter co-ordinates to specify the insertion point for the infill.

*Select the **Option Object** to append an infill to one or more existing bar cross-sections.*

You call the help with the option ?.

Specify rotated angle<0>:

Enter a rotation angle. Press the Enter key to accept the default angle of 0°.

These two input requests are repeated until you press the Enter key to terminate the command.

Option Object

Select first bar cross-section or [Point/?]:

Select an existing bar cross-section to append the infill.

*Select the **Option Point** to insert an infill at any point.*

Select second bar cross-section or [?]:

Select a further bar cross-section (flush with the one previously selected) to fit the infill between the two bar cross-sections.

If you press the Enter key at this point, the infill is positioned with interrupted representation on the first bar cross-section.

These requests are repeated until you press the Enter key to terminate the command.



The infills are inserted on the Infill position of the previous bar cross-section. If none has been defined, an appropriate notice appears in the command line.

The cross sections of the infills, which you insert with this command, are **not** linked to saved library objects and **cannot** be updated.

Notes

You can label the individual layers retrospectively with the command *Parts labeling*.

Associated commands:

- Infill manager
- Apply infill

- Infill

1.4 Wall

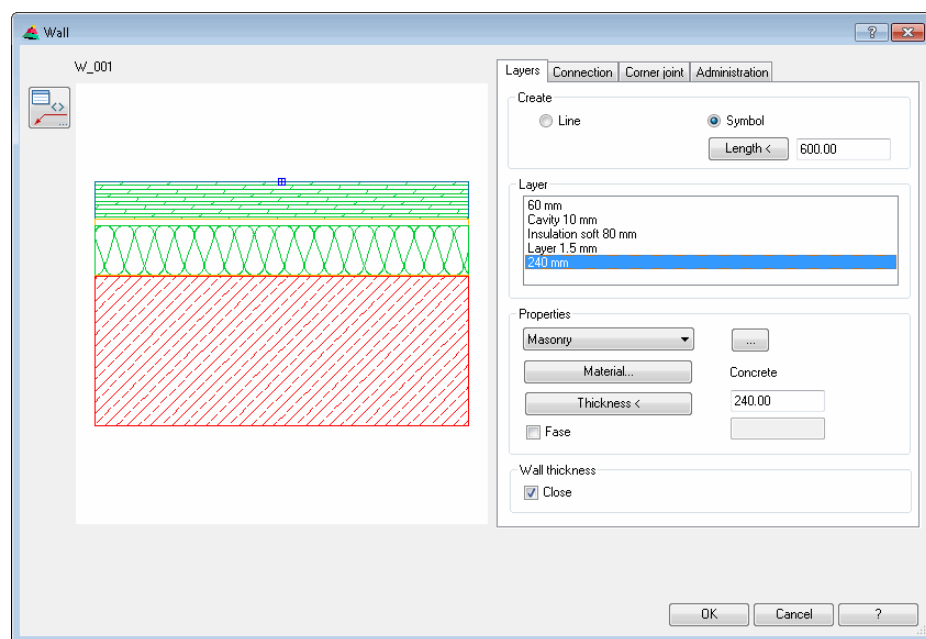


Ribbon: Tab ATHENA > Group Draw > Wall
Menu: ATHENA > Draw > Wall
Toolbar: ATH Draw
Command input: ath_wall

Creates a wall from one or more layers.

The wall structure is defined in a dialog box. Walls can be saved as library objects. When you start the command, the Dialog box Wall manager is started.

Dialog box Wall manager



You will find a detailed description of the dialog box in the chapter *Wall manager* on page 671.

When you terminate the dialog box with OK, an input request follows depending on the type of creation and construction.

Wall as line with Normal type of construction.

Command prompt

Specify starting point of wall or [?]:

Specify the starting point of the wall.

You call the help with the option ?.

Specify next point or [Arc point/Undo/Direction/Length/?]:

Specify the second point of the wall.

Use the option Undo to repeat the previous input request.

Specify side for wall thickness or [Central/Undo/?] <Central>:

Specify on which side of the polyline the wall structure is to be created.

With the option Central the wall structure is positioned on both sides of the polyline.

Specify next point or [Arc point/Undo/Direction/ANgle/Length/?]:

Specify the next point of the wall.

With the **Option Arc point** you can specify an arc point. Further input requests follow.

With the **Option Direction** you can specify the direction for the next wall section. Further input requests follow.

With the **Option Angle** you can specify an angle. Further input requests follow.

With the **Option Length** you can specify the length for the wall section. Further input requests follow.

This input request is repeated. Press the Enter key to terminate the command.

Specify side for wall thickness or [Central/Same/Undo/?] <Same>:

Specify on which side of the polyline the wall structure is to be created.

Press the Enter key to draw the wall on the side previously specified.

Option Arc point

Specify arc point or [Line/Undo/?]:

Specify an arc point.

With the option **Line** you can draw straight wall sections.

Specify endpoint of the arc or [Undo/Line/Arc point/?]:

Enter the end point of the arc or select an option.

Option Direction

Specify direction or [Length/Angle/Point/Undo/?]:

Specify the direction of the wall section or select an option.

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the wall section by pointing with the mouse or entering the length.

Option Angle

Specify angle or [Length/Direction/Point/Undo/?]:

Define the angle relative to the last wall section by pointing with the mouse or entering the angle or selecting an option.

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the wall section by pointing with the mouse or entering the length or selecting an option.

Option Length

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the wall section by pointing with the mouse or entering the length or selecting an option.

Specify angle or [Length/Direction/Point/Undo/?]:

Define the angle relative to the last wall section by pointing with the mouse or entering the angle or selecting an option.

Wall as line with Object type of construction.

Command prompt

Select Polyline or [?]:

Select a polyline to create the wall.

You call the help with the option ?.

Specify side for wall thickness or [Central/?] <Central>:

Specify on which side of the polyline the wall structure is to be created.

With the option **Central** the wall structure is positioned on both sides of the polyline.

Delete polyline [Yes/No/?] <Yes>:

Select the option **Yes** to delete the polylines..

The polyline is retained with the option **No**.

Wall as symbol

Command prompt

Specify insertion point:

Define the insertion point of the wall.

Specify rotated angle<0>:

Specify the rotation angle of the wall or press the Enter key to accept the default angle.



The walls, which are inserted with this command, are **not** linked to saved library objects and **cannot** be updated.

Associated commands

- Wall manager
- Apply wall
- Wall
- Wall parameters

1.5 Facade elevation

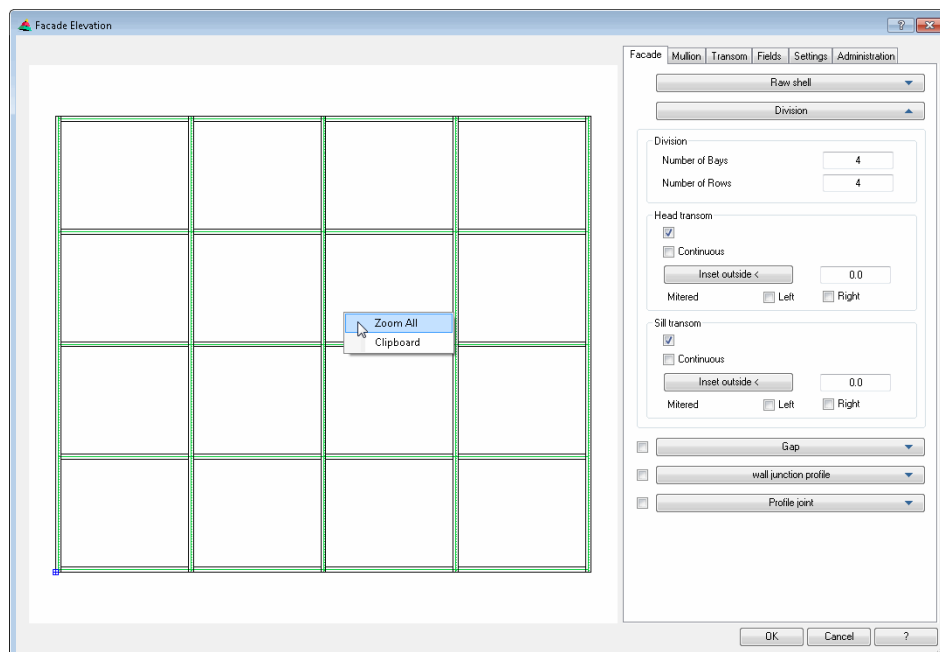


Ribbon: Tab ATHENA > Group Draw > Facade elevation
Menu: ATHENA > Draw > Facade elevation
Toolbar: ATH Draw and ATHENA Functions
Command input: ath_front

Creates a facade elevation.

Span sizes, dimensions, spacing, etc. can be defined in a dialog box. Facade elevations can be saved as library objects.

Dialog box Facade elevation



Display section

On the left side the dialog box contains a preview of the facade elevation. On one hand it provides a visual check and on the other hand offers further functions:

- Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.
- Depending on the active dialog box section, you can select elements of the facade (mullions, transoms or bays) directly in the preview to modify their properties. Selected elements are highlighted in color.

Operating section

On the right side of the dialog box there is the operating section with the tabs:

- Facade
- Mullion
- Transom
- Bays
- Settings
- Manager

The individual tabs in turn contain drop-down menus with the appropriate setting options.

A description of the tabs and drop-down menus is given further below in this section. The Administration section is the same for other objects. You will find a detailed description of this section in the Chapter *Management of objects* on page 134.

Tab Facade

This section contains the following drop-down menus:

- Raw shell
- Division
- Gap
- Wall junction profile
- Profile joint

Drop-down menu Raw shell

Defines the raw shell dimensions.

Width

Defines the width of the raw shell opening. You can write the width directly into the input field or measure off a value from the drawing by clicking the button Width <.

Height

Defines the total height of the raw shell opening. You can write the height directly into the input field or measure off a value from the drawing by clicking the button Height <.

Height left, Height right

Defines the left resp. right raw shell height. You can write the height directly into the input field or measure off a value from the drawing by clicking the button Height left/right <. The entry of the right or left height has the following effects on the shape of the raw shell opening:

- When you specify the Height left **or** right, the raw shell above can be optionally arch-shaped. You control this with the option Arch aligned.
- When you specify the Height left **and** right, the raw shell above is automatically arch-shaped.



The Height left or right must not be greater than the total height of the raw shell.

Arch aligned

With this option you can create a raw shell arch-shaped above.



This option is only available when you have specified Height left **or** Height right.

Rise of the arch

Defines the rise of the arch and creates a raw shell opening which is arch-shaped above. You can write the rise of the arch directly into the input field or measure off a value from the drawing by clicking the button Rise of the arch <. By entering the rise of the arch the Heights left and right are reduced by the entered value. If you deactivate the tick box, the arch above is removed from the raw shell structure.

Drop-down menu Division

Defines the bay division and the properties of the head and sill transoms.

Number of bays

Defines the number of bays in the horizontal direction (columns).

Number of rows

Defines the number of bays in the vertical direction (rows).

Head transom

Switches the top-most transom on or off.

Continuous

Determines whether the head transom is continuous or butts against the mullions.

Inset outside

Specifies the distance of the top edge of the head transom to the top edge of the mullion. You can write the inset directly into the input field or measure off a value from the drawing by clicking the button Height <.



This option is not available when the head transom is continuous.

Mitered left, Mitered right

Results in the head transom being cut for mitering with the left, respectively right mullion.



This option is not available if you have entered an inset for the head transom.

Sill transom

Switches the lowest transom on or off.

Continuous

Determines whether the sill transom is continuous or butts against the mullions.

Inset outside

Specifies the distance of the bottom edge of the sill transom to the bottom edge of the mullion. You can write the inset directly into the input field or measure off a value from the drawing by clicking the button Height <.



This option is not available when the sill transom is continuous.

Mitered left, Mitered right

Results in the sill transom being cut for mitering with the left, respectively right mullion.



This option is not available if you have entered an inset for the sill transom.

Drop-down menu Gap

Gap	
<input checked="" type="checkbox"/>	All equal
Head <	120.0
Sill <	120.0
Left <	80.0
Right <	80.0

The tick box adjacent to the drop-down menu is automatically activated when you define a gap to the raw shell. If you deactivate the tick box, the facade elevation is created without a gap to the raw shell.

All equal

If you activate this tick box, the gap to the raw shell which you have defined for the head, is maintained circumferentially.

Head <, Sill <, Left <, Right <

Specifies the gap from the facade to the raw shell on the relevant sides. You can write the gap directly into the input field or measure off a value from the drawing by clicking the appropriate button.

Drop-down menu Wall junction profile

wall junction profile	
<input checked="" type="checkbox"/>	Alle gleich
Top	<input type="checkbox"/>
Sill	<input type="checkbox"/>
Left	<input checked="" type="checkbox"/>
Right	<input checked="" type="checkbox"/>

Properties Right
 Width: 60.0
 Frame outlet: 30.0
 Depth: 105.0
 Projekt of weather side: 8.0

The tick box adjacent to the drop-down menu is automatically activated when you define a wall junction. If you deactivate the tick box, the facade elevation is created without a wall junction.

Head, Sill, Left, Right

Switches the wall junction profile on or off at the respective point (head, sill, left or right).



You can define wall junction profiles with different dimensions. To do this, mark the appropriate side (head, sill, left or right) and change the dimensions under Properties.

All equal

Accepts the properties for all wall junction profiles.

Width

Specifies the profile width of the junction profile.

Frame reduction

Specifies the reduction dimension for the facade. The facade is reduced by this dimension!

Depth

Specifies the depth of the junction profile.

Weather-side projection

Specifies the projection of the junction profile on the weather side.

Drop-down menu Profile joint

The tick box adjacent to the drop-down menu is automatically activated when you define a profile joint. If you deactivate the tick box, the facade elevation is created without a profile joint.

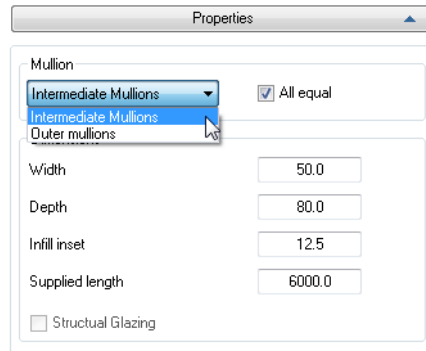
Click in the row to enter the distance. A right click displays a context menu with the option for removing the entered value.

Tab Mullion

This section contains the following drop-down menus:

- Properties
- Axes
- Clearance
- Center distance

Drop-down menu Properties



Determines the mullion properties.

Dialog box section Mullion

Here, you select the posts for which you would like to change the properties.

All equal

Uses the set dimensions for all mullions (intermediate mullions and outer mullions).

Dialog box section Dimensions

Width

Defines the width of the mullion.

Depth

Defines the depth of the mullion.

Infill inset

Defines the infill inset.

Supplied length

Defines the length of the supplied mullion.

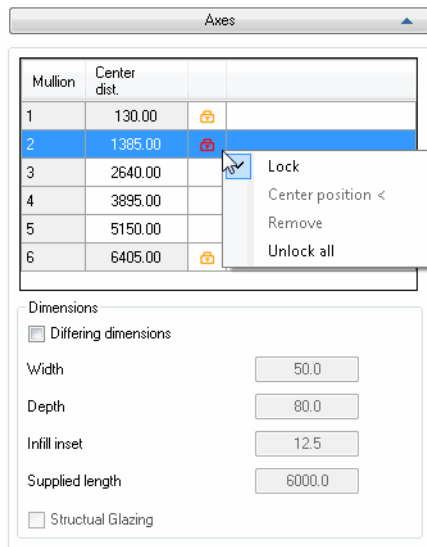
Structural glazing

Activates or deactivates the structural glazing display for all mullions and switches the display of the mullion cover sections on or off.



The structural glazing tick box is only available when the structural glazing tick box has been activated in the Tab Settings under Glazing.

Drop-down menu Axes



Defines the properties of individual mullions and the dimensions of the mullion axes.

Table of mullion axes

In the table the mullions and their center distances are listed. The mullions are numbered from top to bottom. The unit spacings involve the absolute center distances which are each measured from the left edge of the raw shell.

In the table the following additional information is shown:

- When individual mullions are locked, they are identified with a padlock. Directly locked mullions are identified with a red padlock and indirectly locked mullions with a yellow padlock.



Indirectly locked unit spacings arise when you have locked unit spacings in other sections (e.g. Clearance).

The first and last unit spacing is always indirectly locked, because it is defined by the raw shell spacing to the right and left.

- If deviating profile dimensions have been defined, they are marked with an A.

Click a mullion in the table or in the preview to make dimensional changes. The marked mullion is highlighted in color in the preview.

If you click the left mouse key on a dimension, the field is released for editing and you can change the figure.

A right click in the table shows you a context menu with the following command options:

Lock

Locks the marked unit spacing so that it can no longer be changed.

Center position <

Defines a unit spacing by clicking on two points in the drawing. For this the dialog box is temporarily closed.

Remove

Deletes the marked mullion from the list.

Unlock all

Releases the lock of all directly locked mullions.

Dialog box section Dimensions

Differing dimensions

If you activate this tick box, you can change the differing mullion dimensions for the marked axes.

Width

Defines the width of the mullion.

Depth

Defines the depth of the mullion.

Infill inset

Defines the infill inset.

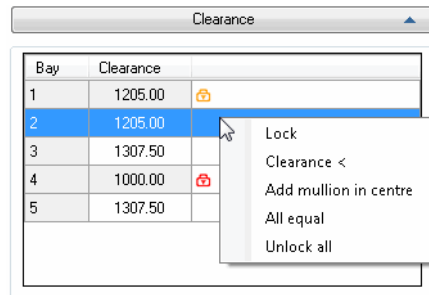
Supplied length

Defines the length of the supplied mullion.

Structural glazing

Activates or deactivates the structural glazing display for individual mullions and switches the display of the respective mullion cover sections on or off.

Drop-down menu Clearance



Defines the clearance mullion dimensions.

Table of clearance dimensions

In the table the bays and their clearance dimensions are shown.

Furthermore, the table also shows whether clearance dimensions are locked or whether they can be changed.

Locked clearance dimensions are identified with a padlock (directly locked clearance dimensions in red, indirectly locked clearance dimensions in yellow).



Indirectly locked clearance dimensions arise when you have locked unit spacings in other sections (e.g. center distances).

If you click the left mouse key on a dimension, the field is released for editing and you can change the figure.

A right click in the table shows you a context menu with the following command options:

Lock

Locks the clearance dimension so that it can no longer be changed.

Clearance <

Defines a clearance by clicking on two points in the drawing. For this the dialog box is temporarily closed.

Add mullion in center

Adds a mullion centrally in the bay.

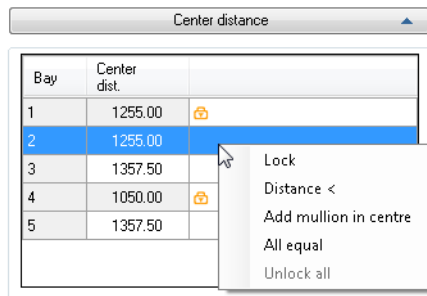
All equal

Sets all unlocked clearance dimensions to the same distance.

Unlock all

Releases the lock of all directly locked clearance dimensions.

Drop-down menu Center distance



Defines the center distances of the mullions.

Table of center distances

In the table the bays and their center distances are shown.

Furthermore, the table also shows whether center distances are locked or whether they can be changed.

Locked distances are identified with a padlock (directly locked clearance dimensions in red, indirectly locked clearance dimensions in yellow).



Indirectly locked center distances arise when you have locked dimensions in other sections (e.g. Clearance).

If you click the left mouse key on a dimension, the field is released for editing and you can change the figure.

A right click in the table shows you a context menu with the following command options:

Lock

Locks the clearance dimension so that it can no longer be changed.

Distance <

Defines a center distance by clicking on two points in the drawing. For this the dialog box is temporarily closed.

Add mullion in center

Adds a mullion centrally in the bay.

All equal

Sets all unlocked center distances to the same dimension.

Unlock all

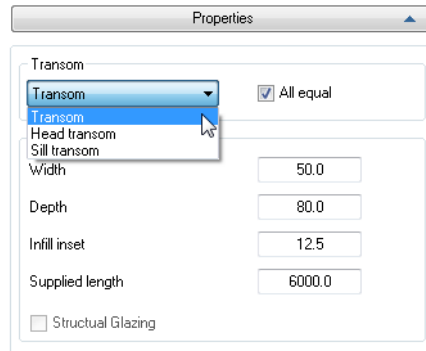
Releases the lock of all directly locked center distances.

Tab Transom

This section contains the following drop-down menus:

- Properties
- Axes
- Clearance
- Center distance

Drop-down menu Properties



Determines the transom properties.

Dialog box section Transom

Here, you select the transoms for which you would like to change the properties.

All equal

Uses the set dimensions for all transoms (head transoms, intermediate transoms and sill transoms).

Dialog box section Dimensions

Width

Defines the width of the transom.

Depth

Defines the depth of the transom.

Infill inset

Defines the infill inset.

Supplied length

Defines the length of the supplied transom.

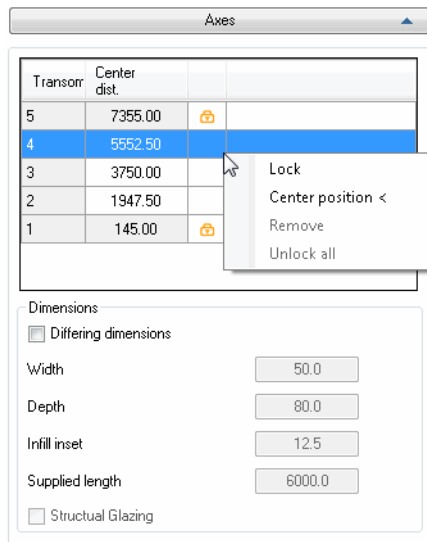
Structural glazing

Activates or deactivates the structural glazing display for all transoms and switches the display of the transom cover sections on or off.



The structural glazing tick box is only available when the structural glazing tick box has been activated in the Tab Settings under Glazing.

Drop-down menu Axes



Defines the properties of individual transoms and the dimensions of the transom axes.

Table of transom axes

In the table the transoms and their center distances are listed. The transoms are numbered from the bottom to the top. The unit spacings involve the absolute center distances which are each measured from the lower edge of the raw shell. In the table the following additional information is shown:

- When individual transoms are locked, they are identified with a padlock. Directly locked transoms are identified with a red padlock and indirectly locked transoms with a yellow padlock.



Indirectly locked unit spacings arise when you have locked unit spacings in other sections (e.g. Clearance).

The first and last unit spacing is always indirectly locked, because it is defined by the raw shell spacing to the right and left.

- If deviating profile dimensions have been defined, they are marked with an A.

Click a transom in the table or in the preview to make dimensional changes. The marked transom is highlighted in color in the preview.

If you click the left mouse key on a dimension, the field is released for editing and you can change the figure.

A right click in the table shows you a context menu with the following command options:

Lock

Locks the marked unit spacing so that it can no longer be changed.

Center position <

Defines a unit spacing by clicking on two points in the drawing. For this the dialog box is temporarily closed.

Remove

Deletes the marked transom from the list.

Unlock all

Releases the lock of all directly locked axes.

Dialog box section Dimensions

Differing dimensions

If you activate this tick box, you can change the differing transom dimensions for the marked transoms.

Width

Defines the width of the transom.

Depth

Defines the depth of the transom.

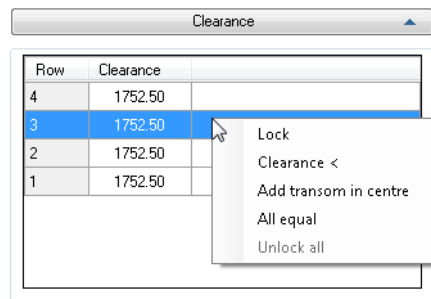
Infill inset

Defines the infill inset.

Supplied length

Defines the length of the supplied transom.

Drop-down menu Clearance



Defines the clearance transom dimensions.

Table of clearance dimensions

In the table the bays and their clearance dimensions are shown.

Furthermore, the table also shows whether clearance dimensions are locked or whether they can be changed.

Locked clearance dimensions are identified with a padlock (directly locked clearance dimensions in red, indirectly locked clearance dimensions in yellow).



Indirectly locked clearance dimensions arise when you have locked unit spacings in other sections (e.g. center distances).

If you click the left mouse key on a dimension, the field is released for editing and you can change the figure.

A right click in the table shows you a context menu with the following command options:

Lock

Locks the clearance dimension so that it can no longer be changed.

Clearance <

Defines a clearance by clicking on two points in the drawing. For this the dialog box is temporarily closed.

Add transom in center

Adds a transom centrally in the bay.

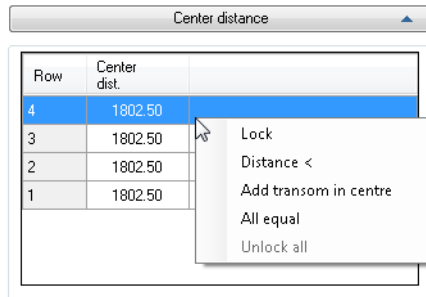
All equal

Sets all unlocked clearance dimensions to the same distance.

Unlock all

Releases the lock of all directly locked clearance dimensions.

Drop-down menu Center distance



Defines the center distances of the transoms.

Table of center distances

In the table the bays and their center distances are shown.

Furthermore, the table also shows whether center distances are locked or whether they can be changed.

Locked distances are identified with a padlock (directly locked clearance dimensions in red, indirectly locked clearance dimensions in yellow).



Indirectly locked center distances arise when you have locked dimensions in other sections (e.g. Clearance).

If you click the left mouse key on a dimension, the field is released for editing and you can change the figure.

A right click in the table shows you a context menu with the following command options:

Lock

Locks the clearance dimension so that it can no longer be changed.

Distance <

Defines a center distance by clicking on two points in the drawing. For this the dialog box is temporarily closed.

Add transom in center

Adds a transom centrally in the bay.

All equal

Sets all unlocked center distances to the same dimension.

Unlock all

Releases the lock of all directly locked center distances.

Tab Bays

This section contains the following drop-down menus:

- Insert
- Knee wall
- Headwall

Drop-down menu Insert

The tick box adjacent to the drop-down menu is automatically activated when you define an insert. If you deactivate the tick box, the facade elevation is created without an insert.

To define an infill or a frame element you must click a bay in the preview. The selected bay is marked in red in the preview and can be appropriately extended by entering numerical values in the input fields Horizontal and Vertical.

When you click a bay where the adjacent bay has already been defined, a dialog box appears with the following options:

New

Defines a new bay.

Join

Joins the bay to the adjacent bay which has already been defined.

Move

Moves the previously defined bay to the selected position.

Dialog box section Bays

Horizontal

Joins the entered number of bays from left to right.

Vertical

Joins the entered number of bays from bottom to top.

Head transom

Switches the head transom on. This option is only available if the bay borders the head transom.

Sill transom

Switches the sill transom on. This option is only available if the bay borders the sill transom.

Door stud

Removes the sill transom below the bay. This option is only available if the bay borders the sill transom.



The door stud also continues if a continuous sill transom has been defined.

Transom continuous

Allows the top and bottom delimiting transoms of the bay to continue.

Infill

Defines the bay as an infill type. You can freely define infill types by entering an infill name into the field or selecting infills already defined from the menu.



The infill type is displayed on the label symbol in the appropriate bay depending on the label setting.

Element

Inserts a frame element into the selected bay. Click the button [...] to define further properties of the frame element in the Dialog box Frame Element Manager.

Multiple

Starts the Dialog box Infill (multiple) where you can insert infills or frame elements in several bays. You will find further information on this in the section *Infill* on page 208.

Remove

Deletes the selected infill or the selected frame element.

Reset all

Deletes all infills or all frame elements.

Drop-down menu Knee wall

The tick box adjacent to the drop-down menu is automatically activated when you define a knee wall. If you deactivate the tick box, the facade elevation is created without a label.



With knee walls the sill transom is omitted.

To define a knee wall you must click a bay in the preview. The knee wall is marked in red in the preview and can be appropriately extended by entering numerical values in the input fields Horizontal and Vertical.

When you click a bay where the adjacent bay has already been defined, a dialog box appears with the following options:

New

Defines a new bay.

Join

Joins the bay to the adjacent bay which has already been defined.

Move

Moves the previously defined bay to the selected position.

Dialog box section Bays

Horizontal

Joins the entered number of knee walls from left to right.

Vertical

Joins the entered number of knee walls from bottom to top.

Remove

Deletes the selected knee wall.

Drop-down menu Headwall

The tick box adjacent to the drop-down menu is automatically activated when you define a headwall. If you deactivate the tick box, the facade elevation is created without a label.



With a headwall the head transom is not needed.

To define a headwall you must click a bay in the preview. The headwall is marked in red in the preview and can be appropriately extended by entering numerical values in the input fields Horizontal and Vertical.

When you click a bay where the adjacent bay has already been defined, a dialog box appears with the following options:

New

Defines a new bay.

Join

Joins the bay to the adjacent bay which has already been defined.

Move

Moves the previously defined bay to the selected position.

Dialog box section Bays

Horizontal

Joins the entered number of knee walls from left to right.

Vertical

Joins the entered number of knee walls from top to bottom.

Remove

Deletes the active headwall.

Tab Settings

This section contains the following drop-down menus:

- Glazing
- Calculation
- Display
- Dimension
- Label

Dialog box section Glazing

Clamping

Defines the clamping thickness.

Height of cover

Defines the height of the cover.

Structural glazing

Activates the structural glazing display for mullions and transoms and switches the display of the covers off.



The settings defined in the glazing section have no effect on the inserted cross-section. These only affect

Drop-down menu Calculation

Specifies how the mullions or transoms are to be distributed.



This is a default setting which only becomes effective when the profile widths of the mullions or transoms are different.

The setting is only effective when you modify the horizontal and vertical division on the Tab Facade in the Drop-down menu Division.

Clearance

Calculates the mullion or transom division according to the clearance dimension.

Center distance

Calculates the mullion or transom division according to the center distance.

Drop-down menu Display

Defines how the facade elevation is inserted into the drawing.

Dialog box section Elevation

Center lines

Creates the profile axes for the output.

Profile

Creates the profile views for the output.

Infill

Creates the infills for the output.

Raw shell

Creates the raw shell outline for the output.

Hidden profiles

Draws the profiles in hidden representation.



This option is only available for structural glazing.

Horizontal section

Creates the horizontal section for the output. This is drawn below the facade elevation.

Vertical section

Creates the vertical section for the output. This is drawn to the right of the facade elevation.

Layer ...

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Dialog box section T-butt joint**Selection menu**

Specifies the transom for which a joint seam is to be defined.

All equal

Uses the same size of joint seam for all transoms.

Joint seam

Specifies the size of the joint seam.

Dialog box section I-butt joint**Selection menu**

Specifies the mullion for which a joint seam is to be defined.

All equal

Uses the same size of joint seam for all mullions.

Joint seam

Specifies the size of the joint seam.

Drop-down menu Dimension

	Dimension style	Suffix
<input type="checkbox"/> Profile Width	ATHENA	
<input type="checkbox"/> Clearance dimension	ATHENA	
<input checked="" type="checkbox"/> Axes	ATHENA	
<input type="checkbox"/> Axis->Raw-shell dimension	ATHENA	
<input type="checkbox"/> Frame dimension	ATHENA	
<input checked="" type="checkbox"/> Raw-shell dimension	ATHENA	

The tick box adjacent to the drop-down menu is automatically activated when you change dimension settings. If you deactivate the tick box, the facade elevation is created without dimensions.

The actual text height of the dimensions which is given by the current dimension settings, is shown above the control elements.

Profile width

Dimensions the elevation width of the profiles.

Clearance dimension

Dimensions the clearance width between the axes.

Center lines

Dimensions the profile axes.

Axis -> Raw-shell dim.

Dimensions the profile axes for the raw shell.

Frame dimension

Dimensions the outer dimensions of the facade elevation.

Raw-shell dimension

Dimensions the raw shell.

Dimension style

Activates the use of a separate dimension style for the dimension option.

If the tick box has been activated, you can select a style for the dimension from the list. Otherwise the current dimension style is used.

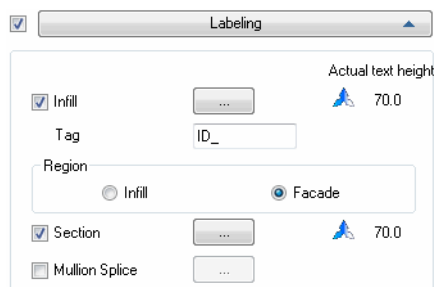
Suffix

Defines an additional text as suffix for the dimension.

Settings

Opens the Dialog box Dimension settings where you can carry out further settings for positioning the dimensions and dimension texts. You will find further information on this in the section *Dimension settings* on page 208.

Drop-down menu Label



The tick box adjacent to the drop-down menu is automatically activated when you make label settings. If you deactivate the tick box, the facade elevation is created without a label.

Infill

Activates labeling of the infill. If you click the button [...], you can adapt the format properties of the label symbol in a dialog box. You will find further information in the Chapter *Leader* on page 419.

Tag

Defines the tag for the label text. The tag is displayed as an additional constituent part of the label.



You have to define infills in the fields so that the tag is displayed for the label. The tag is automatically incremented for different infill types and infill dimensions.

Infill

Causes tags to be separately incremented for different infill types.

facade

Causes tags to be incremented for the complete facade element despite different infill types.

Section

Activates the generation of the section symbols. If you click the button [...], you can adapt the format properties of the section symbol in a dialog box. You will find further information in the Chapter *Section symbol* on page 283.

Mullion splice

Activates labeling of the mullion splice. If you click the button [...], you can adapt the format properties of the splice symbol in a dialog box. You will find further information in the Chapter *Leader* on page 419.

End of program**OK**

Saves the entries and displays an input request to insert the facade elevation into the current drawing.

Cancel

Terminates the dialog box without saving the changes.

Command prompt

Specify insertion point or [?]:

Use the mouse or enter coordinates to specify the insertion point of the facade elevation.

You call the help with the option ?.

Notes

- Facade elevations can be modified with a double click.
- Generation of the section is possible with the command *Generating a section from 2D*. Simplified profiles are produced with a rectangular shape. You will find further information in the section *Generating a section from 2D* on page 219.

1.6 Facade elevation sub-dialog box

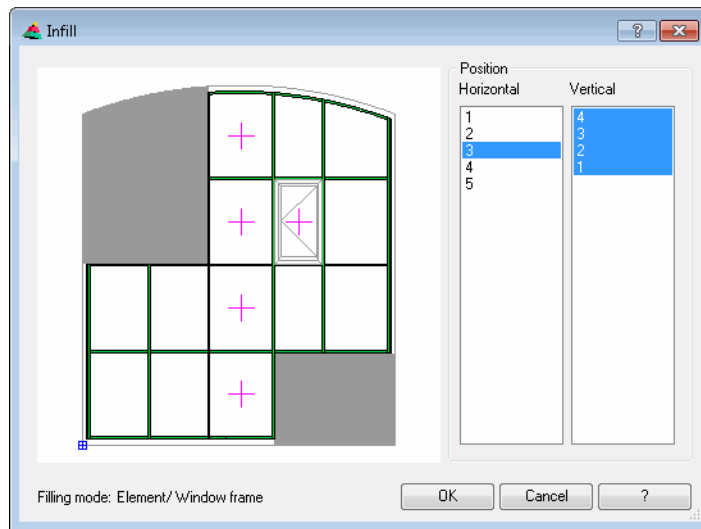
This section describes all sub-dialog boxes which are called exclusively from the facade elevation.

- Infill
- Dimension settings

1.6.1 Infill

In this dialog box you can assign infills to many fields simultaneously.

Dialog box Infill (multiple)



Horizontal

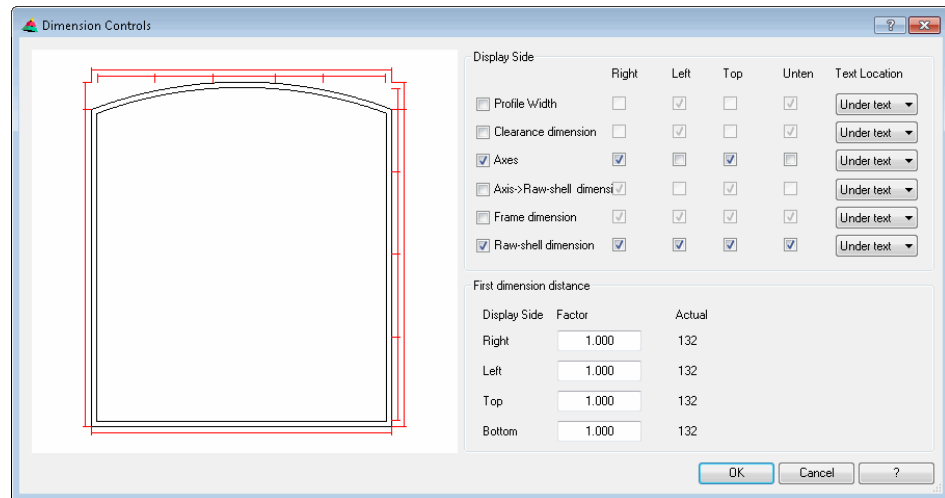
Shows the column numbers from left to right. Here, select the column(s) to which you want to apply infills.

Vertical

Shows the row numbers from top to bottom. Here, select the row(s) to which you want to apply infills.

1.6.2 Dimension settings

In this dialog box you can define additional dimension settings for the facade elevation.

Dialog box Dimension settings**Dialog box section Page**

Right, Left, Top, Bottom

Controls on which side the dimension chain is produced.



Right, Left is used for vertical dimension chains (transom dimensions) and Top, Bottom for horizontal dimension chains (mullion dimensions).

Text location

Specifies the position of the optionally defined addition text.

You can position this text before the dimension figure (prefix), after the dimension figure (suffix) and below the dimension figure.

Dialog box section First dimension distance

Right, Left, Top, Bottom

Defines the baseline distance of the first dimension chain to the facade elevation.

In the input field for the relevant side specify the desired factor to change the distance. The default factor (1.0) causes the baseline distance of the first dimension chain to the facade elevation to be equal to the baseline distance of the dimension chains one to another.



You can set the dimension distance in the Dialog box ATHENA options.

The first dimension distance is multiplied by the dimension distance set in the ATHENA options:

Dimension distance from ATHENA options =
factor*dimscale*(dimgap*2+dimtxt)

First dimension distance = factor*dimension distance from
ATHENA options

The actual distance is shown in the Actual column.

1.7 Facade elevation+



Ribbon: Tab ATHENA > Group > Draw
Menu: ATHENA > Draw > Facade elevation+
Toolbar: ATH Draw
Command input: ath_front+

Applies bar assemblies (mullions, transoms, glazing) and infills (panes, panels) to axes and fields of an existing Facade elevation.

Thus, you can analyze the elements of the facade elevation and generate sections.

The facade elevation must first be created with the command of the same name. You will find further information in the Chapter *Facade elevation* on page 188.

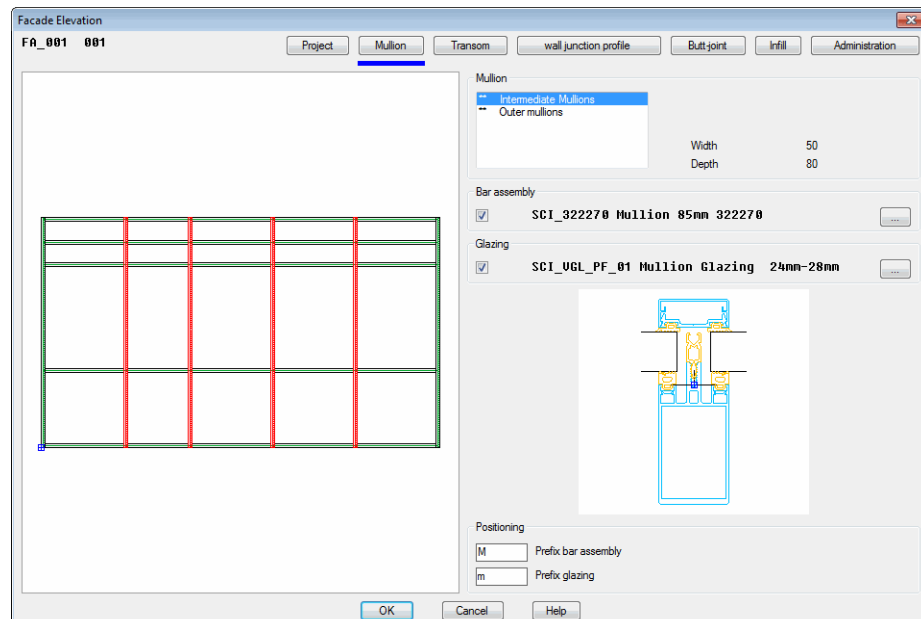
When you execute the command, the following appears:

Command prompt

Select facade elevation or [?]:

Select an existing facade elevation. Once this has taken place, a dialog box appears where you can define the assignments for the various constituent parts of the facade elevation.

Dialog box Facade elevation+



Display section

On the left side the dialog box contains a preview of the facade elevation, providing a visual check. Here, for example, the active profiles are displayed in color.

Operating section

On the right side of the dialog box there is the operating section with the register buttons:

- Project
- Mullion
- Transom
- Junction profile
- Joint component
- Infill

A description of the register buttons can be found further below in this section. The Administration section is the same for other objects. You will find a detailed description of this section in the Chapter *Management of objects* on page 134.

Register button Project

Dialog box section Assign to

Here, the project data of the facade elevation are defined. If you want to analyze the facade elevation, you must assign it to a job. An assignment is only possible if the facade elevation has been assigned a name or an article number in the Administration section.

If you mark an assignment in the list, the job data are shown further below in the dialog box.

Add

Opens the Dialog box Projects, where you can select the project to which the facade elevation is to be assigned. You will find further information on this in the Chapter *Projects* on page 167.

Remove

Deletes the marked project. The assignment of the facade elevation to this project is thus removed.

Dialog box section Bound

Job data

Displays the job data: job, offer and job part for information.

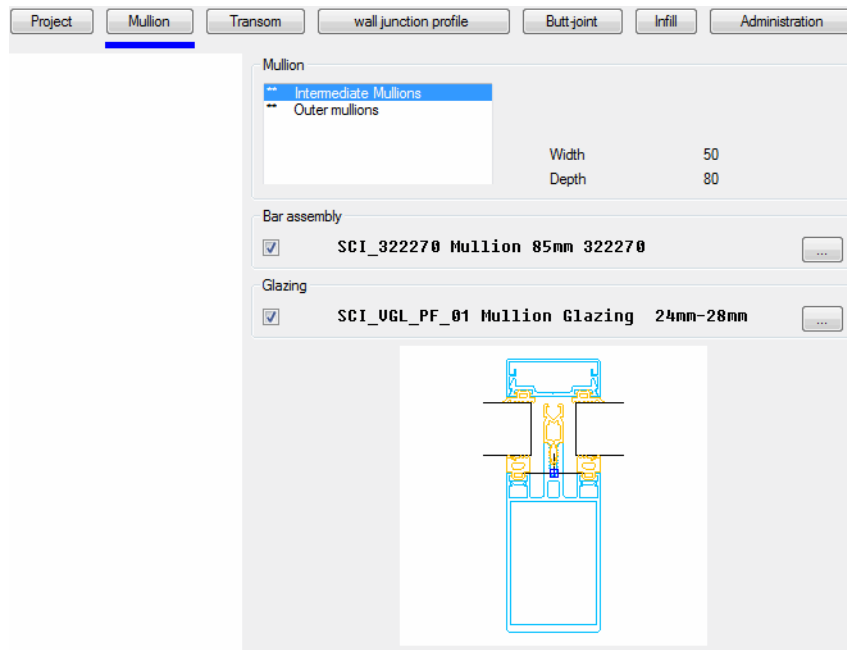
Position

Defines the position number (designation) of the facade elevation.

Quantity

Defines the quantity of the facade elevations (elements) to be produced.

Register button Mullion



Applies bar assemblies and glazings to the mullions.

Dialog box section Mullion

Here, the various types of mullion are listed. Generally, with the facade elevation a differentiation is made between outer mullions and intermediate mullions. If you have defined mullions with different dimensions in the Facade elevation, they are also displayed as separate types of mullion and are designated from the left starting with mullion 0.

Dialog box section Bar assembly

Activate the tick box to assign a bar assembly to the marked mullion. To do this, the Dialog box For object selection is opened.

[...]

Opens the Dialog box For object selection, where you can select a bar assembly to assign it to the marked mullion in the facade elevation.

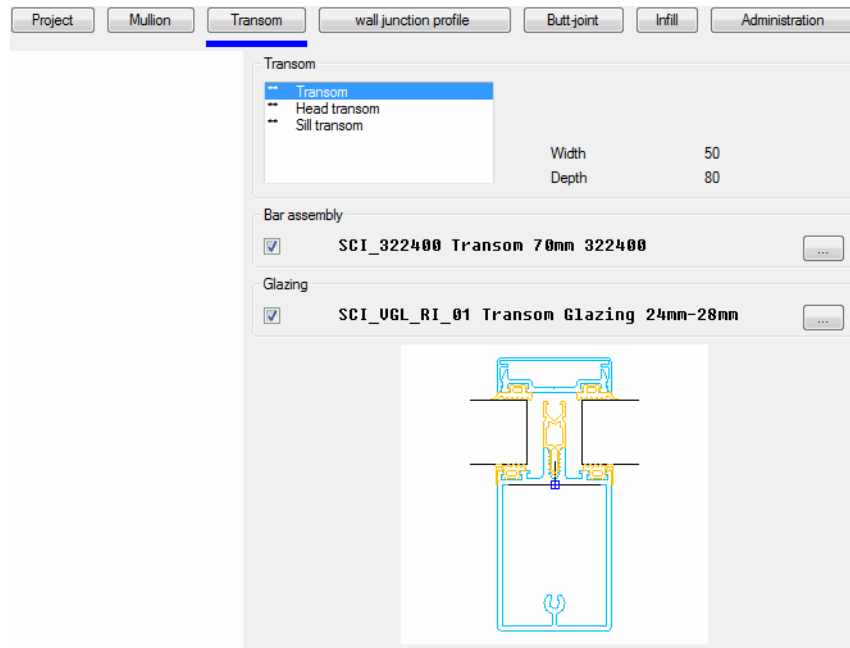
Dialog box section Glazing

Activate the tick box to assign a glazing to the marked mullion. To do this, the Dialog box For object selection is opened.

[...]

Opens the Dialog box For object selection, where you can select a glazing to assign it to the marked mullion in the facade elevation.

A preview of the selected bar assembly and glazing is displayed in the lower section of the dialog box.

Register button Transom

Applies bar assemblies and glazings to the transoms.

Dialog box section Transom

Here, the various types of transom are listed. Generally, with the facade elevation a differentiation is made between head transoms, sill transoms and transoms (intermediate transoms). If you have defined transoms with different dimensions in the Facade elevation, they are also displayed as separate types of transom and are designated from the bottom starting with transom 0.

Dialog box section Bar assembly

Activate the tick box to assign a bar assembly to the marked transom. To do this, the Dialog box For object selection is opened.

[...]

Opens the Dialog box For object selection, where you can select a bar assembly to assign it to the marked transom in the facade elevation.

Dialog box section Glazing

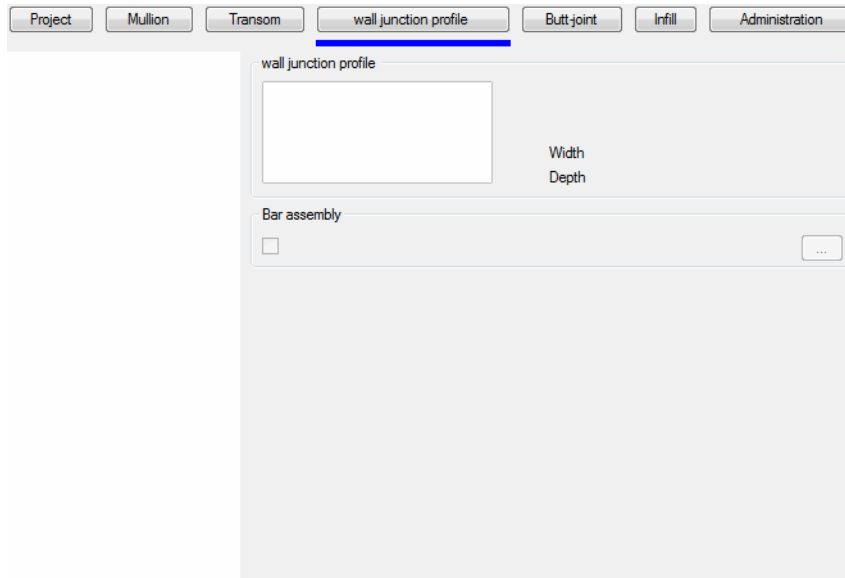
Activate the tick box to assign a glazing to the marked transom. To do this, the Dialog box For object selection is opened.

[...]

Opens the Dialog box For object selection, where you can select a glazing to assign it to the marked transom in the facade elevation.

A preview of the selected bar assembly and glazing is displayed in the lower section of the dialog box.

Register button Junction profile



Applies bar assemblies to the wall junction profiles (if present).



This dialog box section is only active if you have defined wall junction profiles in the Facade elevation.

Dialog box section Wall junction profile

Here, wall junction profiles are listed. There are up to four different wall junction profiles (top, bottom, left, right).

Dialog box section Bar assembly

Activate the tick box to assign a bar assembly to the marked wall junction profile. To do this, the Dialog box For object selection is opened.

[...]

Opens the Dialog box For object selection, where you can select a bar assembly to assign it to the marked wall junction profile in the facade elevation.

A preview of the selected bar assembly is displayed in the lower section of the dialog box.

Register button Joint component

Defines the bar joints of the mullions and transoms in the junctions.

You will find a comprehensive description of the dialog box elements in the chapter *Bar joint manager* on page 709.

Register button Infill

Assigns infills to the bays of the facade elevation.



This dialog box section is only active if you have defined infills as frame elements in the Facade elevation.

Dialog box section Infill

Here, the various types of infill are listed. The number of different types depends on the definition in the facade elevation.

Dialog box section Assign

Activate the tick box to assign an infill to the marked infill type. To do this, the Dialog box Apply infill is opened.

End of program

OK

Saves the entries and assigns the allocated mullions, transoms, wall junction profiles, joints and infills to the constituent parts of the facade elevation.

Cancel

Terminates the dialog box without saving the changes.

1.8 Frame element



Ribbon:	Tab ATHENA > Group Draw > Frame element
Menu:	ATHENA > Draw > Frame element
Toolbar:	ATH Draw and ATHENA Functions
Command input:	ath_elem

Creates a window or door elevation.

Once you have specified the dimensions of the element in the drawing, you can define its properties in a dialog box. Frame elements can be saved as library objects.

When you start the command, the following appears:

Command prompt

Option Area

Specify point in the area or [Corner points/Object selection/?]:

Enter a point within a closed area to specify the element outline.

*Select the **Option Corner points** to specify the corner points of the element.*

*Use the **Option Object selection** to select an object.*

You call the help with the option ?.

Option Corner points

Specify corner point or [Area/Object selection/?]:

Specify the first corner point of the element.

*Select the **Option Area** to specify a point in a closed area.*

Specify corner point or [Area/Object selection/Undo/?]:

Specify the second corner point of the element or select an option.

*Choose the option **Undo** to repeat the last input request.*

Specify corner point or [Area/Object selection/Undo/?]:

Specify the third corner point of the element or select an option.

Specify corner point or [Area/Object selection/Undo/Close/?] <Close>:

Enter the fourth corner point or select an option.

*Input **ENTER** for Close to close the outline.*

Option Object selection

Select object:

Select a closed polyline or lines which form an outline and the end points of which touch in order to define the element outline.

Once you have defined the outline for the window or door element by one of the methods described above, ATHENA starts the Dialog box Frame Element Manager. You will find a description of the functions of the dialog box in the Chapter *Frame Element Manager* on page 676.

End of program

When you terminate the dialog box with OK, the element with the defined properties is created in the specified area.

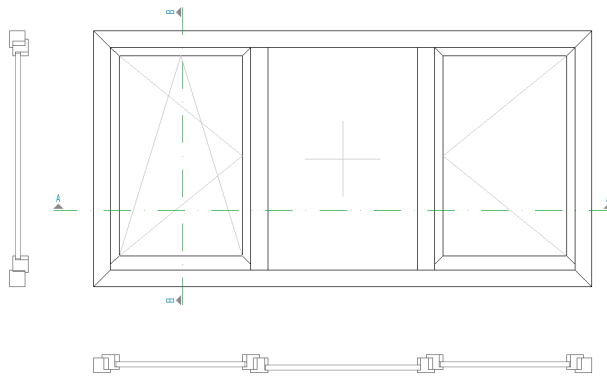


Fig. 1.7: *Window element*

Notes

- The element elevation is an ATHENA object. You can modify dimensions by stretching.
- You can change the element properties with the command **Modify ATHENA** or with double-click editing.
- With the command **Parts labeling** you can label an element with a leader.
- Generation of the section is possible with the command **Generating a section from 2D**. Simplified profiles are produced. You will find further information in the section *Generating a section from 2D* on page 219.

1.9 Generating a section from 2D



Ribbon:	Tab ATHENA > Group Draw > Generate section from 2D
Menu:	ATHENA > Draw > Generate section from 2D
Toolbar:	ATH Draw
Command input:	ath_elem_cs

Generates a section through a facade elevation or a frame element.

In the selected view the section line is drawn as a Section symbol. Along with the section, the section designation is inserted. The section is associatively linked to the selected view.



If you delete the section symbol or the section designation, the section loses its associative link to the view.

When you start the command, the following appears:

Command prompt

Select element or facade elevation or [Settings/?]:

Select an element to generate a section from it.

Select the Option Settings to change the settings of the section symbol.

You call the help with the option ?.

Specify first point of the section line:

Use the mouse or enter coordinates to specify the starting point of the section line.

Specify second point:

Use the mouse or enter coordinates to specify the end point of the section line.

Confirm direction [Invert/?]:

Press the Enter key to adopt the stated direction.



The orientation of the section occurs in dependence of the direction (note the arrow direction of the section symbol) and the element view (viewed from outside or inside).

With the option Invert you modify the viewing direction of the section.

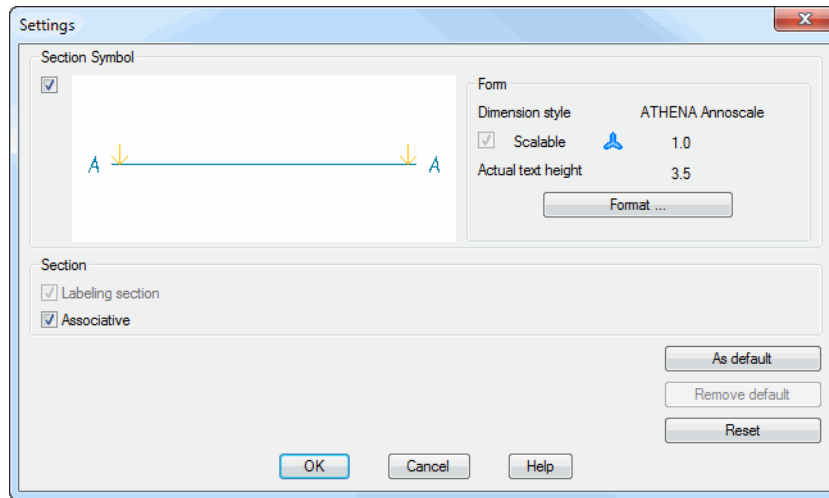
Position section:

The section is now attached to the cross-hair for you to define the insertion point of the section.

Option Settings

Opens the Dialog box Settings.

Dialog box Settings



Dialog box section Section symbol

Switches the section symbol on or off.

Dialog box section Display

Shows the current settings for the section symbol.

Format

Opens the Dialog box Section symbol, where you can configure the symbol settings. You will find further information on this in the section *Section symbol* on page 283.

Dialog box section Section

Label section

Inserts the section designation (e.g. A-A) next to the section.



This option is needed with associative sections and cannot be deactivated as long as Associative is ticked.

Associative

Creates a section which is associatively linked to the selected view.

As default

Saves the current settings as user defaults.

Remove default setting

Removes the user defaults and restores the system defaults. The button is inactive when no user defaults have been saved.

Reset

Restores the default settings (system defaults or user defaults) in the dialog box.

The Dialog box Settings is terminated with OK and the input request follows again.

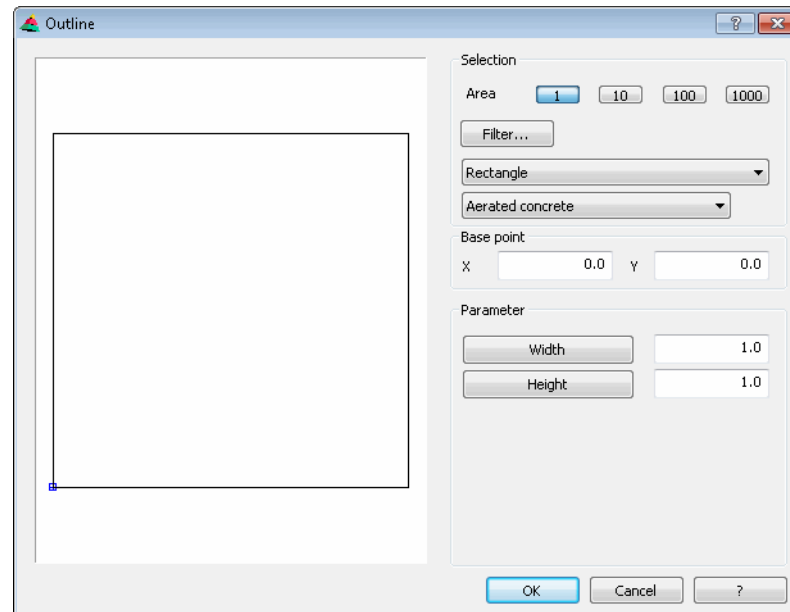
1.10 Outline



Ribbon: Tab ATHENA > Group Draw > Outline
Menu: ATHENA > Draw > Outline
Toolbar: ATH Draw
Command input: ath_outline

With this function you can describe fundamental shapes and insert them into the drawing as closed polylines.

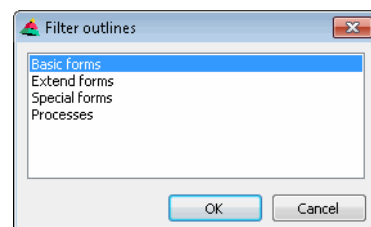
Dialog box Outline



Dialog box section Selection

In this dialog box section you specify the initial shape of the outline. The pull-down list shows you all available basic shapes from which you can choose one.

The button Filter opens a separate dialog box in which you can restrict the selection in the pull-down list. The list shows groupings under which basic shapes are brought together. Mark one or more groups with the aid of CTRL or SHIFT. All marked groups are made available in the pull-down list.



The buttons for the sections 1, 10, 100 and 1000 provide separate memory areas for the describing parameters for a basic shape. Each section is initialized as standard with its correspondingly specified size. The magnitude of the values in the sections is not though prescribed.

The four sections can thus hold different parameter group details for each basic shape. This simplifies the multiple application.

Dialog box section Base point

In this dialog box section you can adapt the insertion point of the outline. You enter positive or negative values according to your required displacement. The displacement of the base point is retained for each basic shape.

Dialog box section Parameter

In this dialog box section you can adapt the selected basic shape via your descriptive parameters. Each shape has its own extent of parameters. With some basic shapes the parameter designations in the dialog box section are buttons which permit optionally different descriptions. These buttons are visually highlighted for identification.

When you click OK, the dialog box is closed and the following input request appears:

Command prompt

Specify insertion point:

The outline is held on the crosshair. Now use the mouse or enter coordinates to specify the insertion point of the outline.

Specify rotated angle<0>:

Define the rotation angle of the outline. You can accept the default angle with Enter or a right click.

Notes

This dialog box is also called by other parts of the program. Therefore it is not always essential to insert the described outline into the drawing. Where applicable, you are returned to the original dialog box.

1.11 Grid division



Ribbon:	Tab ATHENA > Group Draw > Grid division
Menu:	ATHENA > Draw > Grid division
Toolbar:	ATH Draw and ATHENA Functions
Command input:	ath_grid

Divides up an area with a rectangular grid, for example, to produce a placement plan. Optionally, a list as well as the outlines, also fully dimensioned and positioned, can be inserted into the drawing.

When you start the command, the following appears:

Command prompt

Point in the area or [Select objects/?] <Select objects>:
Specify a point within the area which is to be divided with a grid.
*Select **Select objects** to select outlines.*
You call the help with the option ?.



The outlines selected in this way are not associative. This means that: After changing outlines you cannot update the grid division, but must recreate it again.

Specify start point or [?]:
Specify the starting point for the grid division.
Then the Dialog box Grid Division is opened, where you can carry out settings for the grid division.

Select objects

Select outlines

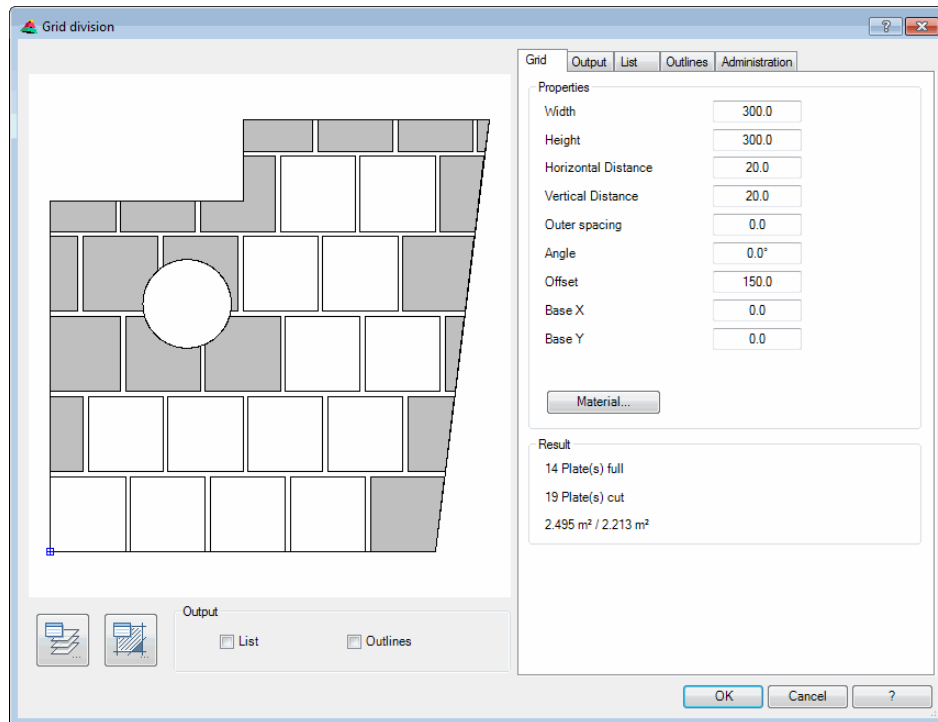
Select objects:

Select the outlines which are to be divided with a grid. Inner outlines are detected as islands and are left out.



The outlines selected in this way are associative. This means that: After changing outlines you can update the grid division.

Dialog box Grid Division



Display section

On the left side the dialog box contains a preview of the grid division. The preview is used as a visual check.

Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.



Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.



Opens the Dialog box Hatch pattern assignment, where you can modify the hatch settings for the plates. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Dialog box section Output

List

Activates or deactivates the list output.

Outlines

Activates or deactivates the outlines output.

Operating section

On the right side of the dialog box there is the operating section with the tabs:

- Grid
- Output
- List
- Outlines

- Manager

Tab Grid

The screenshot shows a software dialog box titled 'Tab Grid'. It has several tabs: 'Grid', 'Output', 'List', 'Outlines', and 'Administration'. The 'Grid' tab is active. Inside, there are two main sections: 'Properties' and 'Result'. The 'Properties' section contains a list of parameters with corresponding input fields: Width (300.0), Height (300.0), Horizontal Distance (20.0), Vertical Distance (20.0), Outer spacing (0.0), Angle (0.0°), Offset (150.0), Base X (0.0), and Base Y (0.0). Below these is a 'Material...' button. The 'Result' section displays the calculated values: '14 Plate(s) full', '19 Plate(s) cut', and '2.495 m² / 2.213 m²'.

Dialog box section Properties

Width

Defines the width of a plate.

Height

Defines the height of a plate.

Horizontal distance

Defines the distance between the horizontal plates.

Vertical distance

Defines the distance between the vertical plates.

Outer spacing

Defines the spacing between the plates and the selected outline.

Angle

Defines the angle of the plate alignment.

Offset

Defines the offset between the vertical plates.



An angle of 0° gives a horizontal offset, an angle of 90° a vertical one.

Base X

Displaces the base point of the first plate in the X direction.

Base Y

Displaces the base point of the first plate in the Y direction.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Dialog box section Result

Shows how many complete and cut plates are needed for the selected outline. Furthermore, the area of the selected outline and the total area of all plates are shown.

Tab Output

Dialog box section Hatched

Complete

Fills all complete plates with a hatching pattern.

Cut

Fills all cut plates with a hatching pattern.

Dialog box section Combine

Consider orientation

Uses different position numbers for the same parts when they are aligned differently.

Dialog box section Position

Prefix

Defines the fixed part of the text at the start of the position number.

Increment

Activates automatic incrementing of the position number.

Start

Defines the variable part of the position number. You can enter figures and also letters. Special characters are not permissible.

Dialog box section Labeling

The tick box activates or deactivates the labeling of the output.

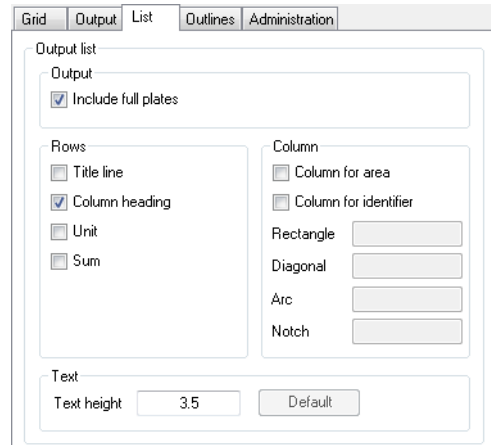
Labeling

Opens the Dialog box Label, where you can modify the settings for the position label. You will find further information on this in the Chapter *Label* on page 117.

The current settings for the dimension style, text height and layer are displayed for information and can be changed by clicking the Format button.

Format

Opens the Dialog box Leader, where you can format the label symbols. You will find further information in the Chapter *Leader*, Section *Symbol/frame* on page 421.

Tab List**Dialog box section Output****Include full plates**

Defines that full plates are also listed in the table output

Dialog box section Rows**Title line**

Switches in the title line of the table output.

Column heading

Switches in the column heading of the table output.

Unit

Displays the units of the plate dimensions in the table.

Sum

Displays the sum of the plates in the table.

Dialog box section Column**Column for area**

Outputs a column for the area of the plates

Column for identifier

Outputs a column for the identifier of the plates In this way differently shaped plates are marked with different identifiers and can be more easily differentiated in the table.

Rectangle

Defines the identifier for rectangular plates.

Diagonal

Defines the identifier for diagonally cut plates.

Arc

Defines the identifier for plates with an arc.

Notch

Defines the identifier for plates with a notch.

Dialog box section Text

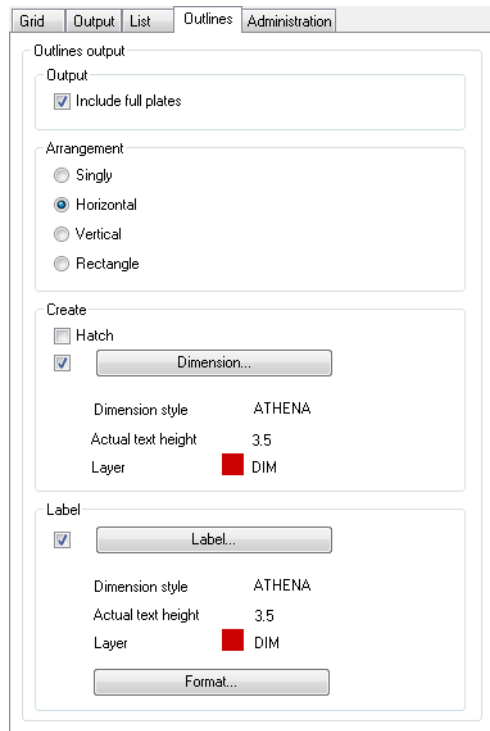
Text height

Defines the text height of the list. The current dimension figure height is used as the default.

Default

Restores the default text height of the list.

Tab Outlines



Dialog box section Output

Include full plates

Also outputs full plates as outline.

Dialog box section Arrangement

Singly

Arranges the plates singly during the output. You must specify an insertion point for each plate.

Horizontal

Arranges the plates horizontally during the output. You only need to specify one insertion point. The base point of the insertion is the lower left corner of the first plate; all other plates are arranged adjacently horizontally to the right.

Vertical

Arranges the plates vertically during the output. You only need to specify one insertion point. The base point of the insertion is the lower left corner of the first plate; all other plates are arranged vertically below it.

Rectangle

Arranges the plates rectangular during the output. You only need to specify one insertion point. The base point of the insertion is the lower left corner of the first plate; all other plates are arranged rectangular to the right and below.

Dialog box section Create

Hatch

Hatches the outlines for the output. The hatching settings of the object are used for the hatching.

Dimensioning

Dimensions the outlines for the output. The current dimension settings are used for dimensions.

The button opens the Dialog box Dimensioning options. For information see "Dialog box Dimensioning options" on page 128.

Dialog box section Labeling

Labeling

Opens the Dialog box Label, where you can modify the settings for the outline label. You will find further information on this in the Chapter *Label* on page 117.

The current settings for the dimension style, text height and layer are displayed for information and can be changed by clicking the Format button.

Format

Opens the Dialog box Leader, where you can format the label symbols. You will find further information in the Chapter *Leader*, Section *Symbol/frame* on page 421.

End of program

When you terminate the program with OK, the grid division and, if necessary, the list and outlines are created in the drawing.

Notes

- You can edit the grid with double-click editing. Lists and dimensioned outlines are however not updated!
- Within an area enclosed boundaries (islands) are detected.
- If an island is to be fully enclosed in a grid area, it is not processed at the position of the island.

1.12 Infills



Ribbon: Tab ATHENA > Group Label > Infills
Menu: ATHENA > Draw > Infills
Toolbar: ATH Draw and ATHENA Functions
Command input: ath_bgla

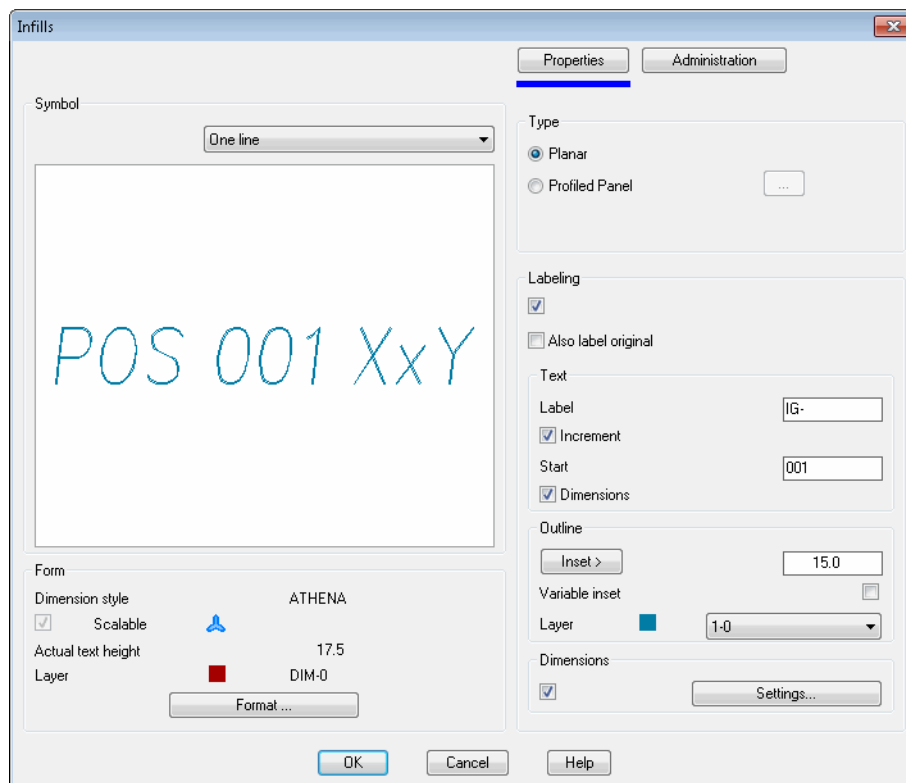
With this routine you can insert diagrams of infills (glazing, panels, ...) into the drawing. The infills are provided with a position number and dimensions and can optionally be completely dimensioned.

Dialog box Infills

In the dialog box you will find on the left side the display section with the preview. On the right side you will see the operating section. You can activate the individual function areas with tab buttons.

You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.

Register button Properties



Display section

Dialog box section Symbol

Above the preview there is a selection menu where you can choose between various position symbols. You can either select a symbol directly from the list or open the Dialog box Selection by clicking the symbol preview and selecting a symbol there.

Dialog box section Display

Dimension style

Displays the current dimension style for information.

Text height

Displays the current text height for information.

Layer

Displays the current layer for information.

Format

Opens the dialog box, where you can adapt the symbol settings. You will find further information in the Chapter *Format settings for labels* on page 106.

Operating section**Register button Properties**

Dialog box section Type

Planar

Creates the diagram of a planar infill.

Profiled panel

Creates the diagram of a profiled panel.



You can edit the profiled panel diagram with the Sheet processing, for example, to supplement punch-outs. To do this, you must bring the diagram into the Sheet processing or select it with Modify ATHENA (double click does not work).

[...]

The button is only active when you select the Profiled panel option. Click the button to open the Dialog box For object selection. Here, the profiled panels defined in the drawing are displayed Select a profiled panel and close the dialog box with OK.

The designation of the selected panel is displayed in the dialog box.

Dialog box section Labeling

Activates the infill label in the form of a position symbol.

Also label original

Additionally creates a label at the origin of the infill.

Dialog box section Text

Label

Defines the name of the Infill as the fixed part of the infill label.

Increment

Activates incrementing the variable extension of the infill label.

Start

Specifies the variable extension of the infill label. You can enter figures and also letters. Special characters are not permissible.

Dimensions

Specifies the dimensions of the infill label.

Dialog box section Outline

Inset

Defines the inset of the infill. You can also indicate the inset with the mouse. To do this, click the Inset < button with the mouse.

Variable inset

Switches in the variable inset. You can now assign different insets to all edges of the infill.

Layer

Here, you decide on which layer the outline is inserted.

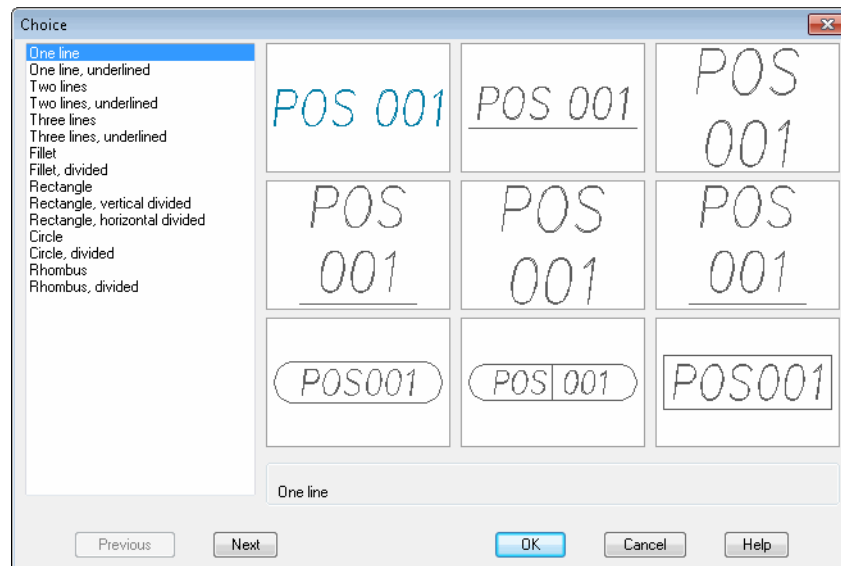
Dialog box section Dimensioning

Switches dimensions on or off.

Settings

Opens the Dialog box Dimensioning options. In this dialog box you can define, using tick boxes, in which manner the infills are to be dimensioned. You will find a detailed description of the Dimension options dialog box in the chapter *Dimensioning options* on page 128.

Dialog box Selection



End of program

If you terminate the dialog box Infills with OK, the following input request appears:

Command prompt

Option Area

Specify point in the area or [Corner points/?]:

Click within a closed area with the mouse.

*You can select corner points with the **Option Corner points**.*

You call the help with the option ?.

The following input requests for the Inset only appear when the Variable inset tick box in the Infill dialog box is set.

Specify inset or [followingSame/?] <14>:

*Specify the inset for the first side of the infill. With **ENTER** you accept the default inset.*

*With the option **followingSame** the default inset is accepted for all the following sides.*

Specify inset or [Undo/followingSame/?] <14>:

*Specify the inset for the next side of the infill. With **ENTER** you accept the default inset.*

*You can change the inset of the previous side with **Undo**.*

This input request is repeated for all sides of the infill. After the last side follows:

Specify insertion point <Ok>:

Use the mouse or enter co-ordinates to specify the insertion point for the infill.

Specify point in the area or [Corner points/?]:

*This query is repeated until the command is terminated with **ENTER** or a **RIGHT CLICK**.*

Option Corner points

Specify corner point or [Area/?]:

Specify the first corner point of the outline with the mouse.

*You can indicate a point in the area of the outline with the **Option Area**.*

Specify corner point or [Area/Undo/?]:

Specify the next corner point of the outline with the mouse.

*With the option **Undo** you can correct the last corner point.*

Specify corner point or [Area/Undo/?]:

*Specify the next corner point of the outline with the mouse. Once you have set the last corner point, you obtain the next input request by entering **ENTER** or by giving a **RIGHT CLICK**.*

Specify insertion point <Ok>:

Use the mouse or enter co-ordinates to specify the insertion point for the infill.

Specify point in the area or [Corner points/?]:

*This input request is repeated until the command is terminated with **ENTER** or a **RIGHT CLICK**.*

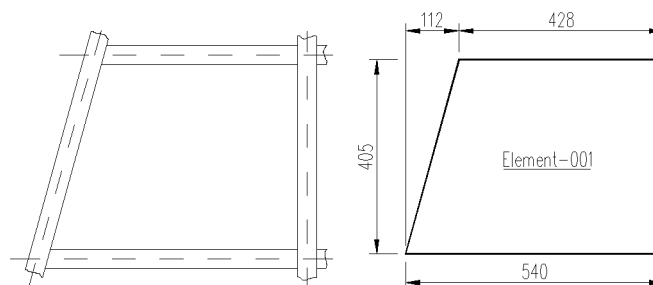


Fig. 1.8: Glass dimensioning

Notes

- The infill dimensioning generally occurs without decimal places.
- The current dimension settings (Layer ...) are used for dimensioning.

1.13 More-centered arch



Ribbon:	Tab ATHENA > Group Draw > More-centered arch
Menu:	ATHENA > Draw > More-centered arch
Toolbar:	ATH Draw
Command input:	ath_kbog

With this routine you can construct a completely dimensioned more-centered arch.

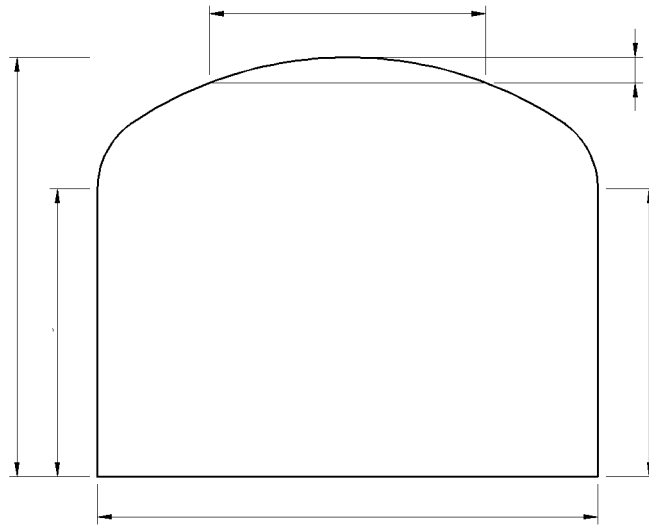


Fig. 1.9: More-centered arch

Command prompt

Specify starting point of more-centered arch or [?]:

Specify the starting point of the more-centered arch.

You call the help with the option ?.

Specify width W or [?]:

*Point out the width **W** with the mouse or enter a width.*

Specify overall height H or [?]:

*Point out the total height **H** with the mouse or enter a height.*

Specify height to arch on left HL or [?]:

Point out the height to the arch on the left with the mouse or enter the height.

Specify height to arch on right HR or [?] <HL>:

Point out the height to the arch on the right with the mouse or enter the height.

*With the option **HL** you can accept the height of the left arch.*

Specify length of the gauge GL or [?] <300>:

*Define the length of the gauge by entry or pointing with the mouse. Input **ENTER** to accept the default (300).*

Specify the rise of the arch or [?] <10>:

*Define the rise of the arch. Input **ENTER** to accept the default (10).*

ATHENA now creates the more-centered arch.

Notes

- ATHENA uses Layer 0 for the outline of the more-centered arch and the current dimension settings for the dimensioning.
- If, due to incorrect geometrical entries, construction of the more-centered arch is not possible, an error message appears and you have the opportunity of correcting the details.

1.14 Spacer



Ribbon:	Tab ATHENA > Group Draw > Spacer
Menu:	ATHENA > Draw > Spacer
Toolbar:	ATHENA Draw
Command input:	ath_klotz

With this command you can create a spacer with just a few mouse clicks. You can use the spacer with or without material specification (e.g. as plastic shim).

Command prompt

This input request only appears when the option Determine on screen has been activated in the Dialog box Spacer properties.

Option Point

[Point Mode]

Specify base point or [Settings/Object/?]: <Object>:

Use the mouse or enter coordinates to specify the base point.

*Use the **Option Settings** to change the properties of the spacer.*

*Use the **Option Object** to create the spacer by object selection.*

You call the help with the option ?.

[Point Mode]

Specify second point or [Width/Thickness/Settings/Undo/?]:

Define the second point of the spacer. With the second point you define the width and direction of the spacer.

*Use the **Option Width** to specify the width of the spacer. A further input request follows.*

*Use the **Option Thickness** to specify the thickness of the spacer. A further input request follows.*

*Use the option **Undo** to repeat the previous input request.*

[Point Mode] Width=XX

Specify third point or [Width/Thickness/Settings/Undo/?]:

Define the third point of the spacer. With the third point you define the thickness and side of the spacer.

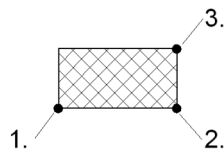


Fig. 1.10: Spacer point mode

Option Object

[Object Mode]

Select edge or [Settings/Point/?] <Point>:

Select an object edge to define the width and direction of the spacer.

*Use the **Option Point** to create a spacer by specifying points.*

[Object Mode] Width=XX

Specify third point or [Width/Thickness/Settings/Undo/?]:

Specify the third point of the spacer or select an option. With the third point you define the thickness and side of the spacer.

Option Width*[Width Mode]**Specify width or [Point/Thickness/Undo/Settings/?]:**Enter the width of the spacer or select an option. You can point out the width with the mouse or enter it in the command line.***Option Thickness***[Thickness Mode]**Specify thickness or [Point/Width/Undo/Settings/?]:**Enter the thickness of the spacer or select an option. You can point out the thickness or enter it in the command line.*

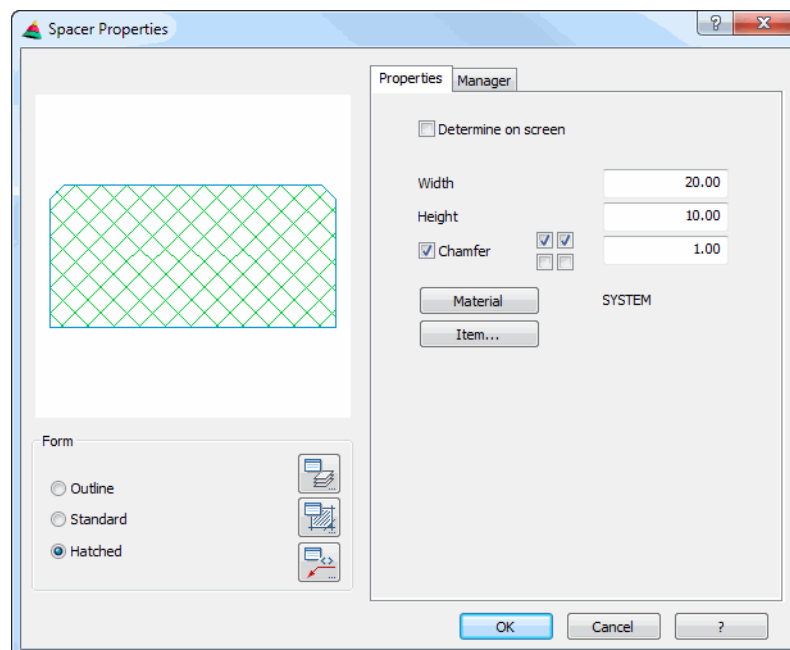
This input request only appears when the option Determine on screen has been deactivated in the Dialog box Spacer properties.

*Specify insertion point or [?]:**Define the insertion point of the spacer.**You call the help with the option ?.**Specify rotated angle<0>:**Define the rotation angle of the spacer.**Press the Enter key to accept the default angle.***Option Settings**

Opens the Dialog box Spacer properties.

Dialog box Spacer properties

The dialog box contains the tabs Properties and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Tab Properties

Determine on screen

Determines the dimensions of the spacer on the screen when you close the dialog box with OK. Further input requests follow.



The option Determine on screen is saved permanently.

If this option has been activated, the input fields Width and Height are deactivated. Otherwise you can specify the dimensions in these input fields.

Width

Defines the width of the spacer.

Height

Defines the height of the spacer.

Chamfer

Creates a spacer with a chamfer. When you activate the tick box, four additional tick boxes are shown with which you can specify in which corner the spacer is to be chamfered.

In the input field you define the chamfer spacing.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Dialog box section Display

Outline

Creates a spacer without infill.

Standard

Creates a standard spacer which is filled with an X.

Hatched

Creates a spacer with hatched infill. When you select a material, the material-dependent hatching is used.

Layer assignment

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch pattern assignment

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Labeling

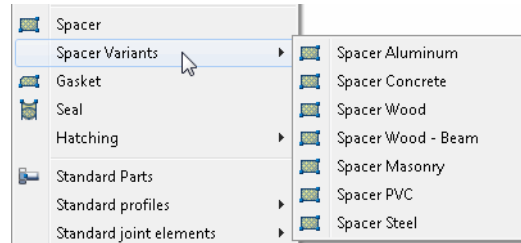
Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Notes

- You can change a spacer by stretching or with grips. The spacer retains its rectangular shape when stretched (optionally with chamfer).
- You can change the spacer properties with the command **Modify ATHENA** or by a double click.
- You can automatically label the spacer with the command **Parts labeling**.

- You can change the layer default settings for the spacer in the Dialog box Layer assignment.

Spacer variants



In the menu ATHENA>Draw>Spacer variants you will find examples of spacers with various materials set as default. These have the advantage that you do not need to activate the dialog box for frequently used materials.

Similarly as with the spacer, you can also predefine materials for Gasket and Seal. In this way you can also insert these objects into the drawing quickly and easily.

In order to define your own spacer variants (or gasket or silicone variants) it is best to create a new command in your user adaptation file (e.g. ath_user.cui). The command macro would then be similar to this:

```
^C^C^PATH_OBJ_MAT_SET KLOTZ WOOD ATH_KLOTZ;
```

This macro activates the material wood and then starts the command **Spacer**, which is then created with the appropriate material settings (hatch and layer).

In the command macro KLOTZ stands for the function. Similarly, you have to use DICHT for gaskets and SILIKON for seals. WOOD stands for the material name. You can view this in the ATHENA options in the sub-dialog box Material properties.

You will find further information about user adaptation in the AutoCAD documentation.

1.15 Gasket



Ribbon:	Tab ATHENA > Group Draw > Gasket
Menu:	ATHENA > Draw > Gasket
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_dicht

With this command you can create a gasket with just a few mouse clicks. You can use the gasket with or without material specification (e.g. as EPDM gasket).

Command prompt

The syntax of the input request is identical to the syntax of the command [Spacer](#) on Page 240.

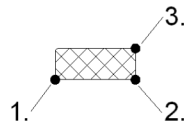
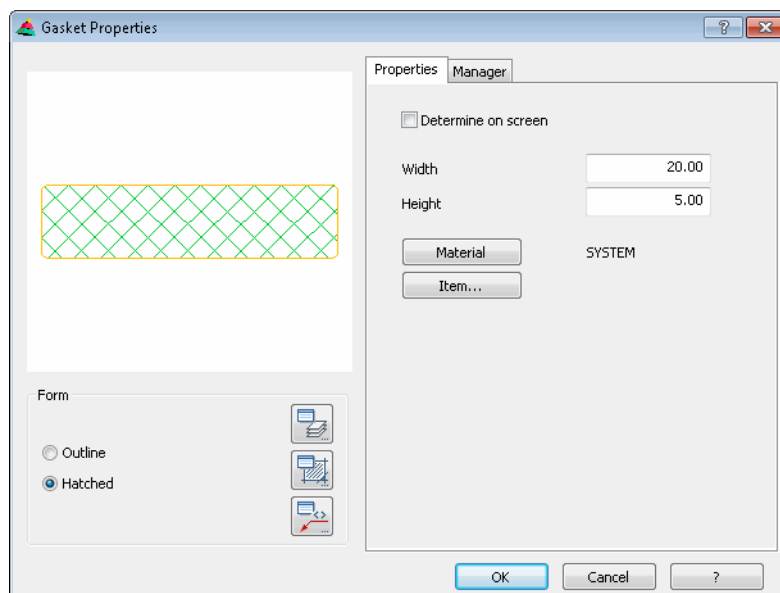


Fig. 1.11: Gasket point mode

Dialog box Gasket properties

The dialog box contains the tabs Properties and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



The dialog box is largely identical to the Dialog box Spacer properties on Page 240.

Notes

- You can change a gasket by stretching or with grips. When stretched, the gasket retains its rectangular shape with rounded corners.

- You can change the gasket properties with the command **Modify ATHENA** or by a double click.
- You can automatically label the gasket with the command **Parts labeling**.
- You can change the layer default settings for the gasket in the Dialog box Layer assignment.

1.16 Seal



Ribbon:	Tab ATHENA > Group Draw > Seal
Menu:	ATHENA > Draw > Seal
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_silikon

With this command you can create a seal with just a few mouse clicks. This can be shown optionally with back-filler.

Command prompt

This input request only appears when the option Determine on screen has been activated in the Dialog box Seal properties.

Option Object

[Object Mode]

Select first joint edge of object or [Point/Settings/?] <Point>:

Select an object to define the first joint edge. In this respect take into account that the side of the seal depends on the selected side of the object.

*Use the **Option Point** to create the seal by specifying points.*

*Use the **Option Settings** to adapt the properties of the seal.*

You call the help with the option ?.

Select second joint edge of object or [Point/Settings/Undo/?] <Point>:

Select an object to define the second joint edge.

*Use the option **Undo** to repeat the previous input request.*

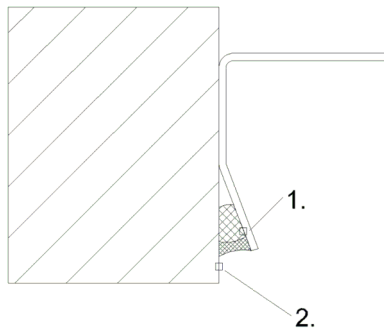


Fig. 1.12: Seal object mode

Option Point

[Point Mode]

Specify first joint point or [Object/Settings/?] <Object>:

Specify the first joint point.

*Use the **Option Object** to create the seal by object selection.*

*Use the **Option Settings** to adapt the properties of the seal.*

You call the help with the option ?.

Specify second joint point or [Object/Settings/Undo/?]:

Specify the second joint point.

*Use the option **Undo** to repeat the previous input request.*

Specify side or [Undo/Apex/?]:

Specify on which side the seal is to be drawn.

*Use **Option Apex** to create a trapezoidal seal.*

Option Apex*Specify apex or [Undo/?]: <Undo>:**Define the apex of the seal.*

This input request only appears when the option Determine on screen has been deactivated in the Dialog box Seal properties.

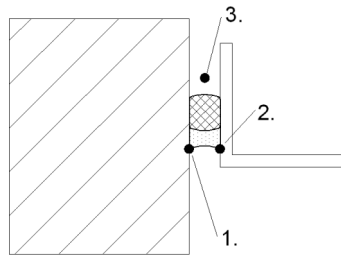
*Specify insertion point or [?]:**Specify the insertion point of the seal.**You call the help with the option ?.**Specify rotated angle<0>:**Define the rotation angle of the seal.**Press the Enter key to accept the default angle.*

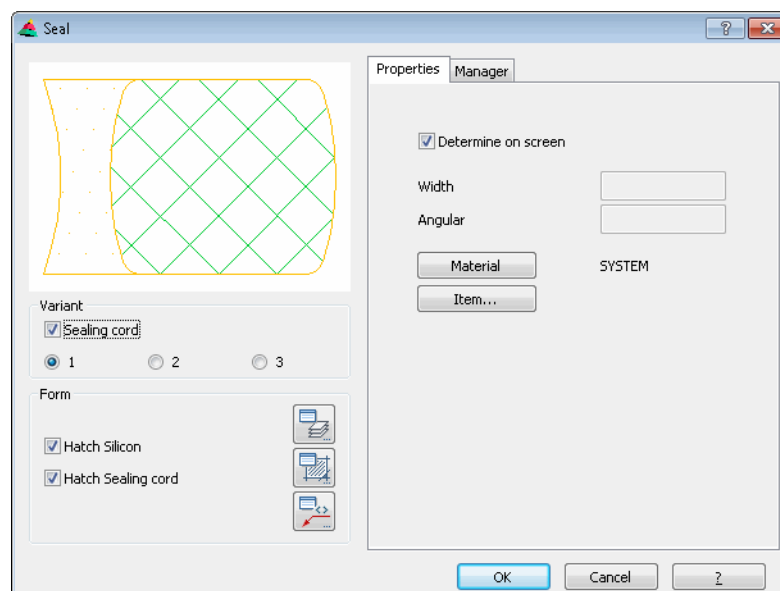
Fig. 1.13: Seal point mode

Option Settings

Opens the Dialog box Seal properties.

Dialog box Seal properties

The dialog box contains the tabs Properties and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Tab Properties

Determine on screen

Determines the dimensions of the seal on the screen when you close the dialog box with OK. Further input requests follow.



The option Determine on screen is saved permanently.

If this option has been activated, the input fields Width and Angle are deactivated. Otherwise you can specify the dimensions in these input fields.

Width

Defines the width of the seal.

Angle

Defines the angle of the seal.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Dialog box section Variant

Sealing cord

Defines whether the seal is to be back-filled with or without sealing cord.

Variant 1

Activates the variant of thin seal and thick sealing cord.

Variant 2

Activates the variant of thick seal and thin sealing cord.

Variant 3

Activates the variant of thick seal and thick sealing cord.

Dialog box section Display

Hatch silicone

Hatches the silicone part of the seal.

Hatch sealing cord

Hatches the sealing cord of the seal.

Layer assignment

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch pattern assignment

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Notes

- You can fit a seal to various shapes by stretching or with grips.
- You can change the seal properties with the command **Modify ATHENA** or by a double click.

- You can automatically label the seal with the command *Parts labeling*.
- You can change the layer default settings for the seal in Dialog box Layer assignment.

1.17 Hatch



Ribbon:	Tab ATHENA > Group Hatch > Hatch name
Menu:	ATHENA > Draw > Hatch > Hatch name
Toolbar:	ATH Hatch
Command input:	ath_hatch

With this command you can fill a bordered area or objects to be selected with an associative hatching pattern. You can select a hatch pattern via icon or on your tablet overlay. The hatching scale is defined by specifying the scale. You will find further information about hatching in your AutoCAD documentation.

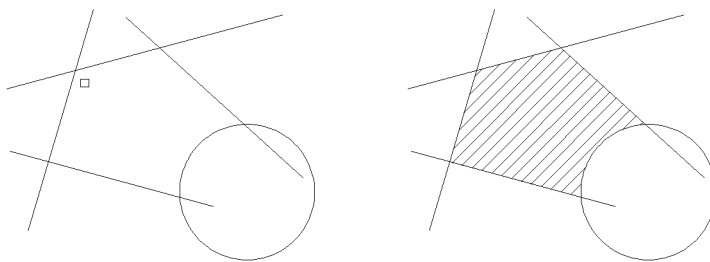


Fig. 1.14: Hatch

Command prompt

Enter scale of hatching <1:1>:

Enter the hatching scale (e.g. 10 for 1:10 to enlarge the hatching distance tenfold) in the command line.

Press the Enter key to accept the default scale. The scale last used is saved for the duration of the drawing session.

Select a point in the area or [Object selection/?]: <Object selection>:

Enter a point within the area to be hatched.

*Use the **Option Object selection** to select objects.*

You call the help with the option ?.

Select a point in the area or [?]:

Enter another point within the area to be hatched. This input request is repeated.

Press the Enter key to terminate the command.

Option Object selection

Select objects:

Using the mouse, select the objects which are to be hatched. This input request is repeated. Press the Enter key to terminate the command.

Notes

The hatchings of the tablet overlay or the icons are defined in the CUI file. You can adapt them to your requirements. You will find further information about hatching and menu adaptation in your AutoCAD documentation.

Below you can see an example of how the hatchings are defined in the CUI file:

^C^C^Path_hatch_layset |3|0 ath_hatch_set ANSI31|0|25|0 ATH_SCHR

- | | |
|--------|---|
| 3 0 | Defines the layer. Here, enter the logical layer name. The first figure defines the color, the second figure defines the line type. You will find further information about layer configuration in the Chapter <i>Layer</i> on page 85. |
| ANSI31 | Defines the hatch pattern. Hatching can be defined according to the pattern. To do this, enter the pattern name. Or enter <code>_U</code> for user hatching (line hatching). |
| 0 | Defines the hatching angle. |
| 25 | Defines the hatching factor. Important: With user hatching (line hatching) this value signifies the line spacing. With pattern hatching this is the scale factor. |
| 0 | Cross hatching. Here, you define whether the hatching is crossed (0) or not (1). This only has an effect with user hatching. |

1.18 Standard Part



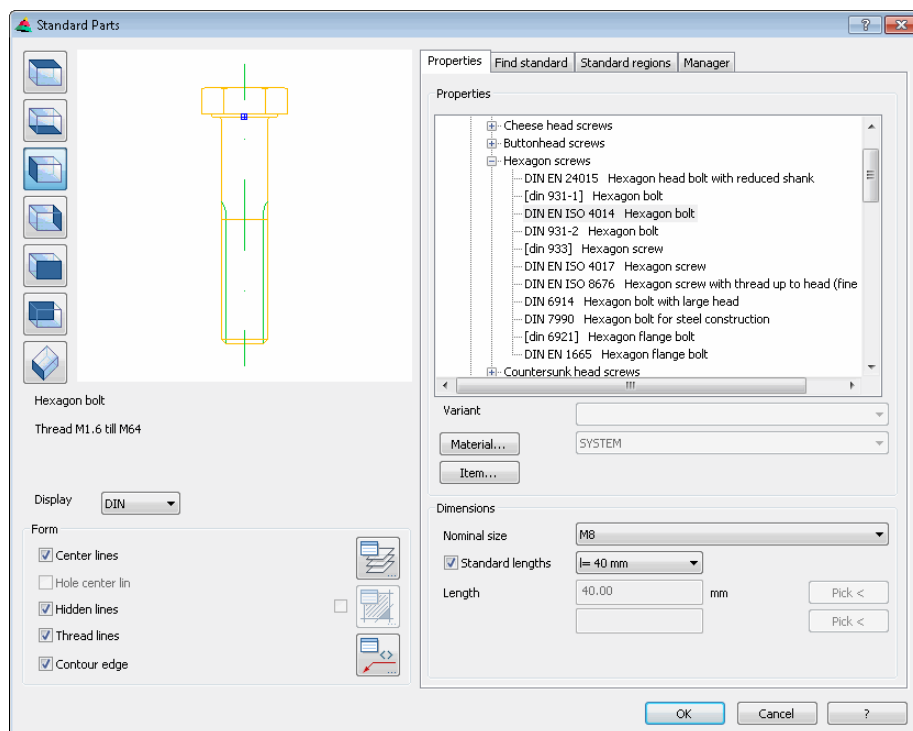
Ribbon:	Tab ATHENA > Group Standard parts > Standard part
Menu:	ATHENA > Draw > Standard part
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_din

With this command you can create standard and manufactured parts in the drawing. Standard parts can be inserted and used in various 2D views or as a 3D object.

Dialog box Standard Parts

In the dialog box you will find on the left side the Display section with the dynamic preview. On the right side you can see the Operating section with several tabs for the various functional areas.

The tabs Properties, Find standard and Standards regions are object-specific. The Administration tab is similar for most dialog boxes. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Display section

The preview shows the chosen standard part or an example part in the selected parts group. The preview is primarily used for a visual check, but offers other functions too:

- The Overview of the selected group is opened using left click on the preview.
- Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.

Below the preview additional information is displayed on the chosen standard, if available.



With many manufactured parts the information text is also a hyperlink to the product page of the displayed part. Click the link to display further information (for example a detailed product description or approval) in your browser.

Display

Defines which standard (DIN, ISO or EN) is to be displayed. If a standard other than DIN is selected and a part is not available in this standard, DIN parts are still displayed, but are identified by pointed brackets, e.g.: <DIN 6914>.



Outdated (withdrawn) standards are displayed in square brackets and lowercase letters, e.g.: [din 933].

To the left of the preview there are buttons with which you can adjust both the preview and also the insertion into the drawing. You will find further information on this in the section *Object views* on page 102.

Notes for insertion into the various views:

- If you insert a joining element as any view or a profile as a side view from the left or right, only the base point and rotation angle are interrogated. For further information see "Command prompt for joining elements and profile sections" on page 253.
- If you insert a profile as 2D plan view, 2D bottom view, 2D front elevation or 2D rear elevation, the profile length is interrogated on insertion. For further information see "Command prompt for profiles" on page 253.
- If you insert a profile as solid, the profile length is interrogated. For further information see "Command prompt for profiles as 3D solids" on page 253.
- Profile solids are "known" bar assemblies, the names of which consist of the TMP and an incremented number. You will find further information on bar assemblies in the sections *Bar Assembly Manager* on page 632 and *Apply bar assembly* on page 738.

Dialog box section Display

Center lines

Switches center lines on or off.

Hole center lines

Switches hole center lines on or off. The option Hole center lines is only available for steel section and is otherwise deactivated.

Hidden lines

Switches the hidden lines on or off. Hidden lines are for example the hole in a washer or the hexagonal socket of a screw.

Thread lines

Switches the thread lines on or off. This options is not available for profiles.

Contour edge

Switches the contour edges for profiles on or off. This option is only available for rounded-off profiles. For example, if you insert the plan view of a steel U-section.

Layer...

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch...

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

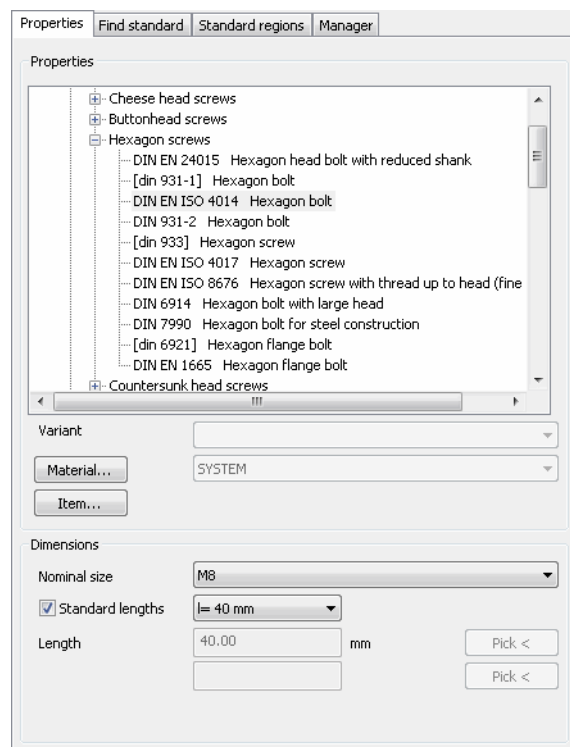
The Hatch button is available if you have ticked the tick box. The tick box also has the effect that the standard part is created with a hatched infill. When you have selected a material, the material-dependent hatching is used. This option is not available for joining elements.

Label...

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Operating section

Tab Properties



Dialog box section Properties

List

Shows the available standard parts in a tree structure. Here you can select the required standard with the mouse or keyboard. Closed branches (standards groups) in the tree structure are identified with +. Open branches are identified with -. When you are navigating with the keyboard, you can use the following keys:

- Upwards arrow changes to the previous standard/group
- Right arrow or + opens the selected branch
- Left arrow or - closes the selected branch
- Downwards arrow changes to the next standard/group

Variant

Shows the variants of a standard, if available.

Material

Opens the Dialog box Materials, where you can assign a material to a standard part. This results in the part being inserted onto a material layer. With Parts labeling the specified material can be automatically quoted on the label.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Dialog box section Dimensions

The screenshot shows a dialog box titled "Dimensions". It has three main sections: "Nominal size" with a dropdown menu showing "M8"; "Standard lengths" with a checked checkbox and a dropdown menu showing "40 mm"; and "Length" with a text input field showing "40.00" and a unit dropdown showing "mm". There are two "Pick <" buttons on the right side of the dialog.

Nominal size

Displays the nominal sizes available for the current part. Here, select the required nominal size.

Standard lengths

Switches the standard lengths on or off. When the standard lengths are switched in, you can select the required length.

Length

Defines the length of the standard part, if the tick box Standard lengths is deselected.

Pick <

Temporarily terminates the dialog box Standard parts and you can specify a length by selecting two points. ATHENA selects the next available standard length if Standard lengths is activated, otherwise the displayed length is used.

Optional input fields:**Clamp size**

Only available for rivets and defines the length of the clamping region. The clamping size available for the selected rivet is displayed.

The functioning principle is the same as for the length. With most parts this section is masked out.

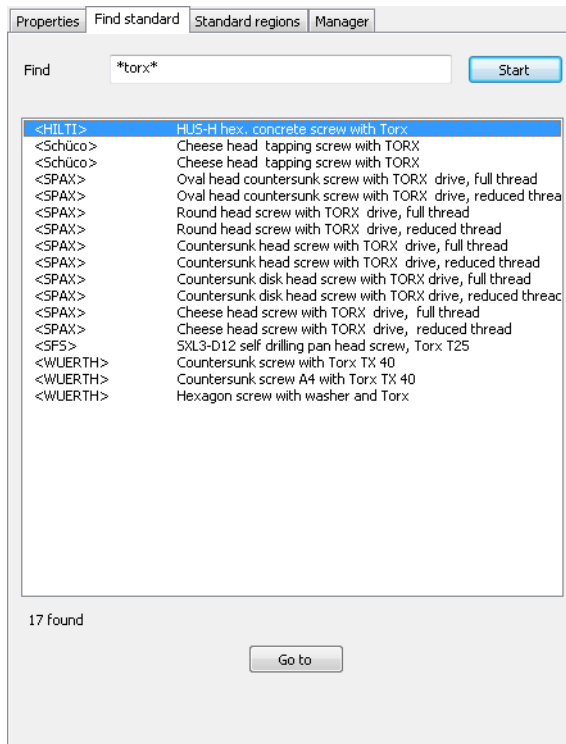
Radius

Only available for rectangular hollow section and square hollow section and defines the outer radius of the tube.

The radii of the rectangular and square tubes are not standardized and may differ from manufacturer to manufacturer. The definition refers just to one section (from - to) which can be defined. The default is always the largest radius.

The functioning principle is otherwise the same as for the length. With most parts this section is masked out.

Tab Find standard



Find

Defines the search term for the standards search. The search term may also include placeholders (* for any character and ? for one character).

Start

Starts the search for standard parts according to the specified search pattern.

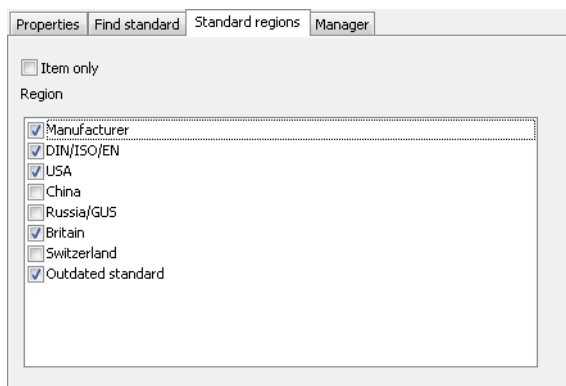
List

Displays the standards groups found. Here, select the standard and click the button Goto.

Go to

Changes to the Tab Properties and displays the selected standard.

Tab Standards regions



Item only

Only shows the standard parts to which an item number has been assigned.

Region

Lists all available regions. Here, using tick boxes, you can control which standards regions are to be displayed.

Tab Manager

You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.

End of program**Cancel**

Terminates the dialog box without inserting a standard part.

OK

Terminates the dialog box, the selected standard part is attached to the cross-hairs and the following input request appears for insertion:

Command prompt for joining elements and profile sections*Specify insertion point:*

Use the mouse or enter coordinates to specify the insertion point of the part.

When you insert the cross section of a profile, you can change between various insertion points by repeated pressing of the Ctrl key.



The grips of the relevant profile cross section can be used as insertion points. The insertion point activated by Ctrl only applies for the duration of the insertion.

Specify rotated angle<0>:

Use the mouse or enter an angle to specify the rotation angle. Press the Enter key to accept the default angle.

Command prompt for profiles*[Object Mode]**Select axis or [Points/?]:*

Choose a system axis with the mouse.

*Select **Option Points** to specify the start and end points of the axis.*

You call the help with the option ?.

*Option Points**[Points mode]**Specify the axis start point or [Object/?]:*

Use the mouse or enter co-ordinates to specify the axis start point.

Choose the option Objects when you want to select an axis.

Specify the axis end point or [Undo/Object/?]:

Use the mouse or enter co-ordinates to specify the axis end point.

You select the option Undo to start the point entry anew.

Bar orientation [Mirror/?]:

The option Mirror mirrors the bar about its own axis.

Command prompt for profiles as 3D solids*[Object Mode]**Select axis or [Points/?]:*

Choose a system axis with the mouse.

*Select **Option Points** to specify the start and end points of the axis.*

You call the help with the option ?.

Option Points

[Points mode]

Specify the axis start point or [Object/?]:

Use the mouse or enter co-ordinates to specify the axis start point.

Choose the option Objects when you want to select an axis.

Specify the axis end point or [Undo/Object/?]:

Use the mouse or enter co-ordinates to specify the axis end point.

You select the option Undo to start the point entry anew.

Choose the option Objects when you want to select an axis.

Bar orientation [Mirror/ Rotate/ surfaceAxis/ surfaceMiddle/ rotationAngle/ Point/?]:

The options for changing the bar orientation are described in the section Apply bar assembly on page 738.

Notes

- The generated standard or manufactured parts can be retrospectively labeled with the ATHENA command **Parts labeling**.
- You can change the properties of the part with the command **Modify ATHENA** or by a double click.
- Ready-to-use fixings (e.g. anchor with nut) are inserted as a screwed joint group. You will find further information in the Chapter *Screwed joint* on page 258.
- Halfen channels include an invisible point which can be used when positioning the associated Halfen screw as snap point. For this, the object snapping point must be switched on.
- With the command **Object to solid body** you can convert the standard parts to solids.
- With the command **Projection objects** you can generate 2D projections of standard parts.
- Similar to 3D bars, 2D projections of profiles can also be trimmed with the command **Cutting**.
- Similar to 3D bars, 2D projections of profiles can also be sliced with the command **Slice Bar**.

Hinweis:

Standard parts and standards are subject to continual modification by the standardization organizations. CAD-PLAN GmbH can in no way be held liable for missing or defective parts and for any errors and losses arising from them. Manufactured parts are subject to continual modification by the manufacturers. CAD-PLAN GmbH can in no way be held liable for defective parts and for any errors and losses arising from them.

References and links:

The published links have been researched and complied with the utmost care. CAD-PLAN GmbH has no influence on the current and future layout and content of the linked pages. CAD-PLAN GmbH is not responsible for the content of the linked pages and does not regard the content as its property. The supplier of the linked web site is solely responsible for illegal, erroneous or incomplete content and for losses arising from the use or disuse of the information. The liability of anyone who only refers to the publication through a link is excluded.

1.19 Standard parts sub-dialog box

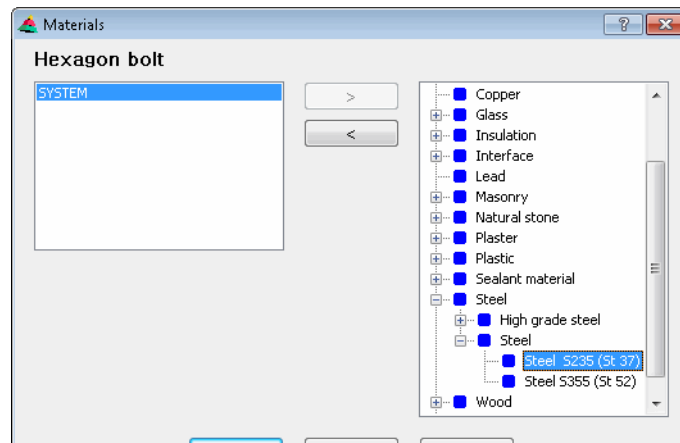
This section describes all sub-dialog boxes which are called exclusively from the standard parts.

- Materials
- Overview of the standards group

1.19.1 Materials

Assigns materials to the selected standard part.

Dialog box Materials



List

The left list shows the materials which have been assigned to the standard. In the default setting most standards (except profiles) have no material specified (Material type: System).

The right list shows the available materials in a tree structure. In both lists you can select materials with the mouse.

[>]

Removes the selected material from the standard.

[<]

Assigns the selected material to the standard.

When you click OK, the setting for the active standard is saved.

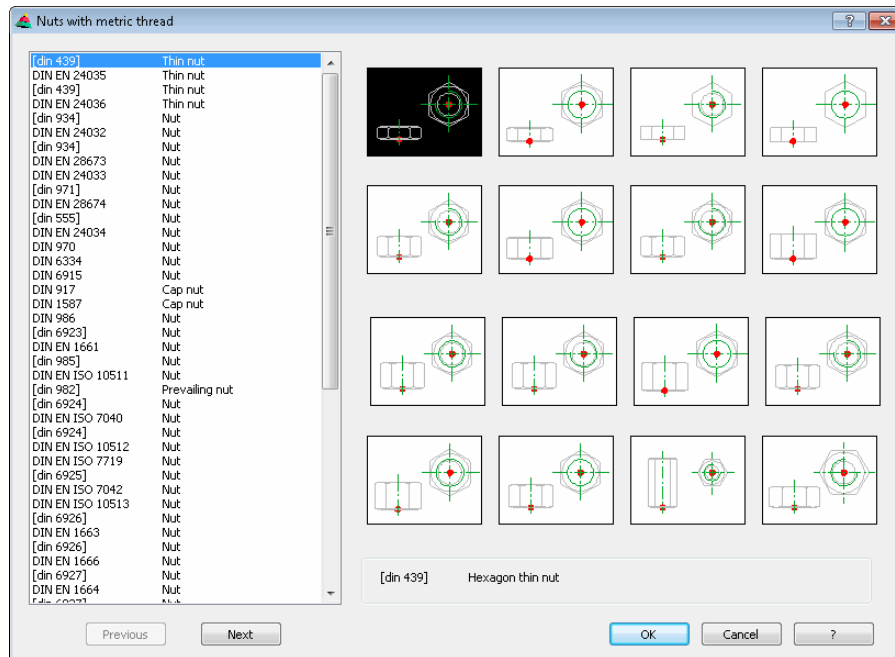


ATHENA saves the material assignment in the file ath_nm_prop.dex.

1.19.2 Overview of the standards group

Shows a graphical overview of the selected standards group.

Dialog box Overview



List

Displays the available standards for the selected group. You can select the required standard here.

Previous

Pages to the previous page of the standards overview. This button is only available for multi-page standards overviews.

Next

Pages to the next page of the standards overview. This button is only available for multi-page standards overviews.

Slide Display

Displays slides of the standards available. When you click on a slide, further information on the standard is superimposed below the slide display.

1.20 Edit standard parts

You can edit standard parts with a double-click. There are the following three variants of editing, depending on the inserted part:

With joining elements and profile cuttings the Dialog box Standard Parts is displayed with the properties of the selected part.

With made-up standard parts, e.g. multi-part anchors, the Dialog box Screwed joint is displayed with the properties of the selected part.

The following input request appears for 2D projections of profiles:

Command prompt

Modify [Orientation/Properties/?] <Properties>:

Select the option Properties to display the Dialog box Standard Parts and to modify its properties.

Choose the option Orientation to modify the alignment of the 2D projection. A further input request follows:

Bar orientation [Mirror/Rotate/?]:

Choose the Mirror option to mirror the 2D projection about its own axis.

Select the option Rotate to change to the next view. By repeated rotation the four views (top, front, bottom, back) can be selected in turn.

1.21 Screwed joint



Ribbon:	Tab ATHENA > Group Standard parts > Screwed joint
Menu:	ATHENA > Draw > Screwed joint
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_scord

You can create, edit and save screwed joints with this command. Screwed joints can be inserted and used in various 2D views or as a 3D object.

Command prompt

Select object or [New/?] <New>:

Select the screwed joint to be changed. A further input request follows.

*Choose the option **New** to create a new screwed joint; the Dialog box Screwed joint is started.*

You call the help with the option ?.

Option Add

Add object or [Remove/?]:

Select the standard part which you want to add to the screwed joint.

*Select the **Option Remove** to remove a standard part from the screwed joint.*

Option Remove

Remove object or [Add/?]:

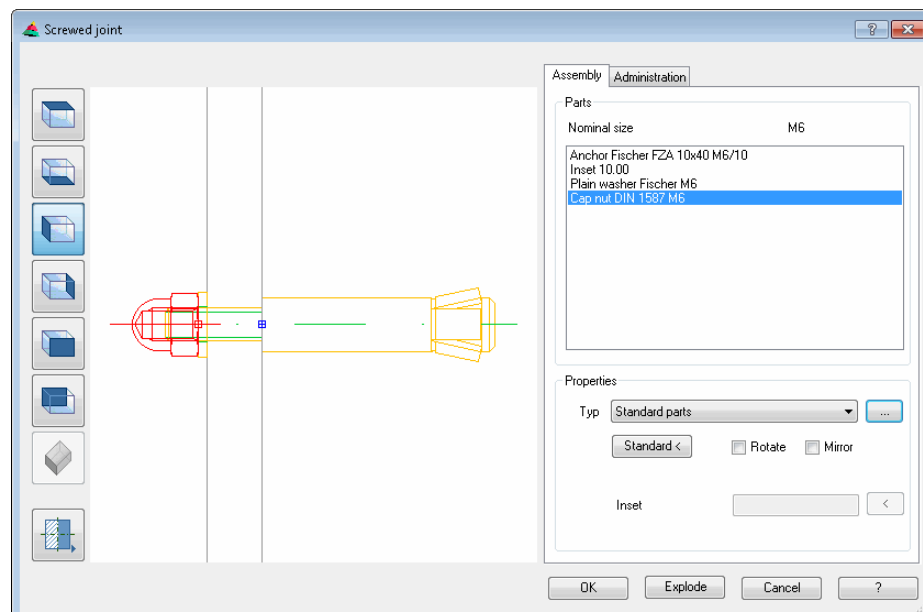
Select the standard part which you want to remove from the screwed joint.

*Select the **Option Add** to add a standard part to the screwed joint.*

Dialog box Screwed joint

In the dialog box you will find on the left side the Display section with the dynamic preview. On the right side you can see the Operating section with several tabs for the various functional areas.

The Properties tab is object-specific. The Administration tab is similar for most dialog boxes. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Display section

The preview displays the defined screwed joint. The preview is used primarily as a visual check. Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.

To the left of the preview there are buttons with which you can adjust both the preview and also the insertion of the screwed joint into the drawing. You will find further information on this in the section *Object views* on page 102.

Display button

If you move the mouse over this button, a menu with the following options opens up:

Hidden

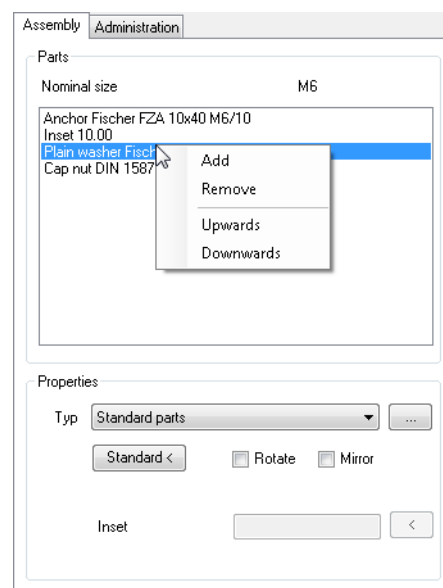
Parts located on top hide the thread below.

Clamping hidden

The clamping hides the thread below.

Operating section

Register button Assembly



Dialog box section Parts

List

Shows the constituent parts of the current screwed joint.

The nominal size of the screwed joint is shown above the list (e.g. M12). The first component in the list defines the nominal size of the complete screwed joint and should be a screw; this then leads to the other components. You can select a part of the screwed joint in the list to change its properties.

If you click in the parts list with the right mouse key, a context menu opens offering the following options:

Add

Copies the selected constituent part of the screwed joint.

Remove

Removes the selected constituent part of the screwed joint.

Upwards

Moves the marked part upwards by one position (to the left in the preview).

Downwards

Moves the marked part downwards by one position (to the right in the preview).

Dialog box section Properties

Type

Changes the type of the selected constituent part. Three types can be used in the screwed joint. Standard part, Drilled hole and Clamping.

[...]

Depending on the type, starts the Dialog box Standard Parts or the Dialog box Drilled hole, where you can change the object properties.

Standard <

Restores the ready-to-use screwed joint, if it has been changed. E.g.: If with a ready-to-use plug/nut screwed joint you have replaced the supplied nut by a cap nut, you can restore the original state with the button Standard.

Rotate

Rotates the selected part through 90 degrees.

Mirror

Mirrors the selected part about its own axis. This is, for example, necessary for single-sided chamfered panes, if the installation situation demands it.

As clamping

Only active for drilled holes and treats them as clamping. This leads to the following parts being displaced by the hole depth (= clamping).

Clamping

Defines the clamping dimension.

[<]

Accesses the clamping dimension in the drawing. The Dialog box Screwed joint is temporarily closed and the input request appears:

Specify clamping or [?]: <10>:

Enter a point to define the clamping or enter a length.

Specify second point:

Enter the second point to define the clamping.

End of program

Click OK to insert the assembly or to accept the changes. Click Cancel to discard the changes.

Explode

Releases the screwed joint into its constituent parts and terminates the dialog box.

Notes

- Ready-to-use fixings (e.g. anchor with washer and nut) are automatically inserted as a screwed joint. You then have, for example, the possibility of replacing the nut by a cap nut.
- With the command **Object to solid body** you can convert the screwed joints to solids.
- With the command **Projection objects** you can generate 2D projections of screwed joints.

1.22 Edit screwed joints

You can edit screwed joints with a double-click. For this, the Dialog box *Screwed joint* is displayed with the properties of the selected part. You will find further information on this in the section *Screwed joint* on page 258.

1.23 Semi-finished product



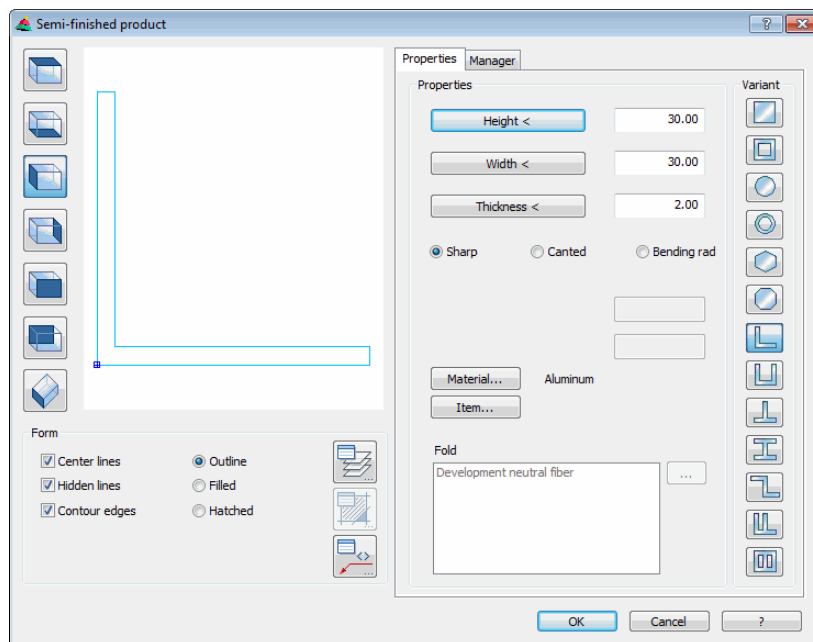
Ribbon:	Tab ATHENA > Group Standard parts > Semi-finished product
Menu:	ATHENA > Draw > Semi-finished product
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_hlbz

You use this routine to insert a semi-finished product into the current drawing. You can choose from twelve different shapes with any dimensions. Semi-finished products can be inserted and used in various 2D views or as a 3D object.

Dialog box Semi-finished product

In the dialog box you will find on the left side the Display section with the dynamic preview. On the right side you can see the Operating section with several tabs for the various functional areas.

The Properties tab is object-specific. The Administration tab is similar for most dialog boxes. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Display section

The preview displays the current semi-finished product. The preview is primarily used for a visual check, but offers other functions too:

Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.

To the left of the preview there are buttons with which you can adjust both the preview and also the insertion into the drawing. You will find further information on this in the section *Object views* on page 102.

Notes for insertion into the various views:

- When you insert a semi-finished product as side elevation from the left or right, only the base point and the rotation angle are interrogated. For further information see "Command prompt for semi-finished product sections" on page 265.
- If you insert a semi-finished product as 2D plan view, 2D bottom view, 2D front elevation or 2D rear elevation, the profile length is interrogated on insertion. For further information see "Command prompt for semi-finished products" on page 265.
- If you insert a profile as solid, the profile length is interrogated. For further information see "Command prompt for semi-finished products as 3D solids" on page 266.
- The solid is a "known" bar assembly, the name of which consists of the TMP and an incremented number. You will find further information on bar assemblies in the sections *Bar Assembly Manager* on page 632 and *Apply bar assembly* on page 738.

Dialog box section Display**Center lines**

Switches center lines on or off.

Hidden lines

Switches the hidden lines on or off.

Contour edge

Switches the contour edges for semi-finished products on or off. This option is only available for semi-finished products with roundings.

Outline

Creates a semi-finished product without infill.

Filled

Creates a filled semi-finished product.

Hatched

Creates a semi-finished product with hatched infill. When you select a material, the material-dependent hatching is used.

Layer

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Operating section**Tab Properties****Height**

Defines the height of the semi-finished product. With this button the dialog box Semi-finished product is temporarily closed and you can define the height by choosing two points.

Width

Defines the width of the semi-finished product. With this button the dialog box

Semi-finished product is temporarily closed and you can define the width by choosing two points.

Thickness

Defines the thickness of the semi-finished product. With this button the dialog box Semi-finished product is temporarily closed and you can define the thickness by choosing two points.

Sharp

Creates a sharp-edged semi-finished product (as extruded profile without radii).

Canted

Creates a canted semi-finished product (folded sheet-metal part). With this option you can define a bending radius. This option is not available for all variants.

Rounded

Creates a rounded semi-finished product (similar to rolled steel section). With this option you can define an inner and outer radius. This option is not available for all variants.

Bending radius

Defines the bending radius. The input field is only available if you have activated the option Edged.

Inner radius

Defines the inner radius. The input field is only available if you have activated the option Edged.

Outer radius

Defines the outer radius. The input field is only available if you have activated the option Edged.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Fold

Shows the folding tables which have been assigned to the material. Here you can select a table with bending allowances, so that the correct development for the folded semi-finished product can be calculated.

[...]

Opens the Dialog box Sheet metal computation values, where you can edit the tables with the allowance values.

You will find further information about this in the Chapter *Sheet metal computation values* on page 114.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Dialog box section Variant

Here you define the type of semi-finished product. You can choose between the following profile variants:

- Square profile section
- Square tube
- Round profile section
- Round tube
- Hexagonal section
- Octagonal profile

- L-section
- U-channel
- T-section
- I-section
- Z-section
- F-section
- Rectangular tube with web

Tab Manager

You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.

End of program

Cancel

Terminates the dialog box without inserting a semi-finished product.

OK

Terminates the dialog box, the semi-finished product is attached to the cross-hairs and the following input request appears for insertion:

Command prompt for semi-finished product sections

Specify insertion point:

Use the mouse or enter coordinates to specify the insertion point of the part.

You can change between different insertion points by repeatedly pressing the Ctrl key.



The grips of the relevant profile cross section can be used as insertion points. The insertion point activated by Ctrl only applies for the duration of the insertion.

Specify rotated angle<0>:

Use the mouse or enter an angle to specify the rotation angle. Press the Enter key to accept the default angle.

Command prompt for semi-finished products

[Object Mode]

Select axis or [Points/?]:

Choose a system axis with the mouse.

*Select **Option Points** to specify the start and end points of the axis.*

You call the help with the option ?.

Option Points

[Points mode]

Specify the axis start point or [Object/?]:

Use the mouse or enter co-ordinates to specify the axis start point.

Choose the option Objects when you want to select an axis.

Specify the axis end point or [Undo/Object/?]:

Use the mouse or enter co-ordinates to specify the axis end point.

You select the option Undo to start the point entry anew.

Bar orientation [Mirror/?]:

The option Mirror mirrors the bar about its own axis.

Command prompt for semi-finished products as 3D solids

[Object Mode]

Select axis or [Points/?]:

Choose a system axis with the mouse.

Select **Option Points** to specify the start and end points of the axis.

You call the help with the option ?.

Option Points

[Points mode]

Specify the axis start point or [Object/?]:

Use the mouse or enter co-ordinates to specify the axis start point.

Choose the option Objects when you want to select an axis.

Specify the axis end point or [Undo/Object/?]:

Use the mouse or enter co-ordinates to specify the axis end point.

You select the option Undo to start the point entry anew.

Choose the option Objects when you want to select an axis.

Bar orientation [Mirror/ Rotate/ surfaceAxis/ surfaceMiddle/ rotationAngle/ Point/?]:

The options for changing the bar orientation are described in the section *Apply bar assembly* on page 738.

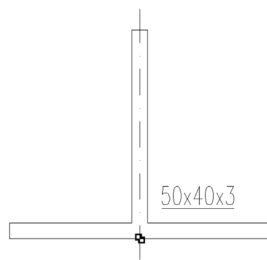


Fig. 1.15: Semi-finished product (T-profile variant)

Notes

- The generated semi-finished products can be retrospectively labeled with the command **Parts labeling**.
- You can change the properties of the semi-finished product with the command **Modify ATHENA** or by a double click.
- With the command **Object to solid body** you can convert the semi-finished products to solids.
- With the command **Projection objects** you can generate 2D projections of semi-finished products. When you modify a semi-finished product, all projections are similarly modified.
- Similar to 3D bars, 2D projections of semi-finished products can also be trimmed with the command **Cutting**.
- Similar to 3D bars, 2D projections of semi-finished products can also be sliced with the command **Slice Bar**.

1.24 Drilled hole



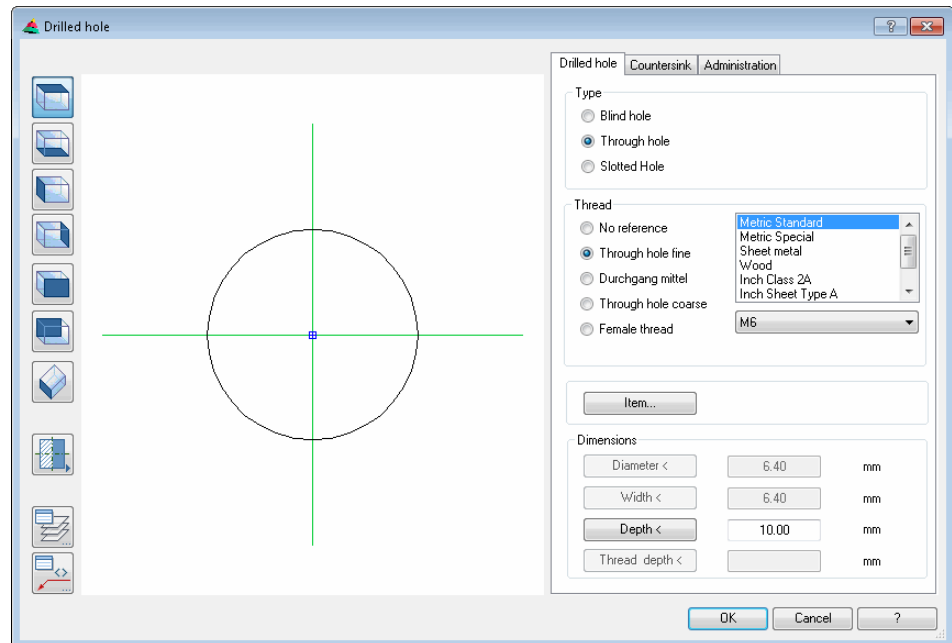
Ribbon: Tab ATHENA > Group Standard parts > Drilled hole
Menu: ATHENA > Draw > Drilled hole
Toolbar: ATH Draw and ATHENA Draw
Command input: ath_drill

You can create various types of hole with this command. Drilled holes can be inserted and used in various 2D views or as 3D objects.

Dialog box Drilled hole

In the dialog box you will find on the left side the Display section with the dynamic preview. On the right side you can see the Operating section with several tabs for the various functional areas.

The Properties tab is object-specific. The Administration tab is similar for most dialog boxes. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Display section

The preview displays the current drilled hole. The preview is primarily used for a visual check, but offers other functions too:

Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.

To the left of the preview there are buttons with which you can adjust both the preview and also the insertion into the drawing. You will find further information on this in the section *Object views* on page 102.

Display button

If you move the mouse over this button, a menu with the following options opens up:

Center lines

Switches hole center lines on or off.

Layer button

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Label button

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Operating section

Tab Drilled hole

Dialog box section Type

Blind hole

Creates a blind hole.

Through hole

Creates a through hole.

Slotted hole

Creates a slotted hole.

Dialog box section Thread

No reference

Creates a drilled hole without any reference. The dimensions can be freely defined.

Through hole fine

Creates a through hole in the fine tolerance class.

Through hole medium

Creates a through hole in the medium tolerance class.

Through hole coarse

Creates a through hole in the coarse tolerance class.

Note on tolerance classes: For metric threads the hole diameter is defined to tolerance classes according to DIN EN 20898. With other types of thread the diameter is subject to 10%, 20% or 30% depending on the tolerance class.

Female thread

Creates a threaded hole.

List of threads

Defines the type of thread. The following are available:

- Metric standard
- Metric special
- Sheet metal
- Wood
- Inch Class 2A
- Inch sheet metal Type A
- Inch sheet metal Type AB
- Inch wood

Pick list Diameter

Defines the thread diameter. This depends on the selected thread.

Dialog box section Dimensions

Diameter

Defines the diameter of the hole. With this tick box the dialog box Drilled hole is temporarily closed and you can define the diameter by choosing two points.

Width

Defines the width of the slotted hole. With this tick box the dialog box Drilled hole is temporarily closed and you can define the width by choosing two points.

Depth

Defines the depth of the hole. With this tick box the dialog box Drilled hole is temporarily closed and you can define the depth by choosing two points.

Thread depth

Defines the thread depth of the hole. With this tick box the dialog box Drilled hole is temporarily closed and you can define the thread depth by choosing two points.

Dialog box section Item

Item

Opens the Dialog box Item, where you can specify an item number (or process number) for the hole. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Register button Counterbore

None

Creates a drilled hole without any counterbore.

Standard

Creates a drilled hole with standard counterbore. Choose the required standard from the list.

Cylindrical

Creates a cylindrical counterbore. The diameter and counterbore depth can be freely defined.

Conical

Creates a conical counterbore. The angle and counterbore depth can be freely defined.

Diameter

Defines the diameter for cylindrical counterbores. With this tick box the dialog box Drilled hole is temporarily closed and you can define the diameter by choosing two points.

Angle

Defines the angle for conical counterbores.

Depth

Defines the depth of the counterbore. With standard counterbores the default depth is suggested. With this tick box the dialog box Drilled hole is temporarily closed and you can define the depth by choosing two points.

Item

Opens the Dialog box Item, where you can specify an item number (or process number) for the counterbore. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Command prompt

Specify insertion point:

Use the mouse or enter co-ordinates to specify the insertion point of the hole.

Specify rotated angle<0>:

Use the mouse or enter an angle to specify the rotation angle.

Press the Enter key to accept the default angle.

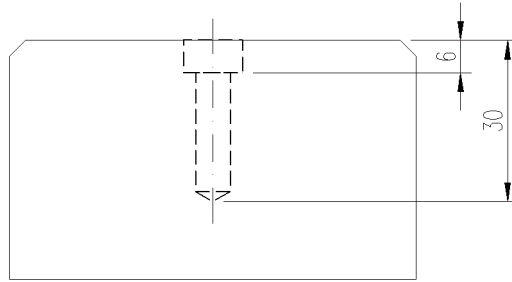


Fig. 1.16: Blind hole with counterbore

Notes

- Drilled holes can be retrospectively labeled with the command **Parts labeling**.
- You can change the hole properties with the command **Modify ATHENA** or by a double click.
- With the command **Object to solid body** you can convert the drilled holes to solids.
- With the command **Projection objects** you can generate 2D projections of drilled holes.

1.25 Pipe

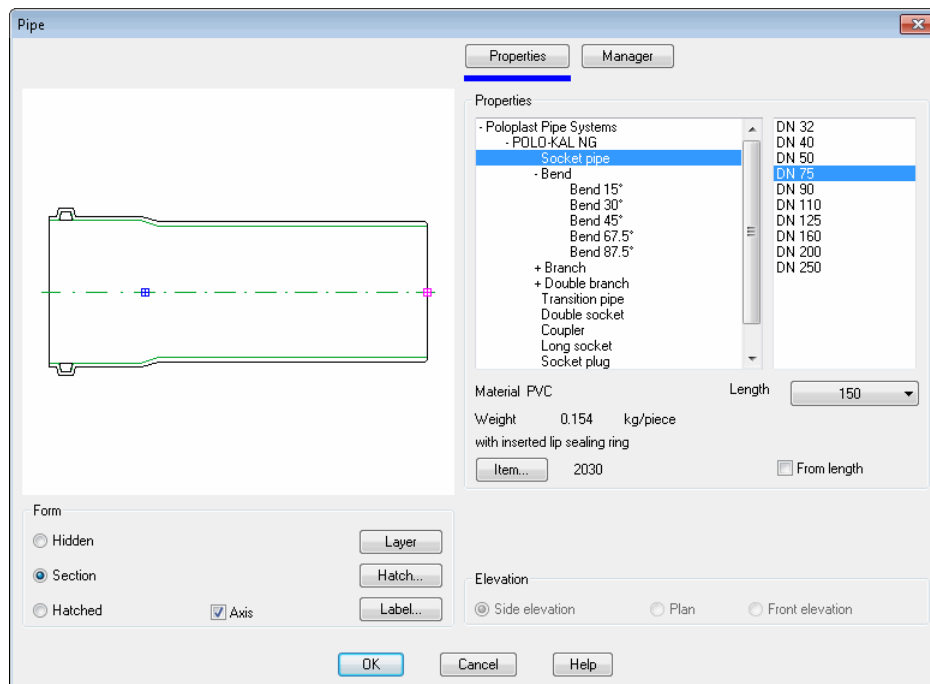


Ribbon: Tab ATHENA > Group Standard parts > Pipe
Menu: ATHENA > Draw > Pipe
Toolbar: ATH Draw
Command input: ath_pipe

You can create pipes with this command. In the dialog box products from various manufacturers are made available and can be inserted into the drawing and combined.

Dialog box Pipe

The dialog box contains the register buttons Properties and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Register button Properties

The selected pipe is illustrated at the upper left as a preview. The preview largely corresponds to the actual pipe in the drawing.

Dialog box section Display

Hidden

Creates a pipe in a hidden view.

Section

Creates a pipe in a sectioned view.

Hatched

Creates a pipe in a sectioned view with a hatch.

Center lines

Turns center lines of the pipe on or off.

Layer

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Dialog box section Properties**List**

Shows the available pipe elements in a tree structure. Here you can select the required pipe element with the mouse. Closed branches in the tree structure are identified with +. Open branches are identified with -. Another list on the right side offers a choice of available nominal sizes.

Length

Defines the length, if you have selected pipe elements.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

From length

Changes the insertion method and the behavior when stretching pipes. When the tick box is deactivated, the pipe is inserted with the selected length and cannot be extended beyond the selected length by stretching. With the tick box activated you can change the length after insertion. The pipe can also be stretched beyond the selected length up to the maximum available length.

Dialog box section Elevation**Side elevation**

Produces the side view of the selected pipe.

Front elevation

Produces the front view of the selected pipe.

Plan

Produces the plan view of the selected pipe.

The dialog box is closed with OK and you can insert the selected pipe into the drawing. Then follows the input request:

Command prompt

Specify insertion point or [Connection point/?]:

Use the mouse or enter coordinates to specify the insertion point of the pipe element.)

*You can connect the pipe element to an existing one with the **Option Connection point**. A further input request follows.*

You call the help with the option ?.

Specify rotation angle or [Connection point/?]: <0>:

Use the mouse or enter an angle to specify the rotation angle.

Specify length [150...3000] or [?] <500>:

Use the mouse or enter co-ordinates to specify the length.

Option Connection point

Select object:

Select an existing pipe element on the desired side.

Specify a point for changing the connection or [Rotate/Move/?]:

Specify a point to change the connection (for example with branching). To do this, with the mouse click the appropriate connection side of the pipe element just inserted. This input request is repeated until you choose another option or terminate the command by pressing the Enter key.

With option Rotate you can rotate bends or sleeves in the other direction.

*With the **Option Move** you can move the pipe element along the pipe alignment.*

Option Move

Specify distance or [?] <0>:

Use the mouse or enter a value to specify a distance.

1.26 Projection Outline



Ribbon:	Tab ATHENA > Group Draw > Projection outline
Menu:	ATHENA > Draw > Projection outline
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_proj

With this command you project the closed outlines in any direction or along a path. ATHENA examines the selected outlines and automatically recognizes external outlines (visible) and internal outlines (hidden).

Command prompt

Select outline or [Settings/?]:

Use the mouse to choose the outline to be projected.

*Use the **Option Settings** to open the Dialog box Projection outline and adapt the projection settings.*

You call the help with the option ?.

Select outline or [?]:

Use the mouse to choose further outlines to be projected. This input request is repeated. Press the Enter key to terminate the object selection.

Specify start point or [Object/?]:

Use the mouse or enter coordinates to specify the projection start point.

*Use the **Option Object** to select an object.*

Specify next point or [Direction/Object/Undo/?]:

Specify a further projection point.

*Use the **Option Direction** to change the projection direction.*

*Use the option **Undo** to repeat the last input request.*

Specify next point or [Direction/Object/Undo/?]:

Define further projection points or select an option. The input request is repeated. Press the Enter key to terminate the object selection.



The length of the projection becomes effective on the stated path. This should be taken into account when specifying the starting point.

Option Direction

Specify direction or [Undo/?]:

Use the mouse or enter coordinates to specify the direction of the projection.

Specify length or [Undo/?]:

Define the length of the projection with the mouse or by entering a length.

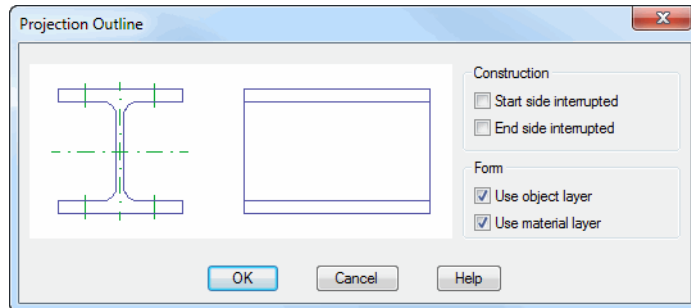
Option Object

Select object for path or [Points/?]:

Select an object (line, polyline or arc), which specifies the path for the projection.

Option Settings

Dialog box Projection outline



Dialog box section Construction

Start side interrupted

Creates a projection with interrupted start side.

End side interrupted

Creates a projection with interrupted end side.

The tick boxes can also both be switched off or on.

Dialog box section Display

Use object layer

Uses the current layer of the object for the projection.

Use material layer

Uses the layer defined for the material for the projection. This option only functions with ATHENA objects if you have assigned a material to them.



When both tick boxes are deactivated, Layer 0 is used for visible edges.

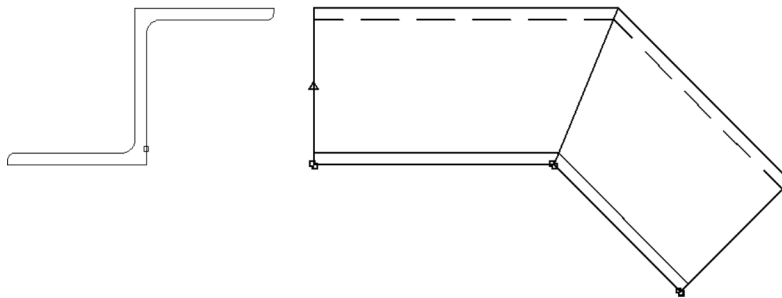


Fig. 1.17: Projection

Notes

- You can project the following object types:
 - Polylines (these must be closed or the end points must touch.)
 - Circles
 - Polylines and circles in blocks, XRefs and groups
 - ATHENA objects (e.g. sheet metal section)
- You can define the layers for the outline and hidden edges in the ATHENA options (see Chapter *Layer* on page 85.)

1.27 Projection objects



Ribbon:	Tab ATHENA > Group Draw > Projection objects
Menu:	ATHENA > Draw > Projection objects
Toolbar:	ATH Draw
Command input:	ath_obj_project

With this command you can generate orthogonal, two-dimensional projections of standard parts, semi-finished products, screwed joints and drilled holes.

Command prompt

Select objects:

Select the objects from which you want to generate an orthogonal projection. This input request is repeated until you terminate the object selection by pressing the Enter key.

Specify the side and starting point for the projection or [?]:

Determine the side and starting point of the projection.



The specified point determines the direction of projection as well as the distance of the projection to the selected object.

You call the help with the option ?.

Specify bar length or [?] <xx>:

Determine the bar length of the projection.



This input request only appears when the selected object is a profile section (standard part or semi-finished product). If the selected object already possesses a projection with length, then this length is used.

Notes

- If you select several objects with different alignment, subsets of the objects are found which possess a common perpendicular. The projection is executed for the largest subset.
- The original and projection are coupled to one another. This means that: If you modify a projection by a double click, all coupled projections are updated. This also applies to the display options (e.g. axes and thread lines).
- When you change the position of a projection (e.g. rotation or displacement), this has no effect on the coupled projections.
- With **Parts count** projected objects are not counted more than once.
- With the command **Object to solid body** you can convert the projections to solids.
- Similar to 3D bars, 2D projections of profile sections can also be trimmed with the command **Cutting**.
- Similar to 3D bars, 2D projections of profile sections can also be sliced with the command **Slice Bar**.

1.28 Space projection



Ribbon: Tab ATHENA > Group > Draw
Menu: ATHENA > Draw > Space projection
Toolbar: ATH Draw
Command input: ath_k3d_project

Projects a plan area (optionally with islands) by entering three height points and creates an area in space.

The can be used, for example, to generate a grid division on a sloping roof surface.

When you start the command, the following appears:

Command prompt

Select external outline:

Select the external outline of the surface to be projected.

Select island:

Select an internal outline of the surface to be projected.

This input request is repeated. Press the Enter key to pass on to the next query once you have selected all the islands.

Specify first point in height:

Select the first point to be defined in height.

Specify height:

Enter the height of the previously selected first point.

Specify second point in height:

Select the second point to be defined in height.

Specify height:

Enter the height of the previously selected second point.

Specify third point in height:

Select the third point to be defined in height.

Specify height:

Enter the height of the previously selected third point.

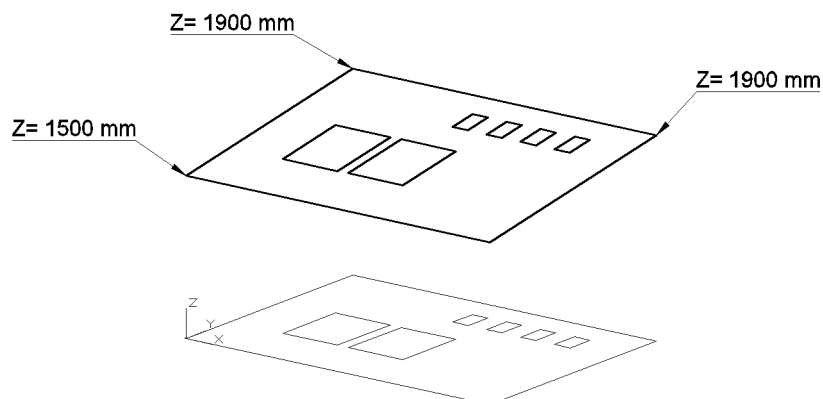


Fig. 1.18: Plan area and projected surface in space

1.29 Center lines



Ribbon:	Tab ATHENA > Group Draw > Center lines
Menu:	ATHENA > Draw > Center lines
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_dimcen

With this command you can produce the center lines of a circle or arc. Center lines are ARX objects which are associatively coupled to the respective circle or arc.

When you start the command, the following appears:

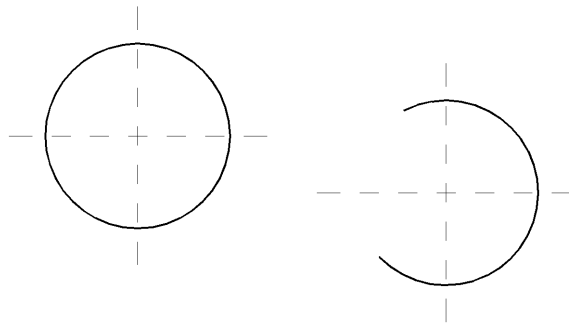


Fig. 1.19: Center lines

Command prompt

Overlap: 4

Select arc or circle or [Settings/?]:

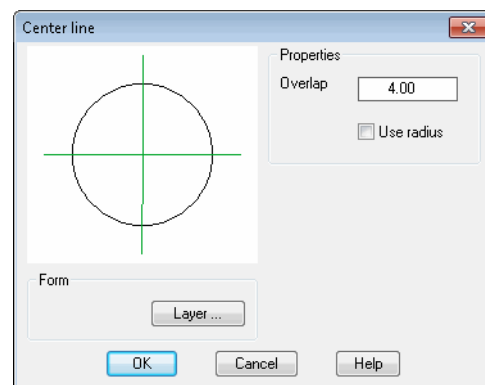
The current overlap is displayed. Select the arc or circle for which center lines are to be created.

*Use the **Option Settings** to change the overlap or the layer of the center lines.*

You call the help with the option ?.

Option Settings

Dialog box Center lines



Overlap

Defines the length of the overlap of the center line.

Use radius

Uses the radius of the selected circle as the overlap.

Layer

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Notes

- If you move the circle or change its radius or diameter, the center lines adapt automatically.
- To carry out changes to the center lines, you can double click them.
- You can define the layer default settings for the center lines in Dialog box System layer under Dimension.
- You can define the default for the overlap in Dialog box ATHENA options under Settings.

1.30 Axis



Ribbon:	Tab ATHENA > Group Draw > Axis
Menu:	ATHENA > Draw > Axis
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_cline

With this command you can create axes with an overlap.

When you start the command, the following appears:

Command prompt

Option Points

Specify start point or [Object/Settings/?]:

Use the mouse or enter coordinates to specify the axis start point.

*Use **Option Object** to convert an object to an axis.*

*Use the **Option Settings** to open the Dialog box Axis and adapt the axis settings.*

You call the help with the option ?.

Specify next point or [Arc point/Undo/Direction/ANGLE/Length/?]:

Use the mouse or enter coordinates to specify the next point of the axis.

*With the **Option Arc point** you can specify an arc point. Further input requests follow.*

Choose the option Undo to repeat the last input request.

*With the **Option Direction** you can specify a direction. Further input requests follow.*

*With the **Option Angle** you can specify an angle. Further input requests follow.*

*With the **Option Length** you can specify a length. Further input requests follow.*

Option Object

Select object or [Points/Settings/?]:

Select the object which you want to convert into an axis.

*With the **Option Points** you can create an axis by specifying points.*

Option Arc point

Specify arc point or [Line/Undo/?]:

Use the mouse or enter coordinates to specify an arc point.

Specify endpoint of the arc or [Undo/Line/Arc point/?]:

Use the mouse or enter coordinates to specify the end point of the arc.

Option Direction

Specify direction or [Length/Angle/Point/Undo/?]:

Use the mouse or enter coordinates to specify the direction of the axis.

Option Angle

Specify angle or [Length/Direction/Point/Undo/?]:

Use the mouse or enter coordinates to specify the axis angle.

Option Length

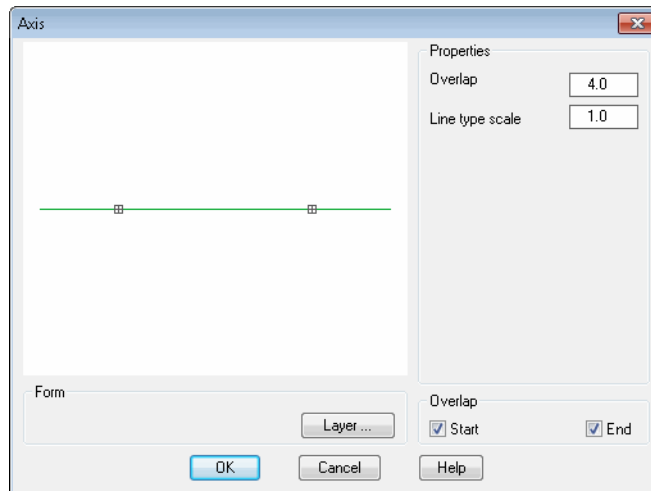
Specify length or [Direction/Angle/Point/Undo/?]:

Specify the length of the axis.

Option Settings

Opens the dialog box Axis, where you can adapt further properties.

Dialog box Axis



Dialog box section Properties

Overlap

Defines the length of the overlap of the axis.

Line type scale

Defines the line type scale of the axis.

Layer

Replaces the previously set layer of the axis.

Dialog box section Overlap

Start

Creates an axis with overlap on the start side.

End

Creates an axis with overlap on the end side.

Notes

- To carry out changes to the axes, you can double click them.
- You can define the layer default settings for the axes in Dialog box System layer under Dimension.
- You can define the default for the overlap in Dialog box ATHENA options under Settings.

1.31 Section symbol



Ribbon:	Tab ATHENA > Group Draw > Section symbol
Menu:	ATHENA > Draw > Section symbol
Toolbar:	ATH Draw
Command input:	ath_cssym

With this function you create the section symbols which can be modified similar to polylines with grips or by stretching.

Command prompt

Option Point

Specify start point or [Object/Settings/?]:

Use the mouse or enter coordinates to specify the start point of the section symbol.

*You can convert a line or polyline into a section symbol with the **Option Object**.*

With the Settings option you open the Dialog box Section symbol, where you can make further settings.

You call the help with the option ?.

Specify next point or [Undo/Direction/ANgle/Length/?]:

Define the next point of the section line. This input request is repeated until you terminate the point specification by pressing the Enter key.

You can repeat the last input with the option Undo.

*With **Option Direction** you can specify the direction for the section line. Further input requests follow.*

*You specify the length of the section line with the **Option Length**. Further input requests follow.*

*You specify the angle of the section line with the **Option Angle**. Further input requests follow.*

Specify side for symbol or [?]:

Define the side on which the section symbol is to be drawn.

Option Object

Select object or [Points/?]:

Select the object which you want to convert into a section symbol.

*With the **Option Point** you define the section line by specifying points.*

Option Direction

Specify direction or [Length/Angle/Point/Undo/?]:

Specify the direction of the section line or select an option. Further input requests follow.

Option Length

Specify length or [Direction/Angle/Point/Undo/?]:

Specify the length of the section line or select an option. Further input requests follow.

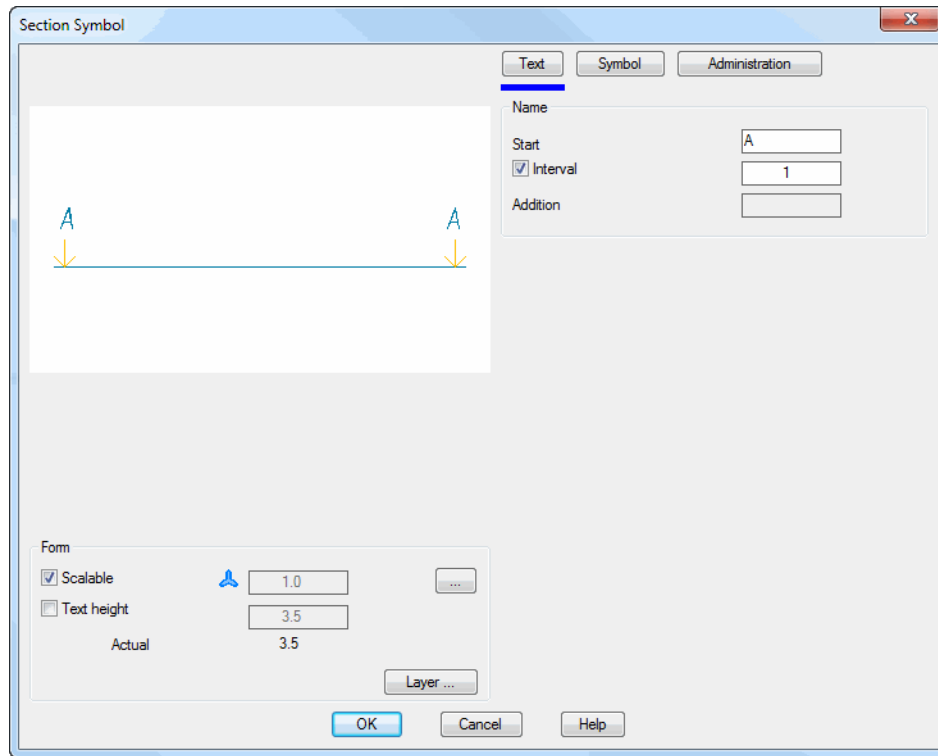
Option Angle

Specify angle or [Length/Direction/Point/Undo/?]:

Specify the angle of the section line or select an option. Further input requests follow.

Dialog box Section symbol

In the dialog box you will find on the left side the display section with the preview. On the right side you can see the register buttons Text, Symbol and Manager. The text and symbol properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Dialog box section Display

Scalable

Switches the scalability of section symbol on or off. You can change the current scale factor in the input field.



With scalability activated, the section symbol is scaled depending on the annotation scale. It is not possible therefore to change the scale factor.

[...]

Opens the Dialog box Label scales, where you can assign scales to the section symbol. You will find further information in the Chapter *Label Scale* on page 120.

Text height

Activates or deactivates the replacement of the text height. If Text height is switched on, you can define the text height in the input field.

The actual text height is shown for information. It is calculated from the text height and the scale factor (labeling scale).

Layer

Opens the Dialog box Layer assignment, where you can adapt the layer properties for the section symbol. You will find further information on this in the section *Layer assignment* on page 108.

Register button Text
Dialog box section Name**Start**

Defines the variable part of the section symbol. You can enter figures and also letters. Special characters are not permissible.

Interval

Defines the intervals between the section symbol numbers. If, for example, you enter "10" here, the numbering occurs with the start "001" as follows: 001, 011, 021...

Addition

Defines a non-variable supplementary text for the section symbol.



Additional texts can only be defined when a two-part symbol has been selected.

Register button Symbol
Dialog box section Symbol

Defines the symbol. The selected symbol is displayed in red. The position and arrow tip can be affected depending on the selected symbol.

Dialog box section Position**Over line**

Positions the section symbol above or below the section line or at the end of the arrow tip.

By line

Positions the section symbol in the extension of the section line.

<->

Reverses the direction (over or under the section line) of the section symbol and of the arrow tip.

Dialog box section Format

Dimension style

Switches in replacement of the dimension style. If you activate this tick box, you can select a dimension style from the list. With the tick box deactivated the current dimension style is set automatically.

Replace style

Text color

Switches in replacement of the text color. When you activate this tick box, you can define a new text color by clicking the color button. With the tick box deactivated the text color set in the dimension style is used automatically.

Arrow tip

Switches in replacement of the arrow tip. With this tick box deactivated the arrow tip set in the dimension style is used automatically.



As standard, the arrow tip defined in the dimension style is not used for the section symbol, but rather an open, right-angled arrow tip!

Arrow size

Switches on replacement of the arrow size. When you activate this tick box, you can define an arrow size in the input field. With the tick box deactivated the arrow size set in the dimension style is used automatically.

Dialog box section Arrow type

Activates the use of an arrow tip.



This option is only available for closed symbols.

Pick list

Defines the shape of the arrow tip.

Dialog box section Representation

Start

Activates the section symbol at the start point of the section line.

End

Activates the section symbol at the end point of the section line.

OK closes the dialog box and the input request is displayed.

1.32 Welded seam

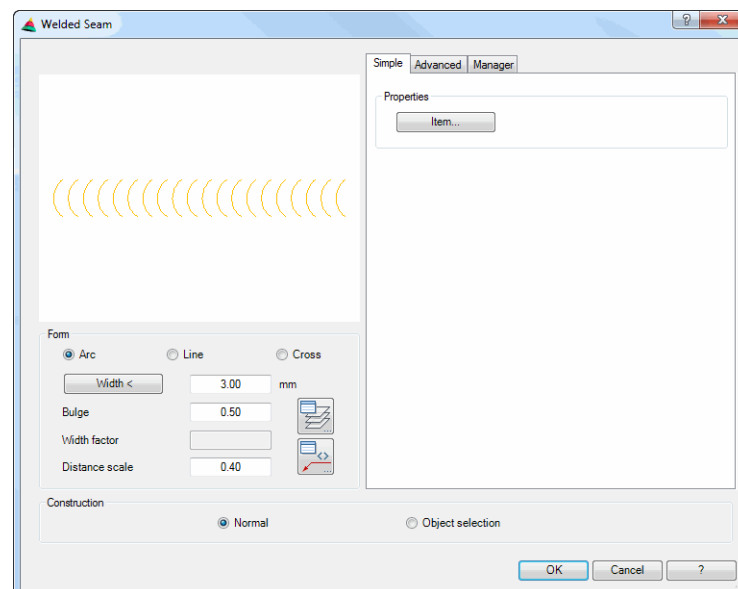


Ribbon:	Tab ATHENA > Group Draw > Welded seam
Menu:	ATHENA > Draw > Welded seam
Toolbar:	ATH Draw and ATHENA Draw
Command input:	ath_weld

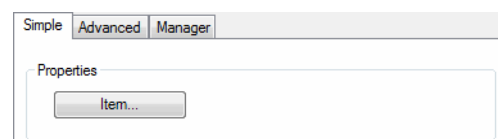
With this function you create the weld seams which can be modified similar to polylines with grips or by stretching.

Dialog box Welded Seam

The dialog box contains the tabs Simple, Expanded and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Tab Simple



Dialog box section Properties

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Tab Expanded

Dialog box section Type

Defines the type of seam. Choose a seam type from the list.

Dialog box section Dimensions

Nominal size

Defines the nominal size of the welded seam.

a

Defines the nominal size of the seam thickness.

z

Defines the nominal size of the limb length.

Interrupt

Creates a non-continuous welded seam.

Length

Defines the length of the welding bead.

Distance

Defines the distance between the individual welding beads.

Dialog box section Alignment

Start

Aligns the welded seam at the starting point.

Centered

Aligns the welded seam centered between the start and end points.

End

Aligns the welded seam at the end point.

Dialog box section Display

Arc

Creates a welded seam with arcs.

Line

Creates a welded seam with lines.

Cross

Creates a welded seam with crosses.

Width

Defines the width of the welded seam. With the button you can access the width of the welded seam in the drawing. For this the dialog box is temporarily closed.

Receptacle

Defines the bulge of the arc-shaped welded seam. The entered value is multiplied by the radius. Values from 0.01 to 1 are permissible.

Width factor

Defines the width factor of the cross-shaped welded seam.

Distance scale

Defines the distance scale of the welded seam.



Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.



Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Dialog box section Construction**Normal**

With the option Normal you create a welded seam by entering points.

Object selection

With the option Object selection you create a welded seam by selecting a polyline object.



A polyline object may be a polyline or also a sheet metal section, welded seam or membrane.

When you terminate the dialog box with OK, the following appears:

Command prompt

Type of construction option Normal

Specify starting point of welded seam or [Settings/?]:

Use the mouse or enter coordinates to specify the starting point of the welded seam.

*Use the **Option Settings** to change the properties of the welded seam.*

You call the help with the option ?.

[Line Mode]

Specify next point or [Undo/Circular arc/Direction/Length/?]:

Use the mouse or enter coordinates to specify the next point.

*Use the option **Undo** to repeat the previous input request.*

*Use **Option Arc** to create an arc-shaped welded seam. Further input requests follow.*

*Use the **Option Direction** to specify a direction. Further input requests follow.*

*Use the **Option Length** to specify a length. Further input requests follow.*

[Line Mode]

Specify side or [Central/Undo/?] <Central>:

Define the side on which the welded seam is to be created.

*Use the option **Central** to create the welded seam centrally.*

[Line Mode]

Specify next point or [Undo/Circular arc/Direction/Length/?]:

Specify the next point of the welded seam or select an option.

Option Arc

[Arc Mode]

Specify next point or [Undo/Close/Direction/second Point/Line/?]:

Specify the next point of the circular arc which is created tangentially to the positive X direction of the previous point.

*Use the **Option Second point** to specify the second point of the arc.*

Option Second point

[Arc Mode]

Specify arc point or [Undo/Line/Direction/?]:

Specify the second point of the arc with the mouse or select an option.

[Arc Mode]

Specify next point or [Undo/Close/Direction/second Point/Line/?]:

Specify the third point of the arc or select an option.

Option Direction

[Line Mode]

Specify direction or [Undo/Circular arc/Length/?]:

Define the direction of the welded seam with the mouse or by entering the coordinates or selecting an option.

[Line Mode]

Specify length or [Circular arc/Undo/?]:

Define the length of the welded seam with the mouse or by entering the length or selecting an option.

Option Length

[Line Mode]

Specify length or [Circular arc/Undo/?]: 30

Define the length of the welded seam with the mouse or by entering the length or selecting an option.

[Line Mode]

Specify direction or [Undo/Circular arc/Length/?]:

Define the direction of the welded seam with the mouse or by entering the coordinates or selecting an option.

Option Settings

Starts the Dialog box Welded Seam.

Command prompt

Type of construction option Object selection

Select Polyline or [?]:

Select the polyline object which is to be converted to a welded seam.

With the option ? you call up the help.

Specify side for wall thickness or [Central/?] <Central>:

Specify on which side the welded seam is to be drawn.

*The wall thickness is produced centrally to the welded seam with the option **Central**.*

Delete polyline [Yes/No/?] <Yes>:

*With the option **Yes** you delete the existing polyline.*

The polyline is retained with the option No.

Notes

- You can change a welded seam by stretching or with grips. When stretched, the welded seam behaves like a polyline.
- You can change the welded seam properties with the command **Modify ATHENA** or by a double click.
- You can automatically label the welded seam with the command **Parts labeling**.
- You can change the layer default settings for the welded seam in Dialog box Layer assignment.

1.33 Welded seam symbol

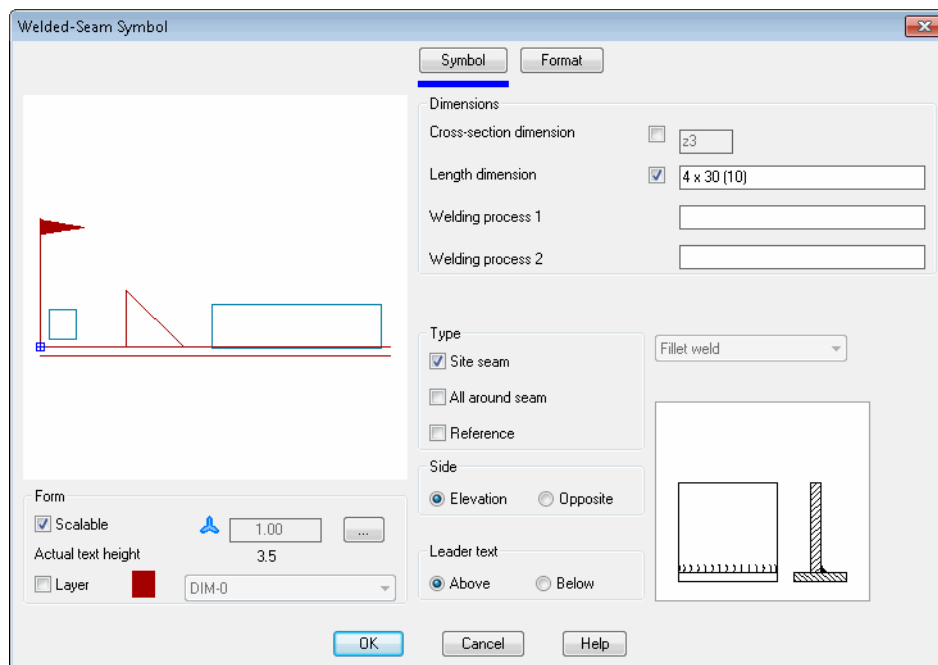


Ribbon: Tab ATHENA > Group Label > Welding symbol
Menu: ATHENA > Draw > Welding symbol
Toolbar: ATH Draw
Command input: ath_swsy

With this program you can create welded seam symbols as labels on a leader. ATHENA creates a welded seam label conforming to the standard EN 22553.

Dialog box Welded seam symbol

In the dialog box you will find on the left side the Display section with the preview. On the right side you will see the Operating section. You can activate the individual function areas with tabs.



Display section

The display section shows a schematic preview of the current welded seam symbol.

Below the preview you will find options for changing the display. You will find further information on this in the section *Display options for labels* on page 105.

Operating section

Register button Symbol

Dialog box section Dimensions

Cross-section dimension

Specifies the seam thickness (a or s) or the limb length (z) of the welded seam. You leave this field open to weld work pieces over the complete length.

Length dimension

Specifies the length of the welded seam.

Welding process

Specifies the code of the welding process on the symbol fork.



The assignment of the codes for the processes conforms to ISO 4063.

You can supplement the process figures in the symbol fork with further details for types of weld and dimensions. They must be separated by forward slashes and are to be specified in the following sequence:

- Process (e.g. according to ISO 4063)
- Assessment group (e.g. according to ISO 5817 and ISO 10042)
- Working position (e.g. according to ISO 6947)
- Additional materials (e.g. according to ISO 544, ISO 2560 and ISO 3581)

Examples



Fig. 1.20: Welding symbol (cross-section dimension and length dimension)



Fig. 1.21: Welding symbol (one/two welding processes)

Dialog box section Type

Site seam

Defines a site seam. This is taken to mean that the welding process is carried out on the construction site. This is indicated with a tag.

All round seam

Defines an all round seam. This is taken to mean that the welding process takes place over the circumference of the part. This is indicated with a circle.

Reference

Reference is taken to mean the reference to the welding process with a number which is explained on the drawing in the vicinity of the text field using legends. Enter the number in the input fields Welding process1 or Welding process 2.

Important: This does not work if you enter something in both fields.

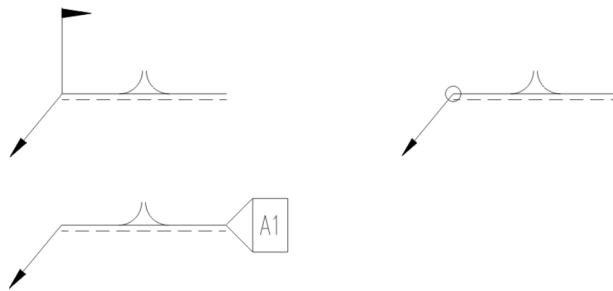


Fig. 1.22: Welding symbol (Site seam, All around seam, Reference seam)

Dialog box section Page

Here you define whether the welded seam is to be applied to the elevation side or opposite side.



Fig. 1.23: Welding symbol (Elevation side, Opposite side)

Dialog box section Leader text

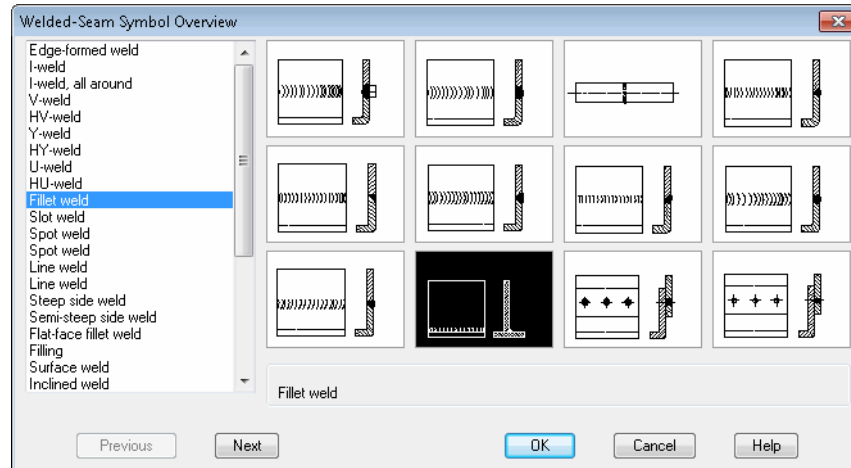
Here you define whether the leader text is to appear above or below the leader line.

Selecting a type of welded seam

When you select a welded seam from the list, it is depicted as a preview.

Alternatively, you can also click the graphical illustration and select a type of seam in the dialog box with the graphical overview. To do this, page through the views using the buttons Previous and Next until you find the desired welded seam. Then click the appropriate picture or the name at the left margin of the dialog box to mark the type of seam.

OK closes the overview and transfers the seam to the previous dialog box.

Dialog box Welded-seam overview**Register button Format**

You will find the functional description of the dialog box section in the section *Format settings for labels* on page 106.

End of program

When you quit the dialog box Welded-seam symbol with OK, the settings are saved for the duration of the drawing session. The positioning of the weld symbol occurs according to the same syntax as described in **Leader**.

Command prompt

Specify starting point of the leader line or [?]:

Specify the starting point (arrow tip of the leader line) of the welding symbol.

You call the help with the option ?.

Specify next point or [Undo/?]:

Define the next point of the welding symbol (kink point of the leader line). This input request is repeated. Press the Enter key to terminate the command.

*Use the option **Undo** to repeat the last input request.*

Notes

In this respect also note the comments on the command **Leader** on page 419.

1.34 Edge symbol

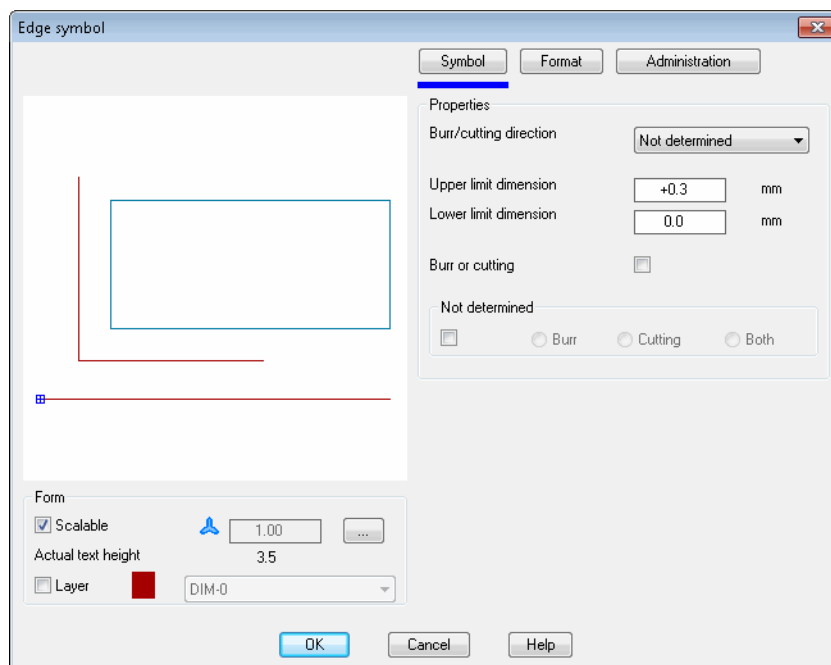


Ribbon: Tab ATHENA > Group Label > Edge symbol
Menu: ATHENA > Draw > Edge symbol
Toolbar: ATH Draw
Command input: ath_edgesym

With this command you can create symbols to define edge conditions of workpieces. The standard DIN ISO 13715 is taken as the basis for the edge symbols.

Dialog box Edge symbol

In the dialog box you will find on the left side the display section with the preview. On the right side you will see the operating section. You can activate the individual function areas with tabs.



Display section

The display section shows a schematic preview of the current edge symbol.

Below the preview you will find options for changing the display. You will find further information on this in the section *Display options for labels* on page 105.

Operating section

The screenshot shows the 'Properties' tab of the 'Edge symbol' dialog box. It contains the following fields and controls:

- Burr/cutting direction:** A dropdown menu currently showing 'Not determined'.
- Upper limit dimension:** An input field containing '+0.3' followed by a unit selector set to 'mm'.
- Lower limit dimension:** An input field containing '0.0' followed by a unit selector set to 'mm'.
- Burr or cutting:** A checkbox that is currently checked.
- Not determined:** A checkbox that is currently checked.
- Radio buttons:** Three radio buttons labeled 'Burr', 'Cutting', and 'Both' are located below the 'Not determined' checkbox. All three are currently unselected.

Register button Symbol

Dialog box section Properties

Burr/cutting direction

You determine whether the burr or cutting direction is to be indeterminate, horizontal or vertical.

Upper/lower limit dimension

Defines the upper or lower limit dimension in the respective input field.

Burr or cutting

If you activate the tick box Burr or cutting, you cannot specify a lower limit dimension.

Not determined

Activates the options Burr, Cutting or Both. If Not determined is switched on, you cannot set any limit dimension.

End of program

When you quit the dialog box Edge symbol with OK, the settings are saved for the duration of the drawing session. The positioning of the edge symbol occurs according to the same syntax as described in [Leader](#).

Command prompt

Specify starting point of the leader line or [?]:

Specify the starting point (arrow tip of the leader line) of the edge symbol.

You call the help with the option ?.

Specify next point or [Undo/?]:

Define the next point of the edge symbol (kink point of the leader line). This input request is repeated. Press the Enter key to terminate the command.

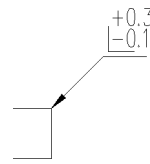
*Use the option **Undo** to repeat the last input request.*

Examples

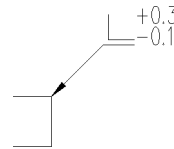
Input in the dialog box

Burr/cutting direction: not determined
Upper limit dimension: +0.3
Lower limit dimension: -0.1

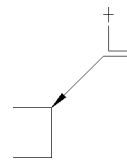
Symbol



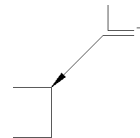
Burr/cutting direction: horizontal
Upper limit dimension: +0.3
Lower limit dimension: -0.1



Burr/cutting direction: vertical
Not determined: Burr



Burr/cutting direction: horizontal
Not determined: Cutting



Notes

In this respect also note the comments on the command *Leader* on page 419.

1.35 Surface symbol



Ribbon: Tab ATHENA > Group Label > Surface symbol
Menu: ATHENA > Draw > Surface symbol
Toolbar: ATH Draw
Command input: ath_surfsym

With this command you generate a symbol for stating the condition of surfaces. The standard DIN ISO 1302 is the basis for the symbols.

Dialog box Surface symbol

In the dialog box you will find on the left side the display section with the preview. On the right side you will see the operating section. You can activate the individual function areas with tabs.

Display section

The display section shows a schematic preview of the current surface symbol.

Below the preview you will find options for changing the display. You will find further information on this in the section *Display options for labels* on page 105.

Operating section

Register button Symbol

The screenshot shows a software dialog box titled 'Symbol' with three tabs: 'Symbol', 'Format', and 'Administration'. The 'Symbol' tab is active. Under the 'Properties' section, there are several configuration options: 'Production' is a dropdown menu set to 'Cutting production'; 'Roughness value' is an input field containing '1.6' with a 'µm' unit; 'to' is another input field with a 'µm' unit; 'Process' is an input field with a button '...'; 'Limit wavelength' is an input field containing '2.5' with a 'mm' unit and a button '...'; 'Symbol for groove direction' is a dropdown menu set to 'Crossing'; and 'Process allowance' is an input field containing '0.8' with a 'mm' unit. A language dropdown menu is also present, set to 'english (GB)'.

Dialog box section Properties

Production

Defines the type of surface production. You can choose as required, machining or non-machining production.

Roughness value

Specifies the maximum mean roughness value (R_a) in μm . In the second input field (to) if you also enter a mean roughness value, then this is taken as the maximum mean roughness value and the value previously entered in the field Roughness value is taken as the minimum mean roughness value.

Processing

Specifies a processing method (e.g ground). You can also save this text in various languages by changing the language. To do this, click the button [...].

Limit wavelength

Defines the part of the sampling length used for the evaluation.

Symbol for groove direction

Specifies a symbol for the groove direction.

Process allowance

Specifies the process allowance in mm.

End of program

When you quit the dialog box Edge symbol with OK, the settings are saved for the duration of the drawing session. The positioning of the surface symbol occurs according to the following syntax:

Command prompt

Specify insertion point:

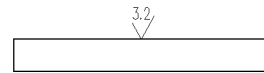
Use the mouse or enter coordinates to specify an insertion point of the surface symbol.

Specify rotated angle<0>:

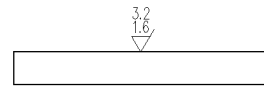
Specify the rotation angle of the surface symbol.

Examples**Input in the dialog box**

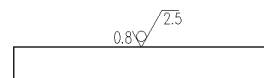
Production: any
 Roughness value: 3.2
 Symbol for groove direction: no
 symbol

Symbol

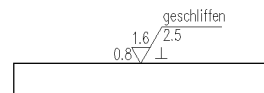
Production: Machined
 Roughness value: 1.6
 to:3.2
 Symbol for groove direction: no
 symbol



Production: Not machined
 Limit wavelength: 2.5
 Symbol for groove direction: no
 symbol
 Process allowance: 0.8



Production: Machined
 Roughness value: 1.6
 Process: ground
 Symbol for groove direction: vertical
 Process allowance: 0.8



2 Sheet metal

Commands in this section:

- Sheet metal section
- Change bending radius
- Window sill
- Profiled sheet
- Sheet processing
- Sheet development of a cross section
- Sheet development of two cross sections
- Sheet development from a primary surface

2.1 Sheet metal section

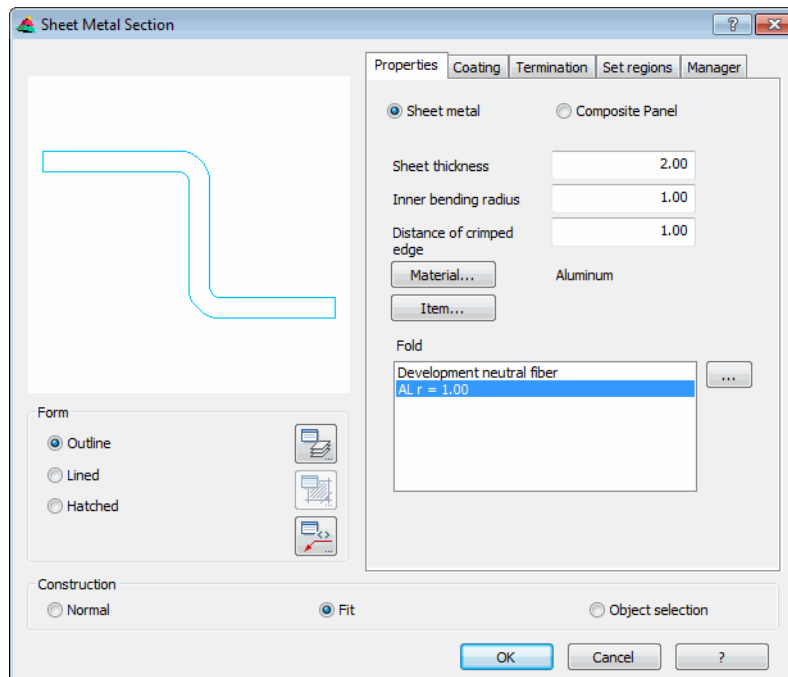


Ribbon: Tab ATHENA > Group Draw > Sheet metal section
Menu: ATHENA > Sheet metal > Sheet metal section
Toolbar: ATH Sheet Metal and ATHENA Draw
Command input: ath_blec_schnitt

With this function you create sheet metal sections or cross sections of composite panels. Similar to polylines, they can be changed with grips or by stretching. There are various construction possibilities available for creating the cross section.

Dialog box Sheet metal section

The dialog box contains the tabs Properties, Coating, Termination and manager. Properties, coating and termination are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Dialog box section Form

Outline

Creates a sheet metal section without infill.

Lined

Creates a lined sheet metal section.

Hatched

Creates a sheet metal section with hatched infill. The material-dependent hatching is used.

Hatch core layer

Turns the hatch infill of the core layer of the composite panel on or off. This option is only shown if composite panels have been selected.

Layer assignment

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch pattern assignment

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Dialog box section Construction**Normal**

Using the option Normal you create a sheet metal section in that you specify the scale, wall thickness, limb lengths and angles by coordinates.

Fit

With the option Fit you create a sheet metal section by clicking on snap points.

Polyline

With the option Polyline you convert a polyline into a sheet metal section.

Tab Properties**Sheet metal**

Activates the sheet metal properties.

Composite panel

Activates the composite panel properties.

Dialog box section Properties of sheet metal

The dialog box has tabs: Properties, Coating, Termination, Set regions, Manager. The 'Properties' tab is selected.

Options: ☒ Sheet metal, ☐ Composite Panel

Sheet thickness: 2.00

Inner bending radius: 1.00

Distance of crimped edge: 1.00

Material...: Aluminum

Item...

Fold:

- Development neutral fiber
- AL r = 1.00

Sheet thickness

Defines the thickness of the sheet metal.

Inner bending radius

Defines the inner bending radius. The bending radius is defined in the folding table. If no folding table is available for the specified bending radius, the development is calculated according to the neutral fiber.

Distance of crimped edge

Defines the distance between the edge of the sheet and the crimped edge.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Fold

Shows the folding tables which have been assigned to the material. Here you can select a table with bending allowances, so that the correct development for the folded semi-finished product can be calculated.

Optionally you can also select development to the neutral fiber here.



The partial length of the bend is corrected when calculating the development according to the neutral fiber. The correction factor (k factor) according to DIN 6935 depends on the bending radius and the sheet thickness and must in each case be calculated or read from a graph.

[...]

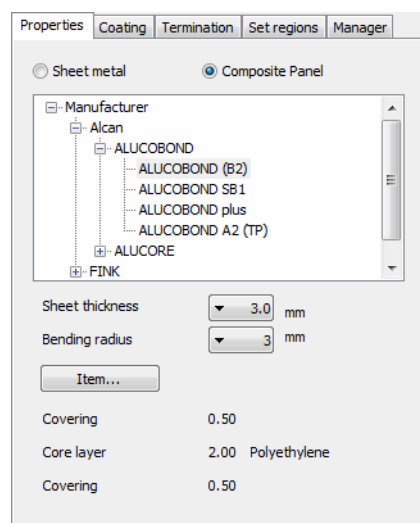
Opens the Dialog box Sheet metal computation values, where you can edit the tables with the allowance values.

You will find further information about this in the Chapter *Sheet metal computation values* on page 114.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Dialog box section Composite panel properties



List

Shows the available manufacturers and their products in a tree structure. Here you can select the required manufacturer with the mouse. Closed branches in the tree structure are identified with +. Open branches are identified with -.

Sheet thickness

Defines the thickness of the composite panel.

Bending radius

Defines the bending radius of the composite panel.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Layers

Here the thicknesses of the inner and outer covering layers are shown and also the core layer.

Tab Coating

Distance of coating line

Activates the replacement of the default distance of the coating line. In the input field you can define the distance of the input field to the sheet.

Area

Switches the coating line on for the viewed side.

Area

Switches the coating line on for the opposite side.

Face side

Switches the coating line on for the face side.

Tab Termination

Dialog box section Elevation side

Change elevation side

Changes the elevation side. This is particularly interesting for composite panels, because they are milled on one side.

Dialog box section Start

The tick box activates the termination at the start of the sheet metal section.



Changes the direction of the fold for the selected termination fold.



Defines a crimped edge as termination.



Defines a rolled edge as termination.



Defines a rolled edge with fold as termination.



Defines an interrupted termination.

Length

Defines the length of the terminating fold. This option is only available for the crimped edge and the rolled edge with fold.

Height

Defines the height of the terminating fold. This option is only available for the rolled edge and the rolled edge with fold.

Dialog box section End

The tick box activates the termination at the end of the sheet metal section.

The function of the buttons is identical to those in the Dialog box section Start.

End of program

When you click OK, the dialog box is terminated. An input request follows depending on the type of construction selected. The input requests for sheets and composite panels are identical.



You can terminate the Dialog box Sheet metal section directly after calling the command by pressing the Enter key to access the input request. In this way you have the possibility of very quickly creating several sheets with identical properties.

E.g.: You have just created a sheet with certain properties. Now press the Enter key twice (once to repeat the sheet cross-section command and a second time to terminate the dialog box immediately) and draw a further sheet with the same properties.

Command prompt *Normal type of construction*

Specify starting point of the sheet metal section or [?]:

Use the mouse or enter coordinates to specify the starting point of the sheet; see figure Sheet metal section option Normal P1.

You call the help with the option ?.

Specify next point or [Direction/Length/Undo/?]:

Specify the next point on the sheet; see figure Sheet metal section option Normal P2.

*With **Option Direction** you can change the direction for the limb. Further input requests follow.*

*With **Option Length** you can specify the length for the limb. Further input requests follow.*

*With the option **Undo** the previous input request is shown again to correct erroneous entries.*

Specify side for wall thickness or [Undo/Central/?] <Central>:

With the mouse click on the side on which the wall thickness of the sheet limb is to be built up (see figure Sheet metal section option Normal P3).

*The wall thickness is produced centrally to the sheet metal section with the option **Central**.*

Specify elevation side or [Undo/?]:

Click on the side which is to be defined as the elevation side. This input request only appears for composite panels, because these are milled on the opposite side.

Specify next point or [Angle/Direction/Length/Crimped edge/Undo/?]:

Specify the next point of the sheet metal section.

*With the **Option Angle** you can specify a fold angle. Further input requests follow.*

*With the **Option Crimped edge** you can specify a crimped edge. Further input requests follow.*

*This input request is repeated until you terminate the routine by entering **ENTER**.*

Option Direction

Specify direction or [Length/Undo/?]:

Point out the direction with the mouse.

Specify limb length or [Direction/Undo/?]:

Point out the length of the sheet limb with the mouse or enter a length.

*With the option **DIRECTION** you can correct the direction.*

Specify side for wall thickness or [Undo/Central/?] <Central>:

With the mouse click on the side on which the wall thickness of the sheet limb is to be built up (see figure Sheet metal section option Normal P3).

*The wall thickness is produced centrally to the sheet metal section with the option **Central**.*

Option Length

Specify limb length or [Direction/Undo/?]:

Point out the length of the sheet limb with the mouse or enter a length.

Specify direction or [Length/Undo/?]:

Point out the direction with the mouse.

Specify side for wall thickness or [Undo/Central/?] <Central>:

With the mouse click on the side on which the wall thickness of the sheet limb is to be built up (see figure Sheet metal section option Normal P3).

*The wall thickness is produced centrally to the sheet metal section with the option **Central**.*

Option Crimped edge

Specify side for crimped edge or [Undo/?] <Undo>:

Define the side on which the crimped edge is to be created.

Specify limb length or [?]:

Specify the limb length of the crimped edge.

Option Angle

Specify angle or [Direction/Length/Undo/?]:

Point out an angle with the mouse or enter an angle.

Specify limb length or [Angle/Direction/Undo/?]:

Point out the length of the sheet limb with the mouse or enter a length.

*With the option **Angle** you can correct the fold angle.*

Specify side for wall thickness or [Undo/Central/?] <Central>:

With the mouse click on the side on which the wall thickness of the sheet limb is to be built up (see figure Sheet metal section option Normal P3).

*The wall thickness is produced centrally to the sheet metal section with the option **Central**.*

Some queries are repeated until you input ENTER to terminate the program.

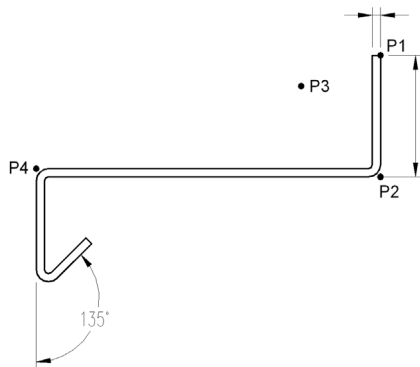


Fig. 2.1: Sheet metal section option Normal

Command prompt **Fit** type of construction

The input requests for the Fit type of construction are mostly identically with those of the Normal type of construction. The only difference is that you can specify the wall thickness anew for each sheet limb.

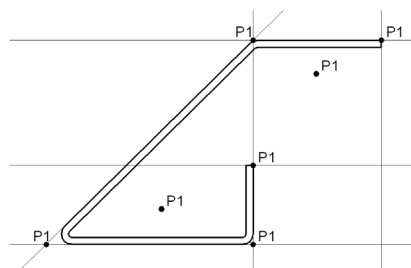


Fig. 2.2: Sheet metal section, Fit option

Command prompt **Polyline** type of construction

Select Polyline or [?]:

*With the mouse, select the polyline from which a sheet metal section is to be created. You call the help with the option **?**.*

Specify side for wall thickness or [Central/?] <Central>:

With the mouse click on the side on which the wall thickness of the sheet limb is to be built up.

*The wall thickness is produced centrally to the sheet metal section with the option **Central**.*

Delete polyline [Yes/No/?] <Yes>:

*With the option **Yes** you delete the existing polyline.*

*The polyline is retained with the option **No**.*

Notes

- You can modify sheet metal sections with grips or with the AutoCAD command **Stretch**.
- To change the properties (e.g. thickness) of a sheet section, use the command **Modify ATHENA** or double click on the sheet with the mouse.
- Use **Parts labeling** to label the sheet with a leader.

- For sheets only the materials are available for which a bending table has been assigned. You will find further information in the Chapter *Material* on page 81.

2.2 Change bending radius



Ribbon:	Tab ATHENA > Group Draw > Change bending radius
Menu:	ATHENA > Sheet metal > Change bending radius
Toolbar:	ATH Sheet Metal
Command input:	ath_sheet_rmod

With this command you can individually change the bending radii of sheet metal sections.

Command prompt

Select fold of the sheet section or [?]:

Select a sheet metal section in the vicinity of the fold whose radius you want to change.

You call the help with the option ?.

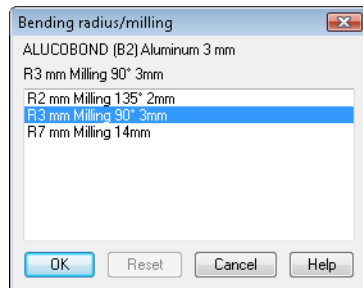
When you have selected a sheet metal section, the following input request appears and when you have selected a composite cross section (e.g. Alucobond), the Dialog box Bending radius/milling is opened.

Specify inner bending radius or [Reset/?] <1>:

Specify a new inner bending radius.

Select the option **Reset** if you want to reset the individually changed bending radii to defined values. This option is only available when bending radii have been individually changed on the selected sheet metal section.

Dialog box Bending radius/milling



List

Shows the available milling cutters for folding the composite panel. Select a milling cutter and click OK.

Reset

Resets the radius to the defined value of the composite panel.

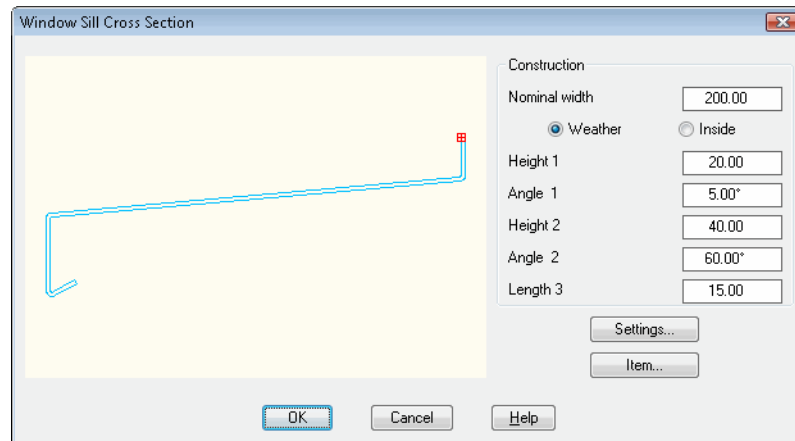
2.3 Window sill



Ribbon: Tab ATHENA > Group Draw > Window sill
Menu: ATHENA > Sheet metal > Window sill
Toolbar: ATH Sheet Metal
Command input: ath_fbank

With this function you create a sheet metal section in the shape of a sheet window sill section in the current drawing.

Dialog box Window sill section



Dialog box section Construction

With the option fields Weather and Inside you specify whether the insertion point of the window sill lies on the inner or outer side of the cross section. This option also affects the nominal width of the window sill. When you select the option Weather, the nominal width from outer edge to outer edge is measured (because the insertion point lies on the outer edge). When you select the option Inside, the nominal width is from outer edge to inner edge (because the insertion point lies on the inner edge).

Here, you define the lengths and angles of the window sill section. A dynamic preview can be seen on the right side of the dialog box. You can see the lengths and angles in the illustration Window sill cross-section option Weather. With obtuse angles the lengths refer to the intersection points and with acute angles they refer to the tangents of the corresponding folds.

Settings

Opens the dialog box Sheet metal section in which you can modify the sheet properties. You will find further information in the Chapter *Sheet metal section* on page 304.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

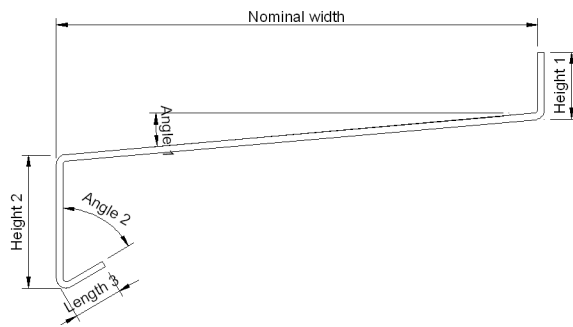


Fig. 2.3: Window sill cross-section option Weather

Command prompt

Specify insertion point:

Use the mouse or enter coordinates to specify the insertion point.

Specify rotated angle<0>:

Use the mouse or enter an angle to specify the rotation angle.

Press the Enter key to accept the default angle.

2.4 Profiled sheet

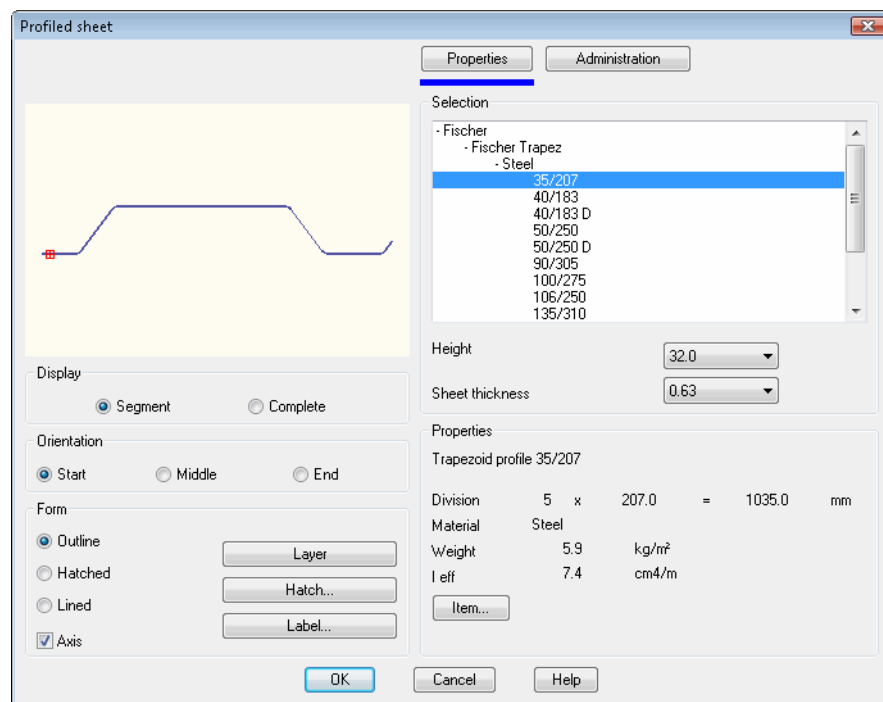


Ribbon: Tab ATHENA > Group Draw > Window sill
Menu: ATHENA > Sheet metal > Window sill
Toolbar: ATH Sheet Metal
Command input: ath_trapez

With this command you can create profiled sheets (trapezoidal or corrugated sheets). There is a selection of various manufacturers and their products.

Dialog box Profiled sheet

The dialog box contains the register buttons Properties and Manager. The properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Dialog box section Display

Segment

Shows the preview of the profiled sheet segment.

Complete

Shows the preview of a complete profiled sheet.

Dialog box section Alignment

Start

Aligns the profiled sheet at the starting point.

Middle

Aligns the profiled sheet in the middle.

End

Aligns the profiled sheet at the end point.

Dialog box section Display

Outline

Creates a profiled sheet without infill.

Hatched

Creates a profiled sheet with hatched infill. The material-dependent hatching is used.

Lined

Creates a lined profiled sheet.

Center lines

Turns center lines on or off.

Layer

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Register button Properties

Dialog box section Selection

List

Shows the available manufacturers and their products in a tree structure. Here you can select the required manufacturer with the mouse. Closed branches in the tree structure are identified with +. Open branches are identified with -.

Height

Defines the height of the selected profiled sheet.

Sheet thickness

Defines the thickness of the selected sheet metal.

Specify the manufacturer of the profiled sheet. The products from the selected manufacturer and their available sheet thicknesses are displayed and can be specified in the corresponding selection menu.

Dialog box section Properties

Displays the properties of the defined product. The main feature is the division and the resulting profile width.



Profiled sheets can be supplied with certain divisions and profile widths. The width of the inserted profile sheet is always divisible by the specified division. When the width is greater than the specified profile width, profile joints (overlaps) are automatically created.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Command prompt

Specify start point:

Use the mouse or enter coordinates to specify the starting point of the profiled sheet.

Specify the end point:

Use the mouse or enter coordinates to specify the end point of the profiled sheet.

Specify the side:

Using the mouse or by entering coordinates specify the side on which the profiled sheet is to be produced.

Notes

- You can modify sheet metal sections with grips or with the AutoCAD command **Stretch**.
- To change the properties (e.g. thickness) of a profiled sheet, use the command **Modify ATHENA** or double click on the profiled sheet with the mouse.
- Use **Parts labeling** to label the profiled sheet with a leader.
- Profiled sheets can be shortened at any point with the command **Slice objects**. After shortening, the profiled sheet can no longer be processed with grips (stretched).

2.5 Sheet processing

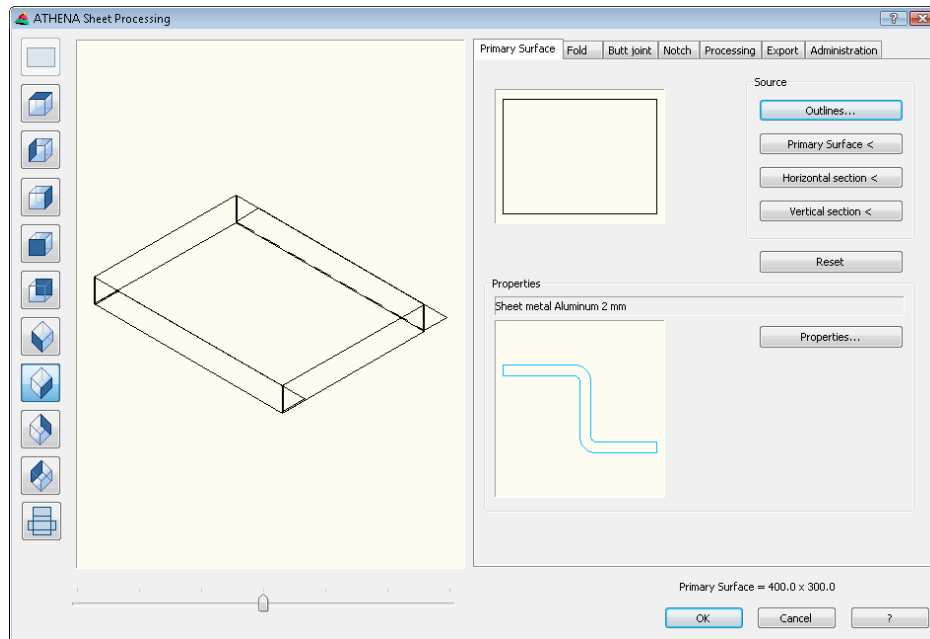


Ribbon: Tab ATHENA > Group Draw > Sheet processing
Menu: ATHENA > Sheet metal > Sheet processing
Toolbar: ATH Sheet Metal
Command input: ath_easy_sheet

With this program you can create, process and output the facade sheets. In a dialog box with various tabs you can define all the properties, such as dimensions, folds, joints and corner variants as well as additional processes. Then you can insert the result as a 3D model, development or sectional view in the drawing or generate DXF data.

Dialog box ATHENA Sheet Processor

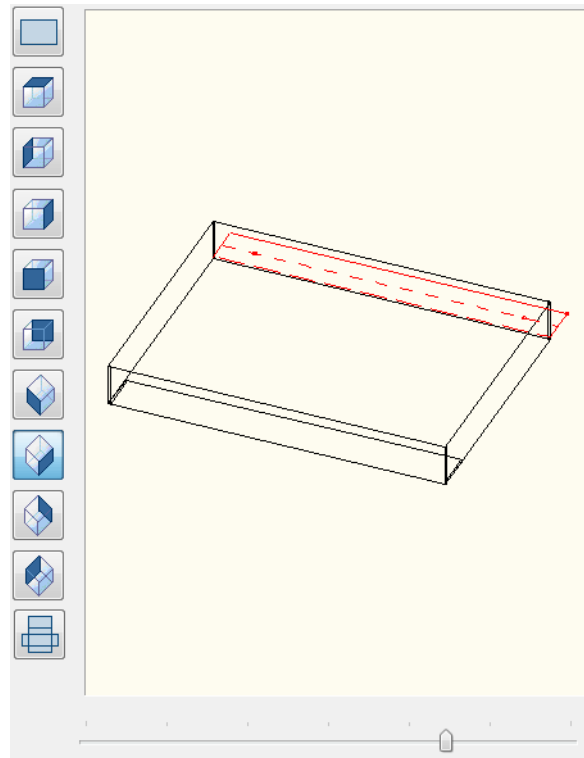
In the dialog box you will find on the left side the Display section with the dynamic preview. On the right side you will see the Operating section. You can activate the individual function areas with tabs.



Display section

The display section shows a schematic preview of the currently described sheet or parts of it. The preview is used for a simple visual check and for the selection of elements for further processing.

Dialog box section Display section

**Changing the view**

Using the buttons on the left side you can switch between various permanently defined views of the sheet model. Apart from the normal and isometric views, the preview of the schematic development is possible at any time.

**Primary surface**

This shows the active area of the sheet viewed from above and the active area in the processing mode (Tab Processing).

**View from above**

Shows the complete sheet as viewed from above. Parts of the sheet may be hidden.

**View from the left**

Shows the complete sheet as viewed from the left. Parts of the sheet may be hidden.

**View from the right**

Shows the complete sheet as viewed from the right. Parts of the sheet may be hidden.

**View from the front**

Shows the complete sheet as viewed from the front. Parts of the sheet may be hidden.



View from behind

Shows the complete sheet as viewed from behind. Parts of the sheet may be hidden.



Isometric view SW

Shows the complete sheet as an isometric view from the southwest. Parts of the sheet may be hidden.



Isometric view SO

Shows the complete sheet as an isometric view from the southeast. Parts of the sheet may be hidden.



Isometric view NO

Shows the complete sheet as an isometric view from the northeast. Parts of the sheet may be hidden.



Isometric view NW

Shows the complete sheet as an isometric view from the northwest. Parts of the sheet may be hidden.



Development

Shows the complete sheet in the developed view.



Dynamic change of viewing angle

In addition to the currently set view, it can be swiveled within a range of 45° on either side. The step increment of the slide control is 1° on the scale. You can also operate the activated slide control with the mouse wheel.

Exceptions for this function are the views primary surface, plan view and development.

Other methods of changing the view are offered by a mouse click in the preview. A small crosshair is displayed instead of the mouse pointer. Similar zoom functions are made available as in a drawing.

Dynamic zoom

To be able to view elements better, the distance to the sheet can be changed. This occurs with the aid of a forwards and backwards movement of the mouse wheel. Here, the distance is changed at the place at which the mouse pointer is located.

Pan

To bring the display field into an optimum position, it can be moved with the aid of the third mouse key (usually the mouse wheel). Move the mouse pointer over the display field and drag it with the third mouse key pressed.

Zoom all

To obtain a quick view of the complete sheet, the view can be brought to the sheet limits. To do this, click with the right mouse key and select Zoom all in the context menu.

Color representation

Various elements are highlighted in color depending on the current register. These elements include folds, surfaces, processes, joints and nodes as well as active selections. Changes to the color assignment can be made in the Dialog box Display options. In Chapter *Display* on page 73 you will find further information about this.

The standard is described as follows.

Active and inactive elements

Active folds, surfaces or processes are displayed in red. Non-selected elements are shown in green. Elements not being viewed are shown in white.

Sheet model

The sheet model is shown in black (white).

Edges of the primary surface

Edges of the primary surface are shown in blue.

Folds

The folds of a model are green.

Selecting elements

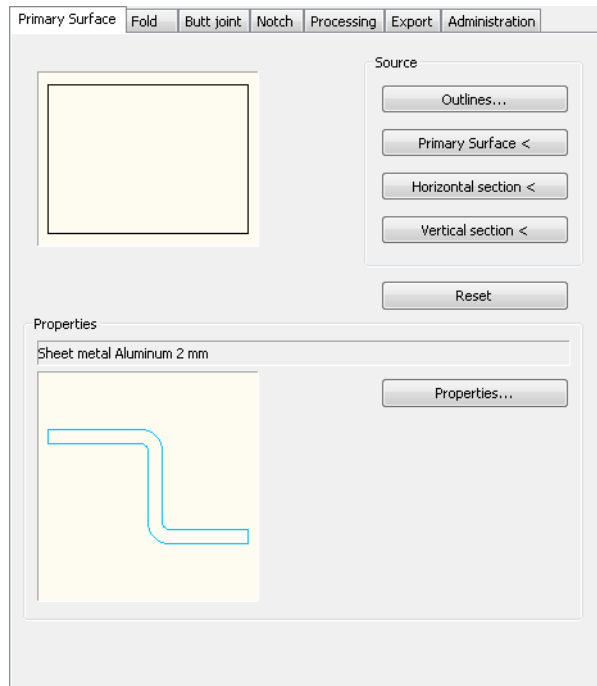
Apart from the sheet preview the display field is also used for selecting elements. Here, the selection possibilities depend on the active tab. You specify an element either by clicking with the left mouse key (Tab Fold) or by paging with arrow keys (Tab Processing, Tab Butt joint, and Tab Notch). The colored highlighting of active elements provides a check.

In its mode each tab allows the selection of certain elements or also dedicated views.

Operating section

This chapter explains the processing section of the program. You can activate the individual function areas using tabs. The arrangement of the tabs is chosen as a practicable sequence in the process, but is not mandatory.

Tab Primary surface



Here you define the properties of the sheet primary surface. The primary surface is the sheet surface from which all folds originate.

The dimensions of the primary surface are shown at the bottom right. With non-rectangular areas the dimensions of the enclosing rectangle are specified.



When you change the primary surface retrospectively, possibly not all assigned elements can be adopted. In this case the program tries to accept elements such as lists of folds and their process steps with the same orientation.

Outlines

Opens the Dialog box Outline, where you can define the outline of the primary surface. You will find further information on this in the section *Outline* on page 221.

Primary surface

Imports the primary surface from the drawing. For this, the dialog box is temporarily closed and the input request follows:

Command prompt

Select primary surface outline:

Use the mouse to choose a polyline outline.

Horizontal section

Imports the horizontal sheet metal section from the drawing. For this, the dialog box is temporarily closed and the input request follows

Command prompt

Select sheet cross section :

Select an ATHENA sheet cross section with the mouse.

Select the elevation side:

Select the sheet elevation side.

Select the segment for the primary surface or [?]:

Select a sheet segment as primary surface.

Specify left side or [?]:

Specify the left side of the sheet.

Vertical section

Imports a vertical metal section from the drawing. For this, the dialog box is temporarily closed and the input request follows

Command prompt

Select sheet cross section :

Select an ATHENA sheet cross section with the mouse.

Select the elevation side:

Select the sheet elevation side.

Select the segment for the primary surface or [?]:

Select a sheet segment as primary surface.

Specify bottom side or [?]:

Specify the bottom side of the sheet.

Reset

Resets the dialog box Sheet metal section to the basic settings. All folds are removed and a rectangular outline is set as the primary surface with the dimensions 400x300.

Properties

Opens the Dialog box Sheet metal section, where you can define its settings. You will find further information on this in the Chapter *Sheet metal section* on page 304.

Tab Fold

Type	Angle	Length	Height	D1	D2	A1	A2
↙	90.0°	50.0				90.0°	90.0°
↘	90.0°	50.0				90.0°	90.0°

Buttons: Opposite same, All same, Import 2D-section <, Fold >

Here, you can add one or several folds to a selected fold of the primary surface and modify its parameters.

In the fold table you generate and process a series of folds for each selected primary surface edge. The selection of an edge of the primary surface occurs via the display section using mouse selection, as described under *Selecting elements* on page 321.

The currently active edge is highlighted in color. Folds can only be applied to straight sheet surfaces. Curved outer edges cannot be selected.

The generation, processing and removal of folds occurs directly in the table using the context menu. The functions of the context menu are described in this section below.

Columns in the fold table

Alignment

Defines the fold alignment. An arrow symbol indicates whether the sheet is folded upwards or downwards. A double click on the symbol inverts the fold direction.

Type

Shows a symbol for the fold type. The following fold types are possible:

Standard fold (without symbol), crimped edge, rolled fold and rolled edge with fold.

Angle

Defines the angle of the fold.

Length

Defines the length of the fold.

Height

Defines the height of the fold. This column is only needed with the types rolled edge and rolled edge with fold.

D1

Shortens the first side sheet fold by the entered distance.

W1

Changes the angle of the first side sheet fold.

D2

Shortens the second side sheet fold by the entered distance.

W2

Changes the angle of the second side sheet fold.



You can edit the dimensions in the cells by direct cell processing. If you click twice consecutively in the cell, the processing mode is activated and you can change the respective value.

Functions in the context menu of the fold table

You open the context menu by right clicking the fold table. The following functions are available:

Add fold

Supplements the list with a fold.

Add fold above

Supplements the list with a fold above the marked fold.

Add fold below

Supplements the list with a fold below the marked fold.

Edit alignment

Changes the fold alignment. Select the appropriate arrow to change the direction of the fold.

Add terminating fold

Supplements the list with a terminating fold.

Edit terminating fold

Changes the type of terminating fold. For this, select the symbol of the respective type of fold.

Remove terminating fold

Removes the marked terminating fold from the list.

Remove fold

Removes the marked fold from the list.

Remove whole list

Deletes the complete fold list.

Get list

Retrieves a saved list of folds (sequence of folds). To do this, the Dialog box For object selection is opened. You will find further information on this in the Chapter *Object selection* on page 138.

Save List

Saves the current fold list. To do this, the Dialog box Save is opened. You will find further information on this in the Chapter *Saving objects* on page 136.

Below the fold table you can see a graphical preview of the current fold list.

Opposite same

Sets parallel primary surface folds the same as the current fold list. A prerequisite is a prevailing folding capability.

Button All same

Sets all folds on the primary surface the same as the current list. A prerequisite is a prevailing folding capability.

Import 2D-section

Imports an ATHENA sheet metal section from the drawing. For this, the dialog box is temporarily closed and the input request follows:

Command prompt

Select sheet cross section:

Use the mouse to choose a sheet metal section.

Select the elevation side:

Specify the sheet elevation side.

Select the segment for the primary surface:

Select a segment of the cross section as primary surface using the mouse.

Fold

Inserts the current fold as section (as shown in the preview) into the drawing. For this, the dialog box is temporarily closed and the input request follows:

Command prompt

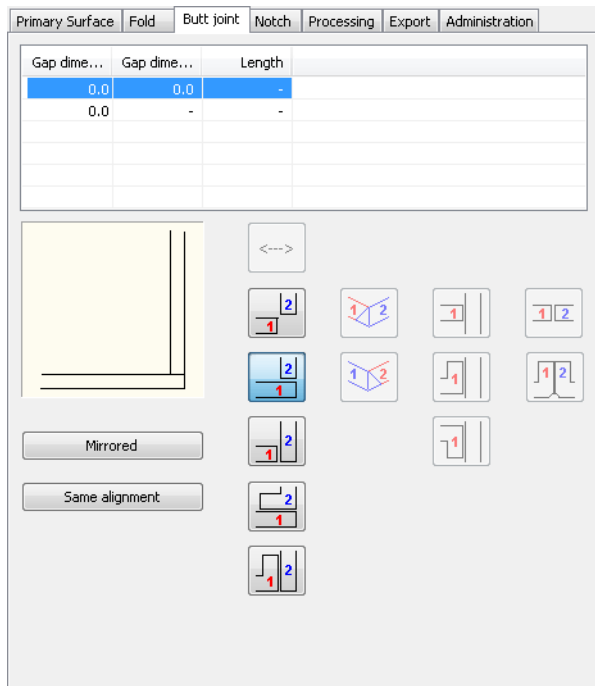
Specify insertion point:

Use the mouse or enter coordinates to specify the insertion point of the section.

Specify rotated angle<0>:

Specify the rotation angle or adopt the default angle.

Tab Butt joint



Sheet edges can be brought together in various ways. The different joint configurations can in each case be implemented variously.

To select the required node use the selection buttons. The active node is highlighted in color. Page forwards or backwards until you reach the required node.

The table shows various joint dimensions of the marked node.



You can edit the dimensions in the cells by direct cell processing. If you click twice consecutively in the cell, the processing mode is activated and you can change the respective value.

Depending on the joint configuration, below the table there are buttons available to define the type of joint. On the buttons the sheet edges are marked with colored figures. The colors in each case match the colored arrows in the dynamic preview. In the counterclockwise direction the "arriving" side is identified in red and the "departing" side in blue.

Apart from the buttons a preview of the selected joint configuration is displayed.

Columns in the joint table

Gap dimension 1

Specifies the gap for the arriving sheet edge (identified with a red 1).

Gap dimension 2

Specifies the gap for the departing sheet edge (identified with a blue 2).

Length

Specifies the length of the side fold. This column is not required with all types of joint.

Buttons



Creates an edge joint the same on both sides.



Creates an edge joint, whereby the departing end is covered.



Creates an edge joint, whereby the arriving end is covered.



Creates an edge joint, whereby the departing end is folded towards the inside.



Creates an edge joint, whereby the arriving end is folded towards the inside.



Creates an edge joint, whereby the departing end has an additional cant. This type of joint can only be selected for composite panels.



Creates an edge joint, whereby the arriving end has an additional cant. This type of joint can only be selected for composite panels.



Creates an orthogonal surface joint.



Creates a surface joint with a fold.



Creates a surface joint with a fold.



Opens the current joint configuration.



Creates a flush planar joint.



Creates a planar joint with a fold at both ends.



Creates a mitered planar joint.



Creates a parallel planar joint, whereby the departing end is the major member.



Creates a parallel planar joint, whereby the arriving end is the major member.

Mirrored

Copies the current joint configuration with mirrored symmetry on the next node in the counterclockwise direction.

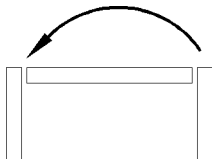


Fig. 2.4: Joint with mirrored symmetry

Same alignment

Copies the current joint configuration with directional symmetry on the next node in the counterclockwise direction.

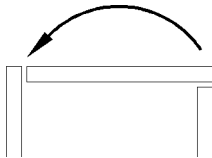
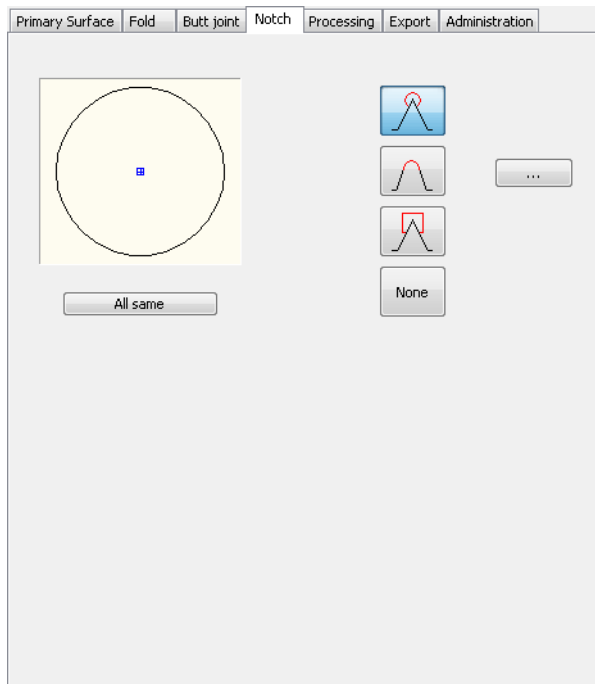


Fig. 2.5: Joint with directional symmetry

Tab Notch



When several surfaces meet, notch processes may be necessary to prevent sheet deformation.

Here you can describe given tool shapes by their parameters and assign single surface nodes.

To select the required surface node use the selection buttons. The active surface node is highlighted in color. Page forwards or backwards through the surface nodes until you reach the one you want.



Creates a circular notch on the active surface node.



Creates a notch in the shape of a circular section on the active surface node.



Creates a rectangular notch on the active surface node.

None

Creates no notch on the active surface node.

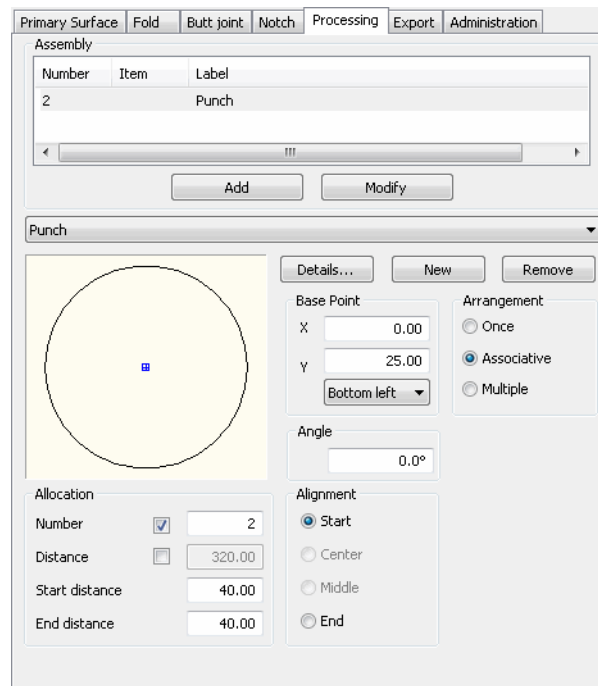
[...]

Opens the Dialog box Outline, where you can define the outline of the notch. You will find further information on this in the section *Outline* on page 221.

All equal

Applies the setting of the current surface node to all other surface nodes.

Tab Processing

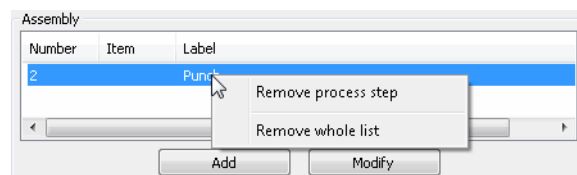


Here you can define processes and assign the surfaces of the sheet singly or using distribution rules.

The extent of the description relates in each case to one surface of the sheet.

To select the required sheet surface use the selection buttons. The active surface is highlighted in color. Page forwards or backwards until you reach the appropriate surface.

Apply processes to surfaces



The list shows the processes which are assigned to the current surface. You can select the processes in the list in order to change their properties. When you click a process with the right mouse key, a context menu appears.

Remove process

Removes the selected process from the list.

Remove whole list

Removes all processes from the list.

Add

Adds the current processing definition to the list (selected sheet surface).

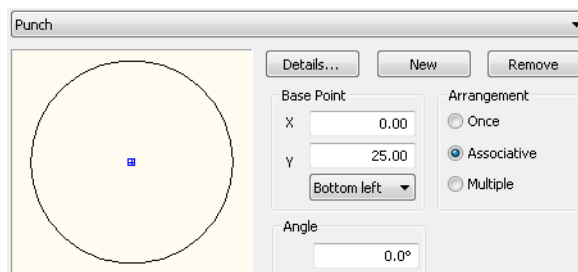
Modify

Replaces the marked entry in the list with the current processing definition.



Processes are written as "References" in the area and can therefore also be entered many times. Here, the type of the arrangement (once, associative or multiple) is not important. If the referenced process changes, the "references" also change!

Processing definition



List

Shows the available processes. Select a process from the list in order to add it to a surface.

Details

Opens the Dialog box Edit Process, where you can modify the processing parameters. For further information see "Dialog box Edit Process" on page 337.



All changes have effects on processes already referenced. Note that with the already distributed processes references of existing processes are involved.

New

Creates a new process.

Remove

Removes the selected process from the list. Then click the button Details to modify the processing properties.

Dialog box section Base point

Specifies the base point used for the process. Enter the X and Y coordinates in the appropriate input fields. Since a process can also be applied many times in a straight line, a line is displayed in the preview. The coordinates refer to the reference point which you can select from the list. The changes only become effective when you click the button Modify.

Angle

Defines the angle of the process. The rotation occurs in the counterclockwise direction. The changes only become effective when you click the button Modify.

Elevation

The display provides a visual check of the current process. The insertion point of the process is identified in color.

Dialog box section Arrangement

Single

Arranges the processes once at the defined base point on the current surface.

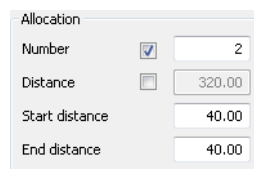
Associative

Arranges processes regularly, according to the defined distribution, on a straight processing line on the current surface. With the associative arrangement additional dialog box sections are displayed where you can define the distribution rules.

Multiple

Arranges processes irregularly, according to the defined distances, on a straight processing line on the current surface. With the multiple arrangement an additional dialog box section is displayed where you can define the distances.

Dialog box section Distribution



Number

Defines the number of processes.

Distance

Defines the spacing between the processes.

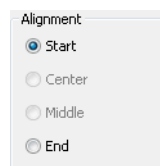
Start distance

Defines the distance of the process from the start of the sheet surface.

End distance

Defines the distance of the process from the end of the sheet surface.

Dialog box section Alignment



Start

Sets the starting point of the process series taking into account the starting distance at the start of the sheet surface.

Centered

Centers an odd number of processes on the surface. The automatic system included prevents the specification of an even number of processes.

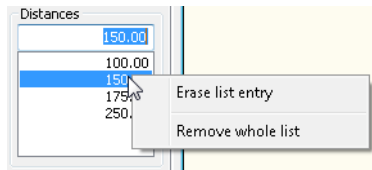
Central

Positions an even number of processes centrally on the surface. The automatic system included prevents the specification of an odd number of processes.

End

Sets the starting point of the process series taking into account the starting distance at the end of the sheet surface.

Dialog box section Distances



Defines the spacing of the processes. Enter in each case the spacing of the processes in the input field and confirm them to transfer them into the list. With a right click a context menu with further functions appears.

Erase list entry

Removes the selected entry from the list.

Remove whole list

Removes all entries from the list.

Tab Output



Dialog box section Output AutoCAD

2D section hor

Inserts a horizontal Sheet metal section into the current drawing. For this, the dialog box is temporarily closed and an input request appears.

2D section ver

Inserts a vertical Sheet metal section into the current drawing. For this, the dialog box is temporarily closed and an input request appears.

3D model

Inserts a 3D model into the current drawing. For this, the dialog box is temporarily closed and an input request appears.

Development

Inserts a development into the current drawing. For this, the dialog box is temporarily closed and an input request appears.

Folds

Consecutively inserts the primary surface and the fold sections of each side into the current drawing. For this, the dialog box is temporarily closed and an input request appears.

The insertion of single fold sections is recommended with non-rectangular primary surfaces, because there are no horizontal or vertical sections.

Projection

Inserts the projection of the current preview into the current drawing. For this, the dialog box is temporarily closed and an input request appears.

Input request for the insertion commands given above:

Command prompt

Specify insertion point or [?]:

Use the mouse or enter coordinates to specify the insertion point for the object to be inserted.

Once the insertion point has been specified, either the following query appears or the Dialog box ATHENA Sheet Processor is reopened.

Specify rotated angle<0>:

Use the mouse or enter an angle to specify the rotation angle of the object to be inserted. This query does not appear with all insertion commands. After specifying the rotation angle the Dialog box ATHENA Sheet Processor is reopened.

Update

Updates a sheet metal object present in the current drawing. For this, the dialog box is temporarily closed and the following input request appears:

Command prompt

Select objects:

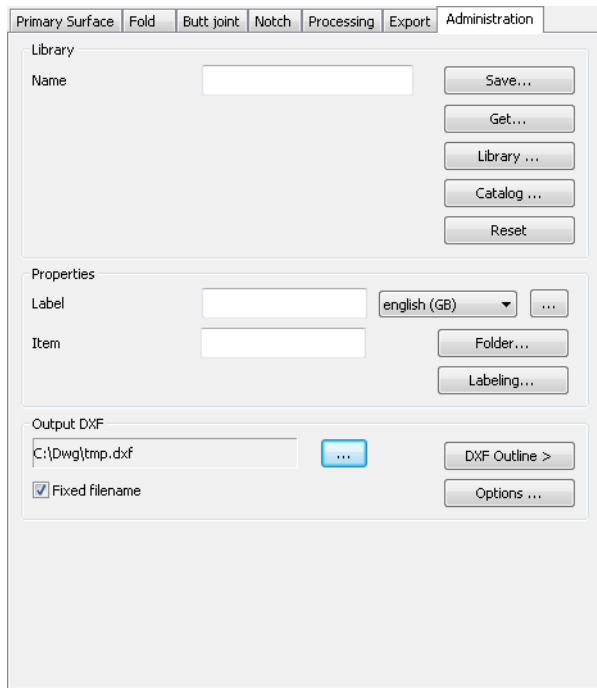
Select the sheet metal object to be updated. After confirming the object selection the Dialog box ATHENA Sheet Processor is reopened.

Dialog box section Output DXF

DXF outline

Creates a DXF file with the current DXF settings. You can make changes to the DXF settings in the Tab Manager.

Tab Manager



Dialog box section Library

Name

Specifies the name of the current sheet.

Save

Opens the Dialog box Save, where you can save the sheet under a name or save it locally without issuing a name. You will find further information in the Chapter *Saving objects* on page 136.

Get

Opens the Dialog box For object selection, where you can load a saved sheet into the dialog box. You will find further information in the Chapter *Object selection* on page 138.

Library

Opens the Dialog box Assembly library, where you can manage sheets saved in the drawing. You will find further information on this in the Chapter *Assembly library* on page 157.

Catalog

Opens the Dialog box Catalog where you can import sheets saved in libraries into the current drawing. You will find further information on this in the Chapter *Assembly catalog* on page 159.

Reset

Resets the dialog box Sheet metal section to the basic settings. All folds are removed and a rectangular outline is set as the primary surface with the dimensions 400x300.

Dialog box section Properties

Label

Defines the sheet label. The label can be specified in various languages. After selecting the language enter the terms in the input field.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels

for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the sheet. The uniqueness of the issued item number is not checked and applies equally to all languages of the sheet.

Folder

Opens the Dialog box Assemblies folder, where you can manage the folder structure for the object within the library. See Folder properties on page 164.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Dialog box section Output DXF**DXF outline**

Creates a DXF file with the current DXF settings.

[...]

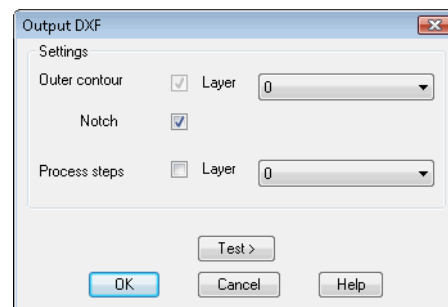
Defines the place of storage for the DXF file. The name of the sheet is used as the file name. If you activate the option Fixed file name, you can also specify a file name here.

Fixed filename

Activates or deactivates the issuing of a fixed file name. If the option is activated, the path specification corresponds to a permanently defined saved address.

Options

Opens the Dialog box Output DXF, where you can modify the settings for the DXF outline.

Dialog box Output DXF**Outer contour**

Activates the output of the outer outline for the DXF output. The outer outline is always output. In the pull-down menu you can select a layer for the outer outline.

Notch

Activates or deactivates the output of the notch for the DXF output.

Process steps

Activates or deactivates the output of the notch for the DXF output. In the pull-down menu you can select a layer for the processes.

Test

Inserts the DXF outline with the current settings into the current drawing.

End of program

OK

Closes the dialog box and saves the current sheet for the duration of the drawing session.

Cancel

Terminates the dialog box without saving the changes.

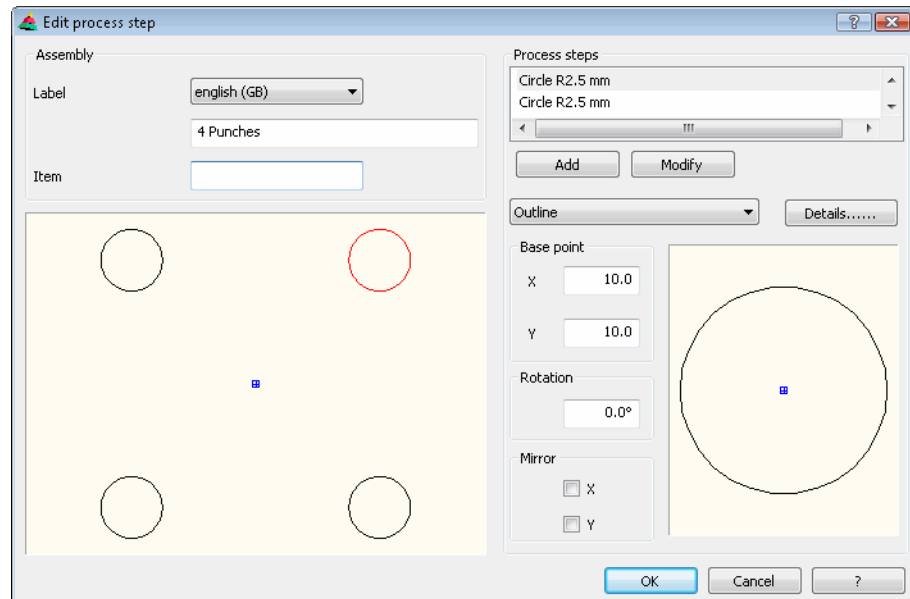
2.6 Sub-dialog boxes Sheet processing

This section describes all sub-dialog boxes which are called from Sheet processing.

- Edit Process

2.6.1 Edit Process

Dialog box Edit Process



A process is always described by one or more outlines. You can define the process parameters in this dialog box.

Dialog box section Assembly

Label

Defines the process label. The label can also be specified in other languages. After selecting the language enter the terms in the input field.

Item

Defines the item number of the process. The uniqueness of the issued item number is not checked and applies equally to all languages of the process.

Preview

Shows the process preview and its base point. If the process consists of several outlines, the active outline is shown in color.

Dialog box section Process steps

List of outlines

Displays all the process outlines. Select a process here to change its properties. When you click an outline with the right mouse key, a context menu appears.

Delete outline

Removes the selected outline from the list.

Delete all outlines

Removes all outlines from the list.

Add

Adds an outline with the current properties to the outline list.

Modify

Modifies the properties of the selected outline. This means that the settings made are transferred to the active outline in the list.

Pull-down list

Defines the outline type. You have the choice between input (drawing object) and outline (parameterized outline).

Details for the type Input

Defines an enclosed outline from the drawing as processing outline. For this, the dialog box is temporarily closed and the following input request appears:

Command prompt

Select outline or [?]:

Select an outline as process.

Specify base point or [?]:

Specify the base point of the outline.

Details for the type Outline

Opens the Dialog box Outline, where you can adapt the processing outline parameters. You will find further information on this in the Chapter *Outline* on page 221.

Base point X

Specifies the X position of the current outline within the group.

Base point Y

Specifies the Y position of the current outline within the group.

Rotation

Specifies the rotation of the current outline about its own base point.

Mirror X

Mirrors the current outline about the X axis.

Mirror Y

Mirrors the current outline about the Y axis.



The position is described relatively from the base point of the current outline to the base point of the group. Changes to the positioning only become effective when the button Modify is clicked.

2.7 Sheet development of a cross section



Ribbon:	Tab ATHENA > Group Draw > Sheet development of a cross section
Menu:	ATHENA > Sheet metal > Sheet development of a cross section
Toolbar:	ATH Sheet Metal
Command input:	ath_blec_ab2

With this routine you create a sheet development incl. dimensions with the aid of a sheet metal section. The sheet metal section must have been created with the program *Sheet metal section*.

The calculation basis for the edge lengths are the bending allowances which are dependent on the sheet thickness and are held in the table assigned to the sheet metal section. You will find information on bending allowances in the Chapter *Sheet metal computation values* on page 114.

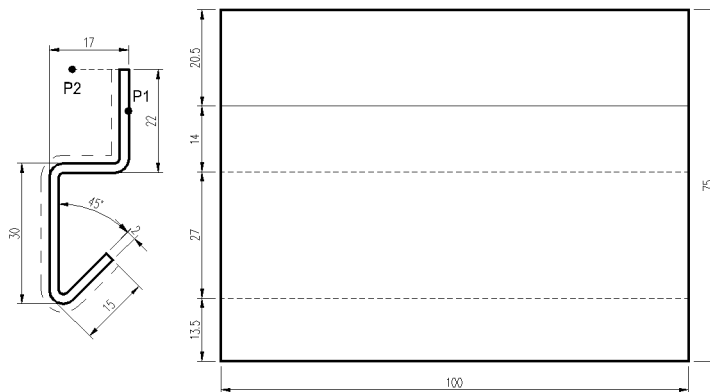


Fig. 2.6: Sheet development of a cross section

Command prompt

Select sheet cross section or [?]:

Click the cross section; e.g. P1.

You call the help with the option ?.

Select the elevation side or [?]:

Specify the elevation side of the section; e.g. P2.

Specify the width of the sheet or [?]:

Define the width of the sheet by pointing with the mouse or entering the width.

Specify insertion point:

Use the mouse or enter coordinates to specify the insertion point of the development.

Specify rotated angle<0>:

*Define the rotation angle of the development. Input **ENTER** to accept the default angle 0°.*

Display elevation side as coated side [Yes/No/?] <No>:

*With the option **No** the elevation side is not labeled.*

*With the option **Yes** the elevation side of the sheet is identified with a dotted and dashed line.*

Notes

- The dimensioning of the sheet metal section occurs with the current dimension settings.
- The outline of the development is drawn in the layer of the developed sheet.
- You control the layer for the fold and coating lines in Dialog box Layer assignment. You will find further information in the Chapter *Layer assignment* on page 108.

2.8 Sheet development of two cross sections



Ribbon:	Not present
Menu:	ATHENA > Sheet metal > Sheet development of two cross sections
Toolbar:	ATH Sheet Metal
Command input:	ath_blch_ab3

With this routine you create the development of two sheet metal sections (horizontal and vertical sections) with dimensions. The sheet metal section must have been created with the program *Sheet metal section*.

The calculation basis for the edge lengths are the bending allowances which are dependent on the sheet thickness and are held in the table assigned to the sheet metal section. You will find information on bending allowances in the Chapter *Sheet metal computation values* on page 114.

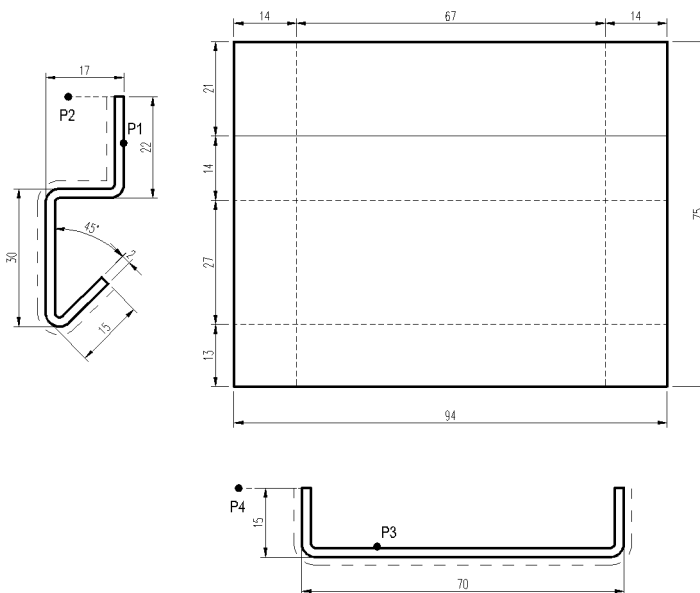


Fig. 2.7: Sheet development of two cross sections

Command prompt

Select first sheet metal section or [?]:

Click the first section; e.g. P1.

You call the help with the option ?.

Select the elevation side or [?]:

Specify the elevation side of the first section; e.g. P2.

Select second sheet metal section or [?]:

Click the second section; e.g. P3.

Select the elevation side or [?]:

Specify the elevation side of the second section; e.g. P4.

Specify insertion point:

Use the mouse or enter coordinates to specify the insertion point of the development.

Specify rotated angle<0>:

*Define the rotation angle of the development. Input **ENTER** to accept the default angle 0°.*

Display elevation side as coated side [Yes/No/?] <No>:

*With the option **No** the elevation side is not labeled.*

*With the option **Yes** the elevation side of the sheet is identified with a dotted and dashed line.*

Notes

- Clinch points are not taken into account with the development.
- The dimensioning of the sheet metal section occurs with the current dimension settings.
- The outline of the development is drawn in the layer of the developed sheet.
- You control the layer for the fold and coating lines in Dialog box Layer assignment. You will find further information in the Chapter *Layer assignment* on page 108.

2.9 Sheet development from a primary surface



Ribbon:	Not present
Menu:	ATHENA > Sheet metal > Sheet development from a primary surface
Toolbar:	ATH Sheet Metal
Command input:	ath_blec_ab1

With this routine you create a sheet development with dimensioning from a primary surface. The sheet metal section must have been previously created with the program *Sheet metal section*.

The calculation basis for the edge lengths are the bending allowances which are dependent on the sheet thickness and are held in the table assigned to the sheet metal section. You will find information on bending allowances in the Chapter *Sheet metal computation values* on page 114.

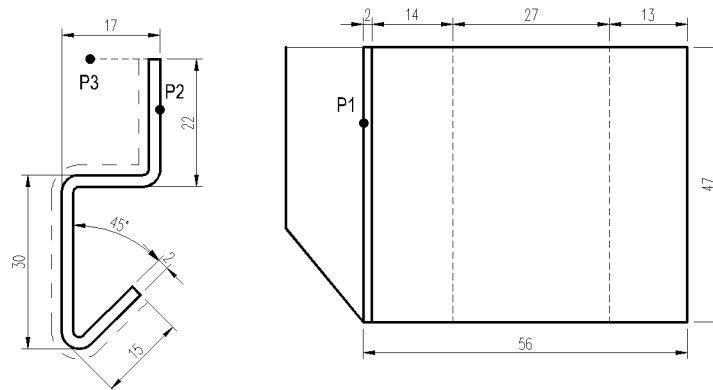


Fig. 2.8: Sheet development from a primary surface

Command prompt

Select edge from which development is to take place [?]:

Click the edge from which the development is to be generated; e.g. P1 in the figure above.

You call the help with the option ?.

Select the side according to which development is to take place [?]:

Use the mouse or enter coordinates to specify the direction in which the development is to occur.

Select sheet cross section or [?]:

Click the section to be developed with the mouse; e.g. P2.

Select the elevation side or [?]:

Specify the elevation side of the first section; e.g. P3.

Display elevation side as coated side [Yes/No/?]<No>:

With the option **No** the elevation side is not labeled.

With the option **Yes** the elevation side of the sheet is identified with a dotted and dashed line.

Notes

- The dimensioning of the sheet metal section occurs with the current dimension settings.
- The outline of the development is drawn in the layer of the developed sheet.

- You control the layer for the fold and coating lines in Dialog box Layer assignment. You will find further information in the Chapter *Layer assignment* on page 108.

3 Stairway

Commands in this section:

- Stairway calculation
- Stairway
- Sub-dialog box Stairway
- Designations and terms

3.1 Stairway calculation



Ribbon: Tab ATHENA > Group Draw > Stairway calculation
Menu: ATHENA > Draw > Stairway calculation
Toolbar: ATH Draw
Command input: ath_fstair

This command divides a line or a distance into stairway steps. The stairway parameters can be set in a dialog box.

Dialog box Stairway calculation

The dynamic preview box shows the true sectional illustration of the stairway as well as its start and end points. Optionally, markings can be switched on when stairway dimensions deviate.

Dialog box section Form

Axis

Creates a stairway section with axis. The axis marks the start and end points of the stairway.

Baseline

Creates a stairway section with baseline.

Mark deviation

Marks deviations if the stairway dimensions deviate from the stairway formula.

Outline

Generates the outline of a stairway section.

Filled

Creates a filled stairway section.

Hatched

Creates a hatched stairway section.

Layer

Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Hatch

Opens the Dialog box Hatch pattern assignment. You will find further information on this in the Chapter *Hatch pattern assignment* on page 110.

Dialog box section Dimensions**Vertical height**

Defines the height of the stairway.

Flight length

Defines the flight length of the stairway.

Input

Temporarily closes the dialog box to adopt a center distance from the drawing. Then the following input request appears:

Command prompt**Option Points**

Specify starting point of stairway line or [Object/?]:

Specify the starting point of the stairway line.

*Select the **Option Object** to specify the stairway line by object selection.*

You call the help with the option ?.

Specify end point of stairway line or [Object/?]:

Specify the end point of the stairway line.

Option Object

Select object or [Points/?]

Select a line as the stairway line.

*Select the **Option Points** to define the stairway line by specifying points.*

When you have specified the stairway line, ATHENA returns to the Dialog box Stairway calculation.

Dialog box section Properties**Rise**

Defines the minimum and maximum values for the rise of the stairway. When you activate this option, the largest possible rise (< maximum rise) is used for the stairway calculation.

Going

Defines the minimum and maximum values for the going (tread depth) of the stairway.

When you activate this option, the largest possible going (< maximum going) is used for the stairway calculation.

Step proportion

The step proportion is calculated from twice the rise plus going ($2s+a$). When you use this option, the stairway is calculated according to the best step proportion.



With this computation method the results in each case from the minimum and maximum rise and minimum and maximum going are compared to the mean step proportion. The result closest to this is accepted.

Number of steps

Carries out the stairway calculation according to the number of steps. You can define the number of steps in the input field.

Slope

The ideal slope of a stairway in the interior is 30°. The current slope is computed and displayed in the results section.

Comfort

The ideal comfort of a stairway is calculated from the going minus rise ($a-s$) and is 120 mm.

The current comfort is computed and displayed in the results section.

Sure-footed safety

The ideal sure-footed safety of a stairway is calculated from the going plus rise ($a+s$) and is 460 mm.

The current sure-footed safety is computed and displayed in the results section.

The stairway calculation is closed with OK and you can insert the stairway section with the settings made into the drawing.

3.2 Stairway

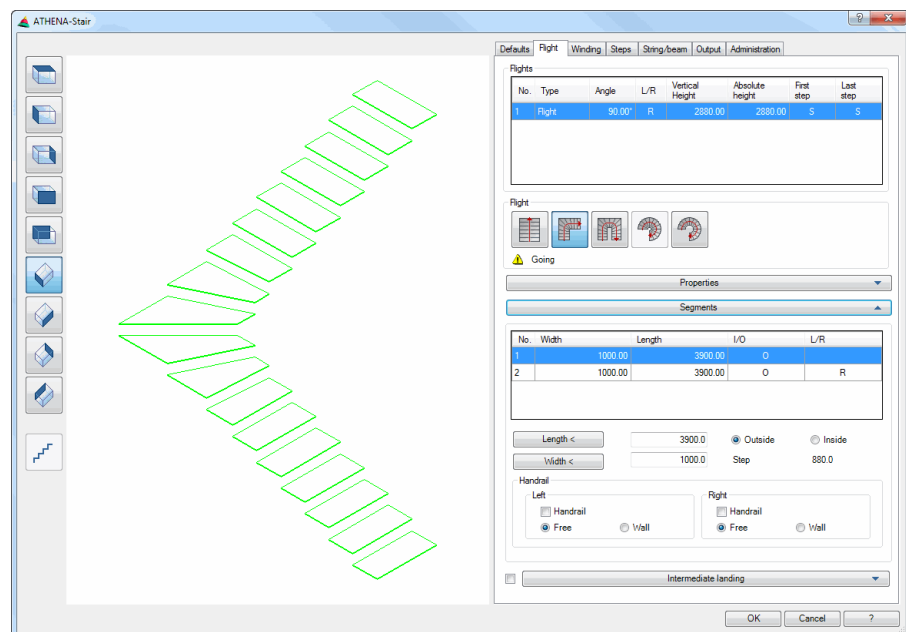


Ribbon: Tab ATHENA > Group Draw > Stairway calculation
Menu: ATHENA > Draw > Stairway calculation
Toolbar: ATH Draw
Command input: ath_fstair

Program for designing stairways. All stairway parameters are defined in one dialog box. You can insert fully dimensioned steps and the plan, 3D model, step diagrams and strings/cross beams into the current drawing.

The basis for the stairway calculation is DIN 18065 (Building stairways).

Dialog box ATHENA Stairway



Display section

On the left side the dialog box contains a preview of the stairway. The preview is primarily used for a visual check, but offers other functions too:

Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.

To the left of the preview there are buttons with which you can adjust both the preview and also the insertion of the projection into the drawing. You will find further information on this in the section *Object views* on page 102.

Operating section

On the right side of the dialog box there is the operating section with the tabs:

- Defaults
- Flight
- Winding
- Steps
- String/beam

- Output
- Manager

The individual tabs in turn contain drop-down menus with the appropriate setting options.

A description of the tabs and drop-down menus is given further below in this section.

The Administration section is the same for other objects. You will find a detailed description of the Administration section in the Chapter *Management of objects* on page 134.

Tab Defaults

This section contains the following drop-down menus:

- Flight
- Steps
- Winding
- Spiral
- Landing
- String
- Cross beam
- Ladder string
- Handrail

Drop-down menu Flight

Dialog box section Basic data

Vertical height

States the vertical height spanned by the stairway.



The vertical height does not necessarily correspond to the height of the stairway. To calculate the stairway height other parameters, e.g. the first and last steps, are taken into account.

Stairway length

Specifies the length of the stairway.

Stairway width

Specifies the width of the stairway.



With string stairways the width including the strings is specified.

First step

Defines whether the first step of the stairway starts with a riser or a tread.

Last step

Defines whether the last step of the stairway starts with a riser or a tread.

Dialog box section Walking line

Distance

Defines the distance of the walking line to the outside or inside of the stairway.

Radius

Defines the radius of the walking line for 1/4 or 1/2 spiral stairways.

Inside

The distance of the walking line specified from the inner side of the stairway.

Outside

The distance of the walking line specified from the outer side of the stairway.

Include handrail

Measures the distance of the walking line from the center of the handrail.

Drop-down menu Steps

Dialog box section Rise-to-tread ratio

Step length

Defines the specified as well as the smallest and greatest admissible step length.

Dialog box section Step

Going

Defines the specified as well as the smallest and the largest admissible going.

Step width, fixed

Activates a fixed width for the step. The width dimension of the step is specified in the input field.

Toe space

Specifies the dimension for the toe space.

Thickness

Specifies the thickness dimension for the step.

Dialog box section Riser

Rise

Defines the specified as well as the smallest and greatest admissible rise.

Thickness

Activates the riser. The thickness dimension of the riser is specified in the input field.

Opposite

Causes the risers to be set from behind against the step.

Intermediate

Causes the risers to be set between the steps.

Drop-down menu Winding

Winding

Radius: 500.0

Going min.:

Method:

- ☒ Radial
- ☐ Line-of-sight method
- ☐ Arc method
- ☐ Angle method
- ☐ Trapezoid method
- ☐ Calculated methode

Dialog box section Winding

Radius

Specifies the radius of the winding for the radial method.

Specifies the radius of the stairwell for the line-of-sight method.

Going min.

Specifies the minimum dimension for the going for the line-of-sight method.

Dialog box section Method

Radial

Calculates the step winding of spiral stairs according to the radial method.

Line-of-sight method

Calculates the step winding of spiral stairs according to the line-of-sight method.

Drop-down menu Spiral

Spiral

Direction: ☒ Right ☐ Left

Internal radius: 500.0

Going min.: 100.0

Spiral angle: 270.0°

Dialog box section Spiral

Direction

Right

Changes the default direction for spiral stairs to the right.

Left

Changes the default direction for spiral stairs to the left.

Internal radius

Specifies the default internal radius for spiral stairs.

Going min.

Specifies the minimum dimension for the going of the steps.

Spiral angle

Specifies the default angle for spiral stairs.

Drop-down menu Landing
Dialog box section Stairwell**Straight**

Sets a straight stairwell as default. The width of the stairwell can be defined in the input field.

Curved

Sets a curved stairwell as default. The radius of the stairwell can be defined in the input field.

Width/radius

Defines the width or the radius of the stairwell.

Dialog box section First step**Floor**

Specifies that the first step of the stairway begins on the floor.
The height of the floor structure can be defined in the appropriate input field.

Ceiling

Specifies that the first step of the stairway begins on the ceiling.
The ceiling dimensions can be defined in the appropriate input fields

Intermediate landing

Specifies that the first step of the stairway begins on the landing.

Dialog box section Total**Thickness**

Defines the total thickness of the ceiling.

Inset

Defines the inset of the ceiling under the first step of the stairs.

Dialog box section Structure**Floor**

Defines the height of the floor structure.

Ceiling

Defines the thickness of the ceiling.

Wall

Defines the wall thickness.

Dialog box section Last step

Analogous to the first step, here corresponding default settings for the last step are defined. You will find the description of this in the dialog box section *First step* on page 353.

Drop-down menu String

Dialog box section String

Width

Defines the width of the string (material thickness).

Distance

Defines the spacing between the step and the string.

Top inset

Defines the distance of the front edge of a step to the upper edge of the string.

Bottom inset/Height

Depending on the switch setting, here you set either the height of the string or the distance of the rear edge of the step to the lower edge of the string.

Dialog box section First step connection

Defines the type of connection of the string in the first step. You can choose between: Standard, Horizontal and Vertical.



The offered possible selections depend on the selected floor connection of the first step in the Drop-down menu Landing.

[...]

Depending on the type of connection selected, a sub-dialog box opens in which you can set additional connection parameters. You will find further information on this in the section *First step string connection* on page 372.

Dialog box section Last step connection

Defines the type of connection of the string in the last step. You can choose between: Standard, Clinched, Horizontal and Vertical.



The offered possible selections depend on the selected floor connection of the last step in the Drop-down menu Landing.

[...]

Depending on the type of connection selected, a sub-dialog box opens in which you can set additional connection parameters. You will find further information on this in the section *String connection, last step* on page 375.

Drop-down menu Cross beam

No yet available.

Drop-down menu Ladder string

No yet available.

Drop-down menu Handrail

Dialog box section Handrail

Height

Defines the height of the handrail.

Diameter

Defines the diameter of the handrail.

Dialog box section Wall

Distance

Specifies the distance of the wall to the center of the handrail.

Dialog box section Free

Distance

Specifies the distance from the center of the handrail alternatively to the stairway, step or wall. The distance which applies is defined with the following option fields:

Stairway

Specifies the distance to the outer side of the stairway.

Step

Specifies the distance to the outer side of the step.

String center

Specifies the distance to the center stairway string.

Tab Flight

The content of this tab is dynamic and changes in dependence of the active type of stairway, which is selected in the table of flights.

When a stairway flight is selected, buttons for changing the flight are displayed. Furthermore, the following drop-down menus are shown with further possible settings.

- Properties
- Segments
- Intermediate landing

If you select a landing in the table, buttons for modifying the landing are displayed. Also, additional dialog box sections with further possible settings are displayed.

Dialog box section Flights

Flights							
No.	Type	Angle	L/R	Vertical Height	Absolute height	First step	Last step
1	Flight	0.00°	R	2880.00		S	S
2	Landing	90.00°	R	0.00	2880.00		
3	Flight	0.00°	R	2880.00	5760.00	S	S

Here, the available flights and landings with the properties are shown in tabular form. The properties of the relevant flight cannot be changed directly in the table. Changes can be made in the input fields further below. For this, select the flight to be changed in the table.

When you click a flight with the right mouse key, a context menu appears with the following options:

Add

Appends a new flight at the end of the table.

Remove

Deletes the selected flight from the table.



The flights and landings always alternate. Consecutive flights of steps or consecutive landings are not possible.

Stairway flight






When you select a flight, the dialog box appears as below:

Defaults Flight Winding Steps String/beam Output Administration

Flights

No.	Type	Angle	L/R	Vertical Height	Absolute height	First step	Last step
1	Flight	0.00°	R	2880.00		S	S
2	Landing	90.00°	R	0.00	2880.00		
3	Flight	0.00°	R	2880.00	5760.00	S	S

Flight

Properties

Segments

☐ Intermediate landing

Dialog box section Flight

You can set the required flight by clicking the appropriate button.



Generates a straight flight of steps.



Generates a quarter spiral flight of steps.



The permissible angular range of this form of flight is 1° to 90°.



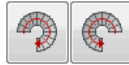
Generates a half spiral flight of steps.



The permissible angular range of this form of flight is 91° to 180°.



This option is however not yet released.



Creates a spiral stairway.



The permissible angular range of this form of flight is 1° to 360°.



If you repeated a click on one of the buttons, the direction (right/left) changes.

Drop-down menu Properties

Properties			
Angle	<input type="text" value="270.0°"/>		
Width	<input type="text" value="1000.0"/>		
Internal radius	<input type="text" value="500.0"/>		
Vertical Height <	<input type="text" value="2880.0"/>	absolute	2880.0
First step	<input checked="" type="radio"/> Rise	<input type="radio"/> Going	
Last step	<input checked="" type="radio"/> Rise	<input type="radio"/> Going	

Angle

Specifies the angle for quarter spiral, half spiral and spiral stairways.



This input field is deactivated for straight flights.

Width

Specifies the width for spiral stairs.



This input field is not displayed for the other types of stairway. Their width is defined in Drop-down menu Segments.

Internal radius

Defines the internal radius of the spiral stairway.



This input field is not displayed for the other types of stairway.

Vertical height

States the vertical height spanned by the stairway.



The difference in height normally corresponds to the story height (top edge of floor to top edge of floor on the next story).

With the Vertical height button < you can pick it up by clicking two points in the drawing. For this, the dialog box is temporarily closed and the input request appears:

Command prompt

Specify vertical height or [?] <2880>:

Specify the first point of the vertical height, e.g. a point on the finished floor of the story.

Press the Enter key to accept the default value in pointed brackets.

Specify second point:

Specify the first point of the vertical height, e.g. a perpendicular point on the finished floor of the next story.

First step

Defines whether the first step of the stairway starts with a riser or a going.

Last step

Defines whether the last step of the stairway finishes with a riser or a going.

Drop-down menu Segments

No.	Width	Length	I/O	L/R
1	1000.00	3900.00	O	
2	1000.00	3900.00	O	R

Length < 3900.0 ☒ Outside ☐ Inside

Width < 1000.0 Step 880.0

Handrail

Left: ☐ Handrail ☒ Free ☐ Wall

Right: ☐ Handrail ☒ Free ☐ Wall

In the table the properties of the segments of the current flight are shown. You can change the properties of the selected flight in the further sections of the dialog box.

Length <

Defines the length of the flight. If you click the button, you can access the length from the drawing. For this, the dialog box is temporarily closed and an input request appears.

Outside

Specifies the length on the outer side of the stairway.

Inside

Specifies the length on the inner side of the stairway.

Width <

Defines the width of the flight. If you click the button, you can access the width from the drawing. For this, the dialog box is temporarily closed and an input request appears.

Dialog box section Handrail

Handrail, left/right

Switches on the handrail on the relevant side for the current stairway section.

Free

Uses the default free distance between the handrail and stairway.

Wall

Uses the default wall spacing for the handrail.



The handrail spacings can be adjusted in each case in the Drop-down menu Handrail of the Tab Defaults.

Drop-down menu Intermediate landing

No.	Width	Depth	Height
1	880.00	880.00	1440.00

☐ Triple-flight

Reference ☒ Start ☐ Central ☐ End

The table shows the properties of the landings. You can change the properties of the selected landings in the further sections of the dialog box.



Intermediate landings can be switched on for single or multi-flight stairways. You interrupt a stairway flight without changing the direction of the flight.

Triple-flight

Generates two landings with half-spiral stairs so that a stairway with three flights is created.

Depth <

Sets the depth dimension of the landing.

Distance <

Sets the horizontal distance of the landing to the stairs.

Reference

Defines the reference to which the distance dimension of the landing refers.

Start

The distance is measured from the front edge of the first step of the stairway to the front edge of the landing.

Central

The distance is derived.



This option is only available for half-spiral stairways with landing.

End

The distance is measured from the rear edge of the last step of the stairway to the rear edge of the landing.

Stairway landing

When you select a stairway landing, the dialog box appears as below:

Defaults Flight Winding Steps String/beam Output Administration

Flights

No.	Type	Angle	L/R	Vertical Height	Absolute height	First step	Last step
1	Flight	0.00°	R	2880.00		S	S
2	Landing	90.00°	R	0.00	2880.00		
3	Flight	90.00°	R	2880.00	5760.00	S	S

Type

First step ☐ Building structure ☒ Ceiling ☐ Intermediate landing

Last step ☐ Building structure ☒ Ceiling ☐ Intermediate landing

Dimensions

Angle

Width

Depth

Total

Thickness

Inset

Offset of last step

Structure

Floor

Ceiling

Wall

Stairwell

☐ ☒ Straight ☐ Curved

Width



In contrast to the intermediate landing which interrupts a flight of steps, a stairway landing is a component which actually separates different flights of steps. Consequently, with the stairway landing the flight direction of the stairway can be changed.

Dialog box section Type

You can set the required flight by clicking the appropriate button.



Generates a straight landing without changing the flight direction.



Generates a landing with a change in the flight direction.



The permissible angular range of this form of landing is 1° to 90°.



Generates a landing with a change in the flight direction.



The permissible angular range of this form of landing is 91° to 180°.



Generates a landing in a spiral newel form.



The permissible angular range of this form of landing is 1° to 269°.



Generates a landing in a spiral form.



The permissible angular range of this form of landing is 1° to 360°.



If you repeat a click on one of the buttons, the direction (right/left) changes.



Generates a landing by choosing a drawn outline. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Select the outline Landing or [?]:

Select the outline which you want to use as the landing.



The outline for the landing must be a closed polyline with at least two straight segments.

Select side for approaching flight or [?]:

Select a polyline segment of the outline for the approaching flight. Here it is sufficient to click in the vicinity of a segment.

Select the left side connection or [?]:

Select a point on the line just selected (polyline segment). At the selected point the left side of the tread of the approaching flight is connected.

Press the Enter key to use the left end (start point of the rubber-band line) of the line.

Select side for leaving flight or [?]:

Select a polyline segment of the outline for the leaving flight. Here too, it is sufficient to click in the vicinity of a segment.

Select the left side connection or [?]:

Select a point on the line just selected (polyline segment). At the selected point the left side of the first step of the leaving flight is connected.

Press the Enter key to use the right end (start point of the rubber-band line) of the line.

First step

Defines whether the first step connects to a building structure, a ceiling or an intermediate landing.

Last step

Defines whether the last step connects to a building structure, a ceiling or an intermediate landing.

Dialog box section Dimensions

Angle

Defines the angle in the change of flight direction.



The permissible angles depend on the selected type of landing.

If you have imported a landing from the drawing, the angle cannot however be changed.

Width

Specifies the landing width.

Depth

Specifies the landing depth.



The width and depth of the landing depends, according to the type, on various parameters (e.g. width of the flight). In this case the dimensions are displayed, but cannot be changed.

Dialog box section Total

Thickness

Defines the total thickness of the ceiling.

Inset

Defines the inset of the ceiling on the first step of the stairs.

Inset of last step

Defines the inset of the ceiling on the last step of the stairs.

Dialog box section Structure

Floor

Defines the height of the floor structure.

Ceiling

Defines the thickness of the ceiling.

Wall

Defines the wall thickness.

Dialog box section Stairwell

Activating the tick box provides a stairwell.



The stairwell can only be manually activated with quarter spiral stairways. With half spiral and spiral newel stairways it is automatically turned on for constructional reasons.

With straight stairways and spiral stairways there is no stairwell.

Straight

Activates a straight stairwell

Curved

Activates a curved stairwell

Width/radius

Defines the width or the radius of the stairwell according to the selected type (straight or curved).

Tab Winding

The screenshot shows the 'Winding' tab of a software interface. It contains the following elements:

- Form kite-winder** section:
 - with radius**: A text input field containing the value 500.0.
 - Going min.**: A text input field containing the value 213.2.
- Steps include** section:
 - ☐ **First step**: A checkbox with an adjacent empty text input field.
 - First area**: A section containing 'Start' and 'End' labels, each followed by three buttons: a circular arrow (reset), a plus sign (+), and a minus sign (-).
 - ☐ **Divide areas**: A checkbox.
 - Second area**: A section containing 'Start' and 'End' labels, each followed by three buttons: a circular arrow (reset), a plus sign (+), and a minus sign (-).
 - ☐ **Last step**: A checkbox with an adjacent empty text input field.
- Bottom navigation**: A row of four buttons (back, forward, left, right) followed by the text '3 Flight'.

Here you can manipulate the winding in the stairs.

Dialog box section Form kite-winder

With radius

Defines the radius for winding the steps.

Going min.

Shows the width of the smallest going.

Dialog box section Steps include

First step

Activates a diagonal first step. In the input field which is enabled on activating the tick box you can define the starting angle.

Dialog box section First area

Start/End

With the relevant buttons you can include additional steps or remove steps when winding steps at the start and end.



Restores the original state of the winding. This means that manually inserted steps are deleted and manually removed steps are added.



Includes a further step in the relevant area (start or end).



Removes a step from the relevant area (start or end).

Divide areas

Divides the sections of the winding for half spiral stairways. In this way you can separately adapt the winding for any change in direction.

Last step

Activates a diagonal last step. In the input field which is enabled on activating the tick box you can define the finishing angle.



Manual inclusion and removal of steps is not possible with all methods of winding. Currently, it only works with the line-of-sight method (see *Winding* on page 352).

Buttons for changing the flight see Page 370.

Tab Steps

Properties		Target		Range	
Vertical Height	2880.0				
Flight length	6471.2				
Rise	151.6	180.0	150.0	210.0	
Going	358.1	260.0	220.0	300.0	
Rises	19				
Goings	18				
Slope	22.9°	30.0°			
Step length	661.3	620.0	590.0	650.0	
Comfort	206.5	120.0			
Sure-footed safety	509.7	460.0			

No. of steps

☐ + -

⏪ ⏴ ⏵ ⏩ 3 Flight

Dialog box section Properties

Here, apart from the calculated values of the current stairway the set default values as well as the minimum and maximum values are displayed for checking.

If the calculated values deviate from the default values, a yellow warning symbol is displayed. In this case you have to decide whether the deviation is acceptable or whether the corresponding values can be improved by the removal or addition of steps.

Dialog box section Number of steps

The tick box is automatically activated when you manipulate the number of steps. If the tick box is deactivated, the number of steps calculated for the flight by the program is used.



Adds a step to the current flight.



Removes a step from the current flight.

Buttons for changing the flight see Page 370.

Tab String/beam

Defaults Flight Winding Steps **String/beam** Output Administration

☒ String

Dimensions

☒ Left ☐ Opposite side ☒ Right

Width 60.0

Distance 0.0

Top inset 100.0

Bottom inset 100.0

First step connection Style **Default** ...

Last step connection Style **Clinched** ...

☐ Cross beam

☐ Ladder side member

3 Flight

This section contains the following drop-down menus:

- String
- Cross beam
- Ladder string

Drop-down menu String

☒ String

Dimensions

☒ Left ☐ Opposite side ☒ Right

Width 60.0

Distance 0.0

Top inset 100.0

Bottom inset 100.0

First step connection Style **Default** ...

Last step connection Style **Clinched** ...

The tick box on the left next to the drop-down menu activates the string for the relevant flight.

Dialog box section Dimensions**Left**

Activates the string on the left side of the flight.

Opposite side

Activates the string on the opposite side of the flight.

Right

Activates the string on the right side of the flight.

Width

Shows the string width which has been defined in the default values.

Distance

Shows the distance of the string to the flight defined in the default values.

Top inset

Defines the distance of the front edge of a step to the upper edge of the string.

Bottom inset/Height

Depending on the default setting, here either the height of the string or the distance of the rear edge of the step to the lower edge of the string (bottom inset) is set.

Dialog box section First step connection

Defines the type of connection of the string in the first step. You can choose between: Standard, Horizontal and Vertical.



The offered possible selections depend on the selected floor connection of the first step in the Drop-down menu Landing.

[...]

Depending on the type of connection selected, a sub-dialog box opens in which you can set additional connection parameters. You will find further information on this in the section *First step string connection* on page 372.

Dialog box section Last step connection

Defines the type of connection of the string in the last step. You can choose between: Standard, Clinched, Horizontal and Vertical.



The offered possible selections depend on the selected floor connection of the last step in the Drop-down menu Landing.

[...]

Depending on the type of connection selected, a sub-dialog box opens in which you can set additional connection parameters. You will find further information on this in the section *String connection, last step* on page 375.

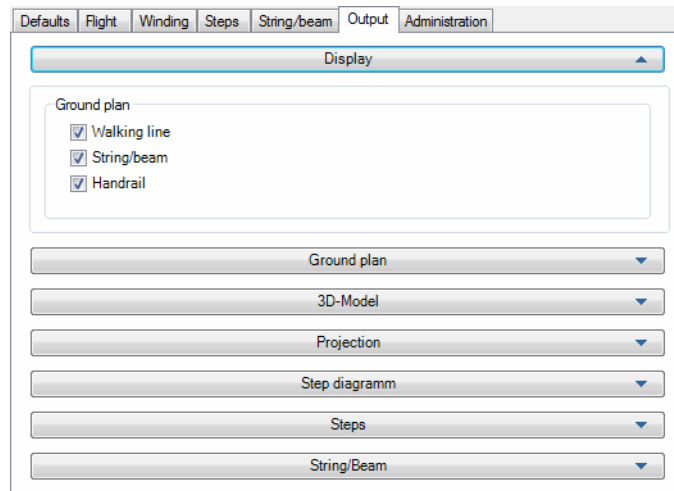
Drop-down menu Cross beam/joist

Cross beams and joists are not currently supported.

Drop-down menu Ladder string

Ladder strings are not currently supported.

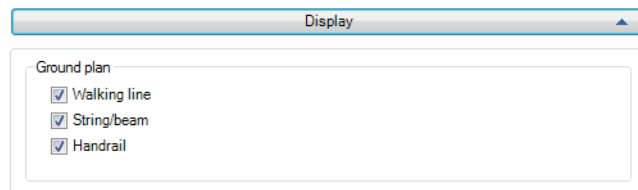
Tab Output



This section contains the following drop-down menus:

- Display
- Ground plan
- 3D model
- Projection
- Step diagram
- Steps
- String/beam

Drop-down menu Display



Dialog box section Ground plan

Walking line

Switches off the preview of the walking line

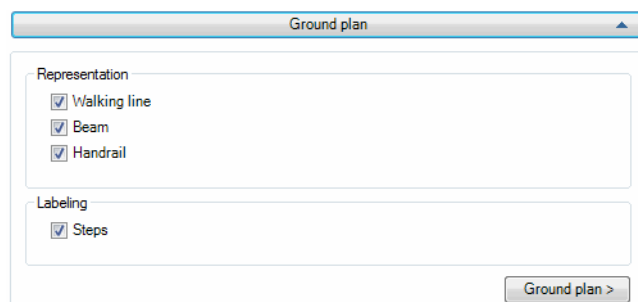
String/cross beam

Switches off the preview of the string and cross beam.

Handrail

Switches off the preview of the handrail

Drop-down menu Ground plan



Dialog box section Representation

Walking line

Displays the walking line when you insert a plan of the stairs.

Beam

Displays the beam when you insert a plan of the stairs.

Handrail

Displays the handrail when you insert a plan of the stairs.

Dialog box section Labeling

Steps

Labels the steps of the inserted plan.

Ground plan >

Inserts the plan with the above defined settings into the drawing. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Specify insertion point or [Update/?]:

Specify the insertion point of the plan.

You can update an existing plan with the Option Update.

You call the help with the option ?.

Specify rotation angle or [Update/?] <0>:

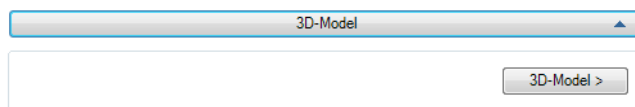
Specify the rotation angle of the plan or press the Enter key to accept the default angle (0°).

Option Update

Select objects:

Select the plan which you want to update.

Drop-down menu 3D model



3D model >

Inserts a 3D model into the drawing. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Specify insertion point or [Update/?]:

Specify the insertion point of the 3D model.

You can update an existing 3D model with the Option Update.

You call the help with the option ?.

Specify rotation angle or [Update/?] <0>:

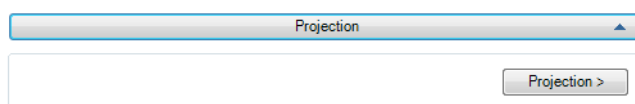
Specify the rotation angle of the 3D model or press the Enter key to accept the default angle (0°).

Option Update

Select objects:

Select the 3D model which you want to update.

Drop-down menu Projection



Projection >

Inserts the current preview picture as a two-dimensional projection into the drawing. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Specify insertion point or [Update/?]:

Specify the insertion point of the projection.

You can update an existing projection with the Option Update.

You call the help with the option ?.

Specify rotation angle or [Update/?] <0>:

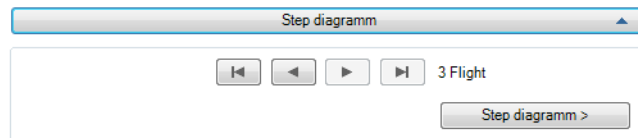
Specify the rotation angle of the projection or press the Enter key to accept the default angle (0°).

Option Update

Select objects:

Select the projection which you want to update.

Drop-down menu Step diagram



Buttons for changing the flight see Page 370.

Step diagram >

Inserts the step configuration of the current flight into the drawing. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Specify insertion point:

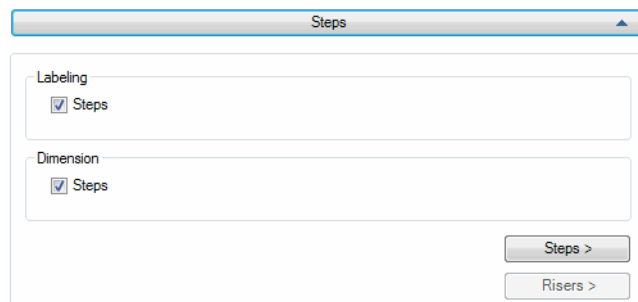
Specify the insertion point of the step configuration.

You call the help with the option ?.

Specify rotated angle<0>:

Specify the rotation angle of the step configuration or press the Enter key to accept the default angle (0°).

Drop-down menu Steps



Labeling Steps

Labels the steps on insertion with a tag and the quantity.

Dimension Steps

Dimensions the steps on insertion. Rectangular steps are not dimensioned!

Steps >

Inserts the steps of the stairway into the drawing. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Specify insertion point or [?]:

Specify the insertion point of the step.

You call the help with the option ?.

This input request is repeated until all steps have been inserted into the drawing.

Risers >

Inserts the risers of the stairway into the drawing. For this, the dialog box is temporarily closed and the following appears:

Command prompt

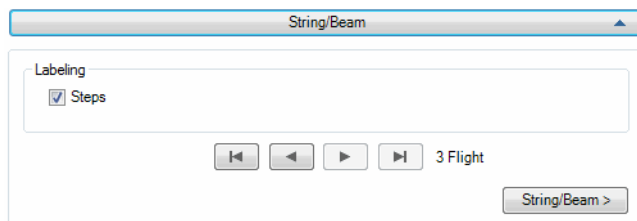
Specify insertion point or [?]:

Specify the insertion point of the riser.

You call the help with the option ?.

This input request is repeated until all steps have been inserted into the drawing.

Drop-down menu String/beam



Buttons for changing the flight see Page 370.

String/beam >

Inserts the string of the stairway into the drawing. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Specify insertion point:

Specify the insertion point of the string.

You call the help with the option ?.

Specify rotated angle<0>:

Specify the rotation angle of the string or press the Enter key to accept the default angle (0°).

3.2.1 Buttons for changing the flight



Changes to the first flight.



Changes to the previous flight.



Changes to the next flight.



Changes to the last flight.

End of program

OK

Saves the settings and terminates the dialog box.

Cancel

Terminates the dialog box without saving the changes.

3.3 Sub-dialog box Stairway

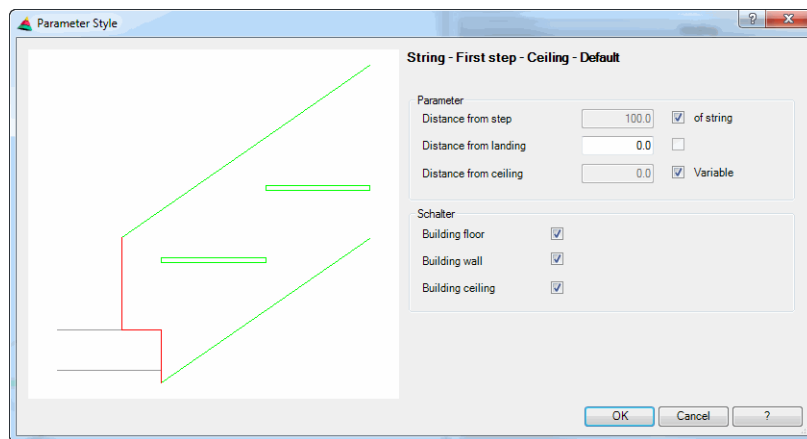
This section describes all sub-dialog boxes which are called from the stairway.

- First step string connection
- String connection, last step

3.3.1 First step string connection

In these dialog boxes you define the parameters for the string connection to the first step. The connection can occur on the floor, on a ceiling and on an intermediate landing. Generally, a differentiation is made between standard, horizontal and vertical connections.

Dialog box Parameters String - First step - Standard



Distance from step

Specifies the horizontal distance of the string to the front edge of the first step, see figure String dimensions: First step - Ceiling - Standard: Dimension 1. The entry field is enabled when you deactivate the From string tick box.

From string

Uses the same distance as to the string top edge (top insert) for the distance from the step.

Distance from landing

Specifies the horizontal distance of the string to the landing, see figure String dimensions: First step - Ceiling - Standard: Dimension 2.

Distance from ceiling

Specifies the vertical distance of the string to the bottom edge of the ceiling, see figure String dimensions: First step - Ceiling - Standard: Dimension 3. The input field is enabled when you deactivate the Variable tick box.

Variable

The bottom edge of the string is formed pointed. The distance to the ceiling is influenced by other parameters (e.g. distance from landing).

Floor structure

With the tick box activated the floor structure is ignored (distances are referenced to the bare floor), see figure String dimensions: First step - Ceiling - Standard: Dimension A.

Wall structure

With the tick box activated the wall structure for the distance from the landing is ignored (distances refer to the raw-shell dimension, see figure String dimensions: First step - Ceiling - Standard: Dimension B.

Ceiling structure

With the tick box activated the ceiling structure for the distance from the ceiling is ignored (distances refer to the raw-shell ceiling), see figure String dimensions: First step - Ceiling - Standard: Dimension C.

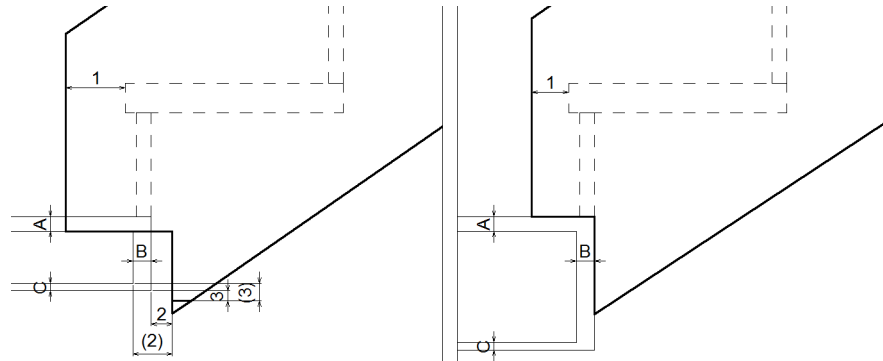
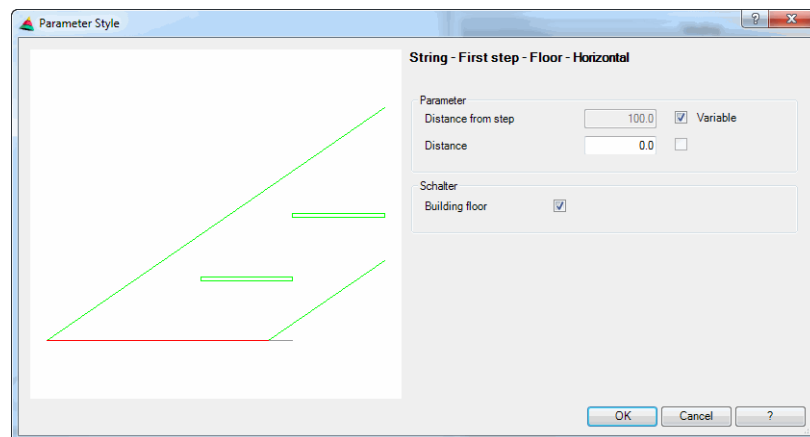


Fig. 3.1: String dimensions: First step - Ceiling - Standard

Dialog box Parameters String - First step - Horizontal**Distance from step**

Specifies the horizontal distance of the string to the front edge of the first step, see figure String dimensions: First step - Floor - Horizontal: Dimension 1. The input field is enabled when you deactivate the From string tick box.

Variable

Uses the distance of the step to the top edge of the string (top inset) also as the horizontal distance to the string. With the tick box activated the string is formed pointed at the front.

Distance

Specifies the vertical distance from the floor. A positive value extends the string downwards, a negative value shortens the string, see figure String dimensions: First step - Floor - Horizontal: Dimension 2.

Floor structure

With the tick box activated the floor structure is ignored (distances are referenced to the bare floor), see figure String dimensions: First step - Floor - Horizontal: Dimension A.

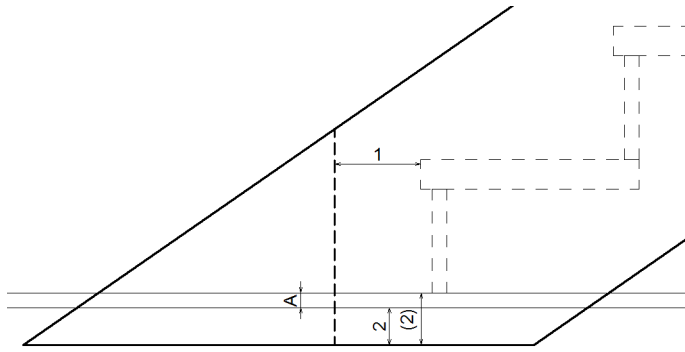
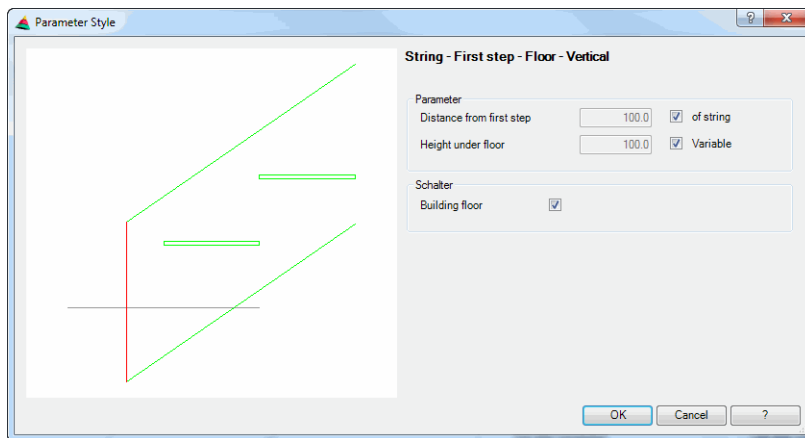


Fig. 3.2: String dimensions: First step - Floor - Horizontal

Dialog box String - First step - Vertical parameters



Distance from first step

Specifies the horizontal distance of the string to the front edge of the first step, see figure String dimensions: First step - Floor - Vertical: Dimension 1. The entry field is enabled when you deactivate the From string tick box.

From string

Uses the same distance as to the string top edge (top insert) for the distance from the first step. With the tick box activated the string is formed pointed at the front.

Height below floor

Specifies the vertical distance from the floor. A positive value extends the string downwards, a negative value shortens the string, see figure String dimensions: First step - Floor - Vertical: Dimension 2.

Variable

The bottom edge of the string is formed pointed. The distance to the ceiling is influenced by other parameters (e.g. distance from landing).

Floor structure

With the tick box activated the floor structure is ignored (distances are referenced to the bare floor), see figure String dimensions: First step - Floor - Vertical: Dimension A.

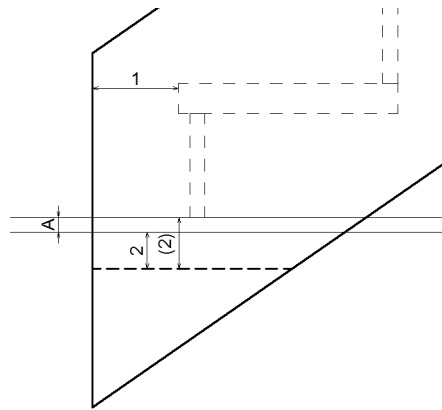
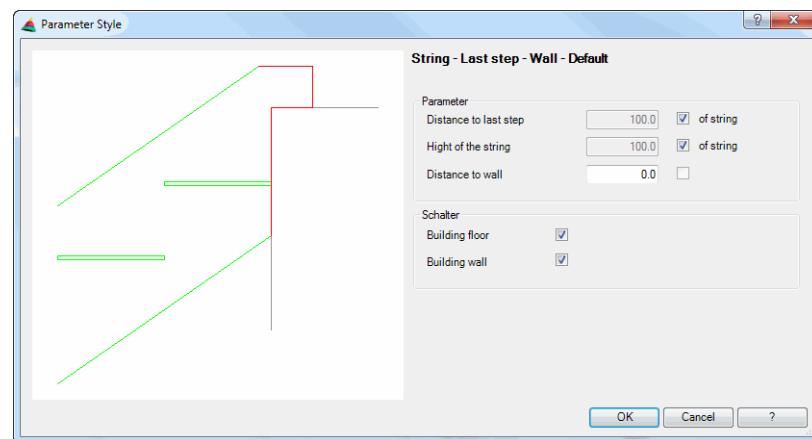


Fig. 3.3: String dimensions: First step - Floor - Vertical

3.3.2 String connection, last step

In these dialog boxes you define the parameters for the string connection to the last step. The connection can occur on a story wall, on a ceiling and on an intermediate landing. Generally, a differentiation is made between standard (clinched), horizontal and vertical connections.

Dialog box Parameters String - Last step - Standard



Distance to last step

Specifies the horizontal distance of the string rear edge to the front edge of the last step, see figure String dimensions: Last step - Wall - Standard: Dimension 1. The entry field is enabled when you deactivate the From string tick box.

From string

Uses the same distance as to the string top edge (top insert) for the distance from the last step.

String height

Specifies the vertical dimension of the string on the last step, see figure String dimensions: Last step - Wall - Standard: Dimension 2.

From string

Uses the same distance as to the string top edge (top insert) for the string height.

Distance to the wall

Specifies the horizontal distance of the string edge to the front edge of the wall, see figure String dimensions: Last step - Wall - Standard: Dimension 3.

Floor structure

With the tick box activated the floor structure is ignored (distances are referenced to the bare floor), see figure String dimensions: Last step - Wall - Standard: Dimension A.

Wall structure

With the tick box activated the wall structure is ignored (distances are referenced to the bare wall), see figure String dimensions: Last step - Wall - Standard: Dimension B.

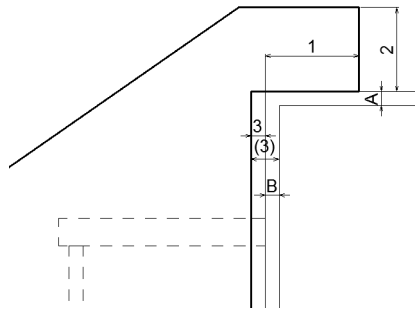
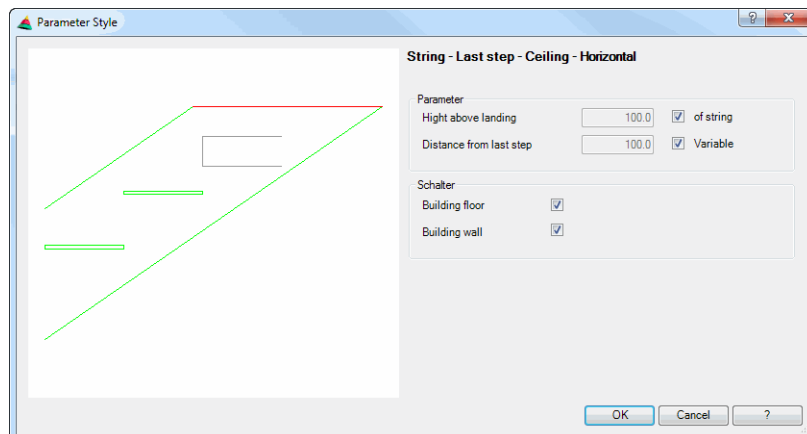


Fig. 3.4: String dimensions: Last step - Wall - Standard

Dialog box Parameters String - Last step - Horizontal



Height above landing

Specifies the vertical distance of the string top edge to the ceiling, see figure String dimensions: Last step - Ceiling - Standard: Dimension 1. The entry field is enabled when you deactivate the From string tick box.

From string

Uses the same distance as to the string top edge (top insert) for the height above the landing.

Distance from last step

Specifies the horizontal distance from the string rear edge to the front edge of the ceiling, see figure String dimensions: Last step - Ceiling - Standard: Dimension 2.

The entry field is enabled when you deactivate the Variable tick box.

Variable

The rear edge of the string is formed pointed. The distance to the ceiling is influenced by other parameters (e.g. distance above landing).

Floor structure

With the tick box activated the floor structure is ignored (distances are referenced to the bare floor), see figure String dimensions: Last step - Ceiling - Standard: Dimension A.

Wall structure

With the tick box activated the wall structure is ignored (distances are referenced to the bare wall), see figure String dimensions: Last step - Ceiling - Standard: Dimension B.

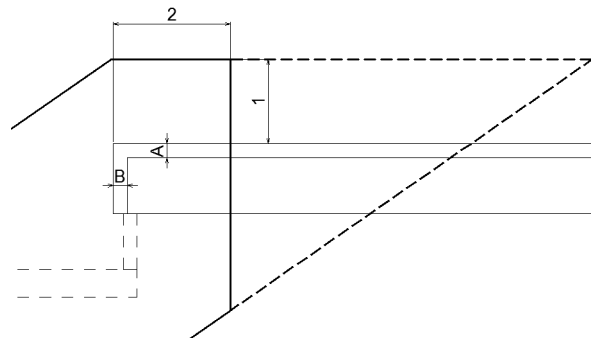
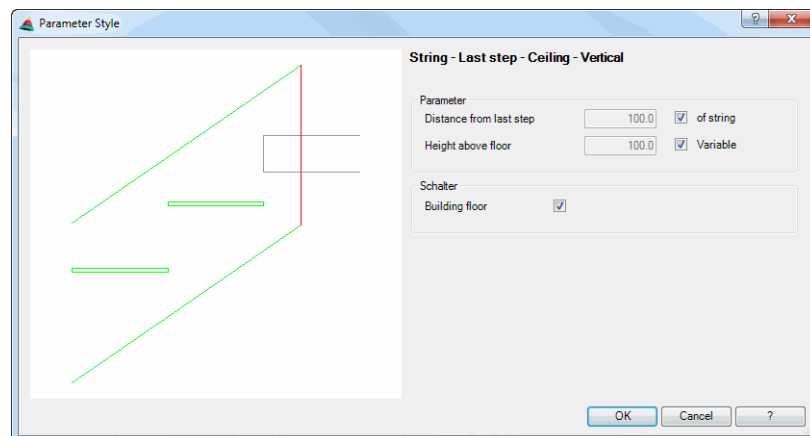


Fig. 3.5: String dimensions: Last step - Ceiling - Standard

Dialog box Parameters String - Last step - Vertical**Distance from last step**

Specifies the horizontal distance from the string rear edge to the front edge of the ceiling, see figure String dimensions: Last step - Ceiling - Vertical: Dimension 1. The entry field is enabled when you deactivate the From string tick box.

From string

Uses the same distance as to the string top edge (top insert) for the distance from the last step.

Height above floor

Specifies the vertical distance of the string top edge to the ceiling, see figure String dimensions: Last step - Ceiling - Vertical: Dimension 2. The entry field is enabled when you deactivate the From string tick box.

Variable

The top edge of the string is formed pointed. The distance to the ceiling is influenced by other parameters (e.g. distance from last step).

Floor structure

With the tick box activated the floor structure is ignored (distances are

referenced to the bare floor), see figure String dimensions: Last step - Ceiling - Vertical: Dimension A.

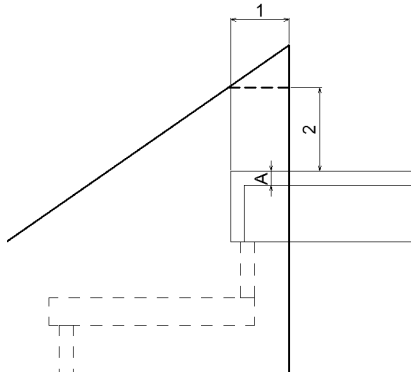


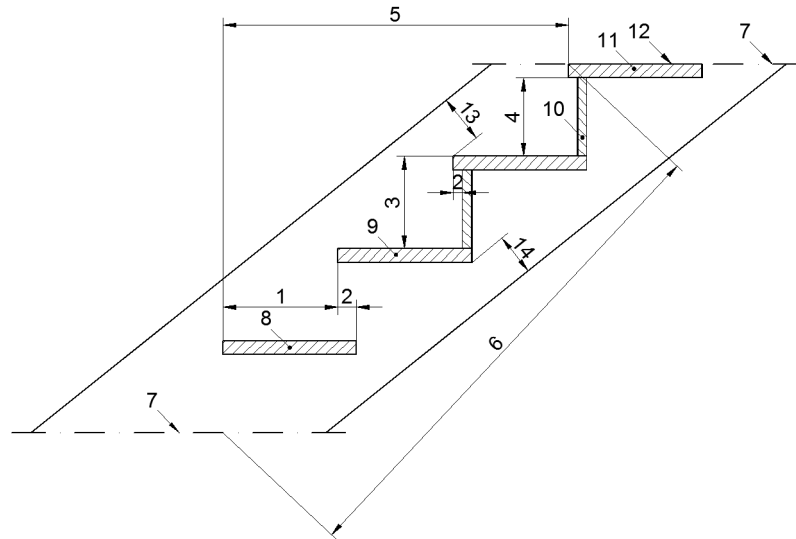
Fig. 3.6: String dimensions: Last step - Ceiling - Vertical



The possible settings of all string parameters depend on the selected type of connection of the string and on the first step or last step which was selected in the Drop-down menu Landing. Furthermore, the options of the dialog box may differ slightly, depending on the type of landing selected (floor, ceiling or intermediate landing).

3.4 Designations and terms

The definition of terms is based on DIN 18065 (Building stairways). For further terms please refer to the standard.



- | | |
|--|------------------|
| 1. Going [a] | 2. Toe space [u] |
| 3. Rise [s] | 4. Step spacing |
| 5. Flight length | 6. Flight |
| 7. Stairway landing, intermediate landing, ceiling | 8. First step |
| 9. Tread | 10. Riser |
| 11. Last step | 12. Tread area |
| 13. Top inset | 14. Bottom inset |

First step

The first step is defined as the first (lowermost) step of a flight of steps after leaving the story level. The first step may be a constituent part of the stairway or of the story level.

Going

The going is the horizontal measurement of the tread surface extending from the front edge of a step to the projection of the front edge of the following step.

Last step

The last step is designated as the last (uppermost) step of a flight of steps, which may also be part of the top landing. The last step is therefore either a constructional part of the stairway and connects to the story level at the same height or it is a constructional part of the story level.

Comfort

The comfort is calculated as the going minus rise and should be about 12 cm ($a-s=12\text{cm}$).

Inset

Distance of the front edge of a tread to the top edge of the string (top inset) or from the rear edge of a tread to the bottom inset of the string (bottom inset).

Sure-footed safety

The sure-footed safety is calculated as the going plus the rise and should be about 46 cm ($a+s=46\text{cm}$).

Walking line

The walking line is an imaginary line indicating the normal path taken by a user of a stairway. It lies in the walking area of the stairway plan.

Slope

The slope of the stairway is the angle between the imaginary line connecting the front edges of the treads and the horizontal. It is given by the rise-to-going ratio (rise/going).

Step length

The step length (step proportion) is calculated from twice the rise plus going and should ideally be 63 cm to 65 cm ($2s+a= 63\text{ cm to }65\text{ cm}$).

Riser

The riser is a perpendicular or almost perpendicular part of the step.

Rise

The rise is the perpendicular distance measured from the tread surface of a step to the tread surface of the following step.

Rise-to-going ratio

The rise-to-going ratio is the quotient of the rise over going. It gives the slope of the stairway.

Winding steps

Winding is the technical term for the course of the spiral steps of a stairway. For the winding of the steps many different projection methods have arisen in the course of centuries. The result of good winding should however always be a comfortable and safe stairway, taking into account the step-length rules and the requirements of building legislation.

Stairwell

The stairwell is the air space in the center of a stairway system. The stairwell is the free space enclosed by the flights, landings and banisters.

Tread surface

See going.

Tread

The tread is the horizontal part of the step.

Step proportion

See step length.

Toe space (undercut)

The toe space is the horizontal measurement by which the front edge of a step protrudes over the width of the tread of the step below.

Winding

See Winding steps.

4 Engineering

Commands in this section:

- Center of gravity and moments
- I_x Required/Deflection/Collapsing stress
- Thickness of Pane / solid
- Thermal analysis
- Thermal resistance
- U_{cw} value for a facade
- R_w value computation

4.1 Center of gravity and moments



Ribbon:	Tab ATHENA > Group Extras > Center of gravity and moments
Menu:	ATHENA > Engineering > Center of gravity and moments
Toolbar:	ATH Engineering and ATHENA Functions
Command input:	ath_schw

With this command you can compute centers of gravity and static moments of an individual profile or of a number of combined profiles. With combined profiles you must specify a composite structural coefficient to define the composition quality.

The program computes or dimensions the following values:

- Moments of inertia (I_x , I_y)
- Section moduli (W_x , W_y)
- Radii of inertia (I_x , I_y)
- Spacings to centroid axis (ex_1 , ex_2)
- Center of gravity
- Area (A)
- Outer contour
- Weight/material

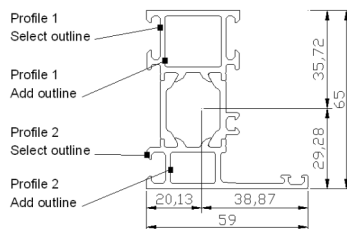


Fig. 4.1: Center of gravity and moments of profile sections

Command prompt

Profile 1

Select outline or [?]:

Click the outer or inner outline of the first profile.

*Input **ENTER** to terminate the program.*

You call the help with the option ?.

Profile 1

Add outline or [Undo/?]:

Click another outer or inner outline of the first profile.

*With the option **Undo** the previous input request is repeated.*

*This input request is repeated until you change to the next input request with **ENTER** or a **RIGHT CLICK**.*

Profile 2

Select outline or [?]:

Click an outer or inner outline of the second profile.

*The input requests are repeated as described above for further outlines and profiles. Once you have selected the last profile outline, input **ENTER** to take you to the next input request.*

Enter composite structural coefficient or [?] <0.7>:

Enter a value between 0 and 1 as Composite structural coefficient.

*Input **ENTER** to accept the default value. The Dialog box Center of gravity and moments opens where you can view the results of the computation and carry out further settings.*

Dialog box Center of gravity and moments

Dialog box section Moments of inertia

With reg. to centroid axes

Outputs the moments of inertia (I_x and I_y) referred to the centroid axes in the results table.

With reg. to principal axes

Outputs the moments of inertia (I_1 and I_2) referred to the principal axes in the results table.

Principal axis position

Outputs the position of the principal axis in the results table.

Composite structural coefficient

Outputs the Composite structural coefficient in the results table.

Dialog box section Section moduli

Outputs the section moduli (W_x and W_y) in the results table.

Dialog box section Radius of inertia

Outputs the radius of inertia (i_x and i_y) in the results table.

Dialog box section Spacings to centroid axis

Outputs the spacing to the centroid axes in the results table.

Dialog box section Cross-section

Outer contour

Outputs the circumference of the outer outline in the results table.

Total area

Outputs the total area in the results table.

Dialog box section Material

Outputs the selected material in the results table. Choose the appropriate material from the list.

Only materials are offered when the density is defined in the physical values. You will find further information on materials in the sections *Material* on page 81 and *Physical values* on page 112.

Weight

Outputs the weight (depending on the selected material) in the results table.

Dialog box section Paste

Text

Inserts a results table into the drawing.

Dimensioning

Dimensions the distance from the outline outer edge to the principal axes.

Principal axes

Draws the principal axes in the outline.

Update

Updates an existing table, which you have to select.

If you click OK with the option Text activated, the following input request appears.

Command prompt when text is switched on.

Specify insertion point:

Specify the insertion point of the results table.

Specify rotated angle or <0>:

Determine the rotation angle of the results table.

*Input **ENTER** to accept the default value.*

Command prompt when update is switched on.

Select table for update or [?]:

Select an existing table to update it.

Composite structural coefficient

The composite structural coefficient reduces the Steiner component in the computation. Consequently, the composite structural coefficient has no effect on the result when the computed profiles have the same center of gravity.

The composite structural coefficient enables you to define the shear strength of various profiles - e.g. composite structural coefficient 1 = shear resistant (rigid) joint (profiles are welded all round). Composite structural coefficients for manufactured profiles must be obtained from the manufacturer.

Notes

- The cross sections to be computed must be drawn in the scale of 1:1.
- The outer and inner contours of the various cross-sections must be circles or polylines (also in blocks). Polylines should be closed. Unclosed polylines are "virtually" closed for the computation. This can lead to inaccurate results.
- With the outline definition of the individual sections (inner or outer outlines) they are displayed in different colors.
- The results are output as a value table. You can modify the display of the table with the command **Table properties**.
- You can define the layer used for the value table in Dialog box System layer.

- The texts in the value table are saved in multiple languages. You can display the text in another language with the command **Set text language**.
- You will find information on materials in the Chapter *Material* on page 81.

Hinweis: This function for the calculation of structural analytical values is an aid for the user. CAD-PLAN GmbH can in no way be held liable for the results and any errors and losses arising from it.

4.2 Ix Required/Deflection/Collapsing stress

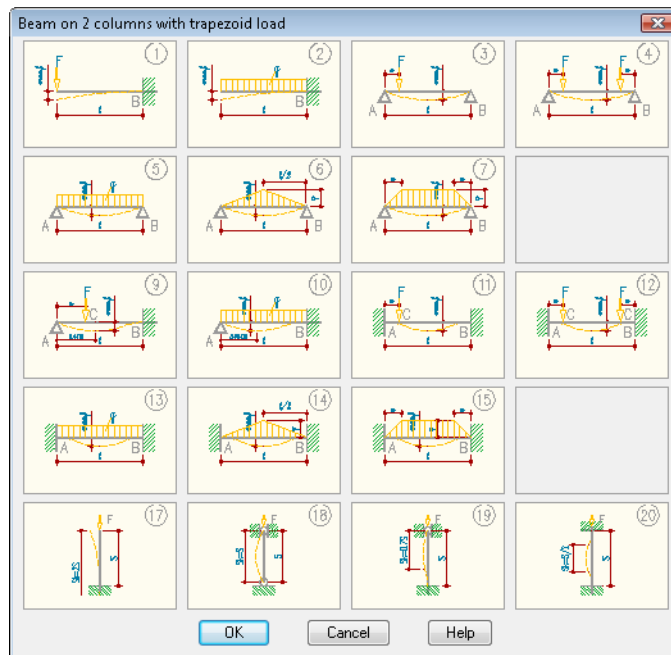


Ribbon: Tab ATHENA > Group Extras > Ix requ. Deflection
Menu: ATHENA > Engineering > Ix requ. Deflection
Toolbar: ATH Engineering and ATHENA Functions
Command input: ath_stik

With this command you determine the required moment of inertia, the maximum deflection and the maximum moment of a beam under 14 different loading cases. Furthermore you can compute the collapsing stress of a support. Here there is a choice of four more loading cases (Euler cases).

For the computation you also need the Ix value of the profile section to be computed. You compute this with the aid of the command [Center of gravity and moments](#). With standard and manufacturers' profiles you can also look up the values in tables or system catalogs from the manufacturer.

Dialog box Overview of the loading cases



You access this dialog box by clicking the graphical preview. To select a loading case you must double click the corresponding picture.

The dialog box for the input of the structural analysis computation behaves dynamically. If you choose a loading case from 1 to 15, the Dialog box for the computation of the required moment of inertia. is started.

If you choose a loading case from 17 to 20, the Dialog box for the computation of the collapsing stress. is started.

Dialog box for the computation of the required moment of inertia.

You can enter a heading or comment for your computation in the input line. Under the input field you will find a pick list from which you can select a loading case.

Dialog box section Profile permissible/chosen

Here, you enter the selected profile values (from the computation, book of tables or system catalog). Permissible values (depending on the geometry and loading case) are displayed and cannot be changed.

Dialog box section Values permissible/actual

Here you enter the maximum deflection. Permissible values (depending on the geometry and loading case) are displayed and cannot be changed.

The computational results are maintained in parallel and can be recognized here. This enables you to compare your results and immediately correct them.

Dialog box Structural analysis (geometry)

Click on the Geometry button to open the dialog box Structural analysis - Geometry. Here, you can enter your geometrical values. Depending on the selected loading case, it may be possible that you cannot complete all the fields.

OK closes the dialog box and you see the main dialog box with the computed results.

Dialog box Structural analysis (loads/material)

Click on the Loads/material button to open the dialog box Structural analysis - loads/material. Here, you can enter the loads and material parameters. Fields not required are masked out (depending on the loading case). After quitting with OK, the results are immediately displayed in the main dialog box.

Dialog box for the computation of the collapsing stress.

You can enter a heading or comment for your computation in the input line. Under the input field you will find a pick list from which you can select a loading case.

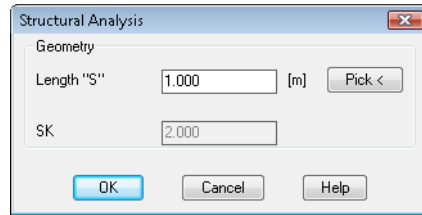
Dialog box section Profile permissible/chosen

Here, you enter the selected profile values (from the computation, book of tables or system catalog). Permissible values (depending on the geometry and loading case) are displayed and cannot be changed.

Dialog box section Values permissible/actual

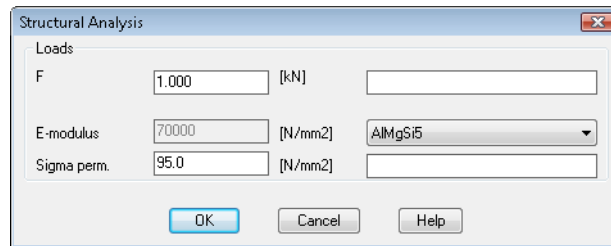
Here you enter the maximum deflection. Permissible values (depending on the geometry and loading case) are displayed and cannot be changed.

The computational results are maintained in parallel and can be recognized here. This enables you to compare your results and immediately correct them.

Dialog box Structural analysis (geometry)

Click on the Geometry button to open the dialog box Structural analysis - Geometry. Here you can enter the length S. The collapsing length SK is computed.

OK closes the dialog box and you see the main dialog box with the computed results.

Dialog box Structural analysis (loads/material)

Click on the Loads/material button to open the dialog box Structural analysis - loads/material. Here, you can enter the loads and material parameters or select a material to automatically apply the specific value. After quitting with OK, the results are immediately displayed in the main dialog box.

Dialog box section Output

This section is identical for all loading cases. With the tick box Insert text you specify whether the results table is inserted into the drawing after OK. If Insert text is deactivated, the computed values are however saved for the duration of the drawing session.

With an activated tick box Insert image, you position a graphical image next to the results table in the drawing. Insert image is masked out when you deactivate Insert text.

The button Default height sets the current dimension figure height as default for the text height in the results table.

Output of results

When you quit the dialog box with OK, you can insert the results table into the drawing (assuming Insert text is activated). The computations are saved for the duration of the drawing session. If you quit the dialog box with Cancel, the computations are reset.

Command prompt

Specify insertion point or [?]:

Specify the insertion point of the results table.

You call the help with the option ?.

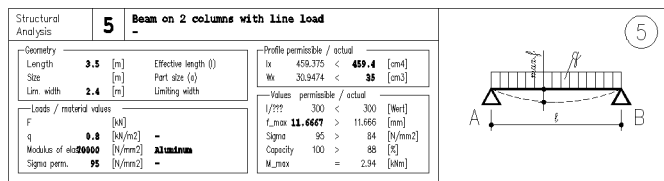


Fig. 4.2: Output of results in the drawing

Notes

- You can add your own materials. You will find further information in the Chapter *Material* on page 81.
- You can change material parameters in the Dialog box Physical Material Properties. You will find information on this in the Chapter *Physical values* on page 112.
- The results table is a block with attributes. With **Modify ATHENA** you open the dialog box Required moment of inertia and can work on the computation.
- The texts in the results table are saved in multiple languages. You can display the text in another language with the command **Set text language**.

Hinweis: This function for the calculation of structural analytical values is an aid for the user. CAD-PLAN GmbH can in no way be held liable for the results and any errors and losses arising from it.

4.3 Thickness of Pane / solid



Ribbon: Tab ATHENA > Group Extras > Pane / solid thickness
Menu: ATHENA > Engineering > Pane / solid thickness
Toolbar: ATHENA Engineering
Command input: ath_bach

With this program section you can carry out a thickness computation of a solid under a defined area load (e.g. wind or snow). The basis for the computation is Bach's plate formula.

Dialog box Solid supported on four sides with area load

Pane / Solid Thickness

Comment:

Input

Width of pane [m]:

Height of pane [m]:

Area load (wind...) DIN 1055 [kN/m²]:

Thickness of pane [mm]:

Type of material: Modulus of elasticity [N/mm²]:

Results

Sigma = 8.44 actual [N/mm²]
 f_{actual} = 3.49 Deflection [mm]
 l/300 = 3.33 for short span [mm]

☒ Insert text Text height:

OK Cancel Help

Comment input field

Enter a text here as heading or comment.

Dialog box section Input

In the input field Width of pane [m] enter the width of the solid in m. In the input field Height of pane [m] enter the height of the solid in m. In the input field Area load (wind) from DIN 1055 [N/mm²] you enter the area loading acting on the solid. You can use the following values as a basis in the vertical method of installation for the pane or solid (wind load): (Impact pressure accord. to Wendehorst):

0 m -	8 m	0.5 kN/m ²
8 m -	20 m	0.8 kN/m ²
20 m -	100 m	1.1 kN/m ²
100 m -	x m	1.3 kN/m ²

In the input field Thickness of pane [mm] enter the thickness of the solid in mm. With the pick list Type of material you choose the material used for the solid. The modulus of elasticity of the selected material is automatically entered into the input field Modulus of elasticity (N/mm²). If required, you can correct the value for the modulus of elasticity. The results of the computation are displayed immediately in the output field Results. When you activate the tick box Insert text, a results table is inserted into the drawing after clicking OK. You can define the text height of the results table in the Text height input field. The default for the text height is the current dimension figure height. When you terminate the dialog box with OK, the current entries are saved for the duration of the drawing session. With Cancel, the entries are discarded.

Command prompt

Specify insertion point:

Specify the insertion point of the results table.

Specify rotated angle<0>:

Determine the rotation angle of the results table.

Press the Enter key to accept the default rotation angle.

Panel or solid figure under defined area load		
CAD-PLAN		
-- Input --		
W	= Span	1.00 m
H	= Span	2.00 m
p	= Aerea load in	0.50 kN/m2
W	= Width Pane/Solid figure	6.00 mm
Modulus of elast. = Material related 73000.00 N/mm2		
-- Result --		
Sigma	= actual	8.44 N/mm2
f_atual	= Deflection	3.49 mm
l/300	= Deflection	3.33 mm
-- Interim result --		
L/K	= Side ratio for Phi	2.00
Phi	= Coefficient from table	2.43
Psi	= Coefficient from table	1.76
K	= shorter dimension * 0,5	0.50 m
-- Info --		
Basic of computation is		
the Bach plate formula		

Fig. 4.3: Pane / solid thickness results text

Notes

- You can add your own materials. You will find further information in the Chapter *Material* on page 81.
- You can change material parameters in the Dialog box Physical Material Properties. You will find further information on this in the Chapter *Physical values* on page 112.
- The result is inserted into the current drawing as a table. You can adapt the display of the table with the command *Table properties*.
- You can recompute the results table with the command *Modify ATHENA*. When you click the table, the dialog box with the default values from the selected table appears. The table is updated after terminating the dialog box with OK.
- You can define the layer used for the results table in Dialog box System layer.
- The texts in the results table are saved in multiple languages. You can display the text in another language with the command *Set text language*.

Hinweis: This function for the calculation of structural analytical values is an aid for the user. CAD-PLAN GmbH can in no way be held liable for the results and any errors and losses arising from it.

4.4 Thermal analysis



Ribbon:	Tab ATHENA > Group Extras > Thermal analysis
Menu:	ATHENA > Engineering > Thermal analysis
Toolbar:	ATH Engineering and ATHENA Functions
Command input:	ath_flixo

With this function you can transfer an existing construction to flixoAT (or to flixo Professional) to conduct a thermal analysis. After the computation with flixoAT you can transfer a block with the temperature trace to ATHENA and insert it into the current drawing.

The material parameters required for the computation (thermal conductivity λ) are, where they are established by ATHENA materials or layers, passed on to flixoAT. You can manually enter missing values or select them from the extensive database included in flixoAT.

In order that material values are automatically transferred to flixoAT, one of the three following conditions must be fulfilled:

- You assign a type of material to an ATHENA object (e.g. thermal insulation).
- You assign to a block an attribute name "mat", the attribute value of which is identical to the name (the "logical" name, not the designation) of a material.
- You assign to an object a layer, the name of which corresponds to the name (the "logical" name, not the designation) of a material.

In each of the three cases a specific material parameter for the thermal conductivity must be assigned to the material.

You will find information on the management of materials and material values in ATHENA in the Chapters *Material* on page 81 and *Physical values* on page 112.

Preparing the drawing

With the transfer of a construction from ATHENA to flixoAT each sub-area is assigned the specific thermal conductivity depending on its layer. Therefore, problems can arise with open outlines and single lines. It is also important that the lines of adjacent constituent parts of the construction lie exactly one above the other.

All constituent parts of the construction consisting of the same material must lie on the same layer.

For the thermal analysis only the actual cut construction should be visible. This means that the lines superfluous for the computation, such as face edges and local constituent parts of the construction are to be deleted (provided the computation is not to take place at just this point).

Areas in the construction, which are enclosed by two layers, are defined as cavities. Cavities must really be closed (e.g. the sealing lips of the EPDM seals must be brought up to the edge of the glass).

The drawing elements superfluous to the thermal analysis, such as standard parts, dimensions and hatching, are automatically filtered out during the transfer of the drawing from ATHENA to flixoAT.

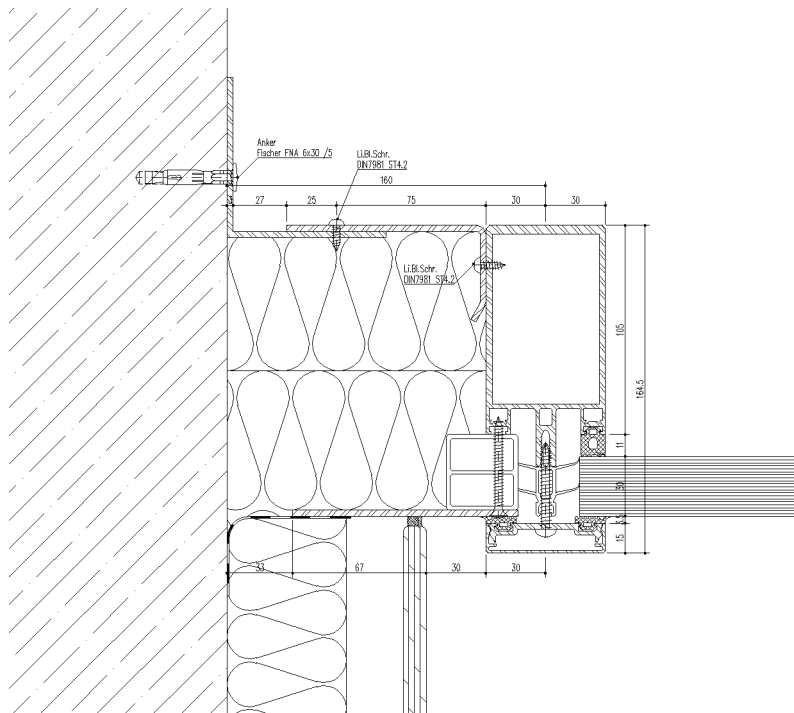


Fig. 4.4: Wall connection

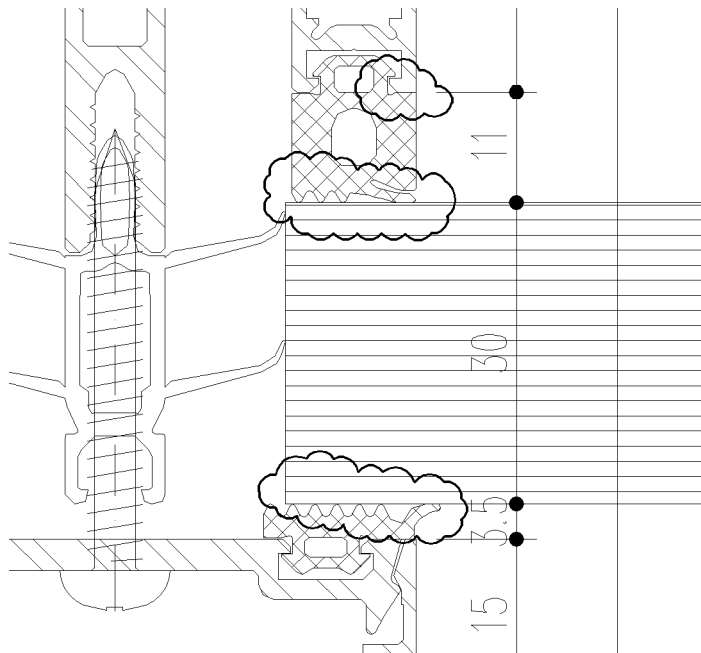


Fig. 4.5: Detail of EPDM seal on a glass infill, closed cavities.

Command prompt

Specify corner point or [Settings/?]:

Specify the first corner point of the arrangement to be computed.

Choose the option **Settings** to start the Dialog box Heat transmission settings.

You call the help with the option ?.

Specify diagonal corner point or [?]:

Specify the diagonal corner point of the arrangement to be computed.

Specify temperature in °C or [?] <20>:

Specify the temperature for the region outlined in green by ATHENA.

Press the Enter key to accept the default temperature.

Specify temperature in °C or [?] <20>:

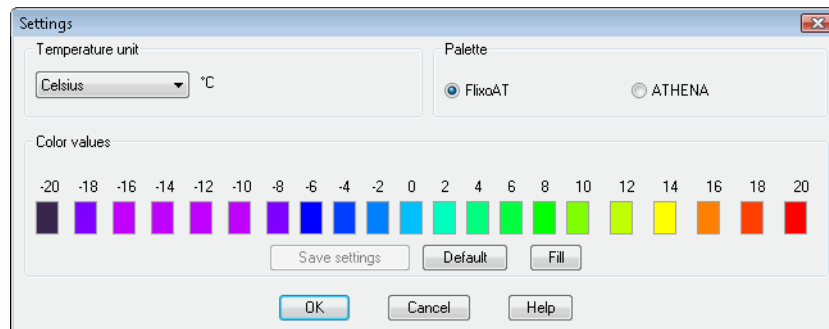
Specify the temperature for the region outlined in green by ATHENA.

Specify pitch size or [?] <4>:

Enter the pitch size (mm) for the color scale of the temperature variation. The smaller the pitch size, the finer the temperature variation. Since the pitch size also has an effect on the speed of transfer, we recommend acceptance of the default value.

Option Settings

Dialog box Heat transmission settings



In this dialog box you can select the required temperature unit from the list.

When you activate the option flixoAT in the Palette section, ATHENA uses the same colors as flixoAT for the temperature variation. If you activate ATHENA, ATHENA uses the colors defined in the section Color values ATHENA for the temperature variation.

You can assign colors to the various temperatures in the section Color values ATHENA. To do this click a color field with the mouse and then select a color in the following dialog box.

When you click the button Save settings, ATHENA saves the changes. Otherwise the changes are only valid for the duration of the drawing session. If you click the button Default, the settings in the dialog box are reset to the last saved settings. If you click the Fill button, you can specify a color for all temperature values. You can then select a different color for a certain temperature (e.g. 10° isothermal) to highlight this temperature curve.

Click OK to save the settings for the duration of the drawing session. Click Cancel to discard the settings. ATHENA terminates the dialog box and continues with the input request: *Specify corner point or [Settings/?]:* (see syntax above).

Notes

- flixoAT is a light version of the thermal analysis and reporting program "flixo Professional" and is installed with ATHENA.
You will find detailed information about its operation in the flixoAT documentation.
- If you have installed flixo Professional, you can use it instead of flixoAT.
ATHENA transfers the data to the flixo version which was last started.
- You will find further information about colors and color selection in your AutoCAD documentation.

Hinweis: This function for conducting a thermal analysis is an aid for the user. CAD-PLAN GmbH can in no way be held liable for the results and any errors and losses arising from it.

4.5 Thermal resistance



Ribbon: Tab ATHENA > Group Extras > Thermal resistance
Menu: ATHENA > Engineering > Thermal resistance
Toolbar: ATHENA Engineering
Command input: ath_htrans

With this command you determine the thermal resistance of components. The components may be composed of any layers. All components are saved in a catalog which can be extended as required.

Dialog box Thermal resistance

Building element components

	d[m]	λ/mK	m²K/W
Glass	0.008	0.800	0.010
Mineral and vegetable fiber insulation WLF 035	0.080	0.035	2.286
Aluminum	0.003	200.000	0.000
Element	0.091		2.296

Edit

Mineral and vegetable fiber insulation WLF 035 d[m]: 0.080 λ/mK: 0.035

Buttons: Edit, Insert, Replace, Remove, Save

Catalog

Sorted (selected) Continuous < > Take on Edit...

5.5 Mineral and vegetable fiber insulation WLF 035 0.035

Output

☒ Insert text Text style header: Tabelle Text height: 3.50 Default height Table: Tabelle

Buttons: OK, Cancel, Help

Heat transmission

R_{si} 0.130 m²K/W
R 2.296 m²K/W
R_{se} 0.040 m²K/W
.....
R₁ 2.466 m²K/W
U 0.406 W/m²K
Internal surface temperature
Air i. 20.00 °C
Air e. -20.00 °C
q 16.22 W/m²
.....
Temp. si 17.89 °C

In the dialog box Thermal resistance you can enter a comment or a title for the computation in the input line at the top. This title can be saved in various languages. Select the required language from the list and enter the text in the chosen language.

Dialog box section Building element components

Building element components			
	d[m]	λ/mK	m²K/W
Glass	0.008	0.800	0.010
Mineral and vegetable fiber insulation WLF 035	0.080	0.035	2.286
Aluminum	0.003	200.000	0.000
Element	0.091		2.296

In the table the selected layers of the component are shown with the appropriate parameters and results as well as the sums of the parameters.

The values from left to right: Component thickness d [m]; nominal value of the thermal conductivity λ_R [W/mK]; thermal resistance of the component layer R [m²K/W]. The sum of all component thicknesses and thermal resistances is shown below the table.

Dialog box section Edit

	d[m]	λ/mK
Mineral and vegetable fiber insulation WLF 035	0.080	0.035

Buttons: Edit, Insert, Replace, Remove, Save

Here you have the possibility of modifying the component thickness and nominal value of the selected component.

With the Edit button you transfer the details of the component, which is marked in the table Building element components, to the input line. You can change the name of the component, the thickness and the nominal value of the thermal conductivity in the input line.

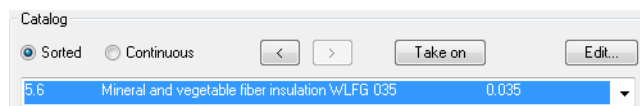
With the Insert button the details from the input line are transferred into the table Building element components. The details are always inserted above the marked line.

With the Replace button the details of the component marked in the table are overwritten.

If you press the button Remove, the marked component is deleted from the table.

With the Save button the details in the input line are saved in the user section of the catalog. You therefore have the possibility of extending the catalog with your own (new or modified) data.

Dialog box section Catalog



Here, you can call up the thermal and moisture protection parameters according to DIN 4108 and use them for the computation.

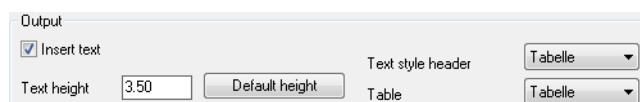
The option buttons Sorted and Continuous affect the display of the catalog entries. Sorted only shows the values or headings of the current chapter or sub-chapter. Continuous shows the parameters of the whole catalog.

With the buttons > you page in the catalog one level lower (e.g. from 1.1 to 1.1.1) and with < one level higher (e.g. from 1.1.2 to 1.1).

With the button Adopt, you transfer the parameter from the catalog into the input line of the Edit section.

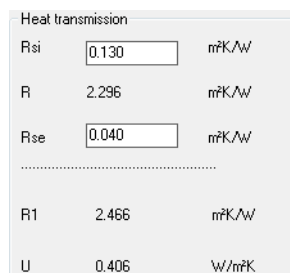
When you click the button Edit ..., the sub-dialog box User catalog is opened for editing. You will find information on the user catalog in the Chapter *User catalog* on page 130.

Dialog box section Output



When the option Insert text is activated, a results text is inserted into the drawing on clicking OK. An input request in the command line follows. You can define the text height for the table in the field Text height. With the button Default the default text height (current height of dimension figures) is set. With the pick lists Text style header and Table you define the text style for the corresponding table elements.

Dialog box section Heat transmission



Here you define the inner and outer heat transmission resistances. These values are dependent on the position of the components and can be read from the appropriate tables. Refer also to the illustration Default values for thermal transmission resistances. Furthermore, the thermal resistances R , R_1 and the thermal transmission coefficient U are computed in this dialog box section.

Dialog box section Internal surface temperature

Internal surface temperature		
Air i.	20.00	°C
Air e.	-20.00	°C
q	16.22	W/m²
.....		
Temp. si	17.89	°C

Here you can enter the internal (Air i) and external (Air e) air temperatures. The thermal conductivity q and the internal surface temperature Temp.si are computed.

When you terminate the dialog box Thermal resistance with OK, a results table is inserted into the drawing. Then follows the query:

Command prompt

Specify insertion point:

Use the mouse or enter coordinates to specify the insertion point for the table.

Specify rotated angle<0>:

Specify the rotation angle of the table or confirm the default angle.

Notes

- You can recompute the results table with the command **Modify ATHENA**. When you click the table, the dialog box with the default values from the selected table appears. The table is updated after terminating the dialog box with OK.
- You can define the layer used for the results table in Dialog box System layer.
- The texts in the results table are saved in multiple languages. You can display the text in another language with the command **Set text language**.
- For this routine the same material catalog is used as for the routine **Rw value computation**. With the command **User catalog** you can save user-specific materials in a catalog.

Hinweis: This function for the calculation of building physics values is an aid for the user. CAD-PLAN GmbH can in no way be held liable for the results and any errors and losses arising from it.

4.5.1 User library for thermal resistance

Tables with physical quantities, default values and component positions relevant to thermal resistance.

Physical Quantities, Symbols and Units

Physical Quantity	Symbol		SI unit
	Current	Outdated	
Thermal energy	Q	Q	J (1J = 1Ws)
Thermal current	Φ	Φ	W
Thermal current density	q	q	W / m ²
Thermal conductivity	λ	λ	W / (m*K)
Thermal transition coefficient	Λ	Λ	W / (m ² *K)
Thermal resistance	R	1 / Λ	m ² *K / W
Thermal transmission coefficient	h	α	W / (m ² *K)
Internal thermal transfer resistance	R _{si}	1 / α_{si}	m ² *K / W
External thermal transfer resistance	R _{se}	1 / α_{se}	m ² *K / W
U value	U	k	W / (m ² *K)
Thermal resistance	R _T	1 / k	m ² *K / W
Specific heat capacity	c	c	J / (kg*K)
Air change rate	n	n, β	1 / h
Joints transmission coefficient	-	a	m ³ / (h*m*dal
degree of thermal transparency	-	g	1 ¹⁾
Reduction factor	-	z	1 ¹⁾
Absolute temperature	T	T	K
Celcius temperature	Θ	θ	°C
Thickness	d	s	m
Length	l	l	m
Area	A	A	m ²
Volume	V	V	m ³
Time	t	t	s
Density	ρ	ρ	kg /m ³

¹⁾ stands for the relationship between two identical units

Indices

internal	i	internal surface	si
external	e	external surface	se
Surface	s		

Fig. 4.6: Quantities, formula symbols, units

Heat Transmission resistance values R_{si,se} ^{1),2)} in m²K/W to DIN 4108

Line	Building Element	R _{si}	R _{se}
1	External wall (except those in line 2)	0.13	0.04
2	External wall with ventilated outer skin ³⁾ , side wall adjoining uninsulated attic		0.08
3	Party wall, stairwell wall, wall between work rooms, partition wall adjoining normally unheated room, side wall adjoining insulated attic		⁴⁾
4	Cellar wall (adjoining soil)		0
5	Ceiling or pitched roof, adjoining open air above (unventilated)	0.13	0.04
6	Ceiling under unused attic, cock loft, or ventilated space (i.e.ventilated pitched roof)		0.08
7	Ceiling between flats, or ceiling between workrooms	Heat flow upwards from below	⁴⁾
		Heat flow downwards from above	
8	Cellar ceiling	0.17	⁴⁾
9	Ceiling, open air below, living space above		0.04
10	Ground floor built directly on soil		0

¹⁾ Apart from lines 4 and 10, this can be simplified in all cases by using R_i=0.13 m²K/W and R_s=0.04m²K/W

²⁾ When checking a building element for the risk of condensation build up, please see the special conditions set out in DIN 4108-3

Fig. 4.7: Default values for thermal transmission resistances

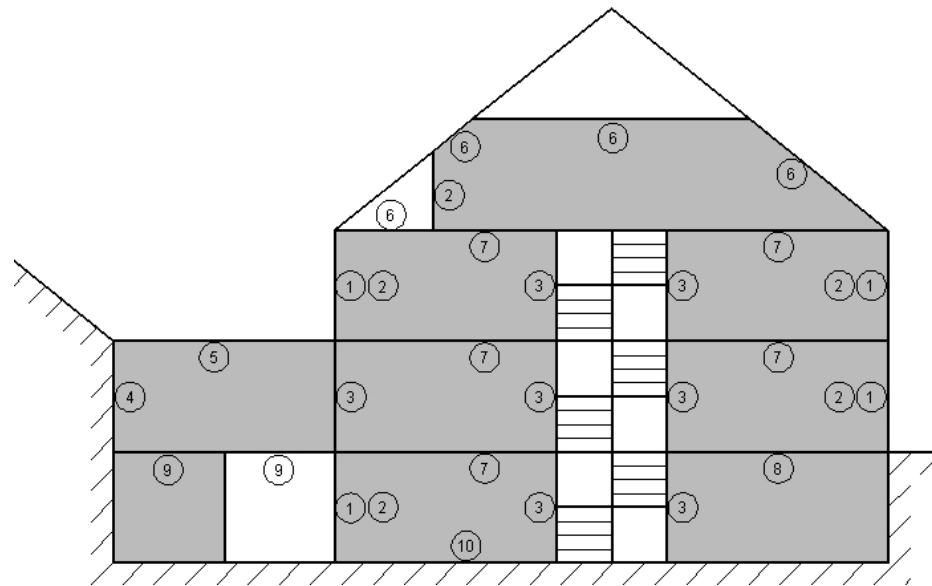


Fig. 4.8: Component positions

4.6 Ucw value for a facade



Ribbon: Tab ATHENA > Group Extras > Ucw value for facade

Menu: ATHENA > Engineering > Ucw value for facade

Toolbar: ATHENA Engineering

Command input: ath_ucw

With this routine you can compute the mean thermal transmission coefficient for a window or a facade.

Dialog box Ucw Value for Facade

Dialog box section Component part

In the dialog box you can enter a comment or a title for the computation in the input line at the top. The title can be saved in various languages. To do this, select the required language from the list and enter the text in the chosen language.

You can define the parameters for the respective position in the input fields. The following values can be entered:

- Number: Number of components.
- Pos.No.: Incremental number or position number of the component.
- Designation: Component name



You can define the designation similarly as with the title in various languages.

- U [W/m²K]: Thermal transmission coefficient of the component
- A [m²]: Area of component
- Psi [W/mK]: Length-referred thermal transmission coefficient for clamped components (e.g. panes).
- I [m]: Length over which the component is clamped (see also the figure below).

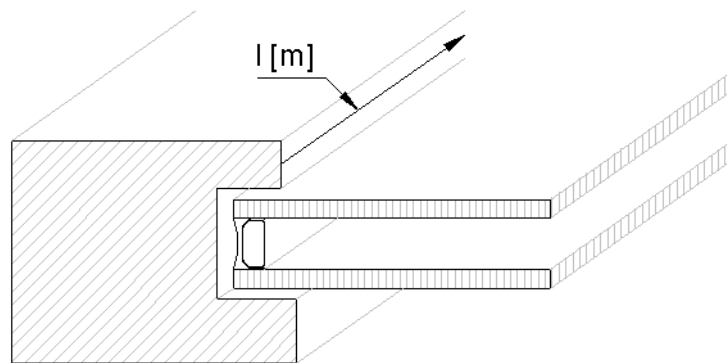


Fig. 4.9: Length of clamping (Psi value)

With the Edit button you transfer the details of the component, which is marked in the table Facade components, to the input fields. You can only change single parameters in the input fields

With the Insert button the details from the input fields are transferred into the table Facade components. The details are always inserted below the marked line.

With the button Replace the marked position in the table is overwritten with the details from the input fields.

If you press the button Remove, the marked position is deleted from the table.

If you click Sort, the positions are sorted.

Dialog box section Facade components

Here the entered elements or facade sections are displayed with their values. Mark a line with the mouse if you want to change it (Replace).

Dialog box section facade

Here the computed results for the area (A_{cw} in m^2) are computed and displayed for the heat transfer coefficient (U_{cw} in W/m^2K).

Dialog box section Output

When the option Insert text is activated, a results text is inserted into the drawing on clicking OK. An input request in the command line follows. You can define the text height for the table in the field Text height. With the button Default the default text height (current height of dimension figures) is set. With the pick lists Text style header and Table you define the text style for the corresponding table elements.

Command prompt

Specify insertion point or [?]:

Specify the insertion point of the results table.

You call the help with the option ?.

Specify rotation angle or [?] <0>

Determine the rotation angle of the results table.

Press the Enter key to accept the default rotation angle.

Hinweis: This function for the calculation of building physics values is an aid for the user. CAD-PLAN GmbH can in no way be held liable for the results and any errors and losses arising from it.

4.7 Rw value computation



Ribbon: Tab ATHENA > Group Extras > Rw value computation

Menu: ATHENA > Engineering > Rw value computation

Toolbar: ATHENA Engineering

Command input: ath_rw

With this routine you can roughly determine the sound insulation factor for a construction.

A classical construction consists of a mass layer (e.g. masonry), an intermediate layer (absorber) and a facing shell.

Dialog box Roughly determined sound insulation factor for a construction.

Building element components

Item no.	d[m]	p[kg/m³]	m'[kg/m²]
Masonry in sand-lime bricks, sand-lime flat bricks (1400)	0.240	1400	336
Intermediate layer S' = 12.000 MN/m²			
Sedimentary rock (sandstone, limestone, slate)	0.020	2600	52
Mass layer	0.240		336

Intermediate layer (absorber) with double-shell construction

S'[MN/m²] 12

Summary Table:

Mass layer	m	336	kg/m²
Rw		56	dB
d		0.240	m
Intermediate layer	s'	1	MN/m²
Facing shell	m	52	kg/m²
d		0.020	m
Construction	f0	24	Hz
Delta Rw		7	dB
Delta Rw		63	dB

In the dialog box you can enter a comment or a title for the computation in the input line at the top. This title can be saved in various languages. Select the required language from the list and enter the text in the chosen language.

Dialog box section Building element components

Building element components			
Item no.	d[m]	p[kg/m³]	m'[kg/m²]
Masonry in sand-lime bricks, sand-lime flat bricks (1400)	0.240	1400	336
Intermediate layer S' = 12.000 MN/m²			
Sedimentary rock (sandstone, limestone, slate)	0.020	2600	52
Mass layer	0.240		336

In the table the selected layers of the component are shown with the appropriate parameters and results as well as the sums of the parameters.

The values from left to right: Component thickness d [m]; density ρ [kg/m³]; area-related mass m' [kg/m²]. Under the table the sums of the mass layer or of the facing shell are displayed.

Dialog box section Edit

Here you have the possibility of modifying the component thickness and density of the selected component.

With the Edit button you transfer the details of the component, which is marked in the table Building element components, to the input line. You can change the name of the component, the thickness and the density in the input line.

With the Insert button the details from the input line are transferred into the table Building element components. The details are always inserted above the marked line.

With the Replace button the details of the component marked in the table are overwritten.

If you press the button Remove, the marked component is deleted from the table.

With the Save button the details in the input line are saved in the user section of the catalog. You therefore have the possibility of extending the catalog with your own (new or modified) data.

In the section Intermediate layer (absorber)... you define the dynamic stiffness S' [MN/m³]. Click the button Insert to insert the intermediate layer above the marked layer. **Important:** All layers above the intermediate layer are treated as mass layers and all layers below the intermediate layer are treated as facing shells.

The following construction (from top to bottom) should be maintained:

1. Mass layer (many components possible)
2. Intermediate layer (only required with double-shell construction).
3. Facing shell (only required with double-shell construction, many components possible).

Dialog box section Catalog

Here you can call the parameters and use them for the computation.

The option buttons Sorted and Continuous affect the display of the catalog entries. Sorted only shows the values or headings of the current chapter or sub-chapter. Continuous shows the parameters of the whole catalog.

With the buttons > you page in the catalog one level lower (e.g. from 1.1 to 1.1.1) and with < one level higher (e.g. from 1.1.2 to 1.1).

With the button Adopt, you transfer the parameter from the catalog into the input line of the Edit section.

When you click the button Edit ..., the sub-dialog box User catalog is opened for editing. You will find information on the user catalog in the Chapter *User catalog* on page 130.

Dialog box section Output

When the option Insert text is activated, a results text is inserted into the drawing on clicking OK. An input request in the command line follows. You can define the text height for the table in the field Text height. With the button Default the default text height (current height of dimension figures) is set. With the pick lists Text style header and Table you define the text style for the corresponding table elements.

Dialog box section Mass layer, intermediate layer and facing shell

In these sections the intermediate results for the layers are displayed. The values are:

- Component thickness d [m]
- Sound insulation factor R_w [dB]
- Area-related mass m' [kg/m²]
- Dynamic stiffness s' [MN/m³]

Dialog box section Construction

In this region the result of the construction is displayed. The values are:

- Resonant frequency f_0 [Hz]
- Sound insulation factor Delta R_w [dB]
- Sound insulation factor R_w tot [dB]

When you terminate the dialog box Thermal resistance with OK, a results table is inserted into the drawing. Then follows the input request:

Command prompt

Specify insertion point or [?]:

Specify the insertion point of the results table.

You call the help with the option ?.

Specify rotation angle or [?] <0>

Determine the rotation angle of the results table.

Press the Enter key to accept the default rotation angle.

Notes

- You can recompute the results table with the command **Modify ATHENA**. When you click the table, the dialog box with the default values from the selected table appears. The table is updated after terminating the dialog box with OK.
- You can define the layer used for the results table in Dialog box System layer.
- The texts in the results table are saved in multiple languages. You can display the text in another language with the command **Set text language**.
- For this routine the same material catalog is used as for the routine **Thermal resistance**. With the command **User catalog** you can save user-specific materials in a catalog.

Hinweis: This function for the calculation of building physics values is an aid for the user. CAD-PLAN GmbH can in no way be held liable for the results and any errors and losses arising from it.

5 Dimension

Commands in this section:

- Dimension object
- Sheet metal dimension
- Divide dimension
- Connected dimensions
- Peripheral dimension
- Coordinate dimensioning
- Leader
- Parts labeling
- Label tags
- Coordinate label
- Position symbols
- Interruption
- Defining interrupted dimensions
- Set interrupted dimension
- Horizontal levels
- Vertical levels
- Highlight associated dimension objects
- Manage dimension styles
- Arrange dimension
- Arrange dimension texts
- Arrange leaders
- Label background color ON
- Label text background color OFF
- Add leader line
- Remove leader line
- Add leader line segment
- Remove leader line segment
- Change label scales
- Modify object label
- Align leaders
- Change dimension settings
- Update dimension
- Dihedral angle
- Length of an object

5.1 Dimension object



Ribbon:	Tab ATHENA > Group Label > Dimension object
Menu:	ATHENA > Dimension > Dimension object
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	<code>ath_dim_obj</code>

With this routine you automatically dimension ATHENA or AutoCAD objects by clicking them. The various Dimensioning options can be set in a dialog box.

Command prompt

Select object or [Settings/?]:

Use the mouse to choose the object to be dimensioned.

*With the option **Settings** you can open the Dialog box Dimensioning options. For information see "Dialog box Dimensioning options" on page 128.*

You call the help with the option ?.

Notes

- Dimensioning is not associative.
- The current dimension settings (e.g. dimension figure height) are used for dimensioning.

5.2 Sheet metal dimension



Ribbon:	Tab ATHENA > Group Label > Sheet metal dimension
Menu:	ATHENA > Dimension > Sheet metal dimension
Toolbar:	ATH Dimension
Command input:	ath_dim_sheet

Dimensions a Sheet metal section.

You can define the dimension settings for sheet metal dimensioning via tick boxes in a dialog box.



This dimension command is only applicable to Sheet metal sections and folded Semi-finished products. The intention is to generate dimensions which are suitable in practice. Consequently, dimensions can be later measured on the finished sheet.

When you execute the command, the following appears:

Command prompt

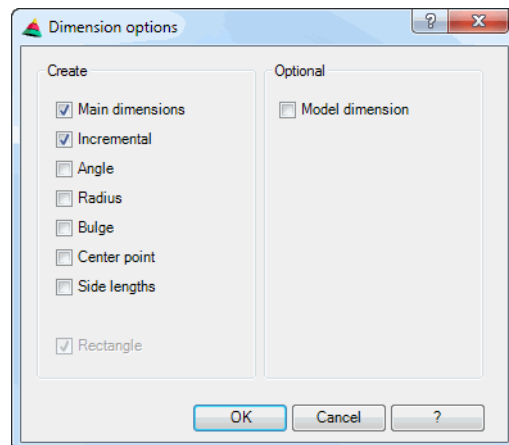
Select sheet metal section or [Settings/?]:

Select the sheet metal section to be dimensioned or select an option.

The Dialog box Sheet metal dimension is opened with the Settings option. Here, you can define which dimensions are to be generated.

You call the help with the option ?.

Dialog box Sheet metal dimension



Edge length

Dimensions the length of the edges in their alignment.

Angle

Dimensions the angle between two adjacent sheet limbs. Right angles are only dimensioned if the tick box 90° has also been activated.

Bending radius

Dimensions the inner bending radius of the sheet.

Sheet thickness

Dimensions the thickness of the sheet metal section.

Maximum dimension

Dimensions the greatest extent (enclosing rectangle) of the sheet orthogonally to the current coordinate system.

Connected dimension

Creates a connected dimension of the sheet orthogonally to the current coordinate system.

Rolling radius

Dimensions the rolling radius of rounded sheet limbs.

OK

Terminates the dialog box, saves the settings made and returns to the input request.

Cancel

Terminates the dialog box, discards the settings made and returns to the input request.

5.3 Divide dimension



Ribbon: Tab ATHENA > Group Label > Divide dimension
Menu: ATHENA > Dimension > Divide dimension
Toolbar: ATH Dimension
Command input: ath_dim_div

With this command you can dimension a linear section with connected dimensions according to certain rules. You define the dimensioning rules in a dialog box.

Command prompt

Specify angle or [Linear/?]: <Linear>:

Specify the angle for the dimension by entering or specifying two points.

Select the option Linear, if you want to generate horizontal or vertical dimensions.

You call the help with the option ?.

Specify second point:

Define the second point of the angle.

Specify starting point of first auxiliary line or [?]:

Specify the first end point of the auxiliary line.

Specify starting point of second auxiliary line or [?]:

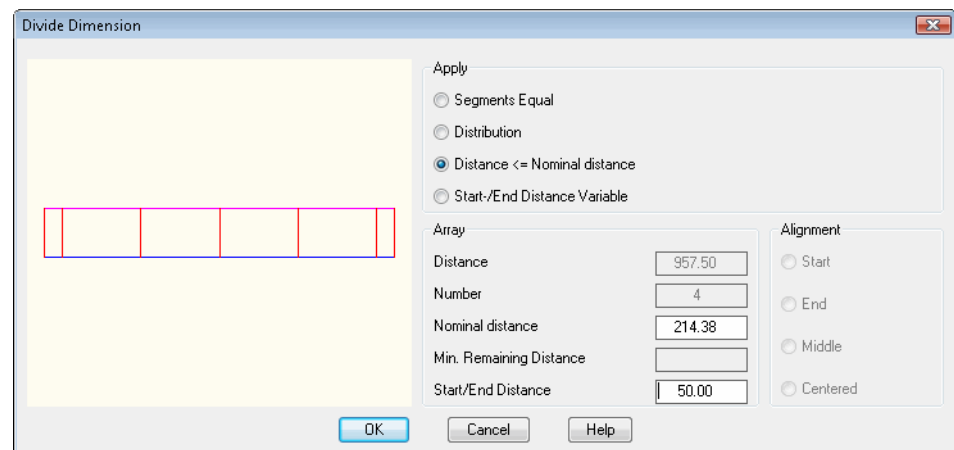
Specify the first end point of the auxiliary line.

Specify position of the dimension line:

Specify the position of the dimension line.

Once you have defined the position, the Dialog box Divide dimension appears.

Dialog box Divide dimension



Dialog box section Apply

Segments equal

Produces a connected dimension with equal segments. The dimensions are calculated from the specified number of segments.

Distribution

Produces a distributed connected dimension. The dimensions are calculated according to the specified array parameters and type of alignment.

Distance \leq Nominal distance

Produces a connected dimension with fixed start and end distances. The dimensions are calculated from the specified start and end distances as well as from the nominal distance.

Start/end distance variable

Produces a connected dimension with variable start and end distances. The dimensions are calculated from the specified nominal distance.

Dialog box section Array

Distance

Shows the distance between the specified points.

Number

Defines the number of dimensions. The number of segments can only be specified when the option Segments equal has been selected.

Nominal distance

Defines the nominal distance of the dimension. The nominal distance cannot be specified when the option Segments equal has been selected.

Minimum remaining distance

Defines the minimum remaining distance of the dimension. The minimum remaining distance can only be specified when the option Distribution has been selected.

Start/end distance

Defines the start or end distance of the dimension. This can only be specified when the option Distribution or Distance $>$ Nominal distance has been applied.

Dialog box section Alignment

The following options are only available when you select the option Distribution.

Start

Aligns the first dimension at the start of the section. The dimension at the end of the section is calculated.

End

Aligns the first dimension at the end of the section. The dimension at the start is calculated.

Middle

Aligns the first dimension in the middle of the section (one dimension is arranged centrally). The dimensions at the start and end of the section are calculated. The specified start/end distance is ignored.

Centered

Aligns the first two dimensions in the middle of the section (the dimension auxiliary line is arranged centrally). The dimensions at the start and end of the section are calculated. The specified start/end distance is ignored.

When you close the dialog box with OK, the dimension chain is generated according to the settings.

5.4 Connected dimensions



Ribbon:	Tab ATHENA > Group Label > Connected dimensions
Menu:	ATHENA > Dimension > Connected dimensions
Toolbar:	ATH Dimension
Command input:	ath_bket

With this routine you can dimension several objects in the form of a connected dimension by specifying an angle or two points as well as the position of the dimension line.

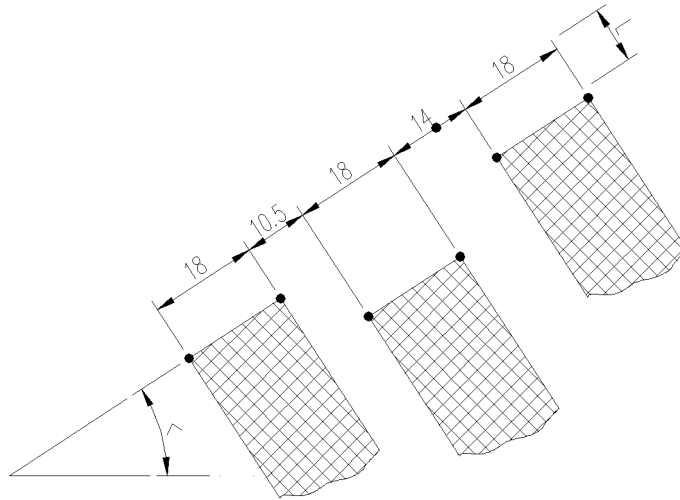


Fig. 5.1: Connected dimensions

Command prompt

Specify angle or [Horizontal/Vertical/?]:

Define the angle of dimensioning by entering two points with the mouse or by entering the angle.

*You can horizontally align the dimension chain with the option **Horizontal**.*

*You can vertically align the dimension chain with the option **Vertical**.*

*You call the help with the option **?**.*

Specify length of the auxiliary line or [Normal/?]: <Normal>:

Set the length of the dimension auxiliary lines by clicking two points with the mouse or entering a length.



Note that the length of the auxiliary line is not an absolute value. It is multiplied by the scale factor of the dimension (*dimscale*)!

*When you use the option **Normal**, the length of the dimension auxiliary line is variable.*

Specify location of the dimension line or [?]:

Use the mouse or enter co-ordinates to specify the location of the dimension line.

Specify first point or [?]:

Enter the start point of the first dimension line.

Specify next point or [?]:

Enter the end point of the first dimension line.

Specify next point or [?]:

*Enter the end point of the second dimension line. This query is repeated until the program is terminated by entering **ENTER**.*

5.5 Peripheral dimension



Ribbon:	Tab ATHENA > Group Label > Peripheral dimensioning
Menu:	ATHENA > Dimension > Peripheral dimensioning
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_dimarc

With this command you create an associative dimension object about the length of an arc, an arc segment or the circumference of a circle for dimensioning. The arcs can also be contained in polylines.

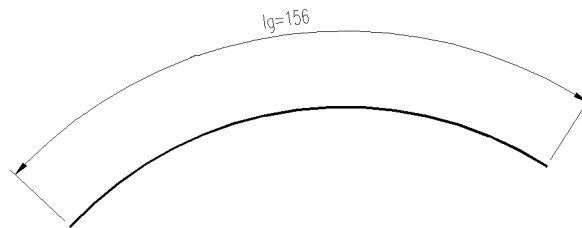


Fig. 5.2: Peripheral dimension, arc length

Command prompt

Select arc or circle or [Arc points/?] <Arc points>:

With the mouse select the arc or circle, the periphery of which you would like to dimension.

*Use the **Option Arc points** to dimension the arc length of a circle or arc segment.*

You call the help with the option ?.

Specify position of dimension arc or [?]:

Choose the position of the dimensional arc with the mouse.

Option Arc points

Specify vertex or [Object/?]:

Define the vertex of the arc dimension. The vertex is located in the center of the circle or arc to be dimensioned.

*With the option **Object** you can select a circle or arc.*

Specify arc point or [Object/Undo/?]:

Enter the first point on the arc.

*With the option **Undo** the previous input request is repeated and you can correct the entry.*

Specify second arc point or [Object/Undo/?]:

Enter the second point on the arc.

Specify position of dimension arc or [?]:

Choose the position of the dimensional arc with the mouse.

Notes

- The length of the desired element is always dimensioned with lg=xx.
- The number of decimal places is determined by the current setting for the dimensioning (system variable *dimdec* = Number of decimal places for the dimensioning).

5.6 Coordinate dimensioning



Ribbon:	Tab ATHENA > Group Label > Coordinate dimension
Menu:	ATHENA > Dimension > Coordinate dimension
Toolbar:	ATH Dimension
Command input:	ath_dim_coordc

With this function you can create incremented absolute dimensions with arrows starting from a specified reference point.
You can use this function when the parts to be dimensioned are to be processed by CNC machine tools.

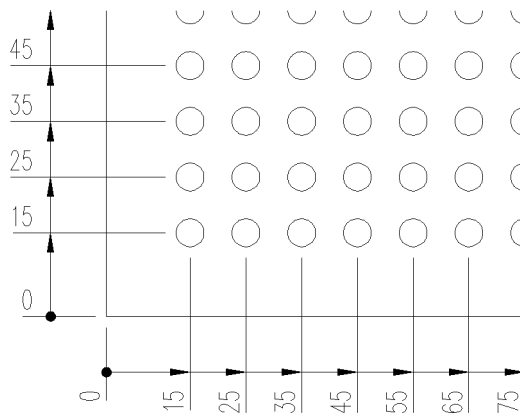


Fig. 5.3: Coordinate dimensioning of a perforated panel

Command prompt

Select dimension chain or [New/?] <New>:

Select an existing dimension chain to extend it.

Choose the option **New** to create a new dimension.

You call the help with the option ?.

Specify next point or [?]:

Use the mouse or enter coordinates to specify the next point to be dimensioned.

Option New

Specify base point of the dimension chain or [?]:

Use the mouse or enter coordinates to specify the reference point for coordinate dimensioning.

Specify position of the dimension line or [?]:

Use the mouse or enter coordinates to specify the position of the dimension line.

Specify next point or [?]:

Use the mouse or enter coordinates to specify the next point to be dimensioned.

Notes

- For coordinate dimensioning ATHENA uses the current dimension layer which you can set up in the Dialog box System layer.
- ATHENA always uses arrows for coordinate dimensioning.
- You will find information on coordinate dimensioning also in your AutoCAD documentation.

5.7 Leader



Ribbon: Tab ATHENA > Group Label > Leader
Menu: ATHENA > Dimension > Leader
Toolbar: ATH Dimension
Command input: ath_leader

With this command you can create single or multi-line and multi-language texts for the labeling of objects. The leader can be formatted in various ways, for example with a frame or symbol.

Command prompt

Specify starting point of the leader line or [?]:

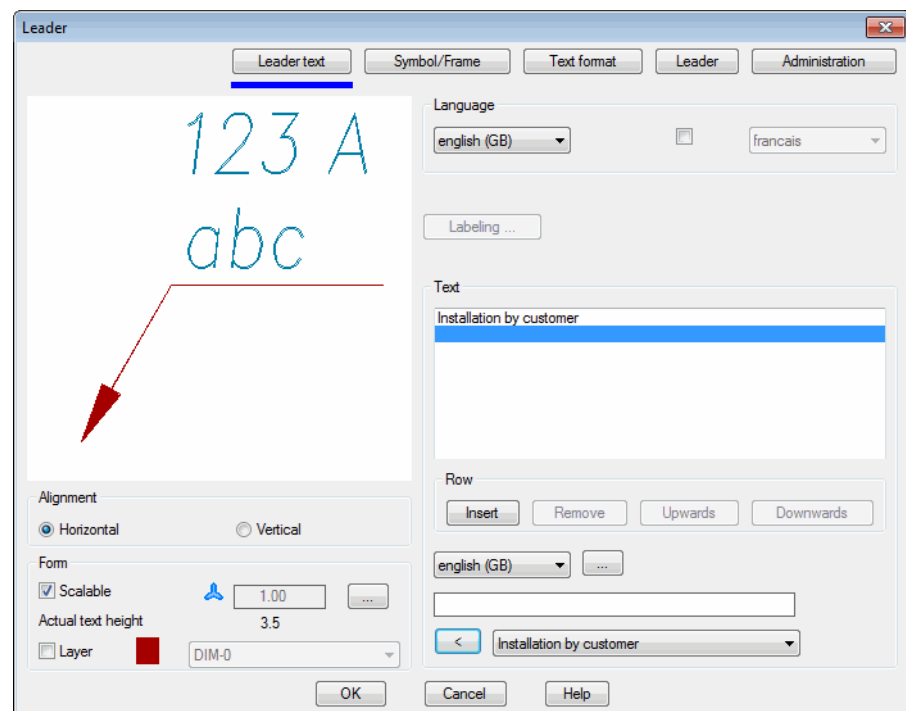
Use the mouse or enter coordinates to specify the starting point of the leader line.

Specify next point or [Undo/?]:

Use the mouse or enter coordinates to specify the next point of the leader line (kink point).

*You can repeat the last input request with the option Back. Enter **ENTER** to write a text on the leader. A dialog box opens for this.*

Dialog box Leader



The dialog box contains the register buttons Leader text, Symbol/frame, Text format, Leader and manager. The text and symbol properties are object specific; the manager is the same for all objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.

Dialog box section Alignment

Horizontal

Aligns the leader text horizontally so that it is legible from the front.

Vertical

Aligns the leader text vertically so that it is legible from the right.

Dialog box section Display

Scalable

Switches the scaling feature of leaders on or off. You can change the current scale factor in the input field.

[...]

Opens the Dialog box Label scales, where you can assign scales to the leader. You will find further information in the Chapter *Label Scale* on page 120.



Leaders with several scales can be aligned differently in different scales.

Actual text height

Specifies the actual text height. This is calculated from the current text height and the scale factor.

Layer

Activates the layer replacement. If you activate the tick box, you can change the layer set as standard.

Register button Leader text

Dialog box section Language

First language

Defines the first language of the leader text.

Second language

Defines the second language of the leader text. The tick box must be activated to display the second language.

Labeling

Opens the Dialog box Label, where you can adapt the label setting of automatically labeled objects. You will find further information in the Chapter *Label* on page 117.

Dialog box section Text

Preview for first language

Displays the leader text of the first language. Here you can mark texts for further editing.

Dialog box section Row

Insert

Inserts a new text row above the marked one. Enter the text in the input line.

Remove

Removes the marked text line.

Upwards

Moves the marked text one row upwards.

Downwards

Moves the marked text one row downwards.

Language selection

Selects the language for the text to be entered.

[...]

Opens the Dialog box Designation, where you can conveniently manage multi-language texts. You will find further information in the Chapter *Designation* on page 122.

Text input

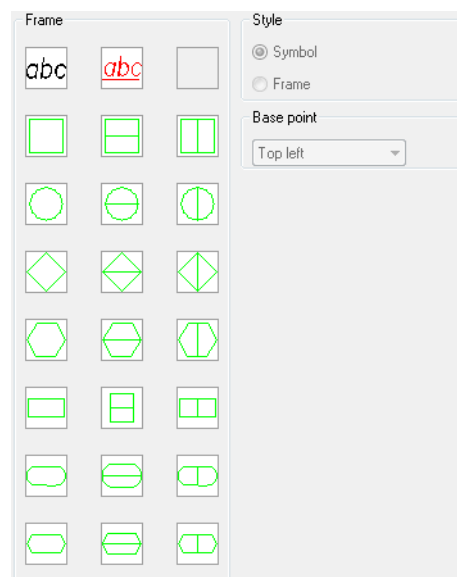
Defines the text in the chosen language. If you press Enter after the text input, the text is transferred into the preview box. The input line is immediately released for the entry of further texts.

[<] and pick list

The pick list displays multilingual standard texts from the text database `ath_user.rsx`. Select a text and click the button [<] to insert the text into the active row.

You will find further information about labeling objects in the Chapter *Multilingual standard texts* on page 60.

Register button Symbol/frame



Dialog box section Frame

Defines the selected frame as border for the leader text. The selected frame is displayed directly in the dynamic preview.

Dialog box section Style

Symbol

Defines the border of the leader text as a symbol. Symbols enclose either the first or first two constituents of the leader text.



With leaders with an arrow, texts are not displayed from the third text constituent.

Frame

Defines the border of the leader text as a frame. A frame always encloses the complete leader text. The Frame option is only available for single-line symbols.

Dialog box section Base point

Defines the base point of the leader. This function is not available for leaders with a leader line.

Register button Text format

Dialog box section Text format

For leader texts the settings of the current dimension style are used as standard. Here, you can overwrite single style settings.

Dimension style

Switches in replacement of the dimension style. If you activate this tick box, you can select a dimension style from the list.

Text style

Switches in replacement of the text style. If you activate this tick box, you can select a text style from the list.

Text height

Switches in replacement of the text height. When you activate this tick box, you can define a new text height in the input field.



The text height defined here is multiplied by the dimension scale factor.

Distance from base line

Switches in replacement of the distance between the text and the base line. When you activate this tick box, you can define a new distance in the input field.



The distance defined here is multiplied by the dimension scale factor.

Text color

Switches in replacement of the text color. When you activate this tick box, you can define a new text color by clicking the color button.

Infill color

Switches in replacement of the infill color of the leader text. When you activate this tick box, you can define a new infill color by clicking the color button.

Dialog box section Symbol

Text style

Switches on replacement of the symbol text style. If you activate this tick box, you can select a text style from the list.

Symbol color

Switches on replacement of the symbol. When you activate this tick box, you can define a new symbol color by clicking the color button.

Infill color

Switches on replacement of the infill color of the symbol text. When you activate this tick box, you can define a new infill color by clicking the color button.

For color changes the dialog box Color is started. You will find further information on this subject in the AutoCAD documentation.

Text alignment

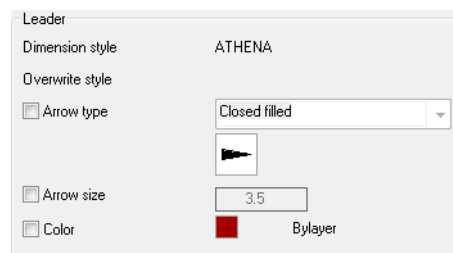
Changes the text alignment. You can align text left justified, centered and right justified.

The text alignment acts on texts in frames as well as on texts arranged below symbols. As standard, texts in symbols are aligned centered.

Placeholder

Defines a text as placeholder for the symbol width. With the placeholder you can influence the width of symbols. In this way it is possible to design symbols with different widths of text with uniform width.

Register button Leader



For leader lines the settings of the current dimension style are used as standard. Here, you can overwrite single style settings.

Dimension style

Displays the dimension style used for information.

Arrow type

Switches in replacement of the arrow tip. If you activate this tick box, you can select a new arrow type from the list.

Arrow size

Switches on replacement of the arrow size. When you activate this tick box, you can define an arrow size in the input field.

Color

Switches on replacement of the arrow color. When you activate this tick box, you can define a new arrow color by clicking the color button.

For color changes the dialog box Color is started. You will find further information on this subject in the AutoCAD documentation.

Notes

- As standard, the leader line is drawn on the current dimension layer. You will find information on layer assignment in the Chapter *Layer* on page 85.
- When stretching leaders with an arrow, the leader arrow always adapts to the new direction.
- You can edit leader texts with **Modify ATHENA** or by double clicking.
- You can adapt the text height to the current dimension figure height with **Update dimension**.

5.8 Parts labeling



Ribbon:	Tab ATHENA > Group Label > Parts labeling
Menu:	ATHENA > Dimension > Parts labeling
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_label

With this command you can automatically label all ATHENA objects (e.g. standard part, sheet metal section, insulation) as well as blocks and their attributes. You can also label objects or blocks, which are contained in blocks.

When you select an object which cannot be automatically labeled (e.g. line, circle), the command **Leader** is started and you can label the object manually.

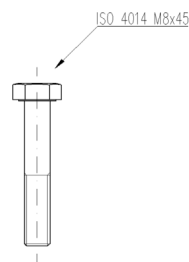


Fig. 5.4: Labeling a screw

Command prompt

Select part or [?]:

Select the object to be labeled. When you select an object contained in a block, the Dialog box **Parts labeling** is started. Here, you can choose whether the block or the selected object (e.g. ATHENA object or nested block) is to be labeled.

You call the help with the option **?**.

Specify starting point of the leader line or [?]:

Use the mouse or enter coordinates to specify the starting point of the leader line (arrow tip).

Specify next point or [Undo/?]:

Define the next point of the leader line (first kink point).

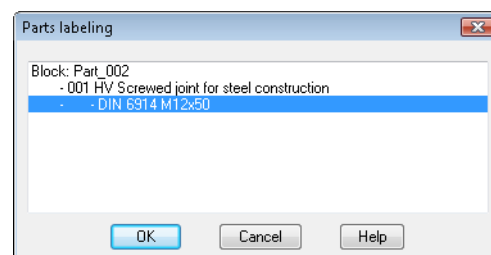
You can repeat the last input request with the option **Undo**.

Specify next point or [Undo/?]:

Define the next point of the leader line (next kink point).

This input request is repeated. Press the Enter key to terminate the command.

Dialog box Parts labeling



Shows the selected nested object in a tree structure. Here select which object you want to label. If you terminate the dialog box with OK, you are requested to specify the starting point of the leader line.

Notes

- You can edit leader texts with **Modify ATHENA** or by double clicking. When you select a parts label, ATHENA displays the texts linked to the labeled part in wavy brackets.
With these label texts you can only supplement whole lines. When you supplement texts, the reference to the labeled part is retained.
- You can set the part label for each part by clicking the button Label ... in the relevant dialog box. You will find information on this in the Chapter *Label* on page 117.
- The leader line is drawn on the current dimension layer. You will find information on layer assignment in the Chapter *Layer* on page 85.
- When stretching leaders with an arrow, the leader arrow always adapts to the new direction.
- You can adapt the text height to the current dimension figure height with **Update dimension**.
- You can change the labeling language by selecting another language in Dialog box ATHENA options. You will find further information in the Chapter *Options* on page 65.

5.9 Label tags



Ribbon:	Tab ATHENA > Group > Label
Menu:	ATHENA > Dimension > Tags labeling
Toolbar:	ATH Dimension
Command input:	ath_tagging_label

Labels objects, the tags of which have been assigned.

You will find information on tags in the Chapter *Assign tags* on page 801.



As standard, tags are labeled without leader arrow.

If you have not yet assigned any tags, an asterisk appears as the label text.
This is replaced by the tag once they are assigned.

Command prompt

Select object:

Select the part to be labeled.

Specify insertion point:

Specify the insertion point of the tag label text.

5.10 Coordinate label



Ribbon: Tab ATHENA > Group Label >
Menu: ATHENA > Dimension >
Toolbar: ATH Dimension
Command input: ath_dim_coord

Creates associative labels of coordinate points which are related to a defined base point (coordinate origin).

Command prompt

Select dimension system or [New] <New>:

Select the origin or a coordinate label of an existing system in order to supplement other coordinate labels. Choose the option New to create a new dimension system.

Enter coordinate origin:

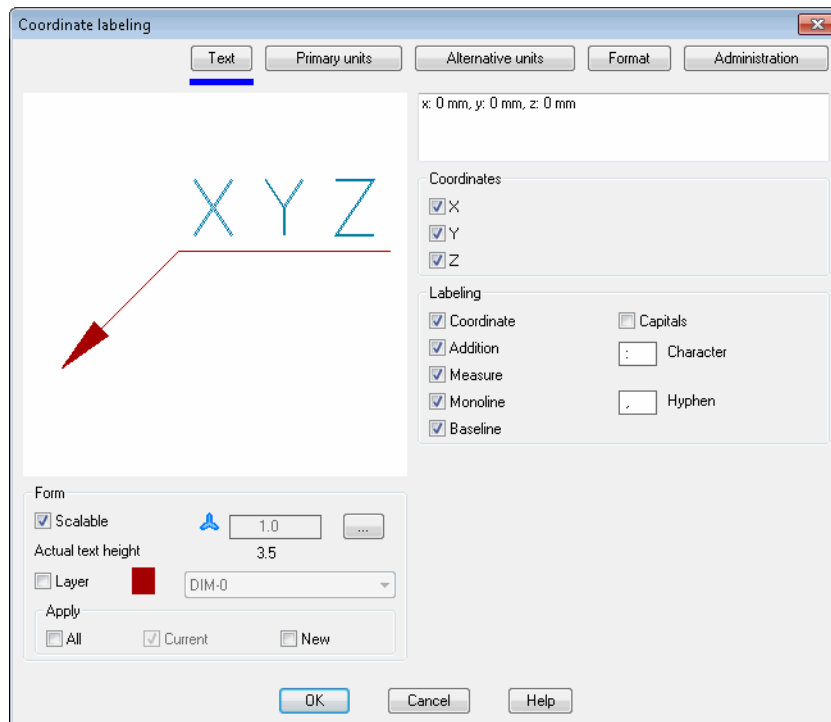
Specify the origin of the coordinate dimension system.

This input request only appears when you have called the option New beforehand. Once you have specified the origin, the Dialog box Coordinate label appears, where you can carry out further settings.

Dialog box Coordinate label

The dialog box appears when you have specified the coordinate origin or when you call the Settings option.

In the dialog box you will find on the left side the display section with the preview. On the right side you will see the operating section. You can activate the individual function areas with tab buttons.



Display section

The display section shows a schematic preview of the current label.

Below the preview you will find options for changing the display. You will find further information on this in the section *Display options for labels* on page 105.

Dialog box section Apply



The tick boxes in this region only affect the display settings (layer and scaling). All other settings (coordinates, units, etc.) have an effect on **all** coordinate labels of the dimension system.

All

Transfers the changes on editing a coordinate label to all objects of the dimension system.

Current

Is always switched on and changes only the selected coordinate label.

New

Changes both the current (Current tick box is always active) and the new coordinate labels of the dimension system.

Operating section

Register button Text

Dialog box section Coordinates

X

Specifies the value of the X coordinate in the coordinate label.

Y

Specifies the value of the X coordinate in the coordinate label.

Z

Specifies the value of the Z coordinate in the coordinate label.

Dialog box section Labeling

Coordinate

Switches the designation of the coordinates (X, Y, Z) on or off.

Capitals

Displays the coordinates as capital letters.

Addition

Switches the additional symbol () on or off.

Character

Defines the delimiting character between coordinates and coordinate value.

Measure

Turns unit of the coordinates on or off.

Monoline

Labels the coordinates in one line.

Separator

Defines the delimiting character between the coordinates.

Baseline

Turns the leader baseline on or off.

Register button Primary units

Dialog box section Primary units

Displays the units set in the drawing.



You can change the units of the drawing with the AutoCAD command Units (_units).

Precision

Defines the accuracy (number of decimal places) for primary units.

Rounding

Determines the rounding rules for the primary units.

If you enter the value 0.25 here, all distances are rounded to the nearest quarter level. If you enter the value 1.0, all distances are rounded to the nearest integer number. The number of figures after the decimal separator depends on the Precision setting.

Decimal separator

Defines the decimal separator.

Dialog box section Suppress zero

Pre-decimal point

Suppresses leading zeros in decimal dimensions. For example, 0.500 becomes .500.

Post-decimal point

Suppresses following zeros in decimal dimensions. For example, 0.500 becomes 0.5.

0 foot

Suppresses stating feet for dimensions in feet and inches when the distance is shorter than one foot. For example, 0'-6 1/2" becomes 6 1/2".

0 inch

Suppresses stating inches for dimensions in feet and inches when the distance is an integer in feet. For example, 1'-0" becomes 1'.

Register button Alternative units

Dialog box section Alternative units

Switches on alternative units.



If you have set decimal units, units with inches are used as alternative units. If you have set units with inches, decimal alternative units are used.

Precision

Defines the accuracy (number of decimal places) for alternative units.

Rounding

Determines the rounding rules for the alternative units.

If you enter the value 0.25 here, all distances are rounded to the nearest quarter level. If you enter the value 1.0, all distances are rounded to the nearest integer number. The number of figures after the decimal separator depends on the Precision setting.

Decimal separator

Defines the decimal separator.

Dialog box section Suppress zero

Pre-decimal point

Suppresses leading zeros in decimal dimensions. For example, 0.500 becomes .500.

Post-decimal point

Suppresses following zeros in decimal dimensions. For example, 0.500 becomes 0.5.

0 foot

Suppresses stating feet for dimensions in feet and inches when the distance is shorter than one foot. For example, 0'-6 1/2" becomes 6 1/2".

0 inch

Suppresses stating inches for dimensions in feet and inches when the distance is an integer in feet. For example, 1'-0" becomes 1'.

Register button Format

Dialog box section Text format

For leader texts the settings of the current dimension style are used as standard. Here, you can overwrite single style settings.

Dimension style

Switches in replacement of the dimension style. If you activate this tick box, you can select a dimension style from the list.

Text style

Switches in replacement of the text style. If you activate this tick box, you can select a text style from the list.

Text height

Switches in replacement of the text height. When you activate this tick box, you can define a new text height in the input field.



The text height defined here is multiplied by the dimension scale factor.

Distance from base line

Switches in replacement of the distance between the text and the base line. When you activate this tick box, you can define a new distance in the input field.



The distance defined here is defined by the dimension scale factor.

Text color

Switches in replacement of the text color. When you activate this tick box, you can define a new text color by clicking the color button.

Infill color

Switches in replacement of the infill color of the leader text. When you activate this tick box, you can define a new infill color by clicking the color button.

Dialog box section Leader lines

For leader lines the settings of the current dimension style are used as standard. Here, you can overwrite single style settings.

Dimension style

Displays the dimension style used for information.

Arrow type

Switches in replacement of the arrow tip. If you activate this tick box, you can select a new arrow type from the list.

Arrow size

Switches on replacement of the arrow size. When you activate this tick box, you can define an arrow size in the input field.

Color

Switches on replacement of the arrow color. When you activate this tick box, you can define a new arrow color by clicking the color button.



For color changes the dialog box Color is started. You will find further information on this subject in the AutoCAD documentation.

End of program

When you quit the dialog box with OK, the settings are saved for the duration of the drawing session. The positioning of the coordinate label occurs according to the same syntax as described in the command **Leader**.

Command prompt

Specify start point or [Settings]:

Specify the point of the coordinate which you want to label. This point corresponds to the arrow tip if you are labeling with leader lines.

Specify the next point of the leader:

Specify the next point (first kink point of the leader line) of the label. Press the Enter key if you only want to produce a text as coordinate label.

Specify the next point of the leader:

Specify the next point (next kink point of the leader line). This input request is repeated until you press the Enter key to terminate the command.

5.11 Position symbols



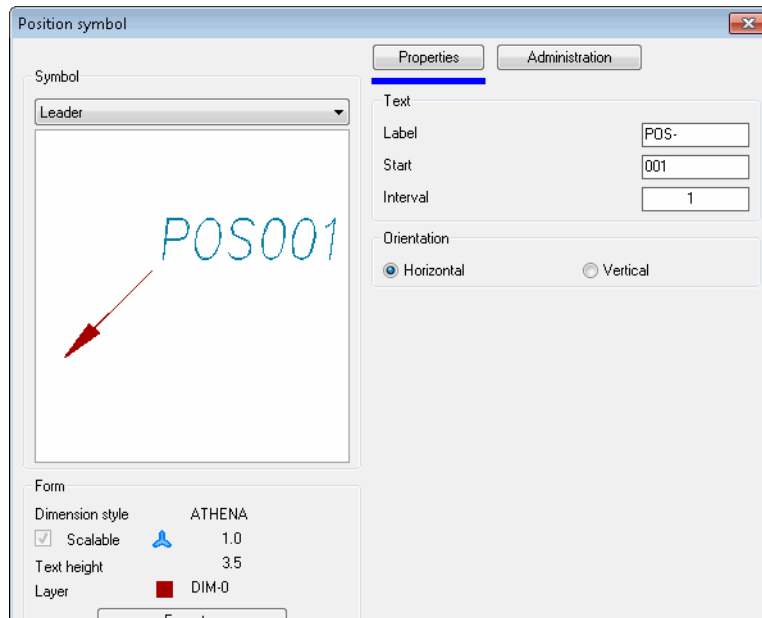
Ribbon:	Tab ATHENA > Group Label > Position symbols
Menu:	ATHENA > Dimension > Position symbols
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_pos

With the command Position symbols you can insert symbols into the drawing for numbering and positioning components. Incrementing of position numbers occurs automatically, whereby you define the counting intervals.

Dialog box Position symbols

In the dialog box you will find on the left side the display section with the preview. On the right side you will see the operating section. You can activate the individual function areas with tab buttons.

You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



Display section

Dialog box section Symbol

Above the preview there is a selection menu where you can choose between various position symbols. You can either select a symbol directly from the list or open the Dialog box Symbol selection by clicking the symbol preview and selecting a symbol there.

Dialog box section Display

Dimension style

Displays the current dimension style for information.

Text height

Displays the current text height for information.

Layer

Displays the current layer for information.

Format

Opens the Dialog box Leader, where you can adapt the symbol settings. You will find further information in the Chapter *Leader* on page 419.

Operating section**Register button Properties**

Dialog box section Text

Label

Defines the fixed part of the position number.

Start

Defines the variable part of the position number. You can enter figures and also letters. Special characters are not permissible.

The variable part of the position designation is incremented with the defined intervals.

Interval

Defines the intervals between the position numbers. If, for example, you enter "10" here, the numbering occurs with the start "001" as follows: 001, 011, 021, ...

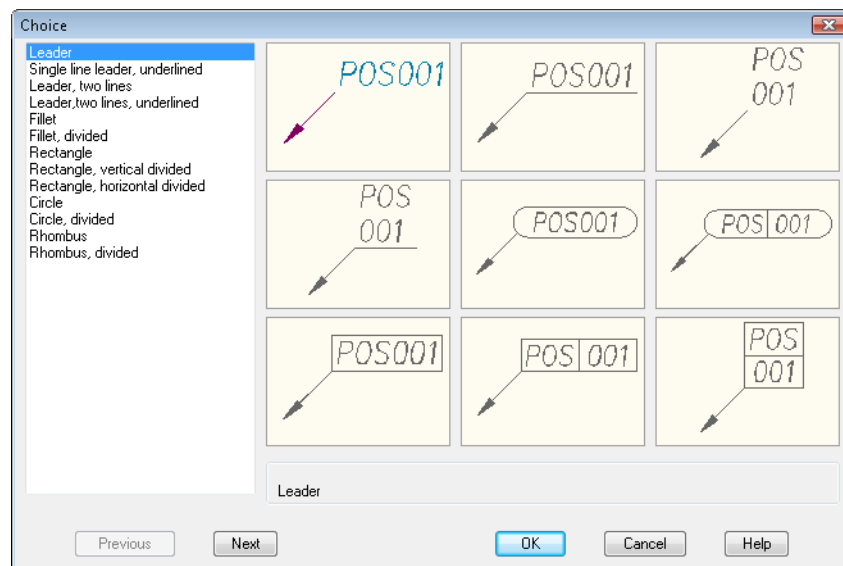
Dialog box section Alignment

Horizontal

Aligns the position symbol horizontally. Texts are legible from the front.

Vertical

Aligns the position symbols vertically. Texts are legible from the right.

Dialog box Symbol selection**End of program**

Once you have carried out the desired settings, you can quit the dialog box by clicking on OK and position the symbols in the drawing.

Command prompt

Specify starting point of the leader line or [?]:

Use the mouse or enter coordinates to specify the starting point of the leader line.

Specify next point or [Undo/?]:

Use the mouse or enter co-ordinates to specify the second point of the leader line (kink point). A symbol without leader line is produced with a right click or by entering ENTER.

These queries are repeated until the command is terminated by entering ENTER or by a RIGHT CLICK.

5.12 Interruption



Ribbon: Tab ATHENA > Group Label > Interruption
Menu: ATHENA > Dimension > Interruption
Toolbar: ATH Dimension and ATHENA Dimension
Command input: ath_abri

With this function you convert with just a few mouse clicks a detail, scaled and dimensioned, into an interrupted detail with interrupted dimension.

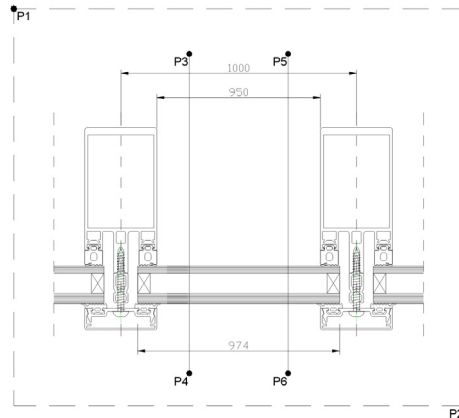


Fig. 5.5: Interruption

Command prompt

Current gap width = 5

Show base dimension or [Gap/?]:

Select a dimension as the base dimension with the mouse (1000 in the illustration above). The base dimension is the definition dimension, forming the basis for calculating all further dimensions.

Select the option **Gap** to change the spacing between the subsequently produced interruption lines (default 5mm).

You call the help with the option ?.

Select objects:

Select all objects with the mouse which are a constituent part of the detail.

Specify starting point of first intersection line or [?]:

Enter the starting point of the first intersection line with the mouse (P3 in the above illustration). Both intersection lines are later pushed together down to the gap (in our case 5 mm).

Specify next point or [?]:

Enter the endpoint of the first intersection line (P4 in the above illustration) with the mouse.

Specify starting point of second intersection line or [?]:

Enter the starting point of the second intersection line (P5 in the above illustration) with the mouse.

Specify next point or [?]:

Enter the end point of the second intersection line (P6 in the above illustration) with the mouse.

Notes

- Changing interrupted dimensions:
You can change dimension texts of interrupted dimensions with the command **Modify ATHENA** or by a double click. To do this the Dialog box Dimension text is started. You will find the functional description in the chapter *Change dimension text* on page 126.
- You can extend the dimensioning retrospectively with the command **Set interrupted dimension**.
- You can define the layer for the interrupted dimensioning in Dialog box System layer.
- When you stretch, move, copy or mirror interrupted dimensions, the dimensions are automatically updated (recalculated).

5.13 Defining interrupted dimensions



Ribbon:	Tab ATHENA > Group Label > Define interrupted dimension
Menu:	ATHENA > Dimension > Define interrupted dimension
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_abdf

With this function you define interrupted dimensions in interrupted drawing details.

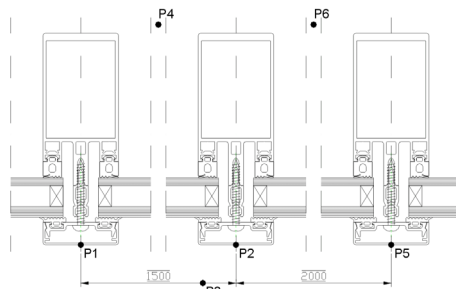


Fig. 5.6: Interrupted definition dimensions

Command prompt

Select level or dimension of definition or [New/?] <New>:

Select an existing level or dimension with the mouse to supplement an existing definition.

Use the option **New** to create a new definition.

You call the help with the option **?**.

Current setting: Decimal places: 1, Scale: 1

Definition [Linear/Aligned/Continue/Scale/?]:

Use the option **Linear** to create a horizontally or vertically aligned dimension. Further input requests follow.

Use the option **Aligned** to create an aligned dimension. Further input requests follow.

Use the option **Option Continue** to continue an existing dimension. Further input requests follow.

Use the **Option Scale** to change the dimension scale. The scale is saved for the definition and is the default for the command **Set interrupted dimension**.

Specify starting point of first auxiliary line or [Options/?] <Options>:

Point to first point on the dimension auxiliary line with the mouse, e.g. P1.

If you select **Options**, the previous input request is repeated.

Specify starting point of second auxiliary line or [?]:

Point to second point on the dimension auxiliary line with the mouse, e.g. P2. As support a rubber band is pulled to the first point.

Specify location of the dimension line or [?]:

Point to the location of the dimension line with the mouse, e.g. P3.

Specify interruption or [?]:

Point to the interruption on the horizontal plane with the mouse; e.g. P4.

Specify length or [?]:

Enter the distance between the dimensioned points; e.g. 1500.

Specify starting point of first auxiliary line or [Options/?] <Options>:

With the mouse specify the first point of the dimension auxiliary line for another horizontal definition dimension.

*Select **Options** to activate the input request with further options.*

Option Continue

Specify starting point of second auxiliary line or [?]:

Point to second point on the dimension auxiliary line with the mouse, e.g. P5. A rubber band is drawn to P2 for support.

Specify interruption or [?]:

Point to the interruption on the horizontal plane with the mouse; e.g. P6.

Enter length or [?]:

Enter the distance between the dimensioned points, e.g. 2000. The last input requests are repeated.

Option Scale

Opens the Dialog box Select scale, where you can change the scale for the interruption definition. You will find further information in the Chapter *Scale* on page 121.

Notes

- Changing interrupted dimensions:
You can change dimension texts of interrupted dimensions with the command **Modify ATHENA** or by a double click. To do this the Dialog box Dimension text is started. You will find the functional description in the chapter *Change dimension text* on page 126.
- You can extend the dimensioning retrospectively with the command **Set interrupted dimension**.
- For interrupted dimensions ATHENA uses its own dimension layer which you can set up in the Dialog box System layer .
- Defined interrupted dimensions are identified with a line below and above the dimension figure.
- You can create any number of interrupted definitions.
- You can copy or mirror interrupted definition dimensions. The previous interrupted definition is extended by the new definition dimension.
- When you stretch, move, copy or mirror interrupted dimensions, the dimensions are automatically updated (recalculated).

5.14 Set interrupted dimension



Ribbon: Tab ATHENA > Group Label > Set interrupted dimension
Menu: ATHENA > Dimension > Set interrupted dimension
Toolbar: ATH Dimension and ATHENA Dimension
Command input: ath_abse

With this function you create interrupted dimensions in interrupted drawing details. You must have previously created at least one interrupted definition dimension to serve as a basis for the calculation of further interrupted dimensioning.

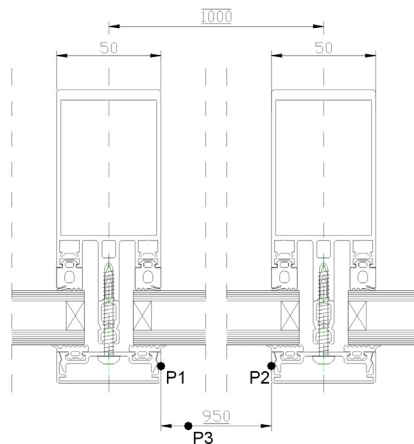


Fig. 5.7: Interrupted dimension

Command prompt

Select level or dimension of definition or [New/?]:

Select the level or dimension of an existing interruption definition (1000 on the illustration). Temporary red lines are now drawn through the interruptions to indicate the interruption.

You call the help with the option ?.

Current setting: Decimal places: 1, Scale: 1

Dimension [Linear/Aligned/Rotated/Decimal places/?]:

Use the **Option Linear** to create a horizontally or vertically aligned dimension. Further input requests follow.

Use the option **Aligned** to create an aligned dimension. Further input requests follow.

Use the **Option Rotated** to create a rotated dimension. Further input requests follow.

Use the **Option Decimal places** to change the number of decimal places.

Option Linear

Specify starting point of first auxiliary line or [Options/?] <Options>:

Point to first point on the dimension auxiliary line with the mouse, e.g. P1.

Select Options to repeat the previous input request and to change the dimension alignment.

Specify starting point of second auxiliary line or [?]:

Point to second point on the dimension auxiliary line with the mouse, e.g. P2.

Specify location of the dimension line or [?]:

Point to the location of the dimension line with the mouse, e.g. P3. The last input requests are repeated.

Option Rotated

Specify starting point of first auxiliary line or [Options/?] <Options>:

Point to first point on the dimension auxiliary line with the mouse, e.g. P1.

Select Options to repeat the previous input request and to change the dimension alignment.

Specify starting point of second auxiliary line or [?]:

Point to second point on the dimension auxiliary line with the mouse, e.g. P2.

Rotation angle or [?]:

Determine the rotation angle of the dimension. Further input requests follow.

Option Decimal places

Specify decimal places or [?] <1>:

Specify the number of decimal places. Further input requests follow.

Notes

- Changing interrupted dimensions:
You can change dimension texts of interrupted dimensions with the command **Modify ATHENA** or by a double click. To do this the Dialog box Dimension text is started. You will find the functional description in the chapter *Change dimension text* on page 126.
- For interrupted dimensions ATHENA uses its own dimension layer which you can set up in the Dialog box System layer .
- Set interrupted dimensions are identified with a line below the dimension figure.
- When you stretch, move, copy or mirror interrupted dimensions, the dimensions are automatically updated (recalculated).

5.15 Horizontal levels



Ribbon: Tab ATHENA > Group Label > Horizontal levels
Menu: ATHENA > Dimension > Horizontal levels
Toolbar: ATH Dimension
Command input: ath_hoho

With this routine you create levels with automatic computation in the horizontal direction. The display of the levels with computation can be made to the left (default) or to the right.

You can also use vertical levels in conjunction with interrupted dimensions. The interruption definitions are taken fully into account.

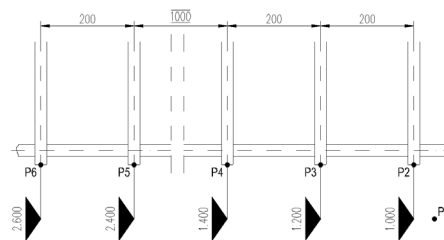


Fig. 5.8: Horizontal levels

Command prompt

Select level or dimension of definition or [New/?] <New>:

Click an existing level or dimension with the mouse to supplement an existing definition.

Use **Option New** to create a new definition.

You call the help with the option ?.

First level already defined.

Levels aligned [Yes/No/?] <No>:

Use the option **No** to place each level vertically as required.

Use the option **Yes** to define an alignment point for the levels.

Specify alignment point or [?]:

Specify an alignment point which determines the vertical position of the levels; see figure Horizontal levels P1.

Option New

Current setting: Decimal places: 0, Scale: 1

Specify first point or [Settings/?]

The first point means the definition level. Click a point, the height of which is known (see figure Horizontal levels P2).

Use the **Option Settings** to change the level settings in a dialog box.

Specify the vertical position of the level or [?]:

Specify a point by clicking with the mouse or entering the co-ordinates to define the vertical position of the levels; see figure Horizontal levels P1.

Enter the height of the first point (mm) or [?] <0>:

Enter the height for the selected point. This value forms the basis of computation for the other levels; see figure Horizontal levels 1000.

Current setting: Decimal places: 0, Scale: 1

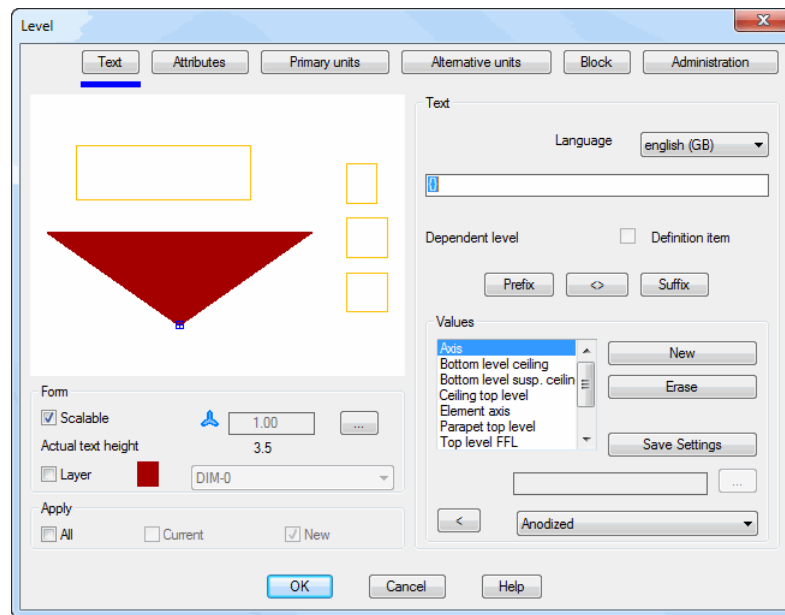
Specify next point or [Settings/?]:

*Specify the next point from which a level is to be created (e.g. P3) or select an option. The input request is repeated until you enter **ENTER** to terminate the command.*

Option Settings

Dialog box Level

In the dialog box you will find on the left side the display section with the preview. On the right side you will see the operating section. You can activate the individual function areas with tab buttons.



Display section

The display section shows a schematic preview of the current level.

Below the preview you will find options for changing the display. You will find further information on this in the section *Display options for labels* on page 105.

Dialog box section Apply



The tick boxes in this region only affect the display settings (layer and scaling). All other settings have an effect on **all** levels in the dimension system.

All

Transfers the changes on editing a level to all levels of the dimension system.

Current

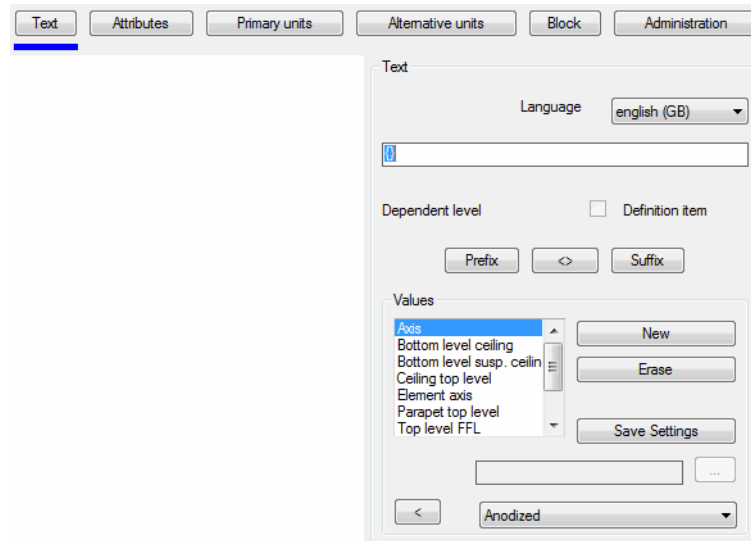
Is always switched on and changes only the selected level.

New

Changes both the current (Current tick box is always active) and the new levels of the dimension system.

Operating section

Register button Text



Dialog box section Text

Language

Defines the language for the optional additional texts (prefix/suffix).

Input line

Shows the dimension text of the current object. You can supplement dimension texts directly in the input line or use the methods described further below.

Below the input line the object type is displayed, e.g. level dependent (calculated level).

Definition dimension

Converts a dependent level into a definition level. The previous definition level is converted in this way into a dependent level.

Prefix

Positions the text selected in the value list before the current dimension text.

[<>]

Restores the original dimension text.

Suffix

Positions the text selected in the value list behind the current dimension text.

Dialog box section Values

Value list

Shows the texts available for the selected object. Here, select the text for further use.

If you select a text, it is displayed in the input line. Here, you can modify the text and adopt it by pressing the Enter key.

New

Enables the input line for the definition of a new text. You write a text in the input line and press the Enter key to insert the text into the list.

Delete

Removes the selected text from the list.

Save List

Saves the texts of the value list.

If you do not save the list, the texts are only available for the duration of the drawing session.

Input line

Defines new texts or displays the text selected in the value list.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.



If you edit texts in the Designation dialog box, you must then save them with the Change button.

[<] and pick list

The pick list displays multilingual standard texts from the text database ath_user.rsx. Select a text and click the button [<] to insert the text into the list. You will find further information about labeling objects in the Chapter *Multilingual standard texts* on page 60.

Register button Attributes

Dialog box section Attributes

Language

Defines the language for the optional additional texts for attributes.

In the three input lines you can directly define the additional texts for the relevant attribute field.

>

Transfers the text marked in the value list to the appropriate attribute line.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Dialog box section Values

Value list

Shows the dimension texts available for the selected object. Here, select the text for further use.

New

Supplements the text from the input line in the list.

Modify

Changes the selected text.

Delete

Removes the selected text from the list.

Save List

Saves the texts of the value list.

If you do not save the list, the texts are only available for the duration of the drawing session.

Input line

Defines new texts or displays the text selected in the value list.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.



If you edit texts in the Designation dialog box, you must then save them with the Change button.

Register button Primary units
Dialog box section Primary units

Displays the units set in the drawing.



You can change the units of the drawing with the AutoCAD command Units (_units).

Precision

Defines the accuracy (number of decimal places) for primary units.

Rounding

Determines the rounding rules for the primary units.

If you enter the value 0.25 here, all distances are rounded to the nearest quarter level. If you enter the value 1.0, all distances are rounded to the nearest integer number. The number of figures after the decimal separator depends on the Precision setting.

Decimal separator

Defines the decimal separator.

Dialog box section Suppress zero

Pre-decimal point

Suppresses leading zeros in decimal dimensions. For example, 0.500 becomes .500.

Post-decimal point

Suppresses following zeros in decimal dimensions. For example, 0.500 becomes 0.5.

0 foot

Suppresses stating feet for dimensions in feet and inches when the distance is shorter than one foot. For example, 0'-6 1/2" becomes 6 1/2".

0 inch

Suppresses stating inches for dimensions in feet and inches when the distance is an integer in feet. For example, 1'-0" becomes 1'.

Dialog box section Dimension scale

Scale factor

Defines the scale factor of the height dimension.



It is recommended that the default of 1.0 is not changed. For example, if you enter the value 2.0, the height dimensions of the dependent levels are doubled.

Register button Alternative units

Switches on alternative units.



If you have set decimal units, units with inches are used as alternative units. If you have set units with inches, decimal alternative units are used.

Precision

Defines the accuracy (number of decimal places) for alternative units.

Rounding

Determines the rounding rules for the alternative units.

If you enter the value 0.25 here, all distances are rounded to the nearest quarter level. If you enter the value 1.0, all distances are rounded to the nearest integer number. The number of figures after the decimal separator depends on the Precision setting.

Decimal separator

Defines the decimal separator.

Dialog box section Suppress zero

Pre-decimal point

Suppresses leading zeros in decimal dimensions. For example, 0.500 becomes .500.

Post-decimal point

Suppresses following zeros in decimal dimensions. For example, 0.500 becomes 0.5.

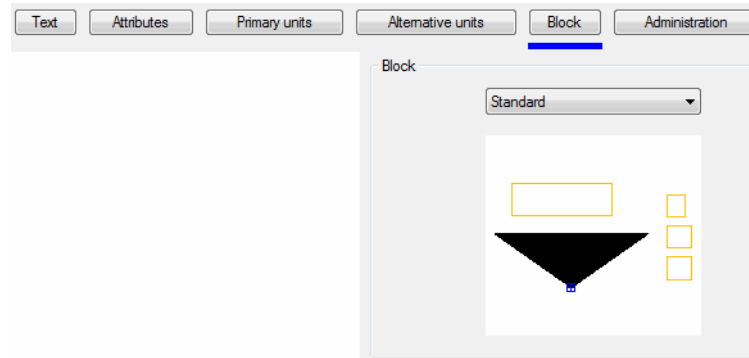
0 foot

Suppresses stating feet for dimensions in feet and inches when the distance is shorter than one foot. For example, 0'-6 1/2" becomes 6 1/2".

0 inch

Suppresses stating inches for dimensions in feet and inches when the distance is an integer in feet. For example, 1'-0" becomes 1'.

Register button Block



Dialog box section Block

Defines the block of level. If you select a block from the list, it is displayed as a preview.

Register button Manager

You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.



The display properties (e.g. the text height) depend on the scale and cannot be saved as default values.

Block assignments also cannot be saved as default. To change the standard level you have to adapt the file `ath_level.dwg`.

Integrating your own blocks for levels:

If you would like to integrate your own blocks, you must observe the following naming conventions:

A drawing is needed for each level.

Attributes for the height (attribute name = HKOTE) and three supplementary texts (attribute names = T1, T2 and T3) must be present.

The drawings must be saved in the folder `ATHENA\DATALOCAL` under the following names.

File name:

Display in the
menu:

ath_level_USER_LEVEL.dwg

USER_LEVEL

Notes about horizontal levels

- The values list is saved in the file ath_obj_prop.dex.
- Modifying levels:
You can change dimension texts of levels with the command **Modify ATHENA** or by a double click. To do this the Dialog box Dimension text is started. You will find the functional description in the chapter *Change dimension text* on page 126.
- When you stretch, move, copy or mirror levels, the levels are automatically updated (recalculated).
- If you mirror the definition levels such that the arrow tips point in the opposite direction, the computation direction of the levels is changed and the levels are automatically updated (recalculated).
- The definition level is underlined in order to identify it as such.
- If you delete the definition level, the next created level automatically becomes the new definition level.
- The display size of the level depends on the current size of the dimension figure.
- In the 3D section the levels are projected into the current UCS.
- The level auxiliary line (line from the level to the defined point) is suppressed when the distance between the defined point and the level insertion point is less than the value of the system variable *dimexo* (distance from the dimension auxiliary line to the edge of the solid). You can set this variable in the *.dim files. You will find further information in the Chapter *Dimension styles* on page 58.
- If you have many level definitions in the drawing, you can show the levels which belong to a group with the command Highlight associated dimension objects.

5.16 Vertical levels



Ribbon: Tab ATHENA > Group Label > Vertical levels
Menu: ATHENA > Dimension > Vertical levels
Toolbar: ATH Dimension and ATHENA Dimension
Command input: ath_hove

With this routine you create levels with automatic computation in the vertical direction.

You can also use vertical levels in conjunction with interrupted dimensions. The interruption definitions are taken fully into account.

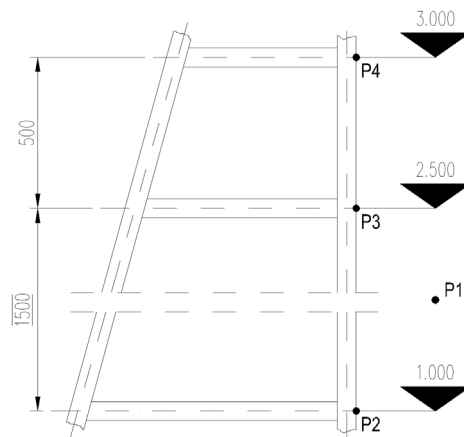


Fig. 5.9: Vertical levels

Command prompt

Select level or dimension of definition or [New/?] <New>:

Click an existing level or dimension with the mouse to supplement an existing definition.

Use **Option New** to create a new definition.

You call the help with the option ?.

First level already defined.

Levels aligned [Yes/No/?] <No>:

Use the option **No** to place each level vertically as required.

Use the option **Yes** to define an alignment point for the levels.

Specify alignment point or [?]:

Specify an alignment point which determines the vertical position of the levels; see figure Vertical levels P1.

Option New

Current setting: Decimal places: 0, Scale: 1

Specify first point or [Settings/?]

The first point means the definition level. Click a point, the height of which is known (see figure Vertical levels P2).

Use the **Option Settings** to change the level settings in a dialog box.

Specify the horizontal position of the level or [?]:

Specify a point by clicking with the mouse or entering the co-ordinates to define the vertical position of the levels; see figure Vertical levels P1.

Enter the height of the first point (mm) or [?] <0>:

Enter the height for the selected point. This value forms the basis of computation for the other levels; see figure Vertical levels 1000.

Current setting: Decimal places: 0, Scale: 1

Specify next point or [Settings/?]:

*Specify the next point from which a level is to be created (e.g. P3) or select an option. The input request is repeated until you enter **ENTER** to terminate the command.*

Notes about vertical levels

Siehe "Notes about horizontal levels" on page 450.

5.17 Highlight associated dimension objects



Ribbon:	Tab ATHENA > Group Label >
Menu:	ATHENA > Dimension >
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_dim_highlight

Highlights dimension objects which belong to the same dimension system as the selected dimension object.

In this way you can check which interrupted dimensions or levels belong to one another. This is in particular helpful if you have created many levels and interrupted dimensions in different dimension systems.

Command prompt

Select level or dimension of definition or [?]:

Select a level or an interrupted dimension.

You call the help with the option ?.

All dimension objects of the dimension system are now emphasized (highlighted). You can terminate the highlighting by regenerating the drawing (AutoCAD command `_regen`).

Notes

- You will find information on levels in the sections *Horizontal levels* on page 443 and *Vertical levels* on page 451.
- You will find information on interrupted dimensions in the sections *Interruption* on page 437, *Defining interrupted dimensions* on page 439 and *Set interrupted dimension* on page 441.

5.18 Manage dimension styles

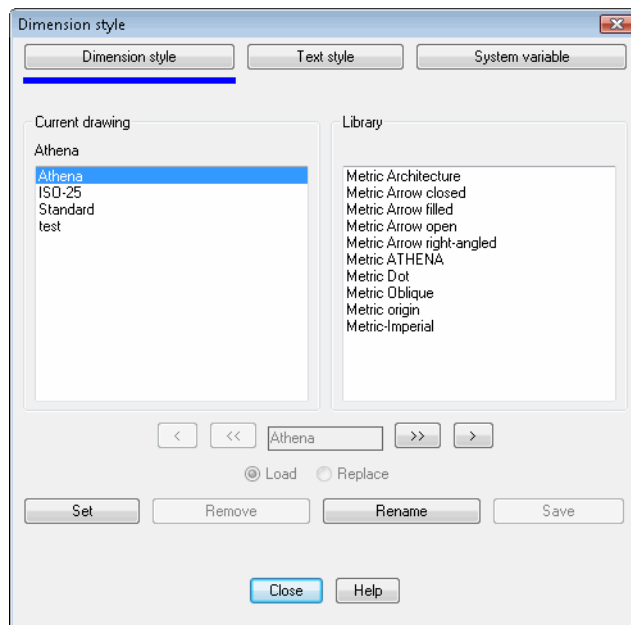


Ribbon:	Tab ATHENA > Group Label > Manage dimension styles
Menu:	ATHENA > Dimension > Manage dimension styles
Toolbar:	ATH Dimension
Command input:	ath_dimstyle

You can manage dimension styles with this command.

You can save dimension styles which are present in the current drawing in a library and load them into other drawings and also set them as current. Furthermore you can extend the library with dimension styles of the current drawing.

Dialog box Dimension style



Register button Dimension style

Activates the Dialog box Dimension style. You will find the functional description further below.

Register button Text style

Activates the Dialog box Text style.

You will find the functional description in the section *Manage text styles* on page 526.

Register button System variable

Activates the Dialog box System variables.

You will find the functional description in the section *Manage system variables* on page 609.

Dialog box section Current drawing

Lists the dimension styles of the current drawing. You can select dimension styles here for further use.

Dialog box section Library

Lists the dimension styles of the library. You can select dimension styles here for further use.



As standard, metric (mm) and British (inch) dimension styles are provided. Which is displayed depends on the current setting for the drawing units.

[<]

Copies the selected dimension style of the library into the current drawing. Depending on the selected option, the selected dimension style is loaded into the drawing or it replaces the current dimension style.

[<<]

Copies all dimension styles of the library into the current drawing.

Input field

Changes the name of the selected dimension style. This is only possible when you click the button Rename.

[>]

Copies the selected dimension style of the current drawing into the library.

[>>]

Copies all dimension styles of the current drawing into the library.

Load

Sets the dimension style copied into the drawing as the current dimension style.

Replace

Replaces the current dimension style in the drawing. The Dialog box Dimension style is then closed.

Set

Sets the selected dimension style of the current drawing as the current dimension style.

Remove

Deletes the selected dimension style from the list.

Rename

Releases the input field for changing the name of the selected dimension style.

Save

Saves the library.

Notes

- The dimension style library is saved in the file ATHENA\DATALOCAL\ath_dim.dex.
- Use the AutoCAD command ***_dimstyle*** to create new dimension styles or to change existing ones. You will find further information on this subject in the AutoCAD documentation.
- You will find further information in the Chapter *Dimension styles* on page 58.

5.19 Arrange dimension



Ribbon: Tab ATHENA > Group Label > Arrange dimensions
Menu: ATHENA > Dimension > Arrange dimensions
Toolbar: ATH Dimension and ATHENA Dimension
Command input: ath_bord

With this routine you can align dimensions by an alignment point that is to be entered.

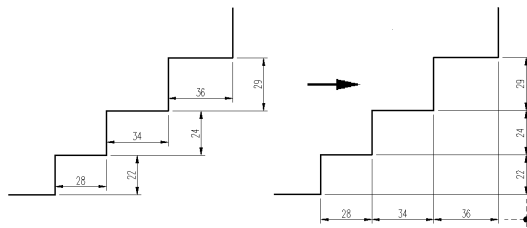


Fig. 5.10: Arrange dimension

Command prompt

Select dimensions:

Select objects:

Select with the mouse all dimensions which are to be arranged.

Specify alignment point or [?]:

Point to the alignment point with the mouse; see figure Arrange dimension.

You can change the positions of all dimensions with this routine. This is practicable, for example, with angular dimensions. With this program you can also dimension the opposing angle or another quadrant.

5.20 Arrange dimension texts



Ribbon:	Tab ATHENA > Group Label > Arrange dimension texts
Menu:	ATHENA > Dimension > Arrange dimension texts
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	<code>ath_dim_txtpos</code>

With this command you can align dimension texts in a row. This can be useful if you work with layouts and want to arrange the dimension texts in the visible section of a viewport.

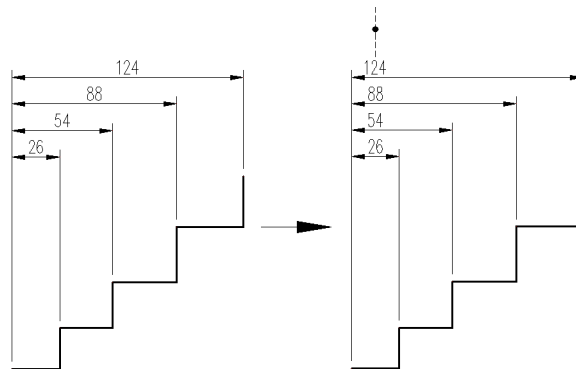


Fig. 5.11: Arrange dimension texts

Command prompt

Select objects:

Select the dimensions whose dimension texts you want to arrange.

This input request is repeated until you terminate the object selection with **ENTER** or a right click.

Specify alignment point or [?]:

Define the alignment point of the dimension texts.

You call the help with the option ?.

5.21 Arrange leaders



Ribbon:	Tab ATHENA > Group Label > Arrange leaders
Menu:	ATHENA > Dimension > Arrange leaders
Toolbar:	ATH Dimension
Command input:	ath_ldr_align

Arranges leaders and labels in a specified alignment.

Horizontal texts (readable from the front) are arranged together on a vertical line.
Vertical texts (readable from the right) are arranged together on a horizontal line.

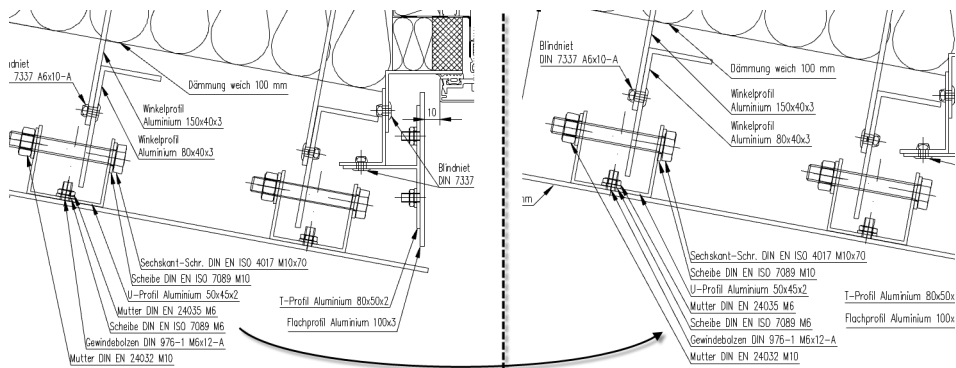


Fig. 5.12: Arrange labels

When the command is called, then follows:

Command prompt

Select objects:

Select the leaders and labels which you want to align.

This input request is repeated until you confirm the object selection by pressing the Enter key.

Specify alignment point or [?]:

Specify the alignment point to arrange the selected labels.

You call the help with the option ?.

x object(s) modified

Notes

You can arrange leaders which have been generated with the following commands:

- Leader
- Parts labeling
- Label tags
- Coordinate label
- Position symbols

5.22 Label background color ON



Ribbon:	Tab ATHENA > Group Label > Label background color ON
Menu:	ATHENA > Dimension > Label background color ON
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_Ldr_hide

Activates the **background** infill color for text for all labels in the current drawing.



Only labels are considered for which no infill color for texts has been turned on.
The command also changes labels in blocks.

5.23 Label text background color OFF



Ribbon:	Tab ATHENA > Group Label > Label background color OFF
Menu:	ATHENA > Dimension > Label background color USN
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_ldr_unhide

Deactivates the **background** infill color for text for all labels in the current drawing.



Only labels are considered for which the **background** infill color for texts has been turned on. Label texts to which an actual color has been assigned as the infill color are not considered.

The command also changes labels in blocks.

Label texts cannot cover any other objects (e.g. hatching), if the label background has been turned off.

5.24 Add leader line



Ribbon:	Tab ATHENA > Group Label > Add leader line
Menu:	ATHENA > Dimension > Add leader line
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_ldr_add

Adds another leader line to a leader. Leader lines can also be complemented for leader labels without a leader line.

When you execute the command, the following appears:

Command prompt

Select label or [?]:

Select the label which you want to complement with another leader line.

Specify next point or [Undo/?]:

Define the next point of the leader line.

Specify next point or [Undo/?]:

Specify a further point to generate a kink point or press the Enter key to terminate the command.

If you press the Enter key directly after the first point request, the arrow tip of the secondary leader line is generated at this point.

You can undo the last input with the option Undo. The input request is repeated.

You call the help with the option ?.



The secondary leader lines are not connected to the original primary leader, but linked such that scale changes, etc. are adapted. Consequently, when to edit a line you can double click the primary leader or a secondary leader line.

Position symbol:

Specify the new position of the label. The arrow tip of the leader line is placed at the previous base point of the label.

This request appears when you select a label without a leader line.

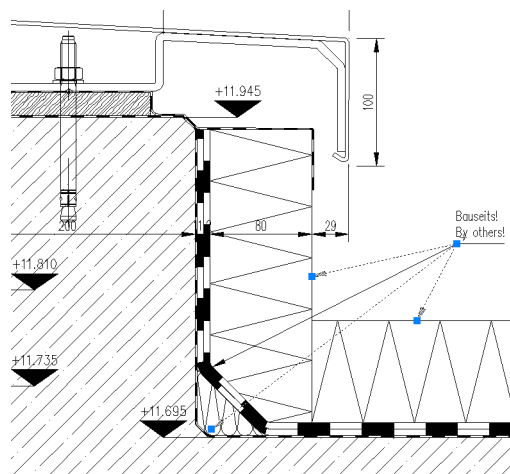


Fig. 5.13: Secondary leader lines



With Scalable leaders the leader lines are always added for all scales, but can be aligned differently retrospectively in various scales.

You will find information on scalability in the section *Scalability (label objects)* on page 105.

Associated commands:

- Leader
- Parts labeling
- Remove leader line

5.25 Remove leader line



Ribbon:	Tab ATHENA > Group Label > Remove leader line
Menu:	ATHENA > Dimension > Remove leader line
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_ldr_rem

With leader labels this removes leader lines.

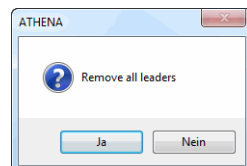
When you execute the command, the following appears:

Command prompt

Select label or [?]:

Select the leader line which you want to remove.

When you select a secondary leader line, it is immediately removed. When you select the primary leader, this input request appears:



Yes: All leader lines are deleted and the leader text is moved to the origin (arrow tip of the primary leader line).

No: Only the leader line is deleted. A secondary leader line becomes a primary one.

You call the help with the option ?.



With scalable leaders the leader lines are always removed for all scales.

You will find information on scalability in the section *Scalability (label objects)* on page 105.

Associated commands:

- Leader
- Parts labeling
- Add leader line

5.26 Add leader line segment



Ribbon:	Tab ATHENA > Group Label > Add leader line segment
Menu:	ATHENA > Dimension > Add leader line segment
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	<code>ath_ldr_edge_add</code>

Appends a leader line segment to a leader line. With leader labels without a leader line a new leader line is produced.

When you execute the command, the following appears:

Command prompt

Select label or [?]:

Select the label which you want to complement with a leader line segment. The selected segment of the label is directly divided and the label is activated such that the segment can be stretched with the grip.

Position symbol:

Specify the new position of the label. The arrow tip of the leader line is placed at the previous base point of the label.

This request appears when you select a label without a leader line.

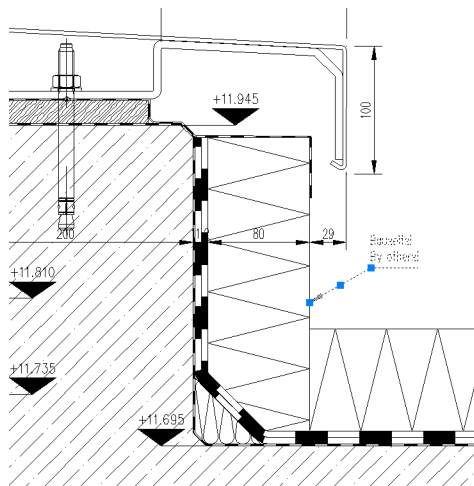


Fig. 5.14: Added (centrally divided) leader line segment



With Scalable leaders the leader line segments are always added for all scales, but can be aligned differently retrospectively in various scales.

You will find information on scalability in the section *Scalability (label objects)* on page 105.

Associated commands:

- Leader
- Parts labeling
- Remove leader line segment

5.27 Remove leader line segment



Ribbon:	Tab ATHENA > Group Label > Remove leader line segment
Menu:	ATHENA > Dimension > Remove leader line segment
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_ldr_edge_rem

Removes segments of leader lines

When you execute the command, the following appears:

Command prompt

Select label or [?]:

Select the label segment which you want to remove. The vertices of the adjacent leader lines are joined.

If only one segment is present, it is deleted.

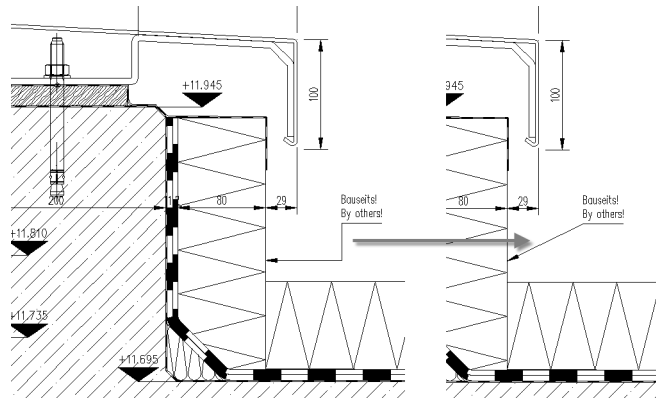


Fig. 5.15: Remove leader line segment



With scalable leaders the leader line segments are always removed for all scales.

You will find information on scalability in the section *Scalability (label objects)* on page 105.

Associated commands:

- Leader
- Parts labeling
- Add leader line segment

5.28 Change label scales



Ribbon:	Tab ATHENA > Group Modify > Change label scales
Menu:	ATHENA > Modify > Change label scales
Toolbar:	ATH Modify
Command input:	ath_ldr_scales

Changes the label scales of scalable objects. These can be:

- ATHENA objects
 - Leader
 - Coordinate label
 - Horizontal levels
 - Vertical levels
 - Welded seam symbol
 - Edge symbol
 - Surface symbol
- AutoCAD objects:
 - Dimensions
 - Text
 - Blocks
 - Hatching
 - Leader
 - Multiple leader line



Changing label scales is only possible for scalable objects or AutoCAD label objects. Non-scalable objects are ignored.

Command prompt

Select objects:

Select the leaders whose scales you wish to change. This input request is repeated until the object selection is concluded by pressing the Enter key.

Then the Dialog box Label scales is opened where you can change the scales of the selected objects. You will find further information on this in the Chapter *Label Scale* on page 120.

Notes

- In the Dialog box Label scales the currently set label scale is displayed (not the scales of the selected leaders). Scales not displayed (which, for example were formerly assigned) are removed!
- You will also find further information about the scalability of objects in the Chapter *Scalability (label objects)* on page 105.

5.29 Modify object label



Ribbon:	Tab ATHENA > Group Label > Modify object label
Menu:	ATHENA > Dimension > Modify object label
Toolbar:	ATH Dimension
Command input:	ath_obj_label

Changes the labeling texts of several objects of the same groups (e.g. label properties of all standard profiles, standard screws, membranes...).

Labels of several selected objects can be updated in this way.

Command prompt

Select object or [Settings/?]:

Select an object of a group, the labels of which you want to change.

You can adapt label settings with the Option Settings.

You call the help with the option ?.

Once you have selected an object, the Dialog box Label is displayed. Here you can adapt the label texts for the object group. You will find further information on this in the section *Label* on page 117.

Close the dialog box with OK once you have adapted the label texts. A further input request follows:

Select object:

Select the objects of which the label texts are to be updated.

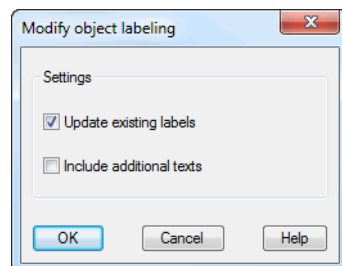


Objects which do not belong to the group (defined by the first object to be selected) are ignored.

Option Settings

Opens the Dialog box Modify object label.

Dialog box Modify object label



Update existing labels

Updates existing labels with the new label texts.

Include additional texts

Takes all additional texts into account when updating the labels

5.30 Align leaders



Ribbon:	Tab ATHENA > Group Label > Align leaders
Menu:	ATHENA > Dimension > Align leaders
Toolbar:	ATH Dimension
Command input:	ath_ldr_view

Aligns leaders in the current view or in the current coordinate system.

Command prompt

Align to [Ucs/View] <View>:

Press the Enter key if you want to align the leaders in the current view.

*With the option **Ucs** you can align the leaders in the current coordinate system.*

Select objects:

Select the leaders which you want to align in accordance with the preceding setting.

With this command you can align the following objects:

- Leader
- Parts labeling
- Coordinate label
- Position symbols

5.31 Change dimension settings

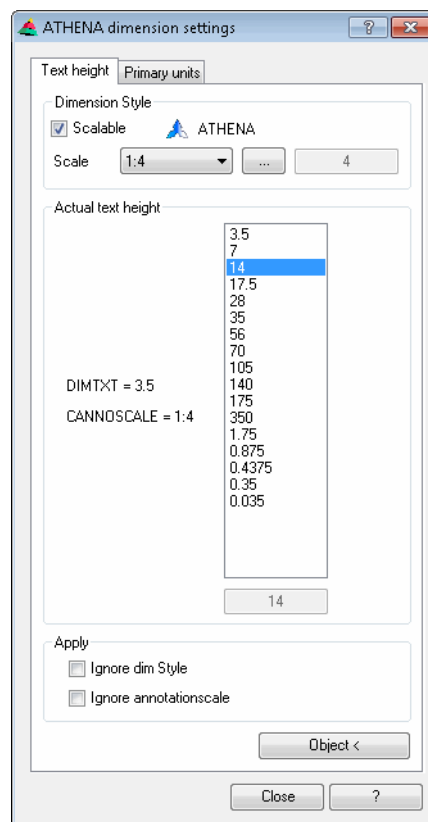


Ribbon:	Tab ATHENA > Group Label > Change dimension settings
Menu:	ATHENA > Dimension > Change dimension settings
Toolbar:	ATH Dimension
Command input:	ath_dim_quickset

Changes various dimension settings (e.g. scaling and accuracy) of dimensions. The desired settings are carried out in a dialog box.

Dialog box ATHENA dimension settings

Tab Text height



Dialog box section Dimension style

Scalable

Switches in dimension scalability. You will find further information in the section *Scalability (label objects)* on page 105.

The name of the current dimension style is displayed.

Scale

Sets the scale for dimension scaling.



If you are using scalable dimensions (AutoCAD labels), with this setting you change the label scale of the viewport (system variable *cannoscale*).

If you are using non-scalable dimensions, you change the scaling of the dimensions (system variable *dimscale*).

[...]

Opens the Dialog box Label scales, where you can add or remove label scales. You will find further information in the section *Label Scale* on page 120.



This button is only available for scalable dimensions.

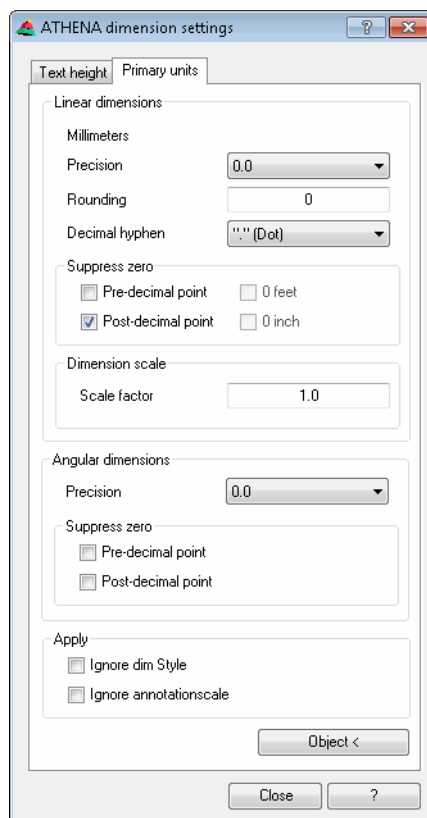
Dialog box section Actual text height

Shows the dimension text height defined in the dimension style (*dimtxt*) and the dimension scale (*cannoscale* or *dimscale*). Furthermore, a list with the actual heights of the scale texts is displayed. You can mark the actual text height in the list to assign it to other dimensions.



The actual text height is calculated from the dimension text height multiplied by the dimension scale.

Tab Primary units



Dialog box section Linear dimensions

Precision

Defines the accuracy of the linear dimensions. You can choose the required accuracy from the list.

Rounding

Defines the rounding rules for the dimensions.

Decimal separator

Determines the separator for decimal figures.

Dialog box section Suppress zero

Pre-decimal point

Suppresses leading zeros in decimal dimensions. For example, 0.5 becomes .5.

Post-decimal point

Suppresses following zeros in decimal dimensions. For example, 12.500 becomes 12.5.

0 foot

Suppresses stating feet for a dimension in feet and inches when the distance is shorter than one foot. For example, 0'-6 1/2" becomes 6 1/2".

0 inch

Suppresses stating inches for a dimension in feet and inches when the distance is an integer in feet. For example, 1'-0" becomes 1'.

Dialog box section Dimension scale

Scale factor

Determines the scale factor of the dimension.

Dialog box section Angular dimensions

Precision

Defines the accuracy of the angular dimensions. You can choose the required accuracy from the list.

Dialog box section Suppress zero

Pre-decimal point

Suppresses leading zeros in decimal dimensions. For example, 0.5 becomes .5.

Post-decimal point

Suppresses following zeros in decimal dimensions. For example, 12.500 becomes 12.5.

Apply

Ignore dimension style

Causes the set properties to be transferred to all selected dimensions irrespective of the dimension style.

Ignore scalability

Causes the set properties to be transferred to all selected dimensions irrespective of the scalability setting.

Object <

Transfers the properties of an existing dimension to the dialog box. In this way you have the possibility of transferring properties from existing dimensions. When you click the button, the dialog box is temporarily closed and the following appears:

Command prompt

Select the dimension or [?]:

Select the dimension whose properties you wish to adopt.

Close

Terminates the dialog box and the following appears:

Command prompt

Select objects:

Select the dimensions, to which you want to assign the dimension settings.

This input request is repeated until you press the Enter key to terminate the object selection.

5.32 Update dimension



Ribbon:	Tab ATHENA > Group Label > Update dimension
Menu:	ATHENA > Dimension > Update dimension
Toolbar:	ATH Dimension and ATHENA Dimension
Command input:	ath_upd

With this command you can update dimensions, interrupted dimensions, levels and leaders. This update command includes all the functions of the original AutoCAD Update command as well as other functions to update ATHENA dimension objects.

You can update the following objects:

- Leaders: The dimension figure height is updated.
- Dimensions: The dimension figure height is updated.
- Interrupted dimensions: The dimension figure height is updated and a new computation is carried out.
- Levels: The dimension figure height (size of the level) is updated and a new computation is carried out.

Command prompt

Select objects:

*Using the mouse select the objects which are to be updated. This query appears until the command is terminated by entering **ENTER**.*

5.33 Dihedral angle



Ribbon:	Tab ATHENA > Group Label > Dihedral angle
Menu:	ATHENA > Dimension > Dihedral angle
Toolbar:	ATH Dimension
Command input:	ath_planang

Using this command you can dimension the angle between two surfaces in space.

You can define the surfaces either by specifying three points or by selecting two objects.

Command prompt

Option Points

Specify the first surface

Specify first point or [Objects/?]:

Enter a point on the first surface.

*Use the **Option Objects** to specify the surface by object selection.*

You call the help with the option ?.

Specify the first surface

Specify the second point or [Undo/Objects/?]:

Enter the second point on the first surface.

Specify the first surface

Specify the third point or [Undo/Objects/?]:

Enter the third point on the first surface.

Specify the second surface

Specify the first point or [Undo/Objects/?]:

Enter a point on the second surface.

Specify the second surface

Specify the second point or [Undo/Objects/?]:

Enter the second point on the second surface.

Specify the second surface

Specify the third point or [Undo/Objects/?]:

Enter the third point on the second surface.

Dihedral angle X°

Specify position of the dimension line:

The dihedral angle is displayed. Specify the position of the dimension line.

Option Objects

Specify the first surface

Select object or [Points/?]:

Select an object to specify the first surface.

*Use the **Option Points** to define the surface by specifying points.*

Specify the first surface

Select second object or [Undo/Points/?]:

Select another object to specify the first surface.

Specify the second surface

Select object or [Undo/Points/?]:

Select an object to specify the second surface.

Specify the second surface

Select second object or [?]:

Select another object to specify the second surface.

Dihedral angle X°

Specify position of the dimension line:

The dihedral angle is displayed. Specify the position of the dimension line.

5.34 Length of an object



Ribbon: Tab ATHENA > Group Label > Length of an object
Menu: ATHENA > Dimension > Length of an object
Toolbar: ATH Dimension
Command input: ath_lobj

With this function you can show the length of a selected object in the AutoCAD text window.

Command prompt

Select objects or [?]:

Use the mouse to choose an object.

Output:

See Outputs.

Insert text [Yes/No/?] <No>:

*Use the **Option Yes** to insert the length as text in the current drawing.*

*Use the option **No** when you want to insert the length not as text.*

You call the help with the option ?.

Option Yes

Specify text height or [?] <XX>:

Specify the text height.

Press the Enter key to accept the default.

Specify insertion point or [?]:

Use the mouse or enter coordinates to specify the insertion point of the text.

Outputs

With object type Line:	<i>Length: ???</i>
With object type Circle or Arc:	<i>Radius: ??? Length: ??? (Circumference)</i>
With object type Polyline or Ellipse:	<i>Polyline (overall length: ???) Segment: Length: ???</i>
With object type Block (Block is not exploded):	<i>Included in block XXXXX (and one of the three above possibilities)</i>

Dimension
Length of an object

Command reference

6 Modify

Commands in this section:

- Modify ATHENA
- Copy/Rotate
- Stretch to dimension
- Copy detail
- Pedit connect
- Objects to polylines
- Optimize objects
- Connect
- Change polyline width
- Join
- Divide objects
- Slice objects
- Explode ATHENA
- Punch objects
- Convert ellipse
- Convert spline
- Hide objects
- Cancel object covering
- Cover mode
- Match ATHENA properties

6.1 Modify ATHENA



Ribbon:	Tab ATHENA > Group Modify > Modify ATHENA
Menu:	ATHENA > Modify > Modify ATHENA
Toolbar:	ATH Modify
Command input:	ath_edit

With this command you can edit ATHENA and AutoCAD objects. After calling the command, if you click an object, the dialog box is opened with which the selected object was created. The default settings of the selected object are displayed in the dialog box and you can change the required parameters. Parameters that cannot be changed are masked out in gray.

Alternatively to this command, you can double click an object without previously starting the command. With some objects however the double click function of Modify ATHENA is different, because AutoCAD dialog boxes are preferably used.

For the following ATHENA objects a dialog box exists which is available to modify the respective object:

- Dimensioning
- Interrupted dimension
- Table

Dimensioning

When you select a dimension with Modify ATHENA or double click it, the Dialog box Dimension text is started and you can edit the dimension text.

You will find information in the Chapter *Change dimension text* on page 126.

The following points should be noted when you modify interrupted dimensions:

- If you delete the original dimension (<>), the dimension texts are always displayed underlined.
- The list of texts is saved in the file
ATHENA\DATALOCAL\ath_obj_prop.dex.

Interrupted dimension

When you select or double click an interrupted dimension with Modify ATHENA, the Dialog box Dimension text starts.

You will find information in the Chapter *Change dimension text* on page 126.

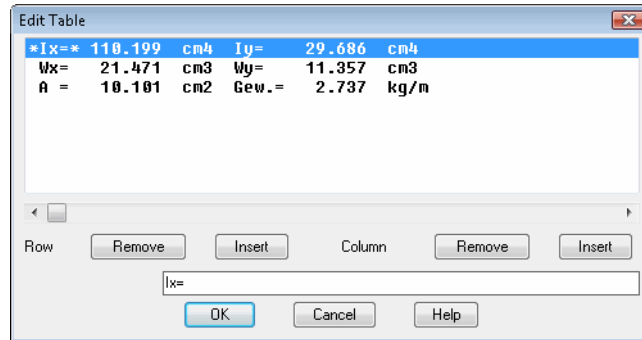
The following points should be noted when you modify interrupted dimensions:

- If you modify (overwrite) a defined dimension, all dependent interrupted dimensions and levels of the definition are recalculated.
- You cannot overwrite dimension figures of computed dimensions, but only supplement them.

Table

When you select an ATHENA table with Modify ATHENA, the dialog box Edit table opens. Editing tables with a double click is not possible.

Dialog box Edit table



Tables are used for the results tables of many calculation programs (e.g. Center of gravity and moments).

This type of table is shown in the dialog box as an example. This table contains five rows and three columns. The first column section in the first row is marked (IX=); this is shown below in the edit line where you can edit it. The marking is indicated by two asterisks (*) which enclose the relevant value. For modification select the required value in each case with the asterisks. Vertically (rows), this occurs by clicking with the mouse. Horizontally (columns), this occurs by moving the horizontal scroll bar or clicking the arrows.

Apart from changing the available values, you can also insert or delete rows or columns. When deleting, the row or column in which a value is marked with asterisks is deleted. You insert a row or column after the point at which a value is marked with an asterisk.

6.2 Copy/Rotate



Ribbon:	Not present
Menu:	ATHENA > Modify > Copy/Rotate
Toolbar:	ATH Modify
Command input:	ath_kodr

With this command you can copy and then rotate objects.

Command prompt

Select objects:

Select the objects which you wish to copy and rotate.

Specify base point for movement or [Multiple/?]:

Use the mouse or enter coordinates to specify the base point of the movement.

*Select the **Option Multiple** to make multiple copies of an object.*

You call the help with the option ?.

Specify second point:

Use the mouse or enter co-ordinates to specify the second point for the movement.

Select option [Reference/roTate/?] <roTate>:

*Select the **Option Rotate** to rotate the objects.*

*Select the **Option Reference** to determine the reference angle.*

Option Rotate

Specify rotation angle:

Use the mouse or enter an angle to specify the rotation angle.

Option Reference

Specify base angle or [?]:

Use the mouse or enter an angle to specify the base angle.

Specify new angle or [?]:

Use the mouse or enter an angle to specify the new rotation angle.

Option Multiple

Specify base point or [?]:

Use the mouse or enter co-ordinates to specify the base point of the movement.

Specify second point:

Use the mouse or enter co-ordinates to specify the second point for the movement.

Select option [Reference/roTaTe/Exit/?] <roTate>:

*Select the **Option Rotate** to rotate the objects.*

*Select the **Option Reference** to determine the reference angle.*

*Choose the option **Exit** to terminate the command.*

The last two input requests are repeated until the command is terminated.

6.3 Stretch to dimension



Ribbon: Tab ATHENA > Group Modify > Stretch to dimension
Menu: ATHENA > Modify > Stretch to dimension
Toolbar: ATH Modify and ATHENA Modify
Command input: ath_stretch_dim

With this command you can stretch or move dimensioned objects. You specify the movement by changing a dimension.

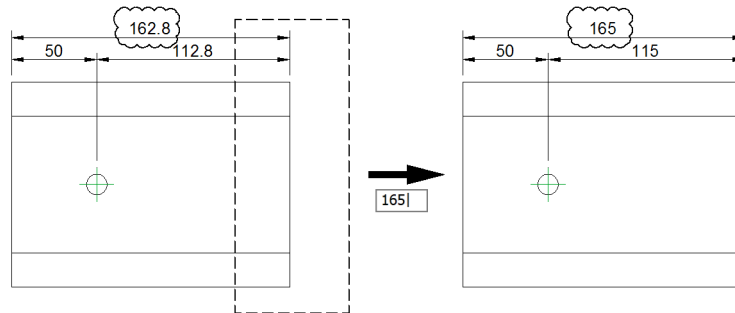


Fig. 6.1: Stretching to a dimension

Command prompt

Specify first corner or [?]:

Specify the first corner of the object selection.

You call the help with the option ?.

Select the opposite corner or [?]:

Specify the diagonally opposite corner of the object selection.

Select the dimension or [?]:

Select the dimension which is to define the movement distance.

Specify new dimension figure or [?] <X>:

Enter a new dimension figure to specify the movement.

6.4 Copy detail



Ribbon:	Tab ATHENA > Group Modify > Copy detail
Menu:	ATHENA > Modify > Copy detail
Toolbar:	ATH Modify
Command input:	ath_deta

With this function you can copy a drawing detail with specification of the scale just with a few mouse clicks. Overlapping objects - e.g. lines (also in blocks) are automatically trimmed.

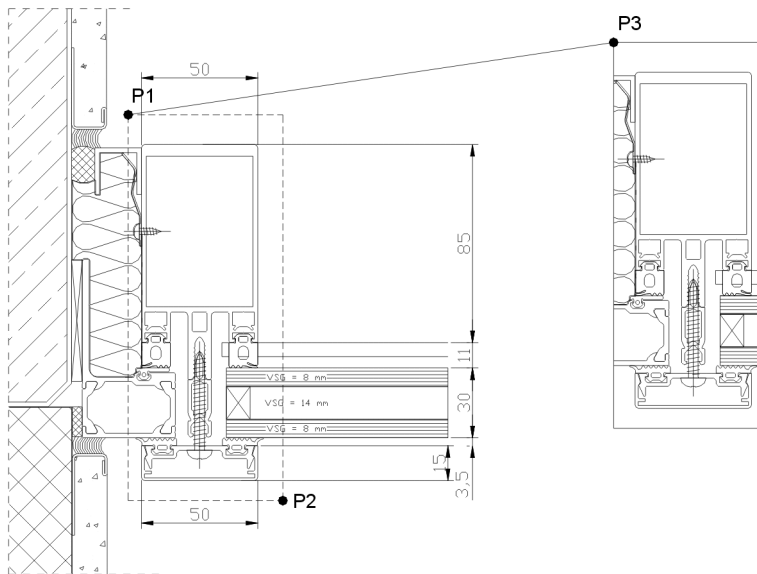


Fig. 6.2: Copy detail

Command prompt

Cross objects or [?]:

Select the objects which you wish to copy by specifying the first diagonal corner point with the mouse; see figure Copy detail P1.

You call the help with the option ?.

Specify second point or [?]:

Specify the second point diagonally opposite with the mouse; see figure Copy detail P2.

Position segment:

Place the copied detail in the drawing; see figure Copy detail P3.

Enter scale factor or [?] <1>:

Enter the scale for the detail.

The default scale (1:1) is accepted by inputting **ENTER** or a **RIGHT CLICK**.

Notes

- When trimming the overlapping elements, the following objects are not taken into account:
 - Dimensions
 - Text
 - Solid surfaces
 - 3-D surfaces

- Attribute definitions.
- Blocks are released before automatic trimming (Explode). Consequently, the layers of the block elements can change. They lose the layer *FromBlock* (on which they were inserted) and are then located on the relevant object layer.
- ATHENA objects (membrane, insulation, ...) are released. In this way these objects lose their smart features.
- Wide polylines which are not completely located in the defined window are released (Explode) and consequently lose their width.

6.5 Pedit connect



Ribbon:	Tab ATHENA > Group Modify > Pedit connect
Menu:	ATHENA > Modify > Pedit connect
Toolbar:	ATH Modify and ATHENA Modify
Command input:	ath_pver

With this function you join single lines or polylines to form an overall polyline.

Command prompt

Select objects:

Use the mouse to choose the objects you wish to join. Press the enter key to confirm the object selection and to connect the selected lines.

Notes

- If the lines have different properties (e.g. layers), the properties of the line first selected are accepted.
- Lines or polylines whose endpoints do not touch, cannot be joined together.

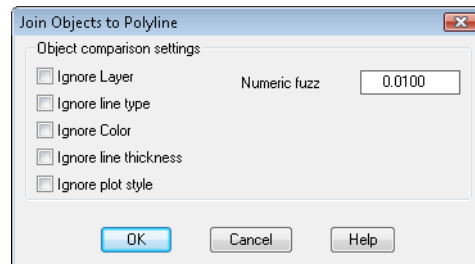
6.6 Objects to polylines



Ribbon:	Tab ATHENA > Group Modify > Objects to polylines
Menu:	ATHENA > Modify > Objects to polylines
Toolbar:	ATH Modify and ATHENA Modify
Command input:	ath_pljoin

With this function you can convert or join a number of objects (lines, polylines and arcs) to form polylines.

Dialog box Objects to polylines



In this dialog box you define under which conditions the objects are to be joined to form polylines.

In the input field Numeric fuzz enter the maximum separation of the line end points. Objects with end points located far apart are not joined.

With the tick boxes Ignore layer, Ignore line type, Ignore color, Ignore lineweight and Ignore plot style you can control when objects are joined and when not. If the corresponding tick box is active, the objects are joined. The properties of the object first selected are adopted.

With OK you terminate the dialog box and can select the objects to be joined. The settings are saved for the duration of the drawing session. With Cancel, the settings are discarded.

Command prompt

Select objects:

Use the mouse to choose the objects you wish to join. Press the Enter key to confirm the object selection.

7 object(s)

0 polyline(s) closed, 2 polyline(s) open

ATHENA indicates how many objects have been processed and joins the selected objects according to the settings. In addition, the open points, where present, are marked with X.

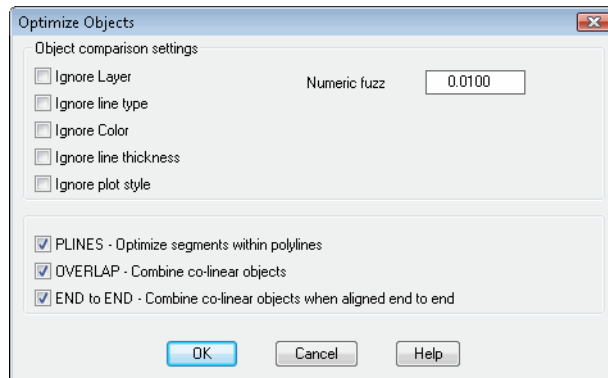
6.7 Optimize objects



Ribbon:	Tab ATHENA > Group Modify > Optimize objects
Menu:	ATHENA > Modify > Optimize objects
Toolbar:	ATH Modify and ATHENA Modify
Command input:	ath_ovk

With this function you can optimize and clean up objects (lines, polylines and arcs).

Dialog box Optimize objects



You define the conditions for the optimization in this dialog box.

Dialog box section Object comparison settings

In the input field Numeric fuzz enter the maximum separation of the line end points. Objects with end points located far apart are not optimized.

With the tick boxes Ignore layer, Ignore line type, Ignore color, Ignore lineweight and Ignore plot style you can control when objects are optimized and when not. If the corresponding tick box is active, the objects are optimized. The properties of the object first selected are adopted.

Dialog box section Optimization

With the tick box Polylines - optimize segments active, you delete segments with a length of 0 from polylines or combine polyline segments located in line.

With the tick box Overlap - Combine colinear objects active, you combine objects which are superimposed or whose parts overlap.

With the tick box End to end you combine objects which touch and are in alignment.

You can delete duplicates with all three options. When ATHENA finds duplicates, the query appears of whether you wish to delete or retain them.

With OK you terminate the dialog box and can select the objects to be optimized. The settings in the dialog box are saved for the duration of the drawing session. With Cancel, the settings are discarded.

Command prompt

Select objects:

Use the mouse to choose the objects you wish to optimize. Press the enter key to confirm the object selection and to terminate the command.

Notes

- When you delete duplicates (superimposed objects of the same length) with this function, the last object selected is always retained.
- If objects have different properties (e.g. layers) and you have ignored these properties, the properties of the last object selected are always accepted.

6.8 Connect



Ribbon:	Tab ATHENA > Group Modify > Connect
Menu:	ATHENA > Modify > Connect
Toolbar:	ATH Modify
Command input:	ath_verb

With this function you connect two lines to form one after checking for the same direction.

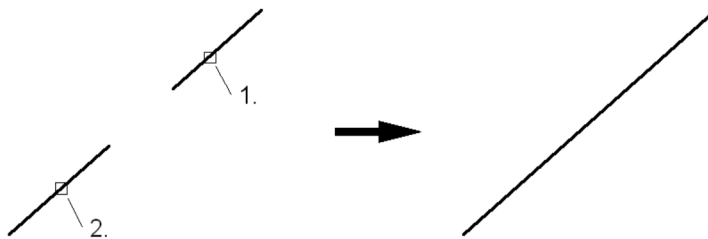


Fig. 6.3: Connecting lines

Command prompt

Select first line or [?]:

Use the mouse to choose the first line.

You call the help with the option ?.

Select second line or [?]:

Use the mouse to choose the second line.

Notes

- If the lines have different properties (e.g. layers), the properties of the line first selected are accepted.
- Polylines are always converted into lines.

6.9 Change polyline width



Ribbon:	Tab ATHENA > Group Modify > Change polyline width
Menu:	ATHENA > Modify > Change polyline width
Toolbar:	ATH Modify
Command input:	ath_pbre

You modify the width of polylines with this function.

Command prompt

Select objects:

Use the mouse to choose the polylines whose width you wish to change.

Specify new width or [?] <0>:

Define the new width by entry or pointing with the mouse.

Press the Enter key to accept the default.

You call the help with the option ?.

6.10 Join



Ribbon:	Tab ATHENA > Group Modify > Join
Menu:	ATHENA > Modify > Join
Toolbar:	ATH Modify and ATHENA Modify
Command input:	<code>ath_join</code>

With this function you join two linear ATHENA objects (sheet metal section, membrane or welded seam) to form one object in the extended intersection.

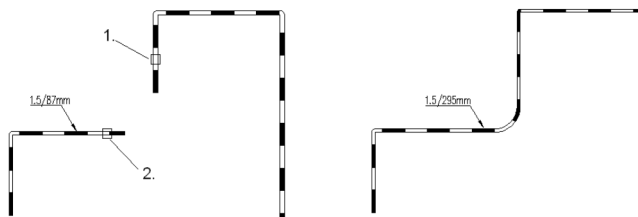


Fig. 6.4: Joining two membranes

Command prompt

Select the limb of the first object or [?]:

Select the limb of the first ATHENA object with the mouse.

You call the help with the option ?.

Select the limb of the second object or [?]:

Select the limb of the second ATHENA object with the mouse.

If the object properties match, the selected objects are immediately joined.

When the selected objects have different properties (e.g. various membrane thicknesses as in the above illustration), the following dialog box is opened with the corresponding notice. If you terminate the dialog box with Yes, the object properties of the object first selected are accepted. If you terminate the dialog box with No, the command terminates without joining the objects.

6.11 Divide objects



Ribbon:	Tab ATHENA > Group Modify > Divide objects
Menu:	ATHENA > Modify > Divide objects
Toolbar:	ATH Modify and ATHENA Modify
Command input:	ath_break

With this function you can break up various AutoCAD and ATHENA objects at any point without having to delete segments. You can break the following objects:

- AutoCAD objects:
 - Line, polyline
 - Circle, arc
- ATHENA objects
 - Sheet metal section, membrane, welded seam

Command prompt

Select objects or [?]:

Select an object to break.

You call the help with the option ?.

Specify break point or [?]:

Use the mouse or enter coordinates to specify the break point.)

Two points are required

This input request only appears when you wish to break up a circle. Circles must be broken at a minimum of two points.

Specify break point or [Undo/?]:

Specify another break point. This input request is repeated. Press the Enter key to terminate the command.

6.12 Slice objects



Ribbon:	Tab ATHENA > Group Modify > Slice objects
Menu:	ATHENA > Modify > Slice objects
Toolbar:	ATH Modify and ATHENA Modify
Command input:	ath_slice

With this function you can slice various AutoCAD and ATHENA objects. Objects to be sliced are:

- AutoCAD objects:
 - Line, polyline
 - Circle, arc
- ATHENA objects
 - Insulation, spacer, gasket
 - Sheet metal section, membrane, welded seam
 - Semi-finished product, standard profile

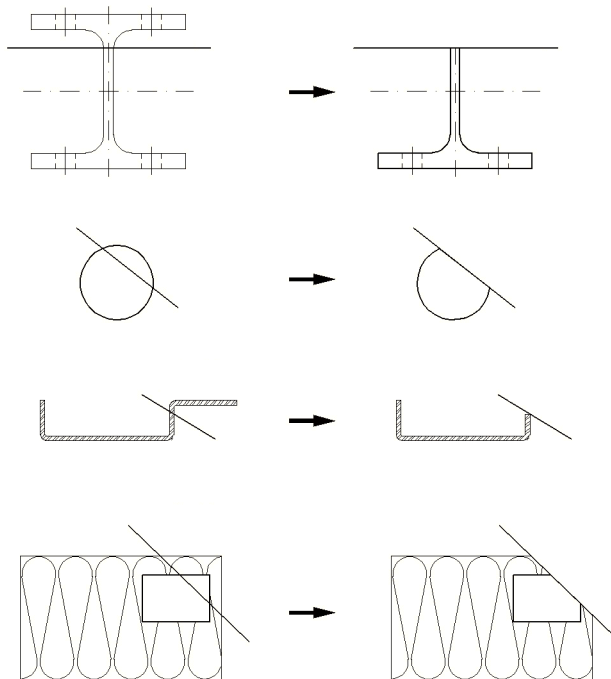


Fig. 6.5: Examples of slicing objects

Command prompt

Select objects:

Use the mouse to choose the objects you wish to slice.

Option Object

Select slice line or [Points/?] <Points>:

Select the line on which you wish to slice the objects.

*Use the **Option Points** to specify two points as the slicing line.*

You call the help with the option ?.

Specify side for slicing or [Undo/?]:

Define the side on which the objects are to be sliced.

*Use the option **Undo** to repeat the last step.*

*Option Points**Specify the slice line start point or [Object/?]:**Use the mouse or enter coordinates to specify the slice line start point.**Use the **Option Object** to select a line as the slice line.**Specify the slice line end point or [Object/Undo/?]:**Use the mouse or enter coordinates to specify the slice line end point.***Notes**

- Some ATHENA objects lose their functional features after slicing. For example, you cannot stretch semi-finished products after slicing.
- Circles are converted to closed polylines after slicing.
- Sheet metal sections, membranes and welded seams are not cut on a diagonal slice line, but instead always at right angles to the relevant limb.

6.13 Explode ATHENA



Ribbon:	Tab ATHENA > Group Modify > Explode ATHENA
Menu:	ATHENA > Modify > Explode ATHENA
Toolbar:	ATH Modify
Command input:	ath_bexp

With this command you release the combined objects into their constituent parts.

The command includes the following extensions:

- You can release ATHENA objects (e.g. standard parts).
- You can release blocks with attributes and simultaneously delete the attributes.

Command prompt

Select objects:

Select the objects to be released (ATHENA objects, blocks, groups or polylines) with the mouse.

6.14 Punch objects



Ribbon: Tab ATHENA > Group Modify > Punch outline
Menu: ATHENA > Modify > Punch outline
Toolbar: ATH Modify and ATHENA Modify
Command input: ath_punch

With this command you can clinch or punch any objects with an outline (stamp).

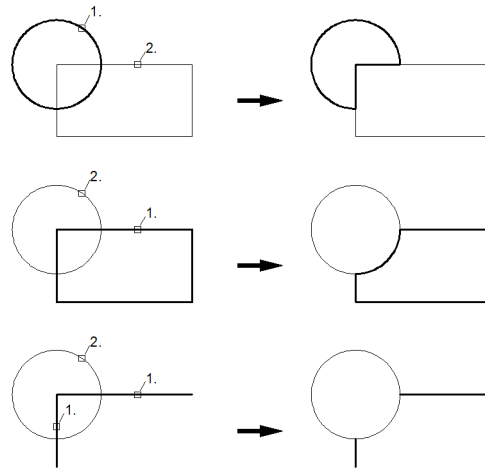


Fig. 6.6: Examples of punched (clinched) outlines

Command prompt

Select objects which are to be punched.

Select objects:

Select the objects which are to be processed (punched or clinched). Confirm the object selection by pressing the Enter key.

Mode: Clinch

Select outline or [Punch/?]:

Select the object which is to clinch the previously selected objects. Choose the option Punch to activate the punch mode.

Mode: Punch

Select outline or [Clinch/?]:

Select the object which is to punch the previously selected objects. Choose the option Clinch to activate the clinch mode.

During clinching the cut amount, which results from the object with the punch outline, is removed from the selected punched object. During punching, only the cut amount, which results from the object with the punch outline, is retained from the selected punched object.

Notes

- When you punch outlines consisting of closed objects (e.g. circles or polylines), closed outlines are again produced.
- When you punch outlines consisting of unclosed objects (e.g. lines or polylines), unclosed objects are produced.

6.15 Convert ellipse



Ribbon:	Tab ATHENA > Group Modify > Convert ellipse
Menu:	ATHENA > Modify > Convert ellipse
Toolbar:	ATH Modify
Command input:	ath_elps

With this command you can convert "genuine" ellipses or elliptical arcs to approximated polyline representations.

Command prompt

Break-up mode: Acad

Select ellipse or [3M/5M/?]:

The currently displayed break-up mode is displayed. Select the ellipse which is to be converted.

Use the options 3M or 5M to modify the break-up mode.

You call the help with the option ?.

The selected ellipse is immediately converted.

Break-up modes

ACAD:

ATHENA produces an approximated polyline representation corresponding to the AutoCAD setting when the system variable *pellipse* has the value 1.

3M:

Approximated polyline representation with three center points per ellipse side.

5M:

Approximated polyline representation with five center points per ellipse side.

Notes

Only "genuine" ellipses can be converted. These are produced when the AutoCAD system variable *pellipse* is set to the value 0. You will find further information about ellipses in the AutoCAD documentation.

6.16 Convert spline



Ribbon:	Tab ATHENA > Group Modify > Convert spline
Menu:	ATHENA > Modify > Convert spline
Toolbar:	ATH Modify
Command input:	ath_spline

With this command you can convert a spline. The spline is converted to an approximated polyline representation.

Command prompt

Select spline or [?]:

Select the spline which you want to convert.

You call the help with the option ?.

Notes

You will find further information about splines in the AutoCAD documentation.

6.17 Hide objects



Ribbon: Tab ATHENA > Group Modify > Hide objects
Menu: ATHENA > Modify > Hide objects
Toolbar: ATH Modify and ATHENA Modify
Command input: ath_hide

With this command you can cover ATHENA objects with other ATHENA objects. This enables you to, for example, show screwed joints better.

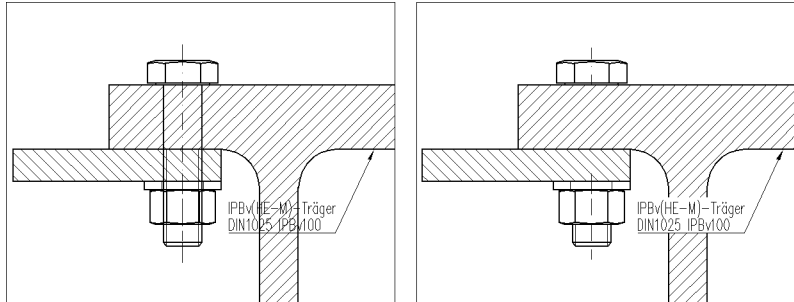


Fig. 6.7: Screwed joint visible and hidden.

Command prompt

Select objects which are to cover other objects.

Select objects:

Select the objects which are to cover other objects.

x object(s) found

The number of the selected objects is shown. This input request is repeated.

Press the Enter key to terminate the object selection.

Select objects which are to be covered.

Select objects:

Select the objects which are to be covered.

Notes

- All leaders produced with **Parts labeling**, automatically cover the labeled parts.
- You can undo the cover function with the command **Cancel object covering**.
- With the command **Cover mode** you can control the display of the surrounding edges.

6.18 Cancel object covering



Ribbon:	Tab ATHENA > Group Modify > Cancel object covering
Menu:	ATHENA > Modify > Cancel object covering
Toolbar:	ATH Modify
Command input:	ath_unhide

With this command you can remove covers of ATHENA objects which have been created with the command *Hide objects*.

Command prompt

Remove cover

Select objects:

Select the objects, the covers of which you want to remove.

6.19 Cover mode



Ribbon:	Tab ATHENA > Group Modify > Cover mode
Menu:	ATHENA > Modify > Cover mode
Toolbar:	ATH Modify
Command input:	ath_hide_mode

Controls the display of surrounding edges for covered objects.

When you cover areas of objects (e.g. the thread of a screw which protrudes into a steel plate), you can control whether the surrounding edges of the object are displayed as covered lines or not.

When you start the command, the following appears:

Command prompt

Display covered surrounding edges [Yes/No/?] <No>:

Choose the option Yes to activate the display of the surrounding edges.

Choose the option No to deactivate the display of the surrounding edges.

You call the help with the option ?.

Select objects:

Select the covered objects of which the cover mode is to be changed.



The cover mode is saved as default for further hidden objects.

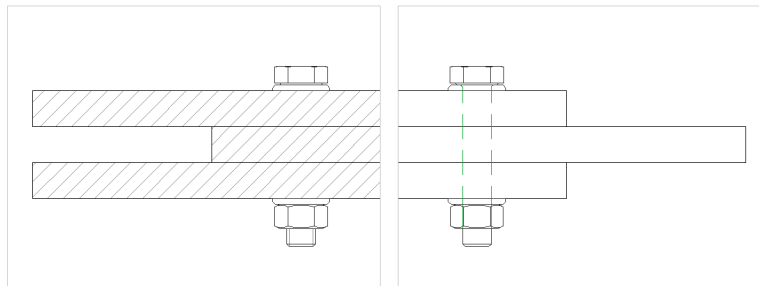


Fig. 6.8: Covered modes: left Off, right On

6.20 Match ATHENA properties



Ribbon:	Tab ATHENA > Group Modify > Match ATHENA properties
Menu:	ATHENA > Modify > Match ATHENA properties
Toolbar:	ATH Modify
Command input:	ath_matchprop

With this function you transfer the properties of an object to one or more other objects.

You can transfer both AutoCAD properties (e.g. layer) as well as ATHENA properties (e.g. screw length).

Command prompt

Select source object or [?]:

Choose a source object with the mouse.

You call the help with the option ?.

Options [Settings/Object selection/?] <Object selection>:

*You can change the properties to be transferred with the option **Settings**. To do this, the Dialog box Match properties is opened.*

*You can select objects which you want to change with the option **Object selection**.*

Target object(s)

Select objects:

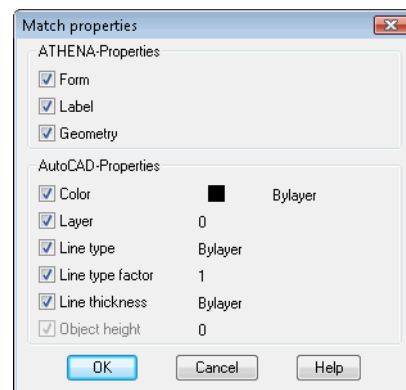
Use the mouse to choose the target objects.

x object(s) found - x object(s) modified

Select objects:

Use the mouse to choose the target objects. This input request is repeated. Press the Enter key to terminate the command.

Dialog box Match properties



In the section AutoCAD properties you select the AutoCAD-specific properties which you wish to transfer to other objects. These properties are valid both for AutoCAD and for ATHENA objects.

In the section ATHENA properties you define by tick boxes which ATHENA-specific properties you wish to transfer to other objects. The properties are not

valid for AutoCAD objects. When you select an AutoCAD object as the source object, this section is masked out.



With standard parts the geometry is only transferred for identical standards. For example, it is not possible to convert a metric screw into a sheet screw.

7 Drawing aids

Commands in this section:

- Horizontal auxiliary line
- Vertical auxiliary line
- Horizontal and vertical auxiliary lines
- Offset auxiliary line
- Auxiliary line on object
- Delete auxiliary lines
- Auxiliary line angle
- Auxiliary line, angle bisecting
- Auxiliary line perpendicular
- Auxiliary line ray infinite
- Auxiliary line ray
- Auxiliary frame DIN A0
- Auxiliary line parallel to cross hairs
- Auxiliary line perpendicular to cross hairs
- Center of
- Object selection according to layer and color
- New point
- Limit line
- Rotate cross hairs

7.1 Horizontal auxiliary line



Ribbon:	Tab ATHENA > Group Drawing aids > Horizontal auxiliary line
Menu:	ATHENA > Drawing aids > Horizontal auxiliary line
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	ath_hiho

With this command you create a horizontal auxiliary line which runs through the specified point.

Command prompt

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

You call the help with the option ?.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.2 Vertical auxiliary line



Ribbon:	Tab ATHENA > Group Drawing aids > Vertical auxiliary line
Menu:	ATHENA > Drawing aids > Vertical auxiliary line
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	ath_hive

With this command you create a vertical auxiliary line which runs through the specified point.

Command prompt

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

You call the help with the option ?.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.3 Horizontal and vertical auxiliary lines



Ribbon:	Tab ATHENA > Group Drawing aids > Horizontal and vertical auxiliary lines
Menu:	ATHENA > Drawing aids > Horizontal and vertical auxiliary lines
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	ath_hihv

With this command you create a horizontal and a vertical auxiliary line through the specified point.

Command prompt

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

You call the help with the option ?.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.4 Offset auxiliary line



Ribbon:	Tab ATHENA > Group Drawing aids > Offset auxiliary line
Menu:	ATHENA > Drawing aids > Offset auxiliary line
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	ath_hivz

With this command you create an auxiliary line which runs parallel to a selected object at the defined distance.

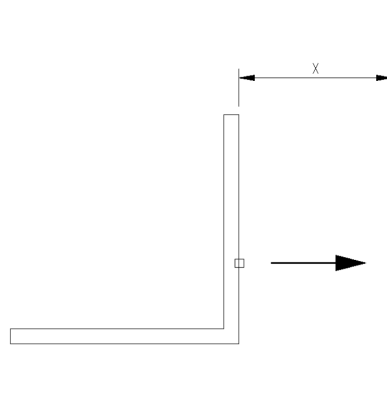


Fig. 7.1: Offset auxiliary line

Command prompt

Specify distance or [Multiple/?]: <5>:

Point with the mouse or enter a length to specify the distance of the auxiliary line to the object.

*With the **Option Multiple** you can offset many auxiliary lines.*

Option Multiple

Specify distances <5, 7, 9, 15>:

Enter the absolute distances of the auxiliary lines. Use a comma to separate the distance values.

Select Polyline or Line or [?]:

Select the line object from which you wish to offset the auxiliary line. You can select a polyline, line, Xline, ray or an edge of a 3D solid. The objects can also be constituent parts of a block.

Specify side of the auxiliary line or [?]:

Specify on which side of the selected object the auxiliary line is to be created.

The last two input requests are repeated. Press the Enter key to terminate the command.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.5 Auxiliary line on object



Ribbon:	Tab ATHENA > Group Drawing aids > Auxiliary line on object
Menu:	ATHENA > Group Drawing aids > Auxiliary line on object
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	ath_hiob

With this command you create an auxiliary line which runs through a selected line object.

Command prompt

Select Polyline or Line or [?]:

Select the line object through which the auxiliary line is to run. You can select a polyline, line, Xline, ray or an edge of a 3D solid. The objects can also be constituent parts of a block.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.6 Delete auxiliary lines



Ribbon:	Tab ATHENA > Group Drawing aids > Delete auxiliary line
Menu:	ATHENA > Drawing aids > Delete auxiliary line
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	<code>ath_hidl</code>

With this command you delete all auxiliary lines from the current drawing.

Important: ATHENA removes all objects which are drawn on the system layer for auxiliary lines (as standard AUX).

7.7 Auxiliary line angle



Ribbon:	Tab ATHENA > Group Drawing aids > Auxiliary line angle
Menu:	ATHENA > Drawing aids > Auxiliary line angle
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	ath_hiwi

With this command you create an auxiliary line which runs through the specified point at the specified angle.

Command prompt

Enter angle or first point or [Points in 3D/?] <Points in 3D>:

Define the angle of the auxiliary line by pointing with the mouse or by entering an angle to create an auxiliary line in the XY plane of the current UCS.

*Use the **Option Points in 3D** to produce an auxiliary line in three-dimensional space.*

You call the help with the option ?.

Option Points in 3D

Specify first point in 3D or [?]:

Point with the mouse or enter coordinates to specify the first point of the angle.

Specify second point in 3D or [?]:

Point with the mouse or enter coordinates to specify the second point of the angle.

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.8 Auxiliary line, angle bisecting



Ribbon:	Tab ATHENA > Group Drawing aids > Auxiliary line, angle bisecting
Menu:	ATHENA > Drawing aids > Auxiliary line, angle bisecting
Toolbar:	ATH Drawing Aids
Command input:	ath_aux_half

Generates an auxiliary line which bisects the angle between two lines which are selected.

Command prompt

Select arc or line or [Specify vertex/?] <Specify vertex>:

Select a line object or an arc. You can select a polyline, line, Xline, ray or an edge of a 3D solid. The objects can also be constituent parts of a block.

If you select an arc, the auxiliary line is produced directly bisecting the lines from the center of the arc to the end points of the arc.

With the **Specify vertex option** you can produce an auxiliary line by specifying three points.

Select second line or [?]:

Select the second line.

You call the help with the option ?.

Specify vertex option

Specify vertex or [?]:

Specify a vertex. The vertex defines the first passing point for the auxiliary line.

Specify first angle point or [?]:

Specify the first point of the angle.

Specify second angle point or [?]:

Specify the second point of the angle.

The middle of the two angle points defines the second passing point for the auxiliary line.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.9 Auxiliary line perpendicular



Ribbon:	Tab ATHENA > Group Drawing aids > Auxiliary line perpendicular
Menu:	ATHENA > Drawing aids > Auxiliary line perpendicular
Toolbar:	ATH Drawing Aids
Command input:	ath_hilo

With this command you create an auxiliary line running perpendicular to the selected object and also running through the specified point.

Command prompt

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

You call the help with the option ?.

Select Polyline or Line or [?]:

Select the line object to which the auxiliary line is to run perpendicular. You can select a polyline, line, Xline, ray or an edge of a 3D solid. The objects can also be constituent parts of a block.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.10 Auxiliary line ray infinite



Ribbon:	Tab ATHENA > Group Drawing aids > Auxiliary line ray infinite
Menu:	ATHENA > Drawing aids > Auxiliary line ray infinite
Toolbar:	ATH Drawing Aids
Command input:	ath_hixl

With this command you create an auxiliary line, infinite at both ends, the angle of which is defined by a second point to the start point.

Command prompt

Specify starting point of the auxiliary line or [?]:

Use the mouse or enter coordinates to specify the start point of the ray. You call the help with the option ?.

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

This input request is repeated. Press the Enter key to terminate the command.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.11 Auxiliary line ray



Ribbon:	Tab ATHENA > Group Drawing aids > Auxiliary line ray
Menu:	ATHENA > Drawing aids > Auxiliary line ray
Toolbar:	ATH Drawing Aids
Command input:	ath_hiry

With this command you create an auxiliary line limited at one end with its start point located at the specified point.

Command prompt

Specify starting point of the auxiliary line or [?]:

Use the mouse or enter coordinates to specify the start point of the ray.

You call the help with the option ?.

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

This input request is repeated. Press the Enter key to terminate the command.

Notes

- A ray has a fixed start point and extends to infinity. (Object type RAY). You will find further information about X Rays in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.12 Auxiliary frame DIN A0



Ribbon:	Tab ATHENA > Group Drawing aids > Auxiliary frame DIN A0
Menu:	ATHENA > Drawing aids > Auxiliary frame DIN A0
Toolbar:	ATH Drawing Aids
Command input:	ath_hira

With this command you create a polyline rectangle as auxiliary line margin in DIN A0 format. The insertion point (left lower corner) is located at the origin of the current UCS.

Command prompt

Command: **AUXILIARY LINE MARGIN**

The auxiliary frame is created directly.

Notes

- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.13 Auxiliary line parallel to cross hairs



Ribbon:	Not present
Menu:	ATHENA > Drawing aids > Auxiliary line parallel to cross hairs
Toolbar:	ATH Drawing Aids
Command input:	ath_hi0s

With this command you create an auxiliary line which runs parallel to the cross hairs and also through the specified point.

Command prompt

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

You call the help with the option ?.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.14 Auxiliary line perpendicular to cross hairs



Ribbon:	Not present
Menu:	ATHENA > Drawing aids > Auxiliary line perpendicular to cross hairs
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	ath_hi9s

With this command you create an auxiliary line running perpendicular to the cross hairs which also runs through the specified point.

Command prompt

Specify passing point of the auxiliary line or [?]:

Using the mouse or by entering coordinates, you define the point through which the auxiliary line is to run.

You call the help with the option ?.

Notes

- Auxiliary lines are infinite lines (object type XLINE). You will find further information about Xlines in your AutoCAD documentation.
- For auxiliary lines ATHENA uses its own system layer (as standard AUX). Use the Dialog box System layer to adapt the layer properties to your own requirements.
- Use the command **Delete auxiliary lines** to remove all auxiliary lines from the current drawing.

7.15 Center of



Ribbon:	Not present
Menu:	ATHENA > Drawing aids > Center of
Toolbar:	ATH Drawing Aids
Command input:	(ath_mitv)

With this command you can snap a point which is located centrally between two other points.

You can use this transparent command like an object snap.

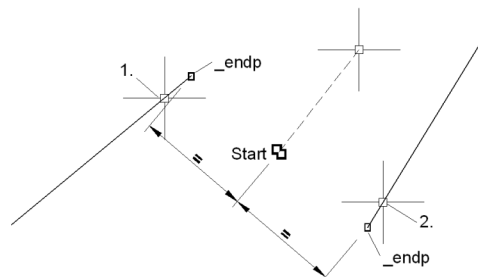


Fig. 7.2: Object snap, Center of

Command prompt

Example line

*Command: specify first point of line:
Run the command Center of.*

*>> Specify first point:
Specify the first point. You can additionally activate the object snap.*

*>> Specify second point:
Specify the second point. You can additionally activate the object snap. The center of the specified points is the start point of the line.*

7.16 Object selection according to layer and color



Ribbon:	Not present
Menu:	ATHENA > Drawing aids > Object selection according to layer and color
Toolbar:	ATH Drawing Aids
Command input:	(ath_slav)

With this command you can select objects with a certain color and a certain layer.

You can use this transparent command like an object filter.

Command prompt

Example of move

Command: move

Select objects:

Run the command Object definition according to layer and color.

OBJECT DEFINITION ACCORDING TO LAYER AND COLOR

Indicate color or [?]:

Select an object with the required color.

You call the help with the option ?.

Point to layer or [Layer name/?] <Layer name>:

Select an object with the required layer.

*Use the **Option Layer name** to enter a layer name.*

Option Layer name

Enter layer name:

Enter the layer name in the command line.

Cross or [?]:

Cross the required area of the drawing. The objects selected according to layer and color are marked and can be moved.

7.17 New point



Ribbon:	Not present
Menu:	ATHENA > Drawing aids > New point
Toolbar:	ATH Drawing Aids
Command input:	(ath_neup)

With this object snap you can define the insertion or base point relative to a point which is to be entered.

You can use this transparent command like an object snap.

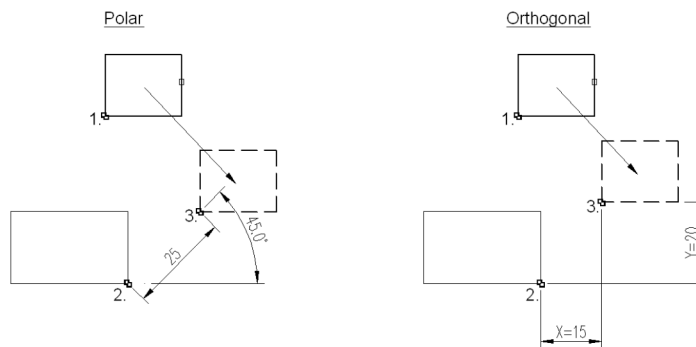


Fig. 7.3: Object snap, New point

Command prompt

Example of move

Command: move

Select objects:

Use the mouse to choose the objects you wish to move.

Base point of the movement:

Run the command New point.

>> Specify old point:

Specify a point.

>> Specify new point relatively [Polar] <Orthogonal>:

*Enter the **Option Orthogonal** to specify the new base point orthogonal to the previous point.*

*Use the **Option Polar** to specify the new point in polar coordinates.*

Option Orthogonal

>> Enter Delta-X <0>:

Enter the X distance between the old and new points.

Press the Enter key to accept the default value.

>> Enter Delta-Y <0>:

Enter the Y distance between the old and new points.

Press the Enter key to accept the default value.

>> Enter Delta-Z <0>:

Enter the Z distance between the old and new points.

Press the Enter key to accept the default value. The new base point is then defined.

Option Polar

>> *Enter angle <0>:*

Enter a polar angle from the old point.

>> *Enter distance:*

Enter a polar distance. The new base point is then defined.

7.18 Limit line



Ribbon:	Not present
Menu:	ATHENA > Drawing aids > Limit line
Toolbar:	ATH Drawing Aids and ATHENA Auxiliary Lines
Command input:	(ath_lbgr)

With this command you can limit a line or polyline at an object (line, polyline, Xline).

You can use this transparent command like an object snap.

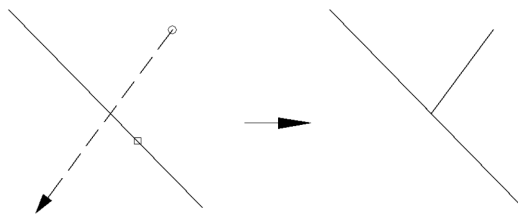


Fig. 7.4: Limit line

Command prompt

Example line

Command: specify first point of line:
Specify the starting point of the line.

Specify next point or [Undo]:
Run the command Limit line.

LIMIT LINE

Specify direction:
Specify the direction of the line.

Select limit line:
Select the limit line.

7.19 Rotate cross hairs



Ribbon: Tab ATHENA > Group Drawing aids > Rotate cross hairs
Menu: ATHENA > Drawing aids > Rotate cross hairs
Toolbar: ATH Drawing Aids
Command input: (ath_snap)

With this command you can align the cross hairs to a selected line, polyline, Xline or ray (both auxiliary lines).

Important: You can use this command transparently.

Command prompt

Select line or polyline:

With the mouse select the object, to whose angle the cross hairs are to be set.

8 Text

Commands in this section:

- Manage text styles
- Change text multiple same
- Change text multiple different
- Text properties
- Convert text
- Text multilingual
- Set text language
- Table import
- Table export
- Table update
- Table properties
- Parts count
- Coating extent

8.1 Manage text styles

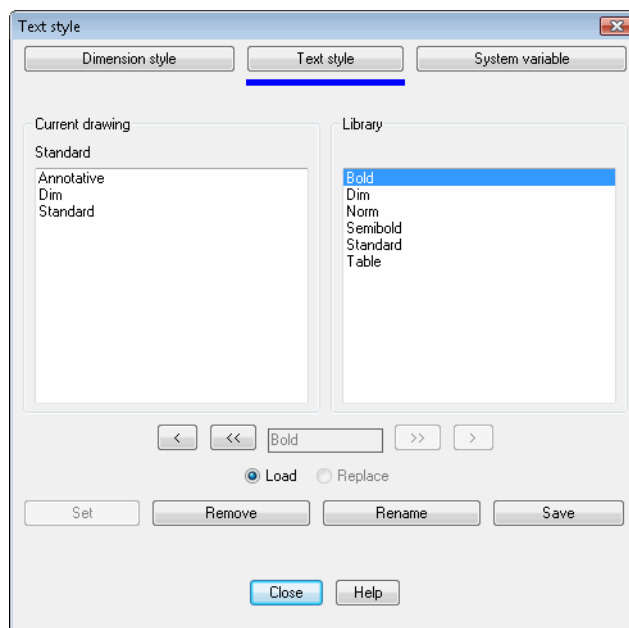


Ribbon:	Tab ATHENA > Group Extras > Manage text styles
Menu:	ATHENA > Text > Manage text styles
Toolbar:	ATH Text
Command input:	ath_txtstyle

You can manage text styles with this command.

You can save text styles which are present in the current drawing in a library and load them into other drawings and also set them as current. Furthermore you can extend the library with dimension styles of the current drawing.

Dialog box Text style



Register button Dimension style

Activates the Dialog box Dimension style.

You will find the functional description in the section *Manage dimension styles* on page 454.

Register button Text style

Activates the Dialog box Text style.

You will find the functional description further below.

Register button System variable

Activates the Dialog box System variables.

You will find the functional description in the section *Manage system variables* on page 609.

Dialog box section Current drawing

Lists the text styles of the current drawing. You can select text styles here for further use.

Dialog box section Library

Lists the text styles of the library. You can select text styles here for further use.

[<]

Copies the selected text style of the library into the current drawing.

[<<]

Copies all text styles of the library into the current drawing.

Input field

Changes the name of the selected text style. This is only possible when you click the button Rename.

[>]

Copies the selected text style of the current drawing into the library.

[>>]

Copies all text styles of the current drawing into the library.

Load

Loads the text style into the current drawing.

Replace

This function is not available.

Set

Sets the selected text style of the current drawing as the current text style.

Remove

Deletes the selected text style from the list.

Rename

Releases the input field for changing the name of the selected text style.

Save

Saves the library.

Notes

- The text style library is saved in the file ATHENA\DATALOCAL\ath_txt.dex.
- Use the AutoCAD command ***_style*** to create a new text style. You will find further information on this subject in the AutoCAD documentation.
- To change the properties of existing texts in the drawing, use the command ***Text properties***.
- You can find out more about text styles in the Chapter *Text styles* on page 59 and in your AutoCAD documentation.

8.2 Change text multiple same



Ribbon:	Tab ATHENA > Group Label > Change text multiple same
Menu:	ATHENA > Text > Change text multiple same
Toolbar:	ATH Text
Command input:	ath_atmg

With this routine you change character strings in a number of text lines simultaneously.

Command prompt

Select objects:

Select all texts in which you wish to change certain character strings.

Enter old text:

Enter the characters or character string which is to be changed.

Enter new text:

Enter a new character or a new character string. The old character string is now replaced by the new one in all the selected texts.

Notes

You **cannot** use this command for text changes in multi-line text objects (MTEXT).

8.3 Change text multiple different



Ribbon:	Tab ATHENA > Group Label > Change text multiple different
Menu:	ATHENA > Text > Change text multiple different
Toolbar:	ATH Text
Command input:	ath_atmv

With this routine you change a number of selected texts in a certain sequence.

You can use this command very effectively, for example, to carry out numerical changes in tables.

Command prompt

Select objects:

Select all texts which you wish to change.

Enter new text:

Enter the new text.

This input request appears once for each text. The text change is always made from top left to bottom right, irrespective of the sequence in which the text objects have been selected.

Notes

You **cannot** use this command for text changes in multi-line text objects (MTEXT).

8.4 Text properties



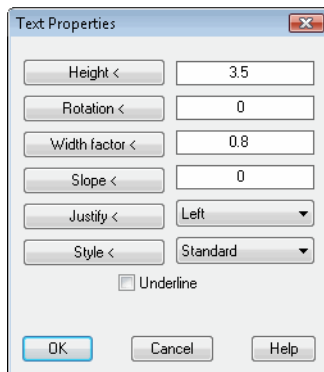
Ribbon:	Tab ATHENA > Group Label > Text properties
Menu:	ATHENA > Text > Text properties
Toolbar:	ATH Text and ATHENA Text
Command input:	ath_atmo

With this command you can change the properties of texts using a clearly laid-out dialog box.

Select objects:

Use the mouse to select the texts to be changed. Press the Enter key to terminate the object selection. The Dialog box Text properties is now opened and you can change the desired properties.

Dialog box Text properties



In the dialog box the properties of the selected text are displayed. If you have selected a number of texts with different properties, the display in the input fields is variable.

Height

You can change the text height by writing the height figure directly into the appropriate input field. With the Height button < you have the possibility of adopting the height of an existing text. When you click the button, the dialog box is temporarily closed. Now you click an existing text to transfer its height to the input field.

Rotation

You can change the text rotation by writing the rotation angle directly into the appropriate input field. With the Rotation button < you have the possibility of adopting the rotation of an existing text. When you click the button, the dialog box is temporarily closed. Now you click an existing text to transfer its rotation to the input field.

Width factor

You can change the text width factor by writing the width factor directly into the appropriate input field. With the Width factor button < you have the possibility of adopting the width factor of an existing text. When you click the button, the dialog box is temporarily closed. Now you click an existing text to transfer its width factor to the input field.

Slope

You can change the text slope by writing the slope directly into the appropriate input field. With the Slope button < you have the possibility of adopting the slope

of an existing text. When you click the button, the dialog box is temporarily closed. Now you click an existing text to transfer its slope to the input field.

Alignment

You can change the justification of a text by clicking a text justification in the drop-down list. With the Justify button < you have the possibility of adopting the justification of an existing text. When you click the button, the dialog box is temporarily closed. You now click an existing text to transfer its justification.

Style

You can change the text style by clicking a style in the drop-down list. With the Style button < you have the possibility of adopting the style of an existing text. When you click the button, the dialog box is temporarily closed. You now click an existing text to transfer its style.

Underline

Activate the tick box to underline all the selected texts.

Notes

- You will find further information about texts and text styles in your AutoCAD documentation.
- ATHENA offers the possibility of defining text styles in a file and of reloading them into the current drawing. You will find further information in Chapters *Text styles* on page 59 and *Manage text styles* on page 526.

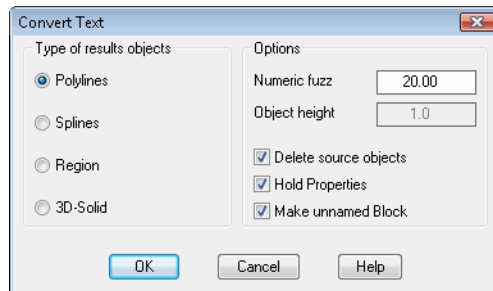
8.5 Convert text



Ribbon:	Tab ATHENA > Group Label > Convert text
Menu:	ATHENA > Text > Convert text
Toolbar:	ATH Text
Command input:	ath_convtext

With this command you can convert texts into polylines, splines, regions or 3D solids.

Dialog box Convert texts



Dialog box section Type of results object

Here you specify the result object of the conversion. You have the choice between the following object types:

- Polylines
- Splines
- Region
- 3D solid

Dialog box section Options

With the Numeric fuzz you specify the resolution when converting text to polylines. The higher the resolution, the longer the text conversion takes. We recommend a value between 5 and 20.

Object height is only available with the result object type 3D solid and defines the height of the object.

When Delete source objects is activated, the selected texts are deleted. When Adopt properties is activated, the layer properties of the text are accepted. Otherwise ATHENA applies the current layer.

When Make unnamed block is activated, the converted texts are saved as anonymous blocks.

When you click OK, the input request follows:

Command prompt

Select objects:

Select the texts to be converted.

This input request is repeated. Press the Enter key to terminate the object selection. The texts are converted according to the entries.

8.6 Text multilingual



Ribbon:	Tab ATHENA > Group Label > Text multilingual
Menu:	ATHENA > Text > Text multilingual
Toolbar:	ATH Text and ATHENA Text
Command input:	ath_mtxt

With this command you can save the text content of an existing text in a number of languages to display another language if required.

This functionality is possible with:

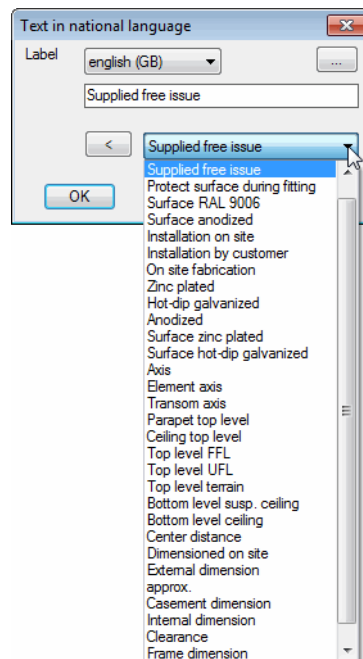
- Single text (DTEXT)
- Paragraph text (MTEXT)
- Attribute text (attribute definition)

Command prompt

Select object:

Click the text to be changed with the mouse. A dialog box is opened depending on the selected text object.

Dialog box Text in national language for single-line text



Label

Specifies the language for the text. Once you have selected a language, you can write the text in the input line or modify an existing text.

[...]

Opens the Dialog box Designation, where you can more conveniently manage the text. You will find a detailed functional description of the this dialog box in the chapter *Designation* on page 122.

Input line

Defines the text in the chosen language.

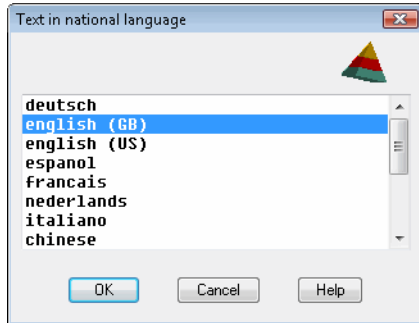
[<] and pick list

The pick list displays multilingual standard texts from the text database

ath_user.rsx. Select a text and click the button [<] to insert the text into the active row.

You will find further information about labeling objects in the Chapter *Multilingual standard texts* on page 60.

Dialog box Text in national language with paragraph or attribute text



Select the language in which you wish to additionally save the text content.

With paragraph text:

When you click OK, ATHENA closes the dialog box and starts the MText Editor defined in AutoCAD, where you can enter the text or modify the existing text. You will find further information about the MText Editor in the AutoCAD documentation.

With attribute text:

When you click OK, ATHENA closes the dialog box and starts the text editor defined in AutoCAD, where you can enter the attribute text or modify the existing text. You will find further information about attributes in the AutoCAD documentation.

Notes

- The texts can also be present in blocks or attributes. For example, texts in drawing frames and text boxes can be displayed in another language. You will find further information in Chapters *Insert drawing frame* on page 552 and *Complete caption* on page 555.
- Text content in other languages is saved invisibly with the text.
- Use the command **Set text language** to display the text in another language.

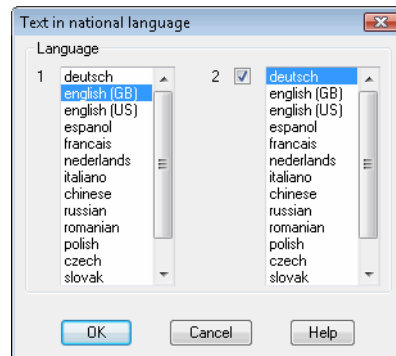
8.7 Set text language



Ribbon:	Tab ATHENA > Group Label > Set text language
Menu:	ATHENA > Text > Set text language
Toolbar:	ATH Text and ATHENA Text
Command input:	ath_mtxt_upd

With this command you can display multilingually saved texts, leader texts or parts labels in another language.

Dialog box Text in national language



Language 1

Defines the language for the text. Select the language in which the text is to be displayed.

Language 2

Defines the second language for the text. Activate the tick box to activate the bilingual features and select the second language.



The bilingual text display is only possible with leaders and parts labels. With single-line texts, paragraph texts and attribute texts this setting has no effect.

When you terminate the Dialog box Text in national language with OK, the following input request appears.

Command prompt

Select objects:

Select the texts which you wish to display in another language.

This input request is repeated until you terminate the object selection with a right click or ENTER.

Notes

- Use the command **Text multilingual** to save text contents in other languages.
- The selected texts can also be present in blocks, attributes or tables. For example, texts in drawing frames, text boxes or tables (e.g. from structural analysis computations) can be displayed in another language. You will find further information in Chapters *Insert drawing frame* on page 552 and *Complete caption* on page 555.
- If the texts have not been saved in the selected language, ATHENA displays the German text.

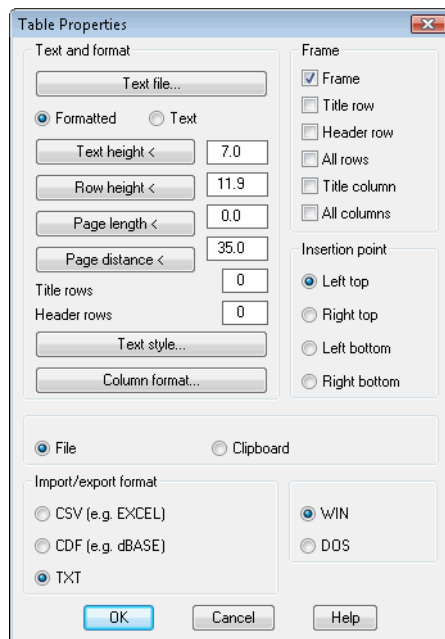
8.8 Table import



Ribbon:	Tab ATHENA > Group Label > Table import
Menu:	ATHENA > Text > Table import
Toolbar:	ATH Text
Command input:	ath_tt_import

With this function you can insert ASCII files into the current drawing as tables.

Dialog box Table properties



Text file

Click this button to select the ASCII file to be imported using the standard dialog box for file selection.

Dialog box section Import/export format

Here, you define whether a file or the Windows clipboard is to be used as the import source.

Furthermore, you define the table import format. You can specify whether the source format for special characters (ä, Ä, ü, Ü, ö, Ö, ß) is DOS or Windows. The other elements of the dialog box are comprehensively described in Chapter *Table properties* on page 541.

When you terminate the dialog box with OK, the following input request appears:

Command prompt

Specify insertion point:

Specify the insertion point of the table.

Specify rotated angle<0>:

Specify the rotation angle of the table or adopt the default angle.

Notes

- When you have made changes to the source file, you can use the command **Table update** to update the imported table.

- Use the command *Table properties* to retrospectively modify the appearance of the table.
- Use the command *Table export* to export the table to a file or to the Windows clipboard.
- You can define the layer used for the table in Dialog box System layer.

8.9 Table export



Ribbon:	Tab ATHENA > Group Label > Table export
Menu:	ATHENA > Text > Table export
Toolbar:	ATH Text
Command input:	ath_tt_export

With this function you can export tables, which are included in drawings, as ASCII files or to the Windows clipboard.

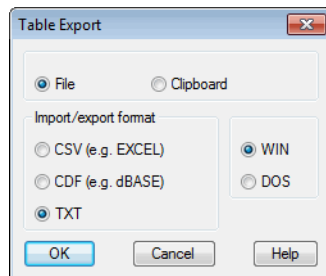
Command prompt

Select table <End>:

Select the table to be exported.

*With **ENTER** or a **RIGHT CLICK** you terminate the command without exporting a table. When you select a table, the dialog box Table export appears.*

Dialog box Table export



Dialog box section File/clipboard

You define with the option buttons whether you wish to export the table in an ASCII file or to the Windows clipboard.

Dialog box section Import/export format

Here, you define the output format of the ASCII file. There is a choice of the following formats:

- csv Interchange format for Excel. The columns are separated with semicolons.
- cdf Interchange format for dBase. The columns are separated with commas. Alphanumerical fields are set in inverted commas.
- txt ASCII text with spaces.

You can also define the display of the special characters:

- WIN Special characters (ö, ü, ä, ß, etc.) are saved in the Windows format.
- DOS Special characters (ö, ü, ä, ß, etc.) are saved in the DOS format.

If you click OK to terminate the dialog box, a standard dialog box for file selection opens and you can specify the storage location and the name for the file to be exported.

Notes

- When you have made changes to the source file, you can use the command **Table update** to update the imported table.
- Use the command **Table properties** to retrospectively modify the appearance of the table.
- Use the command **Table import** to import a table into the current drawing.

8.10 Table update



Ribbon:	Tab ATHENA > Group Label > Table update
Menu:	ATHENA > Text > Table update
Toolbar:	ATH Text
Command input:	ath_tt_upd

With this function you update tables which you have previously inserted with the ATHENA command **Table import**. This is necessary when you have made changes to the source file and wish to incorporate them. The table formatting is not affected.

Command prompt

Select table <End>:

Select the table which is to be updated.

*The function is terminated with **ENTER** or a **RIGHT CLICK**.*

Notes

- Use the command **Table properties** to retrospectively modify the appearance of the table.
- Use the command **Table import** to import a table into the current drawing.
- Use the command **Table export** to export the table to a file or to the Windows clipboard.

8.11 Table properties



Ribbon:	Tab ATHENA > Group Label > Table properties
Menu:	ATHENA > Text > Table properties
Toolbar:	ATH Text
Command input:	ath_tt_prop

With this function you can change the visual and technical properties of a table.

General remarks

A **table** for object types exists in ATHENA. It comprises summarised texts which can be modified in their properties and can be output formatted as ASCII files. Most ATHENA routines which contain a text output (e.g. Parts count) generate these results texts as tables.

Table types

Formatted tables:

Formatted tables contain rows and columns. The values in them are formatted depending on the content (numerical or string). Numerical data is always right justified, strings are always left justified. The text outputs in the structural analysis programs, for example, generate formatted tables.

Text tables:

Text tables only contain rows. The values in them are always arranged left justified. This type should be chosen, for example, when an ASCII file is to be read into the drawing as a table.

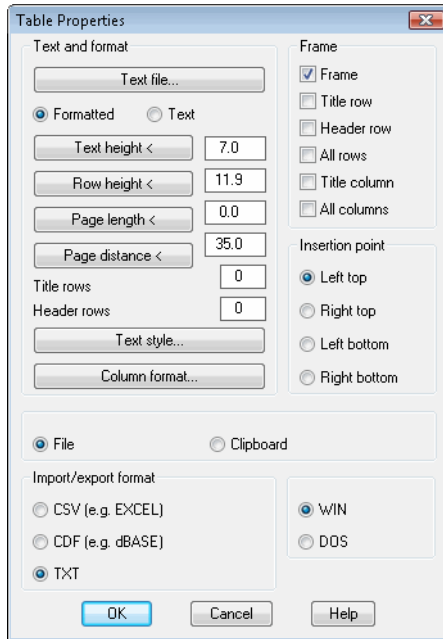
Command prompt

Select table <End>:

Select the table whose properties you wish to change. The dialog box table properties appears.

*You terminate the program with **ENTER** or a **RIGHT CLICK**.*

Dialog box Table properties



Dialog box section Text and format

The button **Text file** is not needed in this routine. See *Table import* on page 536. With the option buttons **Formatted** or **Text** you define the table format (see Table types at the start of this chapter).

Text height

Enter the text height in the input field or access the text height from the current drawing by clicking on two points. The current dimension figure height is used as the default text height.

Row height

Enter the required row height in mm directly or access the row height from the current drawing by clicking on two points. Text height x 1.7 is used as default. We recommend that the default row height is not changed, because the tables are then easiest to read.

Page length

Enter the page length in mm directly or access the page length from the current drawing by clicking on two points. The page length specifies from which length a table break is to occur. With the default value 0 no break is carried out.

Page distance

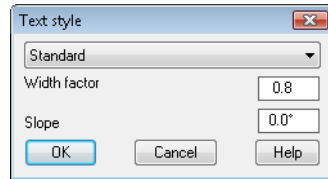
Enter the page distance in mm directly or access the page distance from the current drawing by clicking on two points. Here you specify the spacing of the table blocks when table breaks are present. Text height x 5 is used as default.

Title rows

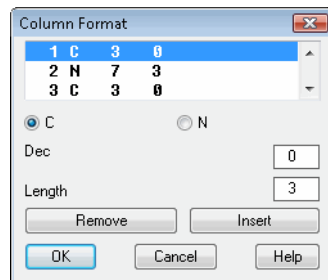
Enter the number of title rows. If you are using tables with breaks, title rows only appear in the first table block. 0 (no title row) is used as default.

Header rows

Enter the number of header rows. Header rows appear in each table block when a break is present. The default is zero (no header rows).

Dialog box Text style

Click Text style to change the properties style, width factor and slope angle in the Text style dialog box. You can find further information on text styles in the Chapter *Manage text styles* on page 526 and in the AutoCAD documentation.

Dialog box Column format

Column format is only active with formatted tables. When you click the button, you can adapt the table column properties in the Column format dialog box.

In the illustration Column format you can see a table with three columns. The first value is the column number. The columns are numbered from left to right. The second value is the specification of whether the column consists of a character string (C as in Character) or numbers (N as in Numerical). The third value determines the number of places. The fourth value determines the number of decimal places.

You can change the settings described above with the toggle switches C and N and with the input fields Dec and Length. You can also supplement or delete columns with the buttons Remove and Insert.

I x=	100. 61	cm4
W x=	18. 32	cm3
I y=	27. 66	cm4
W y=	11. 06	cm3
A =	8. 61	cm2

Fig. 8.1: *Table of centers of gravity and moments*

Dialog box section Frame

Here, you format the table and its rows and columns with frame lines.

- Frame: A frame is drawn around the complete table.
- Title row: The title row (first row) is underlined horizontally.
- Header row: The header row is underlined horizontally.
- All rows: All rows are underlined horizontally.
- Title column: A vertical separation line is drawn between the first and second columns.

All columns: Vertical separation lines are drawn between all the columns.

Dialog box section Insertion point

Here you specify the insertion point of the table.

Dialog box section Import/export format

Here you define the default values for the table export. You will find further information about this dialog box section in the Chapter *Table export* on page 538.

Click OK to adopt the settings for the selected table. Click Cancel to discard the changes. The dialog box is terminated in both cases.

Notes

- When you have made changes to the source file, you can use the command **Table update** to update the imported table.
- Use the command **Table import** to import a table into the current drawing.
- Use the command **Table export** to export the table to a file or to the Windows clipboard.
- You can define the layer used for the table in Dialog box System layer.

8.12 Parts count

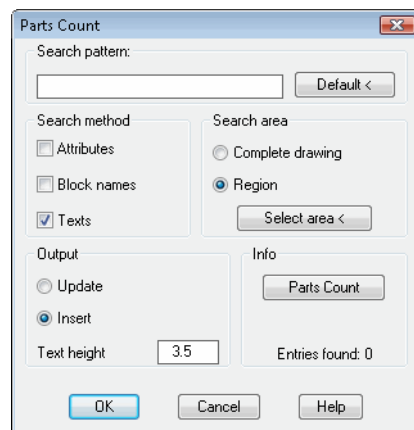


Ribbon:	Tab ATHENA > Group Label > Parts count
Menu:	ATHENA > Text > Parts count
Toolbar:	ATH Text and ATHENA Functions
Command input:	ath_tzae

With this command you can search for attributes, block names or text in a drawing (or part of a drawing) according to a search pattern. The objects found are counted and inserted into the drawing as a table. Consequently, you have the possibility of determining the quantities of individual items in a quick and simple manner.

You can export the parts table optionally with the command **Table export** to other programs (e.g. spread sheets) to use it as a parts list or order list.

Dialog box Parts count



Dialog box section Search pattern

In the Search pattern input field enter the texts or character strings to be sought. Multiple search patterns must be separated with a semicolon. The texts may also include the place holder * for a character string or ? for a character. With the button Default you transfer the texts, block names or attributes into the Search pattern field by clicking. In this respect, the dialog box is temporarily closed and the input request follows:

Command prompt

Select pattern:

Select a text or block as the search pattern.

Examples of search patterns:

- | | |
|-------|---|
| 01 | Finds all texts, block names and/or attributes with the value/ name 01. |
| 01* | Finds all texts, block names and/or attributes whose value/ name begins with 01 and consists of any number of characters. |
| 01;02 | Finds all texts, block names and/or attributes with the value/ name 01 or 02. |

- 01? Finds all texts, block names and/or attributes whose value/ name begins with 01 and consists of three characters.

Dialog box section Search method

Here you specify whether the search is to be made for attributes, texts or block names. You can also activate a number of objects simultaneously.

Dialog box section Search area

Here you specify whether the routine is to search for the selected objects in the whole drawing or just in a part of it. Click the button Select area < and choose the objects to be searched to limit the search area.

Dialog box section Info

Click the button Parts count to start the counting process. The number of objects found is displayed below the button.

Dialog box section Output

Activate Insert to place the table in the current drawing. Activate Update to refresh the existing table. Write the text height for the table in the input field Text height. The current dimension figure height is used as the default value.

OK terminates the dialog box and you can insert the table in the drawing or refresh a table. The settings in the dialog box are saved for the duration of the drawing session.


Command prompt

Specify insertion point:

Use the mouse or enter co-ordinates to specify the insertion point for the infill.

Specify rotated angle<0>:

*Use the mouse or enter an angle to specify the rotation angle. With **ENTER** you adopt the default angle.*



Part	Pieces
Pos 01	8
Pos 02	2
Pos 03	3
Pos 04	1
SUM	14

Fig. 8.2: Inserted table

Notes

- You can search for ATHENA parts (e.g. standard parts, sheet metal section, ...) and count them. Use the Attributes option and choose an existing part with the Default < button.
- You can adapt the appearance of the table with the command **Table properties**.
- With the command **Modify ATHENA** you can change the number of rows and columns or change text in the table.
- You can define the layer used for the table in Dialog box System layer.
- Projections created with the command **Projection objects** are not counted more than once.

8.13 Coating extent



Ribbon: Tab ATHENA > Group Label >
Menu: ATHENA > Dimension >
Toolbar: ATH Dimension and ATHENA Dimension
Command input: ath_coat

Calculates the circumference and partial circumference of an outline and outputs it to a table. The selected circumference of the outline is marked with a coating line.

Command prompt

Select outline or [?]:

Select the outline, the circumference of which is to be calculated.

Specify start point or [?]:

Specify the start point of the partial circumference.

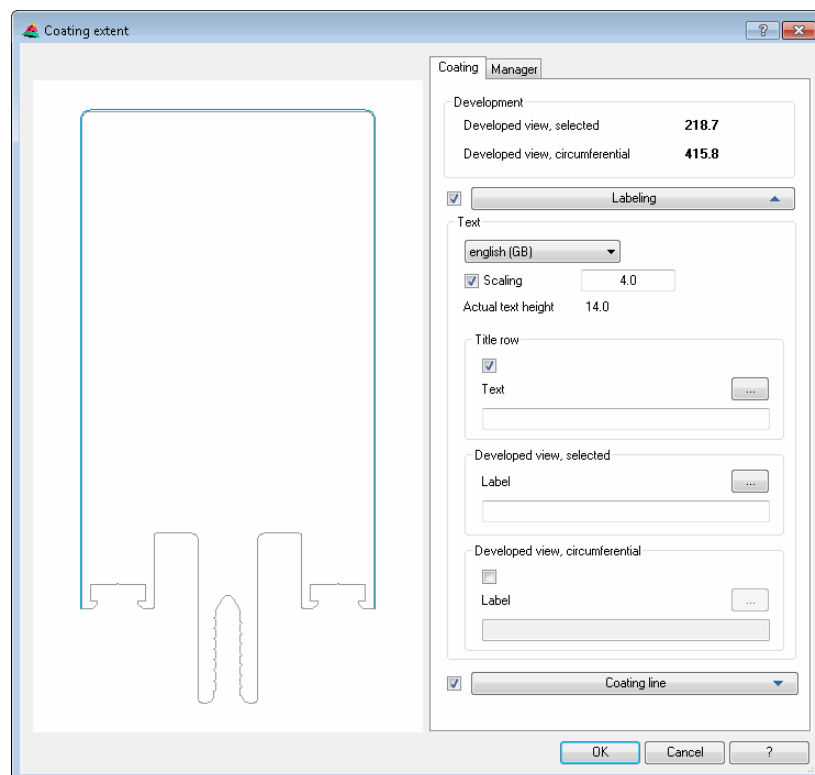
Specify the end point or [?]:

Specify the end point of the partial circumference.

Specify coating side or [?]:

Define the side on which the coating line is to be drawn. Once you have done that, the Dialog box Coating extent is opened, where you can carry out further settings.

Dialog box Coating extent



Display section

On the left side the dialog box contains a preview of the outline with the coating line. It provides a visual check. Additional functions are activated by clicking in

the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.

Operating section

On the right side of the dialog box there is the operating section with the following tabs:

- Coating
- Manager

Tab Coating

Dialog box section Development

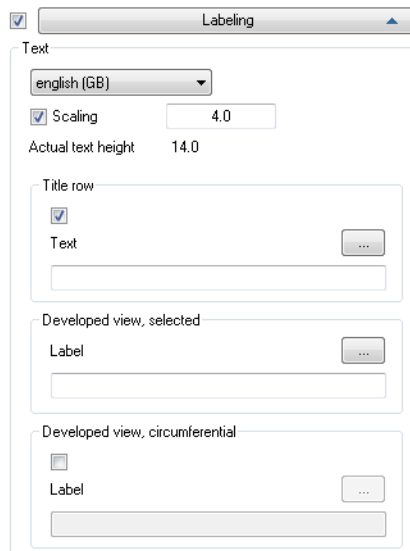
Developed view, selected

Shows the length of the partial circumference of the outline.

Developed view, circumferential

Shows the length of the complete circumference of the outline.

Drop-down menu Labeling



With the switch you can control whether labeling of the coating line is generated or not.

Dialog box section Text

The selection menu for the language defines the table display language.

Scaling

Activates replacement of the text scaling. The scale factor of the dimension style is used for the text size. When you activate the tick box, you can specify the scale factor as required.

The actual text height is shown below the scaling.

Dialog box section Title line

Defines whether a title line is specified in the table.

In the input line you define the text for the title line.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels

for various languages. You will find further information in the Chapter *Designation* on page 122.

Dialog box section Developed view, selected

In the input line you define the designation for the partial circumference (developed view selected).

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

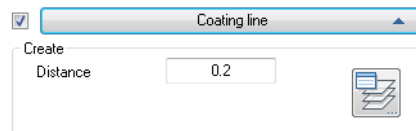
Dialog box section Developed view, circumferential

In the input line you define the designation for the circumference (developed view circumferential).

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Drop-down menu Coating line



With the switch you can control whether a coating line is generated or not.

Dialog box section Create

Distance

Defines the distance of the coating line to the outline.



Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

End of program

When you terminate the dialog box with OK, the coating line is drawn adjacent the selected section of the outline.

Text
Coating extent

Command reference

9 Block

Commands in this section:

- Insert drawing frame
- Complete caption
- Read out caption
- Index
- Insert profiles
- Save intermediate block
- Insert intermediate block
- Create unnamed block
- Name UBlock
- Insert block
- Group
- Explode group
- Save WBlock
- Insert WBlock
- Purge complete drawing
- Save user block
- Insert user block
- WBlock text
- WBlock text frame
- Bind and insert XRefs
- Block manager
- Assign block label
- Block element visibility

9.1 Insert drawing frame

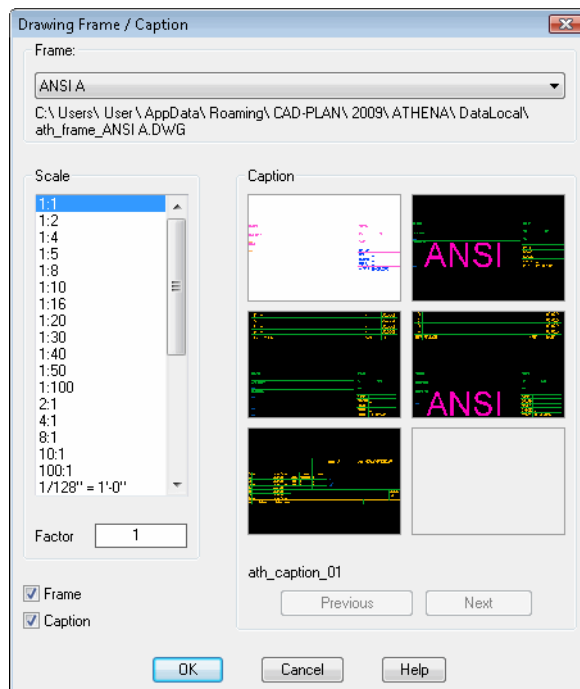


Ribbon:	Tab ATHENA > Group Label > Insert drawing frame
Menu:	ATHENA > Block > Insert drawing frame
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_plan

With this command you can insert a drawing frame with text box into the drawing. There is a choice of drawing frames in the formats DIN A4 to DIN A0 (in each case horizontal and vertical) along with three different text boxes. You have the possibility of integrating your own drawing frames and text boxes.

You can use the ATHENA command *Complete caption* to complete the text box in the drawing frame.

Dialog box Drawing frame/caption



Frame

Defines the drawing frame which you insert into the drawing. The storage destination and file name of the frame drawing are shown below the list.

Scale

Defines the scale for the drawing frame and the caption. Here, the scales from the AutoCAD list of scales are made available. You will find further information on this subject in the AutoCAD documentation.

Caption

Defines the caption which you insert into the drawing. Below the preview images the file name of the caption drawing is shown. With the buttons Next and Previous you can page to the next resp. previous view if you have more than six captions.

Tick box Frame

Activates or deactivates insertion of the drawing frame.

Tick box Caption

Activates or deactivates insertion of the caption.

If you now click the button OK, the dialog box is terminated and the following input request appears in the command line:

Command prompt

Specify insertion point or <0,0>:

Use the mouse or enter co-ordinates to specify the insertion point.

Press the Enter key to accept the default value 0.0 (absolute origin of the UCS).

Please note that the insertion points with drawing frames and captions are different if they are inserted separately; see figure Insertion points for drawing frame and caption.

Integrating your own drawing frames and text boxes:

If you would like to integrate your own drawing frames and text boxes, you must observe the following naming conventions:

A drawing is needed for each drawing frame. They must be saved in the folder ATHENA\DATALOCAL under the following names.

File name:	Display in the menu:	
ath_frame_A0-H.dwg	A0-H	present
ath_frame_A0-V.dwg	A0-V	present
ath_frame_A1-H.dwg	A1-H	present
ath_frame_A1-V.dwg	A1-V	present
ath_frame_A2-H.dwg	A2-H	present
ath_frame_A2-V.dwg	A2-V	present
ath_frame_A3-H.dwg	A3-H	present
ath_frame_A3-V.dwg	A4-V	present
ath_frame_A4-H.dwg	A4-H	present
ath_frame_A4-V.dwg	A4-V	present
ath_frame_A0-Ueberformat.dwg	A0 oversize	Example

When creating a new drawing frame, be sure to identify the insertion point of the text box with a block (base point of the block) with the name *ath_caption_base*. We recommend that an existing drawing frame is copied and adapted.

For each text box a drawing is similarly required which must be saved in the folder ATHENA\DATALOCAL. With text boxes the following conventions should be followed:

ath_caption_01.dwg	present
ath_caption_02.dwg	present
ath_caption_03.dwg	present

ath_caption_04. dwg Example

You can integrate further text boxes into ATHENA. The end numbers of the file names must be incremented.

Notes

- The following must be noted when outputting a drawing with drawing frame to a printer or plotter:
If the drawing frame/caption has been inserted by entering a scale, then at the plotting query Select scale or [Fit] <1>:, the scale, entered via the context menu, must be entered which was used for inserting the drawing frame (e.g. 1:1 for the plotter). Of course, you can also enter Fit here (e.g. for a laser printer).
- If a drawing frame is loaded, no basic settings (*lrfaktor*, *bemtxt*, *bemfktr*) are changed. It is assumed that the designer has already set up his environment.
- The texts in the drawing frames and text boxes are saved in multiple languages. You can display the text in another language with the command **Set text language**.
- You can replace an existing drawing frame with the command **Modify ATHENA**.

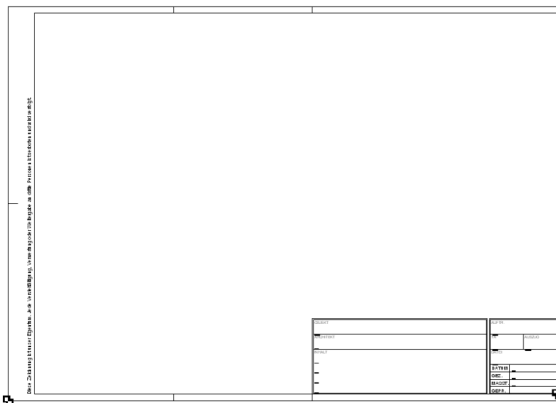


Fig. 9.1: Insertion points for drawing frame and caption

9.2 Complete caption



Ribbon:	Tab ATHENA > Group Label > Complete caption
Menu:	ATHENA > Block > Complete caption
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_plko

You complete a caption with this routine. The file name and current date (optional) are automatically entered into the text box. Furthermore, you can enter the complete drawing path in the drawing frame. You can find further information about configuring the date settings in the chapter *Options* on page 65.

Command prompt

This input request only appears if the date in the text box is not current. It is repeated for each text box in the drawing, if a number have been inserted.

The date has changed!

Old date: XX.YY.ZZZZ

Change date [Yes/No/?] <No>:

Select the option Yes to automatically update the date.

Select the option No to enter the date manually.

You call the help with the option ?.

Dialog box Edit attributes

In the dialog box Edit attributes you can complete and edit the individual fields in the text box. You change between the fields either with the mouse or the tabulator key. In the dialog box only eight lines are displayed. You can display the other lines with the buttons Next and Previous. Click OK to terminate the dialog box and to complete the text box. You will find further information about the Edit attributes dialog box in the AutoCAD documentation.

Notes

The program functions with the attribute definition of the three existing captions which you inserted into the drawing with *Insert drawing frame*. You can also use your own captions. The captions are saved with the names ath_caption_xx.dwg (xx stands for any number) in the folder ATHENA\DATALOCAL. The captions must satisfy the following conditions:

- An attribute for the file name must be present, Attribute name: *FILE*

- An attribute for the date and time must be present, Attribute name: *DATE_TIME*
- A block must be formed with the attributes, Block name: *LEGEND*
- You can define other attributes with any names.
- The texts in the text boxes are saved in multiple languages. You can display the text in another language with the command **Set text language**.

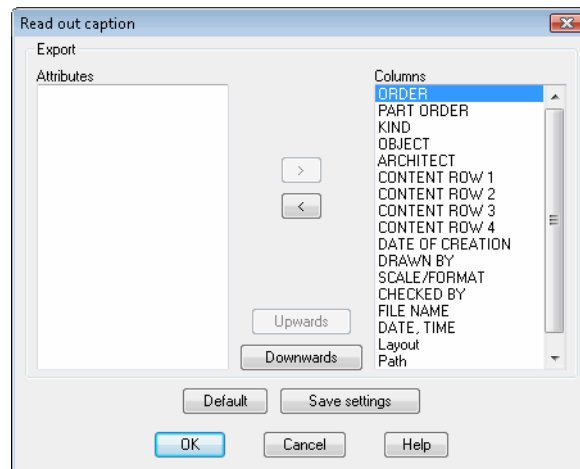
9.3 Read out caption



Ribbon: Tab ATHENA > Group Label > Read out caption
Menu: ATHENA > Block > Read out caption
Toolbar: ATH Block
Command input: ath_plst

With this command you can read out the attributes from one or more captions. The attributes are copied to the Windows clipboard and can then be inserted into a drawing list (e.g. in Excel).

Dialog box Read out caption



There are two pick lists in the Export dialog box section: Attributes and Columns. In the Attributes field the attributes are shown which are not exported. In the Columns field the attributes are shown which are exported as a table column. The basic setting is that all attributes are exported as table columns. With the buttons > and < you can move the displayed attributes between the pick lists. With the buttons Upwards and Downwards you can alter the order of the columns. To do this, mark the required attribute in the Columns field and click Upwards or Downwards until the required position is reached. With the button Save settings you can save the changes in the dialog box as a setup. The button Default restores the basic setup. Click OK to read out the content of the caption and click Cancel to terminate the dialog box without reading out the caption. The settings are however saved.

Conditions for reading out the caption

The following requirements must be fulfilled to be able to read out the content of a caption.

The block name of the caption must start with TEXT or LEGEND, upper/lowercase is not taken into account. Examples: TEXT, LEGEND, SCHRIFT_01, LEGEND_MODEL, ...

In addition the following attributes must be present (attributes which are not present in the drawing are shown in brackets):

German attribute name	Internat. attribute name	Explanation
-----------------------	--------------------------	-------------

AUFTR	ORDER	Order designation
TA	PART	Part order designation
AUSZ	KIND	Type of extract
OBJEKT	OBJECT	Object/project
ARCHITEKT	ARCHITECT	Architect
INHALT1	CONTENT1	Content of row 1
INHALT2	CONTENT2	Content of row 2
INHALT3	CONTENT3	Content of row 3
INHALT4	CONTENT4	Content of row 3
ERST_DATUM	CREATE	Date created
GEZ	DRAW	Drawn by
MASST	SCALE	Format, scale
GEPR	CHECK	Checked by
DATEI	FILE	File name
DATUM_ZEIT	DATE_TIME	Current date
LAYOUT	LAYOUT	Layout name
PATH	PATH	Path for saving
TIME	TIME	Time

Further attributes may be optionally present and can also be read out.

9.4 Index



Ribbon: Tab ATHENA > Group Label > Index
Menu: ATHENA > Block > Index
Toolbar: ATH Block
Command input: ath_plan_index

You use this command to insert an index as a block into the current drawing.

D	-	-	00.00.00
C	-	-	00.00.00
B	-	-	00.00.00
A	-	-	00.00.00
REV.	REVISION	NAME	DATE

CREDIT ARCHITECT DESIGNED CHECKED DATE	RAUPTL TA DATE DATUM GEZ. MASSST. GEGR.
--	---

Fig. 9.2: Index field

Command prompt

Specify insertion point for block:

Use the mouse or enter co-ordinates to specify the insertion point.

Specify scale factors for XYZ axes:

*Specify the size factor. Enter **ENTER** to insert the block in the scale 1:1.*

Specify rotated angle<0>:

*Specify the rotation angle. Enter **ENTER** to accept the default.*

Notes

- Use **Modify ATHENA** to edit the index.
- You can use an index field of your own or modify the existing one. To do this, edit the file ATHENA\DATALOCAL\ath_index.dwg.
- The texts of index fields, are saved in multiple languages. You can display the text in another language with the command **Set text language**.

9.5 Insert profiles



Ribbon:	Tab ATHENA > Group Draw > Insert profiles
Menu:	ATHENA > Block > Insert profiles
Toolbar:	ATH Profile Manufacturers
Command input:	ath_ladeprof

With this command you can insert the drawing of a profile, section or process details from the specified system manufacturer into the current drawing.

For insertion ATHENA uses the AutoCAD DesignCenter or alternatively the standard dialog box for file selection.

Command prompt

INSERT SCHUECO PROFILES

Specify selection <>:*

Enter the profile number (file name) of the file to be inserted. You can use optional wild cards with this input request.

Examples of profile specification

- | | |
|---------------|--|
| 322280 | The profile is attached to the cross hair and can be directly inserted. |
| 322* | ATHENA starts the AutoCAD standard dialog box or the filer and displays all drawings which have file names beginning with 322. |
| * | ATHENA starts the DesignCenter in the folder of the specified system manufacturer. |

Depending on the type of profile specification (see Examples of profile specification), the DesignCenter or the AutoCAD standard dialog box appears. When you have selected a file name, the following input request appears:

Specify insertion point or [?]:

Use the mouse or enter coordinates to specify the insertion point.

You call the help with the option ?.

Confirm the insertion or [Rotate/Move/Xmirror/Ymirror/XVaria/YVaria/Explode/?]:

*Confirm the insertion point with **ENTER** or **RIGHT CLICK** or select an option.*

*You can rotate the block about any angle with the **Option Rotate**. A further input request follows.*

*You can move the block with **Option Move**. A further input request follows.*

*With the option **Xmirror** you can mirror the block about its X axis.*

*With the option **Ymirror** you can mirror the block about its Y axis.*

*You can scale the block in the X direction with the **Option XVaria**. A further input request follows.*

*You can scale the block in the Y direction with the **Option YVaria**. A further input request follows.*

*You can insert the block in its single constituent parts with the option **Explode**.*

Insert designation [Yes/No/?] <Yes>:

*Use the **Option Yes** to insert a file name. Further input requests follow.*

*Use the option **No** in order not to insert the file name. The command is then terminated.*

You call the help with the option ?.

Option Rotate

Specify rotation angle or [?]:

Use the mouse or enter an angle to specify the rotation angle.

Option Move

Specify base point for the movement or [?]:

Use the mouse or enter co-ordinates to specify the base point of the movement.

Specify second point for the movement or [?]:

Use the mouse or enter co-ordinates to specify the second point for the movement.

Option XVaria

Specify X scale factor or [?] <1>:

Define the factor for scaling in the X direction.

Option YVaria

Specify Y scale factor or [?] <1>:

Define the factor for scaling in the Y direction.

Option Yes

Specify text height or [?] <3.5>:

Enter the text height for the file name.

Press the Enter key to accept the default height.

Specify insertion point or [?]:

Specify the insertion point of the file name. The filename is inserted on Layer 2-0.

Notes

You will find further information about the DesignCenter and standard dialog box for file selection in your AutoCAD documentation.

9.6 Save intermediate block



Ribbon:	Tab ATHENA > Group Extras > Save intermediate block
Menu:	ATHENA > Block > Save intermediate block
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_zout

This command enables you to save objects very quickly as intermediate blocks. The objects are retained in the drawing.

The command represents an extension of the Windows clipboard with the advantage that the objects are not lost when the computer is switched off. Furthermore, XRefs can be saved and inserted as intermediate blocks with this command.

Command prompt

Select objects:

Using the mouse, select the objects which are to be saved as intermediate blocks.

Specify insertion base point or [?]:

Use the mouse or enter co-ordinates to specify an insertion point for the objects. You call the help with the option ?.

Intermediate block created

Note that the selected objects have been saved as intermediate block.

Notes

- For insertion use the command **Insert intermediate block**.
- The intermediate block is saved in the temporary folder under the name ath_zb_loginname.dwg. Intermediate blocks are always overwritten again.

9.7 Insert intermediate block



Ribbon:	Tab ATHENA > Group Extras > Insert intermediate block
Menu:	ATHENA > Block > Insert intermediate block
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_zin

With this command you can insert the objects into the drawing which you have previously saved with the command **Save intermediate block**. The objects are not inserted into the drawing as a block.

The command represents an extension of the Windows clipboard with the advantage that the objects are not lost when the computer is switched off. Furthermore, XRefs can be saved and inserted as intermediate blocks with this command.

Command prompt

Specify insertion base point or [?]:

*Use the mouse or enter co-ordinates to specify an insertion point for the objects.
You call the help with the option ?.*

Notes

- Use the command **Save intermediate block** to create intermediate blocks.
- The intermediate block is saved in the temporary folder under the name ath_zb_loginname.dwg. Intermediate blocks are always overwritten again.

9.8 Create unnamed block



Ribbon:	Tab ATHENA > Group Extras > Create unnamed block
Menu:	ATHENA > Block > Create unnamed block
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_ublk

With this command you can combine different objects to form an anonymous block (block without a name).

To break the unnamed block up again into its constituent parts, use the Explode command.

Command prompt

Select objects:

Select with the mouse the objects which are to be constituent parts of the unnamed block.

9.9 Name UBlock



Ribbon:	Tab ATHENA > Group Extras > Name unnamed block
Menu:	ATHENA > Block > Name unnamed block
Toolbar:	ATH Block
Command input:	ath_aux_bnamed

Converts an unnamed block into a named block.

Unnamed blocks cannot be modified. If, for example, you want to modify an unnamed block with the block editor, you must first name it.

Select block:

Choose the unnamed block which you want to name.

Specify the block name <TMP_0000>:

Specify a block name or press the Enter key to accept the default name.

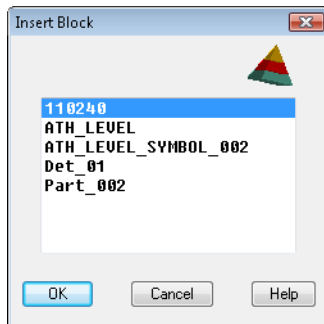
9.10 Insert block



Ribbon:	Not present
Menu:	ATHENA > Block > Insert block
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_bein

With this routine you can insert internal drawing blocks into the drawing in a simple manner.

Dialog box Insert block



The dialog box contains all blocks present in the drawing. To insert a block, mark the block name with the mouse and click the OK button. Alternatively, you can also double click the block name.

Command prompt

Specify insertion point or [Factor/ X/ Y/ Z/ Rotate/ VFactor/ VX/ VY/ VZ/ VRotate]:

Use the mouse or enter co-ordinates to specify the insertion point of the block or select an option. When you have determined the insertion point, another input request follows.

*With the **Option Factor** you determine the scale factor of the X, Y and Z axes. Further input requests follow.*

*With the **Options X/Y/Z** you determine the scale factor of the respective axis. Further input requests follow.*

*With the **Option Rotate** you determine the rotation angle. Further input requests follow.*

*With the **Option VFactor** you determine the preview scale factor for all axes. Further input requests follow.*

*With the **Options VX/VY/VZ** you determine the preview scale factor of the respective axis. Further input requests follow.*

*With the **Option VRotate** you determine the preview rotation angle. Further input requests follow.*

Specify X scale factor, opposite corner or [Corner/XYZ] <1>:

*Specify the X scale factor. Enter **ENTER** to accept the default.*

Enter Y scale factor <Use X scale factor>:

*Specify the Y scale factor. Enter **ENTER** to accept the default.*

Specify rotated angle<0.0>:

*Specify the rotation angle. Enter **ENTER** to accept the default.*

Option Factor

Specify scale factors for XYZ axes:

Enter the scale factor for the X, Y and Z axes. Further input requests follow.

*Options X/Y/Z**Specify X (Y, Z) scale factor:**Enter the scale factor for the respective axis.**Specify insertion point:**Specify the insertion point of the block.**Specify rotated angle<0.0>:**Specify the rotation angle of the block. Enter **ENTER** to accept the default.**Option Rotate**Specify rotation angle:**Specify the rotation angle for the block. Further input requests follow.**Option VFactor**Specify preview scale factors for XYZ axes:**Specify the preview scale factor for all axes. Further input requests follow.**Options VX/VY/VZ**Specify X (Y, Z) scale factor for preview:**Enter the preview scale factor for the respective axis. Further input requests follow.**Option VRotate**Specify preview rotation angle:**Specify the preview rotation angle. Further input requests follow.***Notes**

You will find further information about inserting objects in the AutoCAD documentation.

9.11 Group



Ribbon:	Tab ATHENA > Group Extras > Group
Menu:	ATHENA > Block > Group
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_grup

With this command you can combine different objects to form a group. Nesting is also possible, so that, for example, groups or blocks may form constituent parts of other groups.

Command prompt

Select objects:

Using the mouse, select the objects which are to be constituents of the group.

Notes

- When you select an object in the group, all elements in the group are selected. Single objects in the group can be edited with grips. You can switch groups on and off with the key combination CTRL+A. You will find further information on this subject in the AutoCAD documentation.
- Use the command **Explode group** to break up a group into its constituent parts. If you use the AutoCAD command Explode, polylines and blocks also contained in the group are released into their constituent parts.

9.12 Explode group



Ribbon:	Tab ATHENA > Group Extras > Explode group
Menu:	ATHENA > Block > Explode group
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_grur

With this command you can release a group into its constituent parts.

Command prompt

Select object:

Click the group to be released with the mouse.

Notes

- Use the command **Group** to form a group.
- If you use the AutoCAD command Explode, polylines and blocks also contained in the group are released into their constituent parts.

9.13 Save WBlock



Ribbon:	Tab ATHENA > Group Extras > Save WBlock
Menu:	ATHENA > Block > Save WBlock
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_bdwg

With this command you save WBlocks in the current folder. You also have the possibility of specifying a different path.

The command functions similar to the AutoCAD command WBlock with the difference that the current directory is always suggested as the default.

When you execute the command, the standard dialog box for file selection opens. Here, you choose the save location and enter the file name. If you use a file name which already exists, a confirmation query appears.

Click the Yes button to overwrite the existing file. Click No to enter a new file name. Click Cancel to terminate the command without creating a WBlock.

Command prompt

Select objects:

Select the objects which are to be constituent parts of the block.

Specify insertion base point or [?]:

Use the mouse or enter co-ordinates to specify the insertion point.

Notes

Further information on the subjects of creating drawing files and the standard dialog box for file selection can be found in your AutoCAD documentation.

9.14 Insert WBlock



Ribbon:	Not present
Menu:	ATHENA > Block > Insert WBlock
Toolbar:	ATH Block
Command input:	ath_ins

With this command you select a drawing to insert it as a block into the current drawing. The command offers options of aligning the block during insertion or of replacing an existing block.

The command first opens the standard dialog box for the file selection. Double click the file to be inserted or mark it and click the Open button.

Command prompt

Specify insertion point or [Replace/?]:

Use the mouse or enter co-ordinates to specify the insertion point of the block or select an option. Once you have specified an insertion point, further options follow.

*You can replace an existing block with the option **Replace**. Further input requests follow.*

You call the help with the option ?.

Option Replace

Select block or [?]:

Select the block which you want to replace. Rotation, mirroring or scaling of the original block is retained.

Confirm the insertion or [Rotate/Move/Xmirror/Ymirror/XVaria/YVaria/Explode/?]:

*Confirm the insertion point with **ENTER** or **RIGHT CLICK** or select an option.*

*You can rotate the block about any angle with the **Option Rotate**. A further input request follows.*

*You can move the block with **Option Move**. A further input request follows.*

*With the option **Xmirror** you can mirror the block about its X axis.*

*With the option **Ymirror** you can mirror the block about its Y axis.*

*You can scale the block in the X direction with the **Option XVaria**.*

*You can scale the block in the Y direction with the **Option YVaria**.*

*You can insert the block in its single constituent parts with the option **Explode**.*

Option Rotate

Specify rotation angle or [?]:

Use the mouse or enter an angle to specify the rotation angle.

Option Move

Specify base point for the movement or [?]:

Use the mouse or enter co-ordinates to specify the base point of the movement.

Specify second point for the movement or [?]:

Use the mouse or enter co-ordinates to specify the second point for the movement.

Option XVaria

Specify X scale factor or [?] <1>:

Define the factor for scaling in the X direction.

Option YVaria

Specify Y scale factor or [?] <1>:

Define the factor for scaling in the Y direction.

Notes

Further information on inserting blocks or drawing files and on the standard dialog box for file selection can be found in the AutoCAD documentation.

9.15 Purge complete drawing



Ribbon:	Tab ATHENA > Group Extras > Purge complete drawing
Menu:	ATHENA > Block > Purge complete drawing
Toolbar:	ATH Block
Command input:	ath_bakt

Purges the drawing; optionally unselected objects are first removed before purging.

Command prompt

Selection [Objects/All/?] <Objects>:

*You can select objects with the **Option Objects**. A further input request follows.
With the option **All** the complete drawing is purged.
You call the help with the option **?**.*

Option Objects

The drawing is zoomed to the drawing limits and then zoomed with the scale factor 0.9.

Select objects:

*With the mouse select the objects which you want to leave in the drawing.
Objects which are not selected are removed, then the drawing is purged.*

9.16 Save user block



Ribbon:	Tab ATHENA > Group Extras > Save user block
Menu:	ATHENA > Block > Save user block
Toolbar:	ATH Block
Command input:	ath_uabl

With this command you save selected objects of the current drawing as a new file with the objective of inserting them as a block in other drawings. The files are saved as standard in the user folder. You can use user blocks temporarily or continuously.

Command prompt

When you execute the command, a standard dialog box opens in which you can define a file name. When you click the Save button, the following input request is displayed.

Select objects:

Select the objects which are to be constituent parts of the block.

Specify insertion point or [?]:

Specify the insertion point of the block.

You call the help with the option ?.

A new file is created and the selected objects are deleted from the current drawing.

Notes

- You define the path to the user folder in the ATHENA options. You will find further information in the Chapter *Paths* on page 74.
- You will find further information about creating drawing files in your AutoCAD documentation.

9.17 Insert user block



Ribbon:	Tab ATHENA > Group Extras > Insert user block
Menu:	ATHENA > Block > Insert user block
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_ulad

You use this command to insert a drawing from the user folder into the current drawing.

When you call the command, the AutoCAD standard dialog box is opened in which you can select a drawing file. Mark the file to be inserted with the mouse and then click the Open button. Alternatively, you can also double click the file.

Command prompt

Enter insertion point:

Use the mouse or enter coordinates to specify the insertion point.

Specify X factor <1>:

Specify the X factor.

Press the Enter key to accept the default value.

Specify Y factor <default=X>:

Specify the Y factor.

Press the Enter key to accept the default value.

Specify rotated angle<0.0>:

Specify the rotation angle.

Press the Enter key to accept the default angle.

Notes

- Further information on the subjects of inserting blocks or drawing files and the standard dialog box for file selection can be found in your AutoCAD documentation.
- You can set or change the user file path in the *ATHENA options*.

9.18 WBlock text



Ribbon:	Tab ATHENA > Group Extras > WBlock text
Menu:	ATHENA > Block > WBlock text
Toolbar:	ATH Block
Command input:	ath_bxbl

With this command you save a WBlock in the current folder with the intention of later inserting it in other drawings as XRef. The name of the WBlock is provided by clicking a text.

Command prompt

Select text for filename or [?]:

Click the text which you would like to use as the file name. You call the help with the option ?.

This input request appears when a file with this name already exists.

Filename already exists, replace it? [Yes/No/?] <Yes>:

*Select the option **Yes** to overwrite the existing file. Further input requests follow.*

*Select the option **No** to terminate the command without saving a WBlock.*

Specify insertion base point or [?]:

Use the mouse or enter co-ordinates to specify the base point of the WBlock.

Select objects:

With the mouse, select the objects which you want to save as a WBlock.

Notes

Further information on the subjects of creating drawing files and XRefs and their application can be found in your AutoCAD documentation.

9.19 WBlock text frame



Ribbon:	Tab ATHENA > Group Extras > WBlock text frame
Menu:	ATHENA > Block > WBlock text frame
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_xblk

With this command you save a WBlock in the current folder with the intention of later inserting it in other drawings as XRef. The name of the WBlock is provided by clicking a text. The object selection occurs by clicking on an enclosing polyline.

Command prompt

Select text for filename or [?]:

Click the text which you would like to use as the file name.

You call the help with the option ?.

Specify insertion base point or [Select/?] <Select>:

Use the mouse or enter coordinates to specify the base point.

*You can select an existing AutoCAD point as the base point with the **Option Select**.*

Option Select

Select point or [Point/?] <Point>:

Click on an existing AutoCAD point.

*You can specify an insertion base point with the option **Point**.*

Select the polyline around the objects or [?]:

Click the polyline which encloses the objects.

WBLOCK XXXXX generated with X objects.

Notes

- The insertion point and the enclosing polyline are saved for the WBlock so that when it is recreated only the text has to be clicked as the file name.
Important: The WBlock is overwritten without warning.
- Further information on the subjects of creating drawing files and XRefs and their application can be found in your AutoCAD documentation.

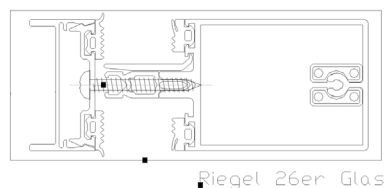


Fig. 9.3: WBlock text frame

9.20 Bind and insert XRefs



Ribbon:	Tab ATHENA > Group Extras > Bind and insert XRefs
Menu:	ATHENA > Block > Bind and insert XRefs
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_bind

With this command all XRefs can be bound and inserted into a drawing. If you want to pass drawings to other AutoCAD users, then you should bind and insert all the XRefs it contains. The XRefs are then saved in the drawings. You do not need to pass on the XRef files, so avoiding problems with the recipient due to incorrect path statements in the XRefs.

In contrast to the AutoCAD command XRef, with `ath_bind` all XRefs located in the drawing are integrated and inserted without calling a dialog box. Consequently, it is possible to couple this command with a script (see also the chapter *Create script* on page 622) to automate the binding and insertion of XRefs in a number of drawings. You will find further information about XRefs in the AutoCAD documentation.

Command prompt

Are all XRefs to be bound and inserted? [Yes/No/?] <Yes>:

Choose the option Yes to bind and insert all XRefs in the drawing.

Select the option No to terminate the command without binding and inserting XRefs.

You call the help with the option ?.

Notes

- If different XRefs contain symbols (blocks, layers, etc.) with the same name but different content, the content of the symbol in the first XRef is transferred to all other symbols with the same name.
- You will find further information about XRefs in your AutoCAD documentation.

9.21 Block manager



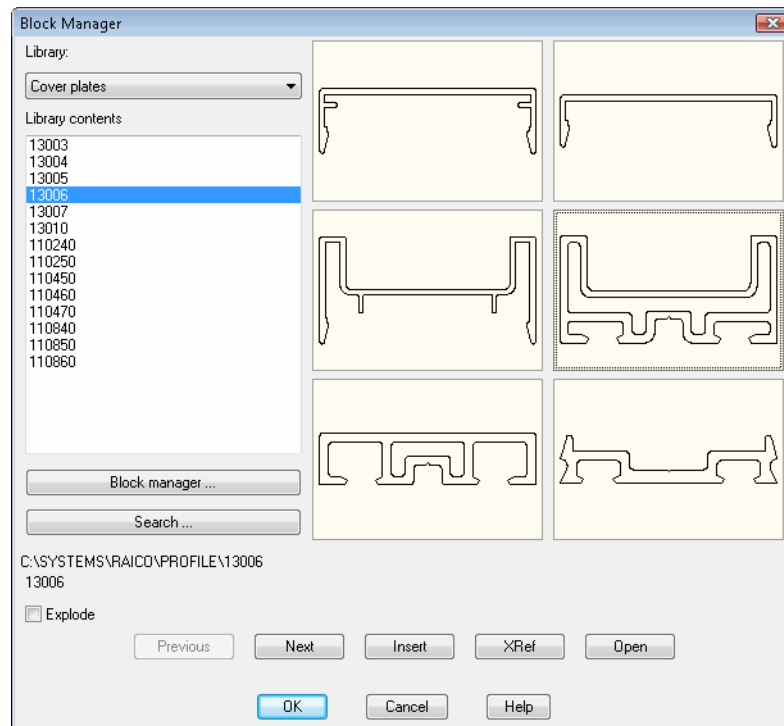
Ribbon: Tab ATHENA > Group Extras > Block manager
Menu: ATHENA > Block > Block manager
Toolbar: ATH Block
Command input: ath_bver

With this routine you can manage drawing files in block libraries. The drawings can be saved in various folders and are saved in a data base file.

The data base file in which the library/block definitions are saved is called blocklib.dat and is located in the directory ATHENA 2014\DATA\.

You can enter a comment for each block. Furthermore, the block manager offers you a preview and a Find function according to block names and comments.

Dialog box Block manager



When you select a library from the library list, all drawing files included in the library content are displayed. On the right side of the dialog box you can see a preview of them. A maximum of six previews can be displayed simultaneously. If your library contains more than six blocks, you can see the next or previous six blocks as a preview using the buttons Previous and Next.

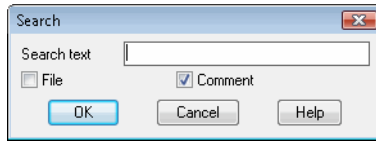
You can mark a drawing file by marking the name in the list or clicking the preview. The path of the marked drawing file is displayed under the button Search.

Click the Insert button to insert the marked drawing file as a block into the current drawing. To release the block on insertion, the Explode tick box must be activated.

Click the XRef button to insert the marked drawing file as an external reference in the current drawing.

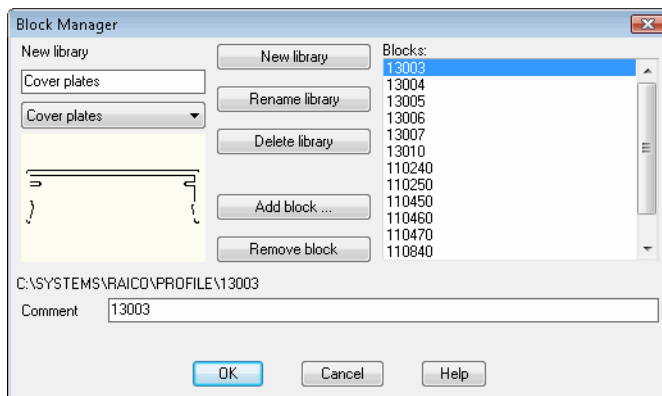
Click the Open button to open the marked drawing file.

Dialog box Search



Click the Search button to search for files in all libraries. In addition, you must specify a search term in the Search text dialog box. Also, you must define whether the search is to occur in files or comment fields. Click OK to start the search. If the search is successful, ATHENA creates a library with the name *SEARCH RESULT*, containing the found drawings. This library is overwritten with each search process.

Dialog box Block manager



To create a new library, enter the name in the input field and click the New library button.

To rename a library, select one from the list and click the Rename library button.

To delete a library, select one from the list and click the Delete library button.

To add a block to a library, select a library from the list and click the Add block button. Now you can select a drawing file with the standard dialog box for file selection. All blocks contained in the library are displayed to the right under Blocks. You can click a block name with the mouse to view a preview to the left.

To add a comment text, mark the required block name with the mouse and write the text in the Comment input line.

Click OK to save the changes and to return to the main dialog box.

9.22 Assign block label



Ribbon: Tab ATHENA > Group Label > Assign block label
Menu: ATHENA > Block > Assign block label
Toolbar: ATH Block and ATHENA Block
Command input: ath_blk_ldr

With this command you can assign label text to a block and configure it. The label texts are displayed when labeling the block with the command **Parts labeling**.

Command prompt

Select block or [?]:

Choose the block to which you want to assign label texts.

After the choice of block the Dialog box Assign block label is started.

Dialog box Assign block label

Write the label texts for the selected block in the appropriate input lines. At least one input line must be filled, otherwise the block name appears as the label text.

Labeling

Opens the Dialog box Label where you can configure the label settings for the selected block. You will find further information in the Chapter *Label* on page 117.

9.23 Block element visibility



Ribbon:	Tab ATHENA > Group Extras > Block element visibility
Menu:	ATHENA > Block > Block element visibility
Toolbar:	ATH Block and ATHENA Block
Command input:	ath_blk_vis

With this command you can control the visibility of dimensions, texts, hatching, and labels contained in blocks.

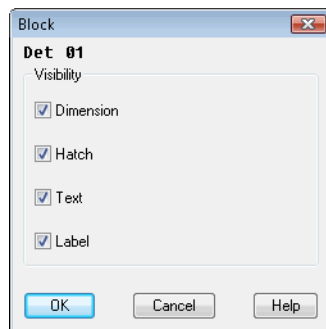
Command prompt

Select block or [?]:

Choose the block whose object visibility you want to change.

After the choice of block the Dialog box Block visibility is started.

Dialog box Block visibility



Dimensioning

Switches the visibility of dimensions contained in the block on or off.

Hatch

Switches the visibility of hatching contained in the block on or off.

Text

Switches the visibility of texts contained in the block on or off.

Labeling

Switches the visibility of labels contained in the block on or off.

10 Viewport

Commands in this section:

- Arrange viewports
- New viewport
- Reset
- Zoom Pan
- Zoom Max
- Change viewport
- Scale
- Zoom viewport
- Zoom limits

10.1 Arrange viewports



Ribbon:	Tab ATHENA > Group Drawing aids > Arrange viewports
Menu:	ATHENA > Viewport > Arrange viewports
Toolbar:	ATH Viewport
Command input:	ath_vp_deta

With this command you create aligned viewports in the selected scale from drawing sections which you define in the model space. ATHENA automatically changes to the layout section so that you can place the viewport.

Command prompt

Specify detail

Specify first point or [?]:

Specify the first corner point of the detail which is to be displayed in the first viewport.

You call the help with the option ?.

Specify second point or [?]:

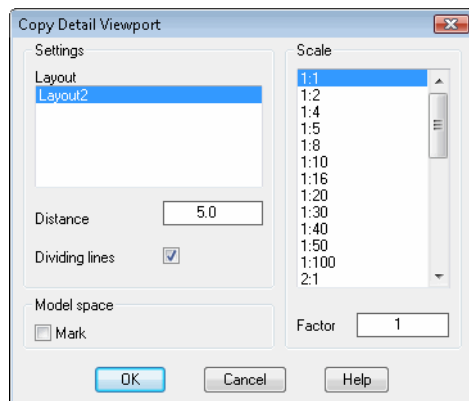
Specify the diagonal corner point of the first detail. The size of the rectangle which you then specify defines the size of the viewport which ATHENA creates.

Specify next detail

Specify the detail which is to be displayed in the next viewport. This input request is repeated so that you can specify further details.

When you press the Enter key, the interrogation is terminated and the Dialog box Detail - Copy viewport is started.

Dialog box Detail - Copy viewport



Dialog box section Settings

Layout

Defines the layout in which the viewports are produced.

Distance

Defines the spacing between the viewports.

Dividing lines

Turns dividing lines between the viewports on or off.

Dialog box section Scale

Specifies the scale of the viewport. In the input field Factor you can define a new scale factor. Here, the scales from the AutoCAD list of scales are made

available. You will find further information on this subject in the AutoCAD documentation.

Dialog box section Modeling section

Mark

Marks the position of the viewports in the modeling section.

When you click OK, the dialog box is closed. ATHENA changes to the specified layout and the input request follows:

Command prompt

Specify insertion point:

Specify the insertion point of the viewports.

Notes

Default layers are used for the viewport and interruption lines. You can adapt this layer in the Dialog box System layer. You will find information about this in the Chapter *Layer* on page 85.

10.2 New viewport

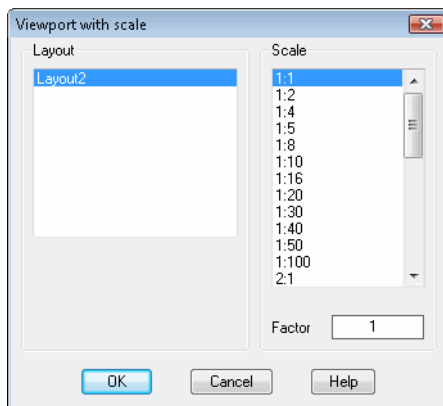


Ribbon:	Tab ATHENA > Group Drawing aids > New viewport
Menu:	ATHENA > Viewport > New viewport
Toolbar:	ATH Viewport
Command input:	ath_mf_vp_neu

You create a new viewport with scale using this command. The scale is set in the viewport using the zoom factor. For each new viewport ATHENA creates a dedicated dimension layer dependent on the scale.

When you execute the command, the Dialog box Viewport with scale is started.

Dialog box Viewport with scale



Layout

Defines the layout in which the viewport is to be produced.

Scale

Specifies the scale of the viewport. In the input field Factor you can define a new scale factor. Here, the scales from the AutoCAD list of scales are made available. You will find further information on this subject in the AutoCAD documentation.

When you close the dialog box with OK, the following input request appears:

Command prompt

Specify first point of window or [?]:

Use the mouse or enter coordinates to specify the first corner point of the viewport.

You call the help with the option ?.

Specify second point of viewport or [?]:

Use the mouse or enter coordinates to specify the second corner point of the viewport diagonally opposite.

Specify center point for viewport or [?]:

Determine the center point of the viewport. For this, first zooming to the drawing limits occurs in the viewport. Once you have specified the center point, the defined scale is set.

10.3 **Reset**



Ribbon:	Not present
Menu:	ATHENA > Viewport > Reset
Toolbar:	ATH Viewport
Command input:	ath_mf_vp_reset

With this command you zoom in all viewports of the active layout to the set scale. In addition the dimension layers are frozen depending on the window.

10.4 Zoom Pan



Ribbon:	Not present
Menu:	ATHENA > Viewport > Zoom Pan
Toolbar:	ATH Viewport
Command input:	ath_mf_vp_pan

With this command you can move the view in the current viewport. You can specify the movement points in another viewport.

Command prompt

Specify movement or [?]:

Use the mouse or enter coordinates to specify the first point of the movement.

You can specify this point in any viewport.

You call the help with the option ?.

Specify second point or [Center/?] <Center>:

Use the mouse or enter coordinates to specify the second point of the movement. You can specify this point in any viewport.

Press the Enter key to accept the first point as the center of the view.

10.5 Zoom Max



Ribbon:	Not present
Menu:	ATHENA > Viewport > Zoom Max
Toolbar:	ATH Viewport
Command input:	ath_mf_f_max

With this command you zoom the current viewport to the maximum screen size.
Use the command [Zoom limits](#) to zoom all viewports.

10.6 Change viewport



Ribbon:	Not present
Menu:	ATHENA > Viewport > Change viewport
Toolbar:	ATH Viewport
Command input:	

With this command you page between the viewports of the current layout. This is important, for example, when the viewports overlap.

10.7 Scale



Ribbon:	Tab ATHENA > Group Drawing aids > Scale
Menu:	ATHENA > Viewport > Scale
Toolbar:	ATH Viewport
Command input:	ath_mf_masstab

With this command you can change the scale of a viewport retrospectively. As with the command **New viewport**, ATHENA also creates here a new dimension layer dependent on the scale. Furthermore, all viewports of the current layout are zoomed to the set scale (see the command **Reset**).

For changes the Dialog box Select scale opens where you can define a scale for the viewport. You will find further information on this in the Chapter *Scale* on page 121.

10.8 Zoom viewport



Ribbon:	Not present
Menu:	ATHENA > Viewport > Zoom viewport
Toolbar:	ATH Viewport
Command input:	ath_mf_vp_zoom

With this command you zoom the current viewport to a new display. You can specify the rectangular window defining the zoom display in any viewport.

Command prompt

Specify first corner or [?]:

Use the mouse or enter coordinates to specify the first corner point of the zoom window. You can specify this point in any viewport.

You call the help with the option ?.

Select second corner or [?]:

Use the mouse or enter coordinates to specify the second corner point of the zoom window diagonally opposite.

10.9 Zoom limits



Ribbon:	Not present
Menu:	ATHENA > Viewport > Zoom limits
Toolbar:	ATH Viewport
Command input:	ath_mf_f_ruck

With this command you zoom all viewports in the current layout. To do this you do not need to quit the active viewport.

11 Layer

Commands in this section:

- Freeze layer
- Layer Off
- Lock layer
- Thaw layer
- Layer On
- Unlock layer
- Change layer
- Delete layer
- Objects invisible
- Objects visible

11.1 Freeze layer



Ribbon:	Tab ATHENA > Group Layer > Freeze layer
Menu:	ATHENA > Layer > Freeze layer
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_lfri

With this routine you freeze a layer by clicking an object on the required layer. Optionally, you can freeze all layers except the selected one. You will find further information about frozen layers in your AutoCAD documentation.

Command prompt

Select object for freezing or [All except?]:

Use the mouse to select an object on the layer to be frozen.

*You can freeze all layers except the selected one with the **Option All except**.*

You call the help with the option ?.

Option All except

Select object:

Use the mouse to select an object possessed by the layer which is not to be frozen. All other layers will be frozen.

Notes

- ATHENA displays the layer name of the selected object in the command line.
- You cannot freeze the current layer. If you select an object which has the current layer, the following message appears in the command line:
XXX = Current layer. It cannot be frozen!
- Use the command **Thaw layer** to thaw out frozen layers.

11.2 Layer Off



Ribbon:	Tab ATHENA > Group Layer > Layer Off
Menu:	ATHENA > Layer > Layer Off
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_laos

With this routine you switch a layer off by clicking an object on the required layer. Optionally, you can switch all layers off except the selected one. You will find further information about switching layers off in your AutoCAD documentation.

Command prompt

Select object for switching off or [All except?]:

Use the mouse to select an object on the layer to be switched off.

*You can switch off all layers except the selected one with the **Option All except**.*

You call the help with the option ?.

Option All except

Select object:

Use the mouse to select an object possessed by the layer which is not to be switched off. All other layers will be switched off.

Notes

- ATHENA displays the layer name of the selected object in the command line.
- You cannot switch off the current layer. If you select an object which has the current layer, the following message appears in the command line:
XXX = Current layer. It cannot be switched off!
- Use the command **Layer On** to switch layers on.

11.3 Lock layer



Ribbon:	Tab ATHENA > Group Layer > Lock layer
Menu:	ATHENA > Layer > Lock layer
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_lspe

With this routine you can lock a layer for editing by clicking an object on the required layer. Optionally, you can lock all layers except the selected one. You will find further information about locking layers in your AutoCAD documentation.

Command prompt

Select object for locking or [All except/?]:

Use the mouse to select an object on the layer to be locked.

*You can lock all layers except the selected one with the **Option All except**.*

You call the help with the option ?.

Option All except

Select object:

Use the mouse to select an object possessed by the layer which is not to be locked. All other layers will be locked.

Notes

- ATHENA displays the layer name of the selected object in the command line.
- You cannot lock the current layer. If you select an object which has the current layer, the following message appears in the command line:
XXX = Current layer. It cannot be blocked!
- Use the command **Unlock layer** to unlock the layer.

11.4 Thaw layer



Ribbon:	Tab ATHENA > Group Layer > Thaw layer
Menu:	ATHENA > Layer > Thaw layer
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_Itau

With this routine you thaw out one or more frozen layers.

Command prompt

Which layers to thaw <>:*

Press the Enter key to thaw all frozen layers or enter the name of the layer which is to be thawed. You can enter the name of the layer or select it on the tablet.

Notes

- You can thaw out a number of specified layers by writing the layer names in the command line separated by a comma, e.g. 0,1-0,2-0,3-0.
- To freeze layers you can use the command **Freeze layer**.

11.5 Layer On



Ribbon:	Tab ATHENA > Group Layer > Layer On
Menu:	ATHENA > Layer > Layer On
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_lein

With this routine you switch one or more frozen layers on.

Command prompt

Which layer to be switched on <>:*

Press the Enter key to switch on all layers that are switched off or enter the name of the layer which is to be switched on. You can enter the name of the layer or select it on the tablet.

Notes

- You can switch on a number of specified layers by writing the layer names in the command line separated by a comma, e.g. 0,1-0,2-0,3-0.
- To switch layers off you can use the command **Layer Off**.

11.6 Unlock layer



Ribbon:	Tab ATHENA > Group Layer > Layer On
Menu:	ATHENA > Layer > Unlock layer
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_lesp

With this routine you unlock one or more locked layers.

Command prompt

Which layers to unlock <>:*

Press the Enter key to unlock all locked layers or specify the name of the layer which is to be unlocked. You can enter the name of the layer or select it on the tablet.

Notes

- You can unlock a number of specified layers by writing the layer names in the command line separated by a comma, e.g. 0,1-0,2-0,3-0.
- To lock layers you can use the command **Lock layer**.

11.7 Change layer



Ribbon:	Tab ATHENA > Group Layer > Change layer
Menu:	Tab ATHENA > Layer > Change layer
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_laen

With this command you can change the layer of any object. The new layer is specified by clicking an object or by stating the layer name.

Command prompt

Select objects:

Use the mouse to choose the objects whose layer you wish to change.

Indicate object on new layer or [Specify/?] <Enter>:

Using the mouse select an object whose layer you wish to adopt.

*With the **Option Specify** you can specify a layer name.*

You call the help with the option ?.

Option Specify

New layer:

Enter a layer name.

Notes

This command does **not** function with interrupted dimensions if in the ATHENA options the option Use ATHENA dimension layer is activated.

11.8 Delete layer



Ribbon:	Not present
Menu:	ATHENA > Layer > Delete layer
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_ladl

With this command you can remove all elements which have the same layer as the selected object from the current drawing.

Command prompt

Define layer:

Select an object which has the same layer as the objects to be deleted. ATHENA now deletes all objects which have been drawn on the selected layer.

11.9 Objects invisible



Ribbon:	Tab ATHENA > Group Drawing aids > Objects invisible
Menu:	ATHENA > Layer > Objects invisible
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_over

With this routine you can render individual objects invisible independent of the layer.

Command prompt

Objects invisible [All except/Select objects]<Select objects>:

Confirm the input request to select objects which are to be rendered invisible.

Select the option All except if you want to select objects which are not to be rendered invisible.

Select objects:

Use the mouse to choose the objects.

Notes

- The number of invisible objects is displayed in the status line if you have activated the tick box Layer info in the status section in the options (see Chapter *Options* on page 65).
- Use the command **Objects visible** to render all invisible objects visible.

11.10 Objects visible



Ribbon:	Tab ATHENA > Group Drawing aids > Objects visible
Menu:	ATHENA > Layer > Objects visible
Toolbar:	ATH Layer and ATHENA Layer
Command input:	ath_oruk

With this routine you can render visible all invisible objects in the current drawing.

Command prompt

x objects have been restored.

ATHENA displays how many invisible objects have been restored.

Notes

- The number of invisible objects is displayed in the status line if you have activated the tick box Layer info in the status section in the options (see Chapter *Options* on page 65).
- Use the command **Objects invisible** to render objects invisible.

12 Service

Commands in this section:

- Load layer
- Manage system variables
- Set default
- Purge layer definition
- Deactivate ATHENA objects
- Activate ATHENA objects
- Show limits
- Zoom limits
- Export items
- Import items
- Plotting
- Adjust plot script files
- Create script
- Lock standards
- ATHENA Filer

12.1 Load layer



Ribbon:	Tab ATHENA > Layer group > Load layer
Menu:	ATHENA > Service > Load layer
Toolbar:	ATH Service and ATHENA Standard
Command input:	ath_layreset

With this command you can reload the defined layers in the current drawing. This is necessary under some circumstances when you receive a drawing from a customer and you wish to use your own layers.

Notes

- Existing layers with the same names are overwritten. This means that their properties change (e.g. the color).
- You can carry out modifications to the layer settings in the Dialog box System layer. You will find detailed information in the Chapter *Layer* on page 85.

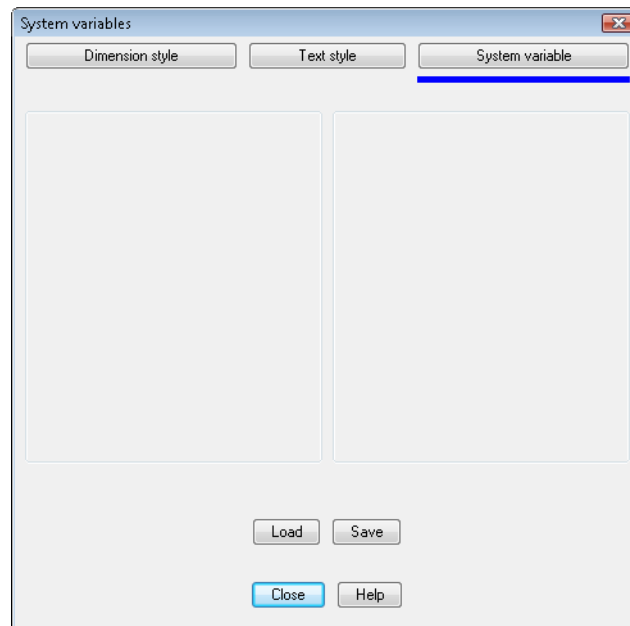
12.2 Manage system variables



Ribbon:	Tab ATHENA > Extras group > Manage system variables
Menu:	ATHENA > Service > Manage system variables
Toolbar:	ATH Service and ATHENA Standard
Command input:	ath_sysvar

With this command you can save and load defined settings through system variables. Loading system settings is practicable when you receive drawings from other people and want to use your own settings.

Dialog box System variables



Register button Dimension style

Activates the Dialog box Dimension style.
You will find the functional description in the section *Manage dimension styles* on page 454.

Register button Text style

Activates the Dialog box Text style.
You will find the functional description in the section *Manage text styles* on page 526.

Register button System variable

Activates the Dialog box System variables.
You will find the functional description below.

Load

Click the button Load to load the system settings from the file `ath_var.dex` into the current drawing.

Save

Click the button Save to save the system settings of the current drawing to the file `ath_var.dex`.

Notes

The system settings are saved in the file `ATHENA\DATALOCAL\ath_var.dex`. You will find further information in the Chapter *AutoCAD system variables* on page 57.

12.3 Set default



Ribbon:	Not present
Menu:	ATHENA > Service > Set default
Toolbar:	ATH Service
Command input:	ath_vorg

With this routine you save the properties of the selected object as a default setting. When you then start the appropriate command, the properties of the previously selected object are displayed in the dialog box.

You can use this command with all objects which have been inserted into the drawing via a dialog box (e.g. sheet, insulation, standard part ...).

Command prompt

Select object:

Using the mouse select the object whose properties are to be saved as default.

Default has been set

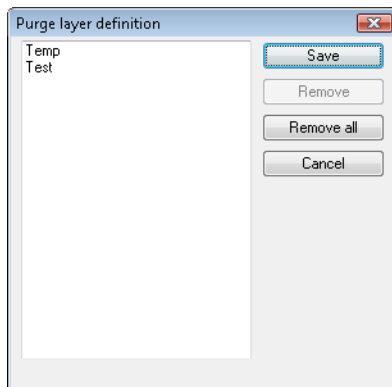
12.4 Purge layer definition



Ribbon:	Tab ATHENA > Layer group > Purge layer definition
Menu:	ATHENA > Service > Purge layer definition
Toolbar:	ATH Service
Command input:	ath_layer_purge

With this command you can purge unreferenced layer definitions. Unreferenced layer definitions are not used as Systemlayer nor as a layer for Materialeigenschaften.

Dialog box Purge layer definition



All unreferenced layers are displayed in the left section of the dialog box. You can mark one or more layers with the mouse and delete them by clicking the button Remove. The button Remove all deletes all (also those not marked) layers from the list. The changes only become effective when you click the button Save. With Cancel, the changes are discarded.

Notes

Further information about layers and layer assignments can be found in the sections *Layer* on page 85 and *Material* on page 81.

12.5 Deactivate ATHENA objects



Ribbon:	Tab ATHENA > Extras group > Deactivate ATHENA objects
Menu:	ATHENA > Service > Deactivate ATHENA objects
Toolbar:	ATH Service and ATHENA Standard
Command input:	ath_deactivate

With this command you deactivate all ATHENA ARX objects. All ARX objects in the drawing are automatically converted to anonymous blocks.

You can, for example, use this command before you pass on a drawing to a customer who uses different CAD software.

Notes

- You can activate ATHENA objects again. For this use the command [Activate ATHENA objects](#).
- Also, deactivated objects can be labeled with a leader using the command [Parts labeling](#).

12.6 Activate ATHENA objects



Ribbon:	Tab ATHENA > Extras group > Activate ATHENA objects
Menu:	ATHENA > Service > Activate ATHENA objects
Toolbar:	ATH Service and ATHENA Standard
Command input:	<code>ath_activate</code>

With this command you activate all ATHENA ARX objects in the current drawing which were previously deactivated.

Notes

You can deactivate ATHENA objects. For this use the command *Deactivate ATHENA objects*.

12.7 Show limits



Ribbon:	Not present
Menu:	ATHENA > Service > Show limits
Toolbar:	ATH Service
Command input:	ath_zlim

With this command you display the drawing limits using yellow temporary lines.
This display is deleted when regenerating or redrawing the view.

Command prompt

Limiting size: 1189.0 x 841.0.

The limiting size is shown in the command line.

12.8 Zoom limits



Ribbon:	Not present
Menu:	ATHENA > Service > Zoom limits
Toolbar:	ATH Service
Command input:	ath_zoli

With this command you zoom to the limits of the current drawing. A once-only screen formation takes place.

With this zoom command the second regeneration of the drawing is, in contrast to the commands Zoom extents or Zoom all, omitted.

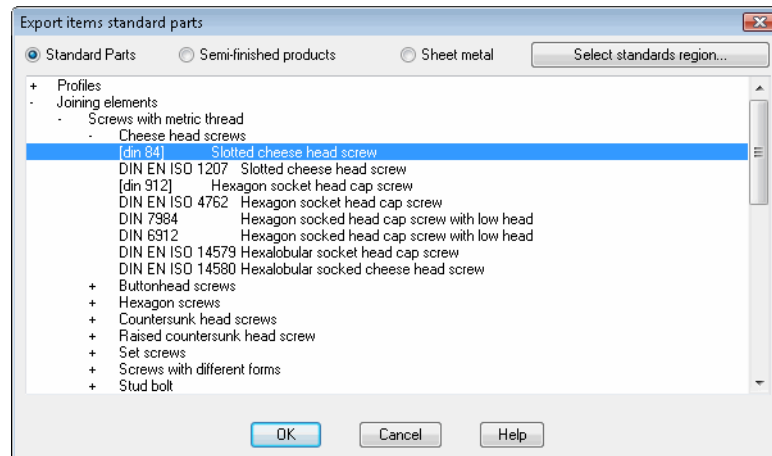
12.9 Export items



Ribbon: Tab ATHENA > Extras group > Export items
Menu: ATHENA > Service > Export items
Toolbar: ATH Service
Command input: ath_nm_export

With this function you export a group of standard parts to be selected to the clipboard.

Dialog box Export items standard parts



Standard parts

Selects standard parts for item export.

Semi-finished products

Selects semi-finished products for item export.

Sheet metal

Selects sheet metal for item export.

Select standards regions

Opens the Tab Standards regions, where you can define from which regions the standards are to be displayed.

List

Shows the available standard parts in a tree structure. Here you can select the required standard with the mouse. Closed branches (standards groups) in the tree structure are identified with +. Open branches are identified with -.

Click OK to execute the export. ATHENA returns in a dialog box the number of records written.

Then insert the content of the clipboard into any spread-sheet program (e.g. Microsoft Excel) so that you can add your own item numbers in column C. Enter an * (asterisk) to remove an existing item number.

In the other columns you can supplement any information required for internal use.



Column B must not be changed, because it involves the standard part code. During import ATHENA only takes into account columns B and C.

	A	B	C	D	E	F
7	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø10.2x1.4\	ART 123 465	<DIN EN	Rundes na	ø10.2x1.4
8	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø10.2x1.6\	ART 123 466	<DIN EN	Rundes na	ø10.2x1.6
9	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø10.2x1.8\	ART 123 467	<DIN EN	Rundes na	ø10.2x1.8
10	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø10.2x2.0\	ART 123 468	<DIN EN	Rundes na	ø10.2x2.0
11	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø10.2x2.3\	ART 123 469	<DIN EN	Rundes na	ø10.2x2.3
12	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø10.2x2.6\	ART 123 470	<DIN EN	Rundes na	ø10.2x2.6
13	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x0.5\	ART 123 471	<DIN EN	Rundes na	ø12x0.5
14	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x0.6\	ART 223 465	<DIN EN	Rundes na	ø12x0.6
15	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x0.8\	ART 223 466	<DIN EN	Rundes na	ø12x0.8
16	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x1.0\	ART 223 467	<DIN EN	Rundes na	ø12x1.0
17	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x1.2\	ART 223 468	<DIN EN	Rundes na	ø12x1.2
18	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x1.4\	ART 223 469	<DIN EN	Rundes na	ø12x1.4
19	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x1.6\	ART 223 470	<DIN EN	Rundes na	ø12x1.6
20	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x1.8\	ART 223 471	<DIN EN	Rundes na	ø12x1.8
21	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x2.0\	ART 223 472	<DIN EN	Rundes na	ø12x2.0
22	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x2.3\	ART 223 473	<DIN EN	Rundes na	ø12x2.3
23	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x2.6\	ART 223 474	<DIN EN	Rundes na	ø12x2.6
24	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x2.9\	ART 223 475	<DIN EN	Rundes na	ø12x2.9
25	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12x3.2\	ART 223 476	<DIN EN	Rundes na	ø12x3.2
26	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12.7x0.5\	ART 223 477	<DIN EN	Rundes na	ø12.7x0.5
27	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12.7x0.6\		<DIN EN	Rundes na	ø12.7x0.6
28	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12.7x0.8\		<DIN EN	Rundes na	ø12.7x0.8
29	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12.7x1.0\		<DIN EN	Rundes na	ø12.7x1.0
30	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12.7x1.2\		<DIN EN	Rundes na	ø12.7x1.2
31	R2008	DIN\INDIN02448\INDIN2448\STEEL\ø12.7x1.4\		<DIN EN	Rundes na	ø12.7x1.4

Fig. 12.1: Standard parts group imported into Excel.

Notes

- If you use a word processor or a text editor instead of a spread-sheet program, the columns are separated by tabulators.
- Use the command **Import items** to import groups of standard parts into ATHENA.
- You will find information about the clipboard in your Windows documentation.

12.10 Import items



Ribbon: Tab ATHENA > Extras group > Import items
Menu: ATHENA > Service > Import items
Toolbar: ATH Service
Command input: ath_nm_import

With this function you import a group of standard parts from the clipboard into ATHENA.

In your spread sheet mark the standard parts which you wish to import into ATHENA and copy them to the Windows clipboard (ATHENA only takes columns A and B into account during import). Then start the command **Import items** to import the new item numbers into ATHENA. ATHENA returns the number of items transferred.

	A	B	C	D	E	F
7	R2008	DIN\DIN02448\DIN2448\STEEL\ø10.2x1.4\	ART 123 465	<DIN EN	Rundes na	ø10.2x1.4
8	R2008	DIN\DIN02448\DIN2448\STEEL\ø10.2x1.6\	ART 123 466	<DIN EN	Rundes na	ø10.2x1.6
9	R2008	DIN\DIN02448\DIN2448\STEEL\ø10.2x1.8\	ART 123 467	<DIN EN	Rundes na	ø10.2x1.8
10	R2008	DIN\DIN02448\DIN2448\STEEL\ø10.2x2.0\	ART 123 468	<DIN EN	Rundes na	ø10.2x2.0
11	R2008	DIN\DIN02448\DIN2448\STEEL\ø10.2x2.3\	ART 123 469	<DIN EN	Rundes na	ø10.2x2.3
12	R2008	DIN\DIN02448\DIN2448\STEEL\ø10.2x2.6\	ART 123 470	<DIN EN	Rundes na	ø10.2x2.6
13	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x0.5\	ART 123 471	<DIN EN	Rundes na	ø12x0.5
14	R2008	DIN\DIN02448\	ART 223 465	<DIN EN	Rundes na	ø12x0.6
15	R2008	DIN\DIN02448\	ART 223 466	<DIN EN	Rundes na	ø12x0.8
16	R2008	DIN\DIN02448\	ART 223 467	<DIN EN	Rundes na	ø12x1.0
17	R2008	DIN\DIN02448\	ART 223 468	<DIN EN	Rundes na	ø12x1.2
18	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x1.4\	ART 223 469	<DIN EN	Rundes na	ø12x1.4
19	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x1.6\	ART 223 470	<DIN EN	Rundes na	ø12x1.6
20	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x1.8\	ART 223 471	<DIN EN	Rundes na	ø12x1.8
21	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x2.0\	ART 223 472	<DIN EN	Rundes na	ø12x2.0
22	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x2.3\	ART 223 473	<DIN EN	Rundes na	ø12x2.3
23	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x2.6\	ART 223 474	<DIN EN	Rundes na	ø12x2.6
24	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x2.9\	ART 223 475	<DIN EN	Rundes na	ø12x2.9
25	R2008	DIN\DIN02448\DIN2448\STEEL\ø12x3.2\	ART 223 476	<DIN EN	Rundes na	ø12x3.2
26	R2008	DIN\DIN02448\DIN2448\STEEL\ø12.7x0.5\	ART 223 477	<DIN EN	Rundes na	ø12.7x0.5
27	R2008	DIN\DIN02448\DIN2448\STEEL\ø12.7x0.6\		<DIN EN	Rundes na	ø12.7x0.6
28	R2008	DIN\DIN02448\DIN2448\STEEL\ø12.7x0.8\		<DIN EN	Rundes na	ø12.7x0.8
29	R2008	DIN\DIN02448\DIN2448\STEEL\ø12.7x1.0\		<DIN EN	Rundes na	ø12.7x1.0
30	R2008	DIN\DIN02448\DIN2448\STEEL\ø12.7x1.2\		<DIN EN	Rundes na	ø12.7x1.2
31	R2008	DIN\DIN02448\DIN2448\STEEL\ø12.7x1.4\		<DIN EN	Rundes na	ø12.7x1.4

Fig. 12.2: Copying the items to the clipboard.

Notes

- Use the command **Export items** to export groups of standard parts.
- You will find further information about the procedure in the Chapter *Export items* on page 617.
- You will find information about the clipboard in your Windows documentation.

12.11 Plotting



Ribbon:	Tab ATHENA > Extras group > Plot 01 to Plot 06
Menu:	ATHENA > Service > Plot 01 to Plot 06
Toolbar:	ATH Service and ATHENA Standard
Command input:	ath_plot01 - ath_plot06

You can plot drawings in an uncomplicated manner with these plotting commands. The routine automatically detects the orientation (portrait or landscape format) of the drawing and takes it into account during plotting. You can serve six different output media (printers or plotters with different formats and plotting styles). For this you must appropriately configure the plot script files (see Chapters *Plotter configuration* on page 61 and *Adjust plot script files* on page 621).

Command prompt

Specify first corner or [?] <0.0>:

Use the mouse or enter coordinates to specify the first corner of the plotting area.

Press the Enter key to accept the default values.

Specify second corner or [?] <1189.841>:

Use the mouse or enter coordinates to specify the second corner of the plotting area.

Press the Enter key to accept the default values.

Enter the setting for shaded plot [As shown/ Wire frame/ Hidden / Rendered] <As shown>:

Use the option As shown to plot the view as displayed.

Use the option Wire frame to plot the view as a wire frame.

Use the option Hidden to leave the hidden lines unplotted.

Use the option Rendered to plot the view rendered.

Select scale: or [Adapt/?] <Adapt>:

At this point a context menu is opened where you can select the plotting scale.

Press the Enter key to enter a scale (e.g. 2 for 2:1 or 0.5 for 1:2). Press the Enter key again to accept the default scale.

Execute plotting [Ok/Exit]<OK>:

Press the Enter key to start the plotting process.

*Select the option **Exit** to cancel plotting.*

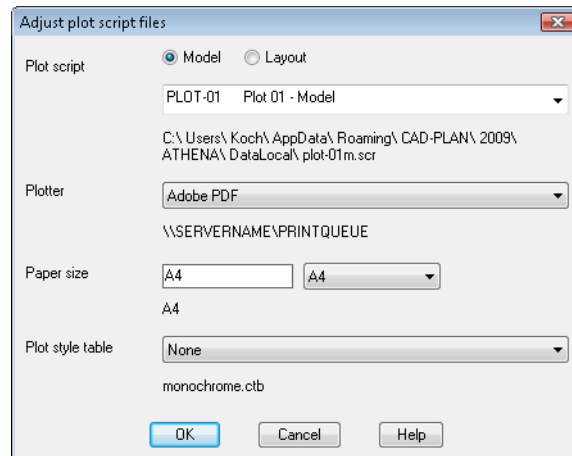
12.12 Adjust plot script files



Ribbon: Tab ATHENA > Extras group > Adjust plot script
Menu: ATHENA > Service > Adjust plot script
Toolbar: ATH Service
Command input: ath_plotscr

With this command you can set up the plot script files for the ATHENA plot function conveniently in a dialog box.

Dialog box Adjust plot script files



Using the option buttons choose whether the script files are to be displayed for the model space or the layout section.

Select the plot script file which you want to adapt from the list Plot script. The file path is shown below the pick list.

Choose an output device from the Plotter list. The currently assigned plotter is shown under the list.

Choose a paper format from the list. The currently assigned paper format is shown below the pick list.

Choose a plot style table from the list. The currently assigned plot style table is shown below the pick list.

Repeat these steps for all available plot script files. Click OK to save the plot script files. Click Cancel to discard the settings.

Notes

- Plot script files can be copied. You will find further information in the Chapter *Plotter configuration* on page 61.
- When saving plot script files, a backup copy (plot*.bak) is produced.

12.13 Create script



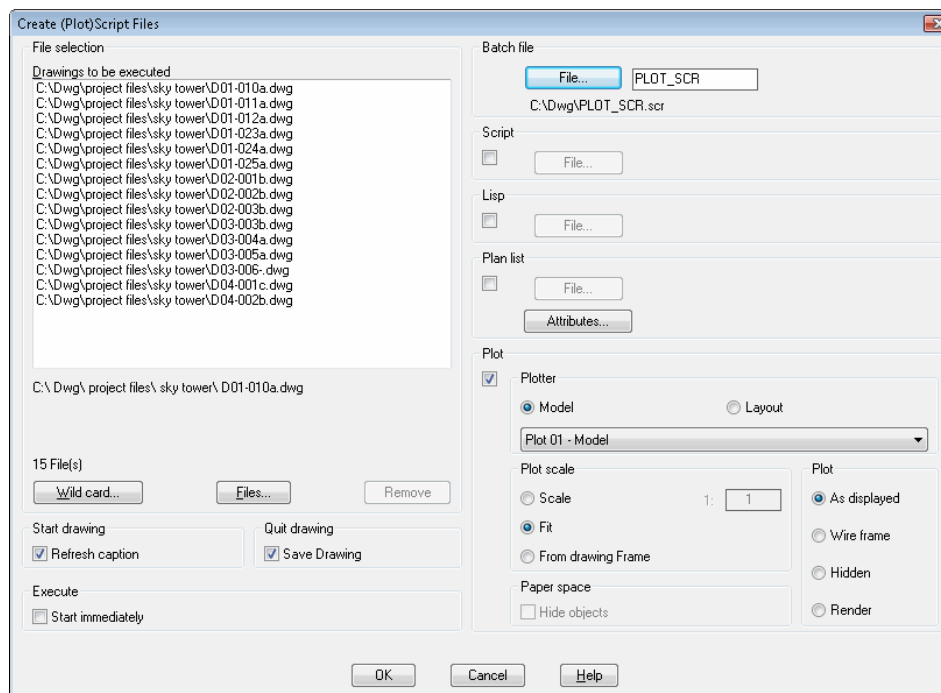
Ribbon:	Tab ATHENA > Extras group > Create script
Menu:	ATHENA > Service > Create script
Toolbar:	ATH Service
Command input:	ath_batch

With this command you can create script files to automate commands or command sequences in a number of files. For example, you can print out or clean up all the files in a folder or generate a drawing list.

If you wish to use the batch plotting function, you must first configure the plot script files. You will find further information on this in the Chapter *Plotter configuration* on page 61.

If you wish to execute commands or command sequences, you must first define the operating steps in a file as Lisp or script instructions. You can find script files for immediate use in the folder \ATHENA\SAMPLE\JOB_SCR.

Dialog box Create (plot) script files



Dialog box section File selection

In the field Drawings to be executed all files selected for batch processing are listed.

When you click the button Wildcard ..., a dialog box opens in which you can define files using the place holders (* or ?).

With the button Files ... you can select one or more (with held down CTRL or SHIFT key) files in a dialog box.

With the Remove button you can delete the marked files from the list Drawings to be executed. If you have not marked any file, this button is masked out.

Dialog box section Start drawing

Activate the tick box Refresh caption if the current date is to be written to the caption. In this respect certain conditions are required; you can obtain further information in the section *Insert drawing frame* on page 552.

Dialog box section Quit drawing

Activate the tick box when you want to save the drawing after the batch processing.

Dialog box section Execute

When you activate Start immediately, the script is started immediately after terminating the dialog box with OK.

Dialog box section Batch file

Here you can enter the file name for the script file which links the operating instructions to the executing files. The script file is saved in the current folder. If you want to change the saving location, you must click the button File A standard dialog box is opened where you can change the saving location.

Dialog box section Script

Activate the tick box when you want to link a script to the executing files. Click the button File to select the script file with the operating instructions.

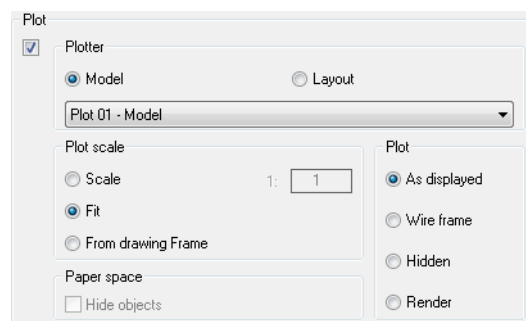
Dialog box section Lisp

Activate the tick box when you want to link a Lisp file to the executing files. Click the button File to select the Lisp file with the operating instructions.

Dialog box section Plan list

Activate the tick box when you want to generate a plan list of the executing files. Click the button File to specify the saving location and file name for the list. Click Attributes ... to define which plan details are to appear in the list. To do this the Dialog box Read out caption is started. You will find further information in the Chapter *Read out caption* on page 557.

Dialog box section Plot



Activate the tick box to plot the drawings to be executed.

In the section Plotter you define the section to be printed (model or current layout). Furthermore, you can choose an output device from the list.

In the section Plot scale you can specify a plotting scale with the option Scale. With the option Fit the plotting scale is adapted to the current page size. With the option From drawing frame the plotting scale is read from the scaling of the

drawing frame. We recommend the option From drawing frame for large-format plotting scripts if the model space is to be plotted.



To be able to use this option, the drawing frame must fulfill certain requirements. You will find further information in the Chapter *Insert drawing frame* on page 552.

Click OK to create the script file. Click Cancel to discard the settings and close the dialog box.

Notes

Use the AutoCAD command `_script` to start a script. You will find further information about scripts in the AutoCAD documentation.

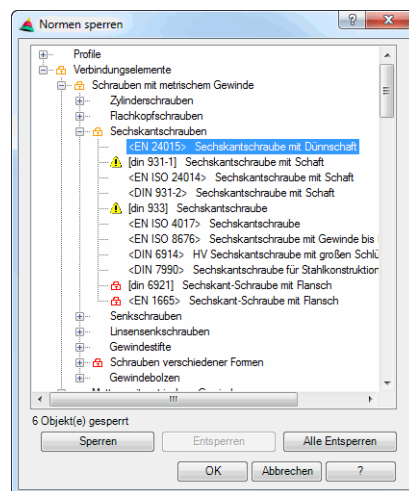
12.14 Lock standards

Ribbon:	Not present
Menu:	Not present
Toolbar:	Not present
Command input:	ath_norm_lock

Locks standards groups and standards which are not needed.

Locked standards are not offered for selection in the Dialog box Standard Parts. This reduces the display to standards and standards groups which you actually need.

Dialog box Lock standards



List of standards

Presents the list of all existing standards groups and standards in a tree structure.

You open the relevant branch by clicking on [+]. A click on [-] closes the opened branch. You can select a single standard or a complete group of standards to lock or unlock them.

Locked standards are marked with a symbolic padlock. A red padlock indicates: This standards group or standard is locked. A yellow padlock indicates: This standards group includes locked standards.

Outdated or withdrawn standards are still available for reasons of compatibility and are marked with an exclamation mark.

Lock

Locks the selected standards group or standard.

Unlock

Unlocks the selected standards group or standard.



You can only unlock the standards, which were directly locked. This means those which are marked with a red padlock.

Unlock all

Unlocks all locked standards groups and standards.

End of program

If you quit the dialog box with OK, the settings which have been made are saved.
If you do not want to save the settings, press the Cancel button.

12.15 ATHENA Filer

Ribbon:	Not present
Menu:	Not present
Toolbar:	Not present
Command input:	ath_filer

The ATHENA filer is no longer documented in this version. For reasons of compatibility it is still available, but must be activated with the command `ath_filer`.

Command prompt

Activate filer [On/Off] <Off>:

*Choose the option **Yes** to activate the filer.*

*Choose the option **No** to deactivate the filer.*

Produce a slide on saving [On/Off] <Off>:

*Use the option **On** to cause the filer to produce a slide on saving drawings.*

*Use the option **Off**, if the filer is not to produce any slides when saving.*

If you have activated the filer, you can start it with the command `ath_ende`.

G Modeling Command Reference

This section explains the functions of ATHENA 2014 for designing mainly in the third dimension.

The chapters are subdivided into functional sections. The functional sections correspond to the submenus in the Modeling pull-down menu.

1 **Manager**

Commands in this section:

- Bar Assembly Manager
- Infill manager
- Wall manager
- Frame Element Manager
- Assembly Manager
- Bar joint manager
- Process manager
- Arrangement manager
- Display modes
- Visibility of bar components
- Model Inspection
- Visualize axis model

1.1 Bar Assembly Manager



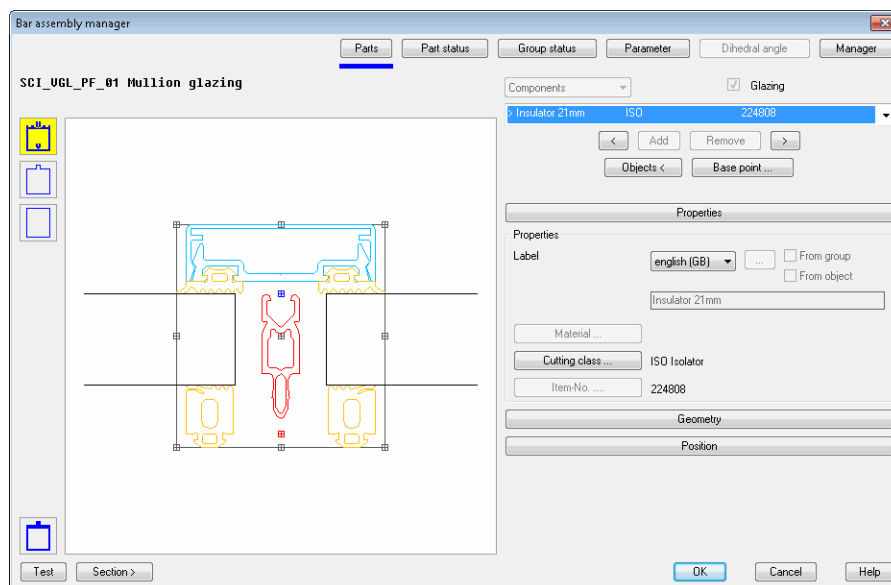
Ribbon: Tab ATHENA > Group Model > Bar assembly manager
Menu: Modeling > Manager > Bar assembly manager
Toolbar: ATH Manager
Command input: ath_bar_edit

This function serves in creating and editing qualified bar assemblies, which can be used as a bar in space (3D) or as a section in the drawing.

Bar assemblies can consist of one or more component parts and be provided with additional properties. These are in turn dependent on the type of bar assembly. The following types are possible:

1. Bar assemblies without further intelligence, i.e. those which only contain outline information.
2. Bar assemblies which are formed from references. References are pointers to other bar assemblies (similar to external references in drawings). The following variants and combinations can be defined from references.
 - Simple variants. Application example: Mullions with or without mullion reinforcement.
 - Glazing, i.e. assemblies whose components are modified or displaced depending on the glass thickness. Application example: Transoms in which variously thick panes can be fitted.
 - Variable dihedral angles. Application example: Variable corner mullions.
3. Bar assemblies which include stretchable outlines. Application example: Wooden mullions (stretchable) with aluminum attachments.
4. Profiles with stiffening (zig-zag or traverse).

Dialog box Bar Assembly Manager



Display section

In the upper section the name of the bar assembly is displayed if available.

The preview of the current bar assembly is dynamic. The active component is shown in red as standard. Inactive parts are displayed in their object color (for example, the color of the assigned material). Furthermore, a rectangle is displayed which encloses all components in the bar assembly.

The preview can be displayed in various ways:



Full display

Shows the complete components in the preview.



Cutting outline

Shows the cutting outlines of the components in the preview. If no cutting outline has been defined, the outer outline is displayed.



Simplified outline

Shows the simplified outlines of the components in the preview.



The simplified outline displays components of up to eight sides. With parts with more than eight sides the enclosing rectangle is shown.



Settings for section

Opens the Dialog box Bar cross section, where you can define settings for the section. You will find further information in the section *Bar cross section* on page 153.

Test

Opens the Dialog box Parameters for bar assembly, where you can test the functions of the assembly. You will find further information in the section *Parameters for bar assembly* on page 140.

Section >

Inserts the current bar assembly as a section into the current drawing. For this, the dialog box is temporarily closed and the input request follows:

Command prompt

Specify insertion point:

Define the insertion point of the section.

Specify rotated angle<0>:

Define the rotation angle of the section. You can accept the default angle with Enter or a right click.



If the bar assembly includes variants, the Dialog box Parameters for bar assembly precedes the insertion.

Operating section

The operating section of the Bar assembly manager contains the following sections:

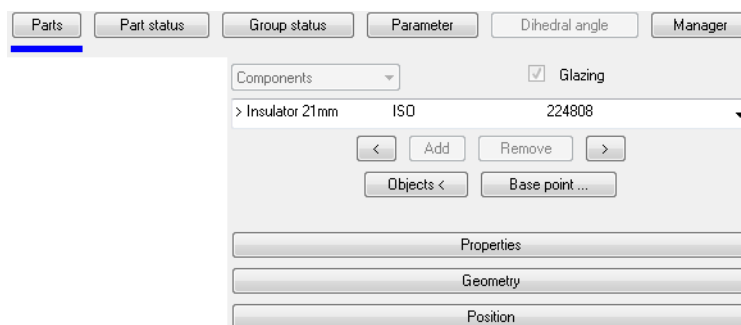
- Register button Parts:
In this section you define and change the outline properties of the components. For example you can specify how the part is to be aligned within the bar assembly.

- Register button Part status:
Here you can influence the additional information of component properties. You can for example define whether the component is listed in the parts list or not.
- Register button Group status:
Here you define information and properties of the complete bar assembly. You can for example save structural analysis values or a variable dihedral angle.
- Register button Parameter:
This section is available when the bar assembly contains references and varies in dependence of the assembly properties. For bar assemblies with variable lengths stretch parameters, for example, are displayed.
- Register button Dihedral angle:
This section is active when variable dihedral angles have been defined for the bar assembly. Then for example, you can define how the components are to behave when the angle is changed.
- Manager
In the Manager section the functions here include the saving and loading of bar assemblies. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.

Register button Parts

The parts section consists of the component section and the drop-down menus:

- Properties
- Geometry
- Position



Dialog box section Component parts

Component list

Shows all components in the current bar assembly and their properties. Here, you can select a component to modify its properties. The selected component is shown red in the preview.

Referenced components are identified with an arrow (>).

If you have defined stiffening, you can select its constituent parts (components, strut or bottom boom) in the left list.

<

Activates the previous component in the list. The active component is shown red in the preview.

Add

Adds a new component to the list. The new component initially has no outline. This means that you must assign an outline and define the other component properties.

Remove

Removes the active component from the list.

>

Activates the next component in the list. The active component is shown red in the preview.

Objects

Adds objects from the drawing as components to the assembly. Objects that can be selected are components which have been inserted into the drawing as sections and ATHENA objects (for example semi-finished products). When you click the button, the dialog box is temporarily closed and the input request appears:

Command prompt

Select objects:

*Select the objects which you want add as components for the assembly.
Terminate the object selection by pressing the Enter key.*

Specify the base point for the assembly or [?]:

Use the mouse or enter coordinates to specify the base point of the assembly.



If the current assembly already contains components, the selected components can either be supplemented or replace the current component. For this, a query is issued which you must confirm appropriately.

Base point ...

Opens the Dialog box Base Point, where you can modify the base point of the assembly. You will find further information in the section *Base Point* on page 654.

Drop-down menu Properties

Here you define the properties of the current component.



If components are used by referencing in a complex bar assembly, then only the cutting and the position can be modified in the assembly definition. You must modify the other properties on the original component.

Label

Defines the designation of the component. You can save the component designation in various languages. To do this, choose the required language from the list.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

From group

Adopts the designation of the assembly for the component.

From object

Adopts the designation of the object for the component. This option is only available for ATHENA objects.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Cutting

Opens the Dialog box Cutting class, where you can assign a cutting type to the component.

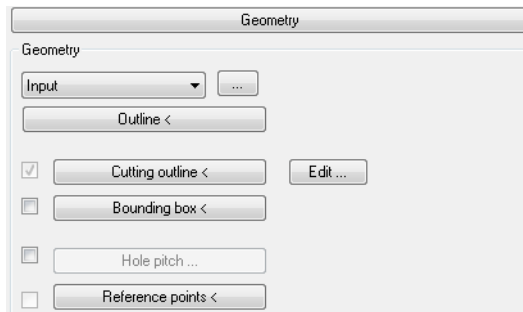
Component parts sharing the same cutting are modified in the Cutting dialog box according to the type of joint. If a transom and mullion butt against each other, they must be assigned the same cutting.

You will find further information in Sections *Cutting class* on page 124 and *Bar joint manager* on page 709.

Item

Opens the Dialog box Item, where you can select an item number. You will find a detailed explanation on items in the Chapter *Item* on page 125.

Drop-down menu Geometry



Above the dialog box section Geometry ATHENA shows warnings and notices. For example, when a component has not yet been assigned any outline.

Type list

Defines the component types. Depending on the type, by clicking the button [...] a dialog box is opened where you can modify the properties of the active component. The following component types are possible:

Input

Block or enclosed polyline outline from the drawing.

Outlines

You will find further information in the Chapter *Outline* on page 221.

Reference

Refers to another bar assembly. If you want to use a reference, the Dialog box For object selection is opened. You will find further information in the Chapter *Object selection* on page 138.

Standard Part

You will find further information in the Chapter *Standard Part* on page 248.

Semi-Finished Products

You will find further information in the Chapter *Semi-finished product* on page 262.

Sheet metal

You will find further information in the Chapter *Sheet metal section* on page 304.

Gasket

You will find further information in the Chapter *Gasket* on page 240.

Spacer

You will find further information in the Chapter *Spacer* on page 236.

Insulation

You will find further information in the Chapter *Insulation* on page 177.

Seal

You will find further information in the Chapter *Seal* on page 242.

Membrane

You will find further information in the Chapter *Membrane* on page 172.

Glazing axis

You will find further information in the Chapter *Axis Symbol* on page 745.

Outline <

Assigns an outline to the current component. By clicking the button the dialog box is temporarily closed and the following input request appears:

Command prompt*Select objects:*

Use the mouse to select the objects whose outlines are to be assigned to the component part. These can be blocks, ATHENA objects (sheet metal section...), circles or polyline outlines.

Specify component part base point:

Choose a base point for the component with the mouse. This input request does not appear if you have chosen a block when selecting an object, because it has a base point.

Specify assembly base point or [Adopt] Adopt:

Use the mouse to select the base point of the assembly.

Choose the option Adopt to accept the base point of the component part.

Cutting outline <

Assigns a cutting outline to the component.

The cutting outline has two functions:

Firstly, it is used to process adjacent components with the same cutting class during the profile cutting.

Secondly, the cutting outline is used for calculating the bar diagrams. For this reason the external dimensions of the cutting outline should never be larger or smaller than the enclosing rectangle of the component outline.



If you do not change the cutting outline of components, then the following cutting outlines are used as standard:

- External outline - for components with less than eight sides
- Enclosing rectangle - for components with more than eight sides

By clicking the button the dialog box is temporarily closed and the following input request appears:

Command prompt*Specify assembly base point:*

Use the mouse to select the base point of the assembly.

Select cutting outline:

Use the mouse to choose the outline to be used as the cutting outline for the component.

Edit ...

Opens the Dialog box Modify cutting outline, where you can define the cutting outline of the component by parameters. You will find further information on this

in the Chapter *Cutting outline* on page 654.

If the component has been defined as a notched part in Drop-down menu Status, the Dialog box Edit notch is opened. Here, the notch distances can be specified to enlarge the notch. You will find further information in the section *Edit notch* on page 655.

Bounding box <

Assigns a cutting outline to a bounding box.

A bounding box is then needed when neither the enclosing rectangle of the outline nor the cutting outline are suitable for the cutting.



In contrast to the cutting outline the bounding box can be smaller than the enclosing rectangle of the component outline. If a bounding box is specified, it has priority for the cutting before all other outlines.

By clicking the button the dialog box is temporarily closed and the following input request appears:

Command prompt

Specify assembly base point:

Use the mouse to select the base point of the assembly.

Select bounding box:

Use the mouse to choose the outline to be used as the bounding box for the component.

Hole pitch

Assigns a regular hole pitch to the component. The button Hole pitch opens the Dialog box Component hole pitch where you can define the hole pitch settings. You will find further information in the section *Component hole pitch* on page 657.

Reference points

Defines further reference points, to which, with variable assemblies, other components can be fixed. For this, the dialog box is temporarily closed and the following input request appears:

Command prompt

Specify assembly base point:

You determine the base point of the assembly.

Specify the reference point or [Next/Remove/eXit] <eXit>:

Define an additional reference point or select an option.

The option Next shows the next reference point.

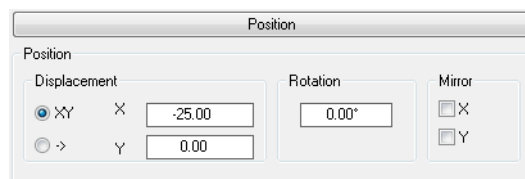
The option Remove deletes the current reference point.

The option eXit terminates the input request.



In the preview only the reference points of the current component are shown.

Drop-down menu Position



Changes the position of the current component within the assembly.

Displacement XY

Activates the Cartesian coordinate input for the component displacement.

X

Defines the absolute X distance between assembly base point and component base point.

Y

Defines the absolute Y distance between assembly base point and component base point.

Displacement ->

Activates the polar coordinate input for the component displacement.

--

Defines the absolute distance between assembly base point and component base point.

<

Defines the angle between assembly base point and component base point.

Rotation

Defines the rotation angle of the component.

Mirror X

Mirrors the component in the X direction (not about the X axis!)

Mirror Y

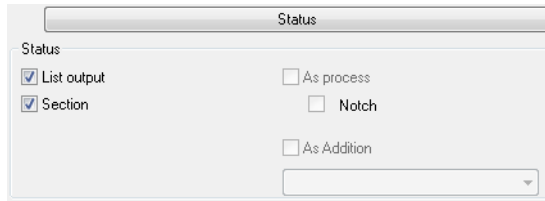
Mirrors the component in the Y direction (not about the Y axis!)

Register button Part status

The section Part status includes a selection menu for selecting a component and the following drop-down menus:

- Status
- Dihedral angle
- Stretch
- Supplied
- Surface
- Production

Drop-down menu Status



List output

Controls the list output for the active component. If you deactivate the tick box, the selected component is not written to the parts list.

Section

Controls the generation of the section for the active component. If you deactivate the tick box, the selected component is not taken into account when the section is generated.

As process

Controls the processing mode for the active component. When the tick box is activated, the component outline of the components having the same cutting type is subtracted. The contour acts practically as a longitudinal cut.



You can also define the process as a variant and therefore switch it on or off!

Notch

Shows the notch mode of the current component. The tick box is activated automatically when you activate the tick box As process for a component with a cutting which is not used for other components.

The notch distances can be adapted in the Dialog box Edit notch. You will find further information in the section *Edit notch* on page 655.

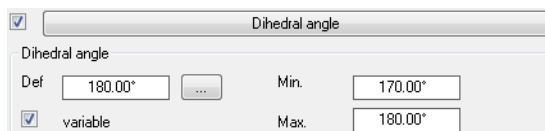
As addition

Assigns the current component to another component as an additional part. In the list you can select the component to which you want to assign the additional part.



If a part has been assigned to another one as an addition, both parts are treated as one part. The additional part is not taken into account for the bar cutting.

Drop-down menu Dihedral angle



The tick box to the left of the button of the drop-down menu shows whether a component with a dihedral angle has been defined. If you deactivate the tick box, the definition of the dihedral angle is deleted.

Def

Defines the current angle of the component.

Variable

Activates variable dihedral angle for the component. When the tick box is activated, the button [...] is released.

[...]

Opens the Dialog box Dihedral Angle Component, where you can define its

settings. You will find further information in the section *Component dihedral angle* on page 659.

Min

Defines the smallest angle of the component. This is only possible if the tick box Variable is activated.

Max

Defines the largest angle of the component. This is only possible if the tick box Variable is activated.

Drop-down menu Stretch

Here, you define components with variable width (definition dimension X) or height (definition dimension Y).

The tick box links from the button of the drop-down menu and shows whether a component has been defined with variable dimensions. If you deactivate the tick box, the dimension definition is deleted.

Dialog box section Definition dimension X and definition dimension Y

Def

Defines the actual (current) component dimension.

Variable

Activates variable dimensions for the component. When the tick box is activated, the button [...] is released.

[...]

Opens the Dialog box Stretch component, where you can define its settings. You will find further information in the section *Stretch component* on page 656.

Min

Defines the smallest admissible component dimension.

Max

Defines the largest admissible component dimension.

Drop-down menu Supplied

Total dimensions		
X	50.00	mm
Y	50.00	mm
Cross-section	23.974	cm²
Weight		

Dialog box section Supplied

Shows supplied lengths and prices. This information is displayed when the components have been imported via an interface from a spread-sheet program.



The additional information of dimensions, development and supply are purely informative and have no technical significance.

Dialog box section Total dimensions

Shows the dimensions of the enclosing rectangle as well as the cross section and weight of the current component.

Drop-down menu Surface

The screenshot shows a window titled "Surface". Inside, there is a section labeled "Development". Below it, there are two rows of input fields. The first row is labeled "Summary" and contains a text box with the value "325.000" followed by a unit "mm". The second row is labeled "Pol." and contains an empty text box followed by a unit "mm".

Dialog box section Development

Defines values for the development, which, for example, can be used for calculating the surface treatment. A differentiation is made between the total surface (Total) and the visible surfaces (Pol).

Drop-down menu Production

The screenshot shows a window titled "Production". It contains two main sections: "Machine support" and "Saw". Under "Machine support", there is a "Machining centre" section with radio buttons for 0°, 90°, 180°, and 270°, and a checkbox for "Mirror". The "Saw" section also has radio buttons for 0°, 90°, 180°, and 270°, and a checkbox for "Mirror".

Defines how the component is supported on the machining center. You can define two separate machine supports. One for machining centers and one for the saw.

You can specify for the output which support is actually used. You will find further information in Sections *Bar diagram* on page 812, *Bar list* on page 806 and *Export CNC* on page 826.

Register button Group status

The section Group status contains the following drop-down menus:

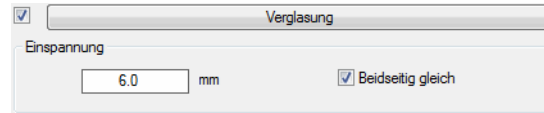
- Glazing
- Dihedral angle
- Stretch
- Strut
- Structural analysis

The screenshot shows a dialog box with several tabs at the top: "Teile", "Teilestatus", "Gruppenstatus" (which is selected and highlighted with a blue underline), "Parameter", "Flächenwinkel", and "Verwaltung". Below the tabs, there is a "Beschriftung ..." button. Underneath, there is a checked checkbox followed by a dropdown menu showing "Verglasung". Below that, there is a section labeled "Einspannung" with a text box containing "6.0" and a unit "mm", and a checked checkbox labeled "Beidseitig gleich". Further down, there are several rows, each with a checkbox and a dropdown menu: "Flächenwinkel", "Strecken", "Versteifung", and "Statik".

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Drop-down menu Glazing



The tick box to the left of the button of the drop-down menu shows whether glazing has been defined. If you deactivate the tick box, the glazing definition is deleted.

Dialog box section Clamping

Defines the assembly as glazing which in which infills can be mounted. You can enter the thickness of the defined infill in the input field. It must be identical to the clamping thickness of the Infill position. You will find further information in the Chapter *Axis Symbol* on page 745.

If you have activated the tick box Clamping, you can define the properties of the glazing (e.g. various clamping thicknesses) in the section Glazing.

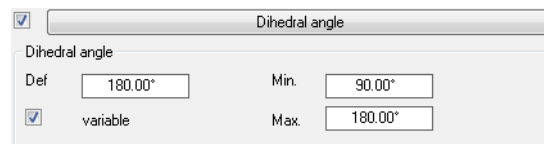


The following conditions must be fulfilled to define glazing. The assembly must consist of referenced components and include an infill position.

Both sides same

Prevents different clamping thicknesses for infills being used for right and left. In the Dialog box Parameters for bar assembly with the tick box activated only one menu field is shown where you can specify the clamping for the right and left clamping.

Drop-down menu Dihedral angle



The tick box to the left of the button of the drop-down menu shows whether a dihedral angle has been defined. If you deactivate the tick box, the definition of the dihedral angle is deleted.

Def

Defines the current angle of the assembly.

Variable

Activates variable dihedral angle for the assembly. When the tick box Dihedral angle variable is activated, you can define in the section Dihedral angle how the components are to relate to one another for variable dihedral angles.

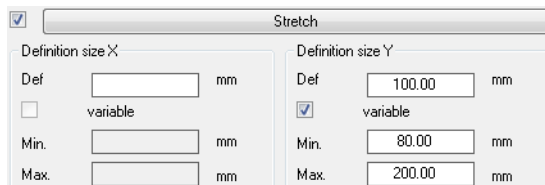
Min

Defines the smallest angle of the assembly. This is only possible if the tick box Variable is activated.

Max

Defines the largest angle of the assembly. This is only possible if the tick box Variable is activated.

Drop-down menu Stretch



Here, you define assemblies with variable width (definition dimension X) or variable height (definition dimension Y).

The tick box to the left of the button of the drop-down menu shows whether a stretchable assembly has been defined. If you deactivate the tick box, the stretch definition is deleted.

Dialog box section Definition dimension X and definition dimension Y

Def

Specifies the defined (actual) width (X) or height (Y) of the assembly.

Variable

Activates variable dimensions for the assembly. When the tick box Variable is activated, you can define in the section Stretch how the components are to relate to one another for variable dimensions.

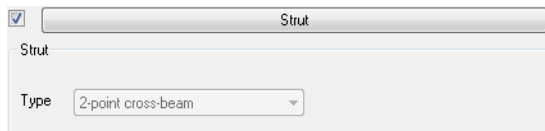
Min

Defines the smallest admissible assembly dimension.

Max

Defines the largest admissible assembly dimension.

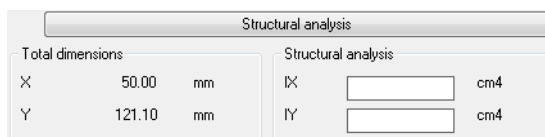
Drop-down menu Strut



Defines an assembly with a strut.

If you have activated the tick box Strut, you can define the properties of the strut in the section Strut.

Drop-down menu Structural analysis



Dialog box section Total dimensions

Displays the dimensions of the enclosing rectangle of the assembly.

Dialog box section Structural analysis

Defines the Ix and Iy values of the assembly.



This information is purely informative and currently has no technical significance.

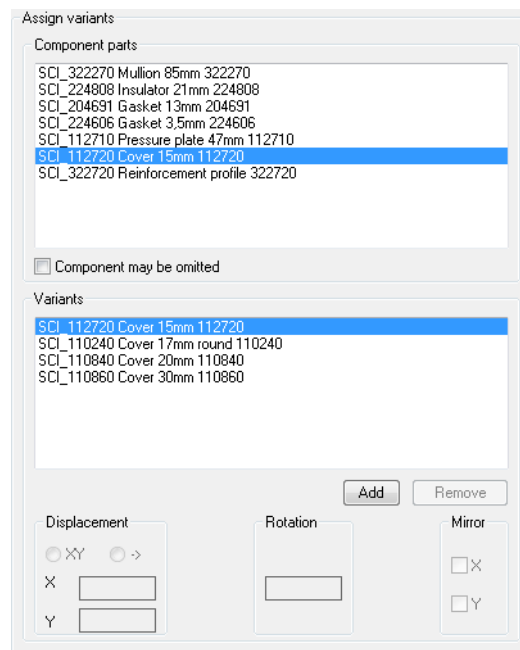
Register button Parameter



The section Parameter is dynamic. There are different types of assemblies having different parameters which can be set:

- Assemblies with variants
For parameters see Dialog box section Assign variants.
- Assemblies with glazing
For parameters see Dialog box section Glazing.
- Assemblies with a strut
For parameters see Dialog box section Strut.
- Assemblies with stretchable components
For parameters see Dialog box section Stretch.

Dialog box section Assign variants



Dialog box section Component parts

Shows all components in the assembly. Here, you mark the component for which you want to define variants.

Component may be omitted

Defines that the component can be omitted when used as a variant.

Dialog box section Variants

Shows the variants for the part marked in the dialog box section Component parts.

Add

Specifies a component as a variant. For this, the Dialog box For object selection is opened, where you can select an assembly. You will find further information on this in the Chapter *Object selection* on page 138.

Remove

Removes the marked variant from the list.

Dialog box section Displacement

Displacement XY

Activates the Cartesian coordinate input for the component displacement.

X

Defines the absolute X distance between the defined component and the variant.

Y

Defines the absolute Y distance between the defined component and the variant.

Displacement ->

Activates the polar coordinate input for the component displacement.

--

Defines the absolute distance between the defined component and the variant.

<

Defines the angle between the defined component and the variant.

Dialog box section Rotation

Defines the rotation of the variant.

Dialog box section Mirror

X

Mirrors the variant in the X direction.

Y

Mirrors the variant in the Y direction.

Dialog box section Glazing

Glazing

Inset

28 28 Remove

> Cover 15mm 112720
> Pressure plate 47mm 112710
> Gasket 3.5mm 224606
> Gasket 3.5mm 224606
> Gasket 13mm 204691
> Gasket 13mm 204691
> Insulator 21mm 224808
Infill position
Infill position

Include

☐ Always
☒ If inset
☐ No

Assign

☒ Left
☐ None
☐ Right

Replace...
Add
Remove

Component part

Def: SCI_204691 Act: SCI_204534

Position

Displacement

☒ XY ☐ ->

X: 0.00 Y: 0.00

Rotation

0.0°

Mirror

☐ X ☐ Y

Relative ...

Dihedral angle

☐ Define ...

Variant

☒ Assign to

Dialog box section Clamping

Shows the available clamping thicknesses in a list. Here, you select an clamping thickness to adapt further properties. In the input field you can specify new clamping thicknesses.

Remove

Removes the current clamping thickness from the list.

The component list lists all parts defined in the assembly. Here, you select the component whose properties you want to adapt to the current clamping thickness.

Replace

Replaces the component for the current clamping thickness. The Dialog box For

object selection is opened to select the component. You will find further information on this in the Chapter *Object selection* on page 138.

Add

Supplements a component for the current clamping thickness. The Dialog box For object selection is opened to select the component. You will find further information on this in the Chapter *Object selection* on page 138.

Remove

Removes the selected component for the current clamping thickness.

Dialog box section Include**Always**

Defines that the marked component is always used.

If clamping

Defines that the marked component is used only for clamping.

No

Defines that the marked component is not used for the current clamping thickness.

Dialog box section Assign**Left**

Assigns the current component to the left clamping section.

None

Does not assign the current component to any side.

Right

Assigns the current component to the right clamping section.

Dialog box section Component part**Def**

Shows the name of the component selected for the defined clamping thickness.

Act

Shows the name of the component selected for the current clamping thickness.

Dialog box section Position

Moves the component for the current clamping thickness.

Dialog box section Displacement**Displacement XY**

Activates the Cartesian coordinate input for the component displacement.

X

Defines the absolute X distance between the defined component and the component selected for the current clamping thickness.

Y

Defines the absolute Y distance between the defined component and the component selected for the current clamping thickness.

Displacement ->

Activates the polar coordinate input for the component displacement.

--

Defines the absolute distance between the defined component and the component selected for the current clamping thickness.

<

Defines the angle between the defined component and the component selected for the current clamping thickness.

Dialog box section Rotation

Defines the rotation of the component selected for the current clamping thickness.

Dialog box section Mirror

X

Mirrors the component selected for the current clamping thickness in the X direction.

Y

Mirrors the component selected for the current clamping thickness in the Y direction.

Dialog box section Dihedral angle

Define

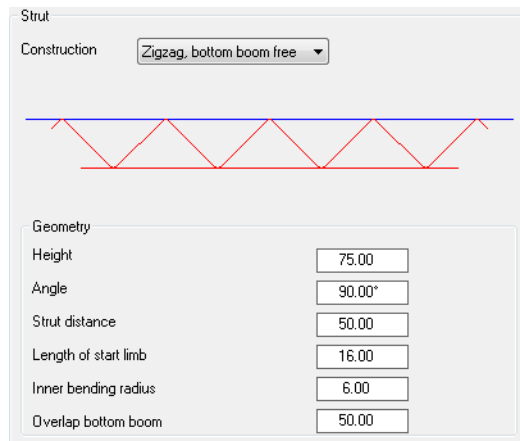
Opens the Dialog box Define dihedral angle, where you can define dihedral angles. This button is only released when the components within the assembly have been inserted as references. You will find further information in the Chapter *Define dihedral angle* on page 661.

Dialog box section Variant

Assign to

Opens the Dialog box Assign variants, where you can define component variants. This button is only released when the components within the assembly have been inserted as references. You will find further information about variants in the chapter *Variants* on page 663.

Dialog box section Strut



Construction

Here, there is the possibility of selecting the desired type of construction from the pull-down menu.

Previously defined types are:

Zigzag, bottom boom free

This type consists of two parallel axes which are joined together by a zigzag shaped strut (braces). With this type of construction the two axes can be of different length. This is set via specification of the overlap for the bottom boom.

Zigzag, bottom boom fixed

This type consists of two parallel axes which are joined together by a zigzag

shaped strut (braces). With this type of construction the two axes are of equal length.

Cross-beam

This type consists of an axis which is joined to the bottom boom by a strut in the middle. The bottom boom and the bracing adapt automatically via Height and Inset bottom boom, irrespective of the length of axis.

Cross-beam 3-part

This type consists of an axis which is joined to the bottom boom by two struts. The struts are distributed in the ratio 1:3.

Dialog box section Geometry

Here, the design parameters for the selected construction can be adapted appropriately to the structural requirements. The respective required input fields are released depending on the selected construction.

Height

This designates the distance between the main axis (bar) and the bottom boom.

Angle

This designates the included angle of the zigzag shaped strut.

Strut distance

This designates the distance from the end of the axis to the start of the zigzag shaped strut.

Length of start limb

This is the length of the first element of the zigzag shaped strut.

Inner bending radius

This is the inner bending radius of the zigzag shaped strut.

Overlap bottom boom

This is the distance from the end of the axis of the bottom boom to the start of the zigzag shaped strut.

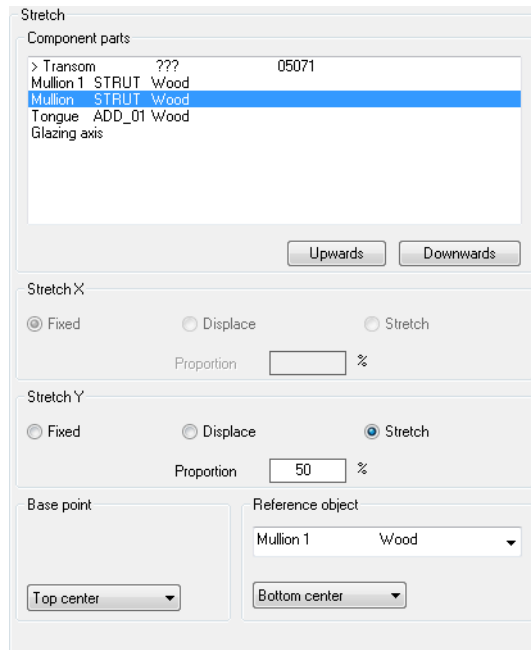
Bottom boom inset

This is the distance between the end of the axis and the start of the bottom boom.

Notes

- This only affects the types of construction Cross-beam and Cross-beam 3-part.
- The values entered here are implemented in the symbolic graphics.

Dialog box section Stretch



Dialog box section Component parts

Lists the components of the assembly. Here, you mark a component to adapt its stretching properties.

Upwards

Moves the marked component upwards.

Downwards

Moves the marked component downwards.



The order of the components affects the behavior of the assembly. The component which does not change its position in the assembly forms the main reference object and must be located in the first position. All components located beneath it can only draw on components situated above it.

Dialog box section Stretch X

Here the behavior of the selected component for a change in width of the assembly in the X direction is defined.

Fixed

This button causes the component to remain unchanged in its dimensions and position.

Move

This causes the component to be displaced by the factor specified under Proportion during a width change of the assembly in the X direction.



Only components can be displayed which themselves are not variable.

Stretch

If this tick box is activated, the selected component is scaled by the factor specified under Proportion during a width change of the assembly in the X direction.

Proportion

This specifies the proportion in %, possessed by the selected component, of the total dimensional change of the assembly.

For Stretch Y the procedure is analogous to Stretch X.

Dialog box section Base point

Defines the base point of the selected component within the assembly. The action defined under Stretch X/Y becomes effective from this point onwards.

Dialog box section Reference object

Selects the component on which the selected component is directly dependent during a change of dimension of the assembly.

Furthermore, using the pull-down menu, the point is selected here from which the action defined under Stretch X/Y acts on the subordinate component.

Using Test in the Dialog box Parameters for bar assembly a check can be made of whether the parameters defined in the dialog box for the desired reaction of the assembly have been correctly entered.

Register button Dihedral angle

Dihedral angle

Defined 180.0°

Rotate plane

☐ Left ☒ Both sides ☐ Right

Base point

☒ Correction

Rotational axis spacings

< 180.0° > 180.0°

X 12.60

Y 7.60

Component parts

- > cover cap 0°-5° both sides 323330
- > cover cap 323390
- > Gasket 224606
- > Gasket 224606

Reaction

☒ Fixed ☐ Rotate

Direction of rotation

☐ Positive ☐ Negative

Centre of rotation

☐ XY ☐ Y

Reference point

☐ XY ☐ Y

Rotation from

- > cover cap 0°-5° both sides 323330
- > cover cap 323390
- > Gasket 224606
- > Gasket 224606

Adopt Bind

Dialog box section Dihedral angle

The defined dihedral angle is shown at the upper left.

Left

Rotates only the left plane.

Both sides

Rotates the left and right planes.

Right

Rotates only the right plane.

Correction

Activates the angle-dependent correction of the base point.

< 180°

Defines the distance of the base point from the rotation point of the plane for angles less than 180°.

>180°

Defines the distance of the base point from the rotation point of the plane for angles greater than 180°.

Dialog box section Component parts

Shows all components defined in the assembly. Here you can select the components for defining their reaction.

Dialog box section Reaction

Fixed

Causes the marked part to retain its position.

Rotate

Causes the marked part to be rotated. When you activate this option, further input fields are released.

Dialog box section Direction of rotation

Positive

Rotates the marked component in the positive direction of rotation.

Negative

Rotates the marked component in the negative direction of rotation.

Dialog box section Center of rotation

XY

Defines the rotation point by the entry of Cartesian coordinates. Enter the X and Y values in the appropriate input fields.

->

Defines the rotation point by the entry of polar coordinates Enter the length and angle in the appropriate input fields.



The dimensions entered here describe the position of the center of rotation relative to the base point of the assembly. This base point is located at the point of intersection which arises through the dihedral angle of the defined reference points of the components.

You should first access the dimensions required for the definition in the assembly inserted into the drawing.

Dialog box section Reference point

XY

Defines the reference point by the entry of Cartesian coordinates. Enter the X and Y values in the appropriate input fields.

->

Defines the reference point by the entry of polar coordinates Enter the distance and angle in the appropriate input fields.



The reference point describes the point of the center of rotation of the assembly relative to the system point of the component. With many profile systems the system point lies at the level of the glass support without gasket.

The base point of the assembly is determined anew from the reference point depending on the dihedral angle of the assembly, so that the assembly is correctly positioned.

Dialog box section Rotation from

If the parameters of the primary components have been defined and tested, dependent parts of the assembly can be conveniently integrated here. To achieve this, first the primary component is selected in the dialog box Components. Then the part dependent on the primary component is marked in this dialog box.

Adopt

Transfers the parameters of the primary component to the selected component.

Bind

Binds the marked component to the primary component.

The reaction of the assembly can be checked at any time with the Test function.

End of program

OK

Saves the current settings and terminates the dialog box.



Only the settings are saved in the dialog box so that you find the same status on restarting the Bar assembly manager.

The current bar assembly is not automatically saved!

Cancel

Terminates the dialog box without saving changes.

1.2 Bar Assembly Manager Sub-Dialog Boxes

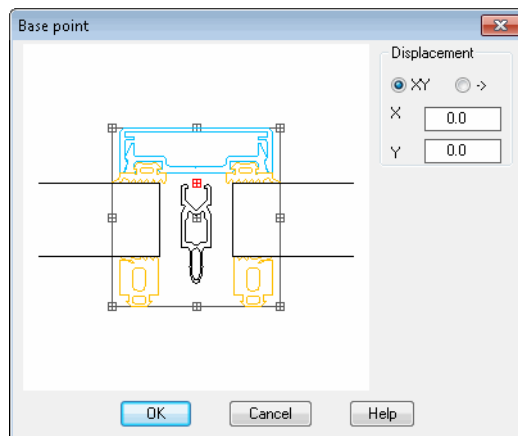
This section describes all sub-dialog boxes which are called exclusively from the bar assembly manager.

- Base Point
- Cutting outline
- Edit notch
- Stretch component
- Component hole pitch
- Component dihedral angle
- Define dihedral angle
- Variants

1.2.1 Base Point

Changes the base point of the bar assembly.

Dialog box Base Point

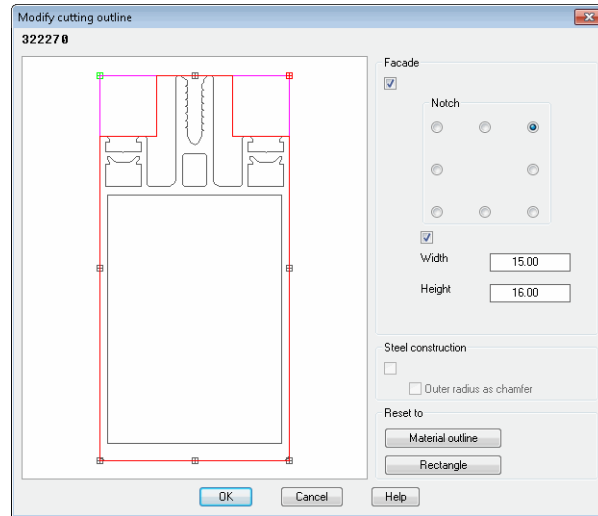


By entering co-ordinates in the X and Y fields, you can displace the insertion point of the current assembly relative to the present base point. A positive X value shifts the insertion point to the right, a negative X value to the left. A positive Y value shifts the insertion point upwards, a negative Y value downwards.

The displacement can also be specified by a distance and an angle by changing to the polar input mode.

1.2.2 Cutting outline

Changes the cutting outline of a component.

Dialog box Modify cutting outline**Dialog box section facade**

Activates the facade mode where you can define rectangular notches by parameters.

The current cutting outline is displayed in the color of magenta. When you define a notch, the cutting outline is recalculated and shown in red.

Notch

Defines the notch position.

Width

Defines the notch width.

Height

Defines the notch height.

Dialog box section Steel construction

Activates the steel construction mode. When you activate this tick box, all internal radii are replaced by chamfers in the cutting outline.

Outer radius as chamfer

Replaces all external radii by chamfers.

Dialog box section Reset to**Material outline**

Deletes the existing cutting outline and resets it to the material outline (outer outline of the component).

Rectangle

Deletes the existing cutting outline and resets it to the enclosing rectangle of the component.

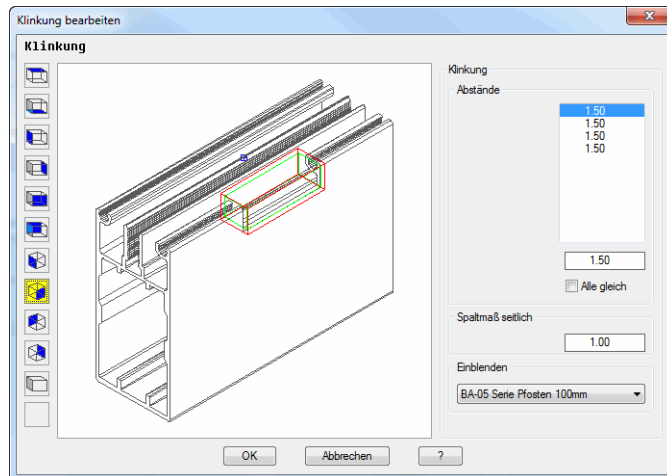


Previously defined notches cannot be removed or manipulated individually. To carry out changes you must reset the outline (to the material outline or rectangle) and redefine the notches.

1.2.3 Edit notch

Changes the distances of components which have been defined as notches.

Dialog box Edit notch



To the left of the preview there are buttons with which you can adjust the preview. You will find further information on this in the section *Object views* on page 102.

Dialog box section Notch

Distances

Lists the distances for each side of the notch. The side belonging to the marked value is identified with an x in the front elevation.

To change the distance you can mark the value in the list and change it in the input field.

All equal

Accepts the entered distance for all sides of the notch.

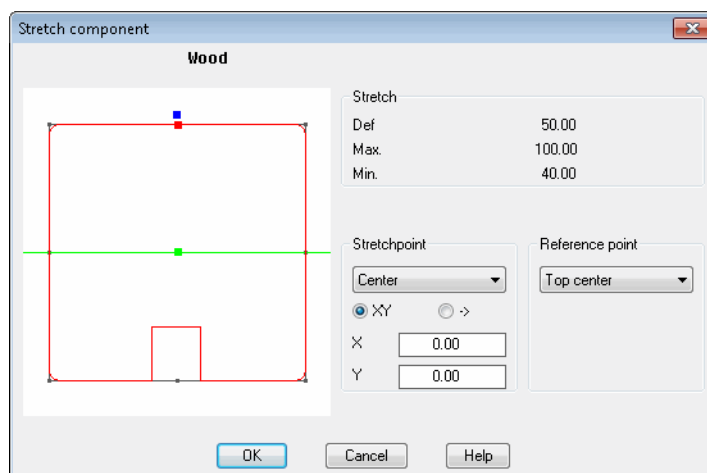
Side gap dimension

Changes the side gap dimension of the notch.

1.2.4 Stretch component

Defines the rules for stretching components.

Dialog box Stretch component



Here the point can be defined, depending on the situation, at which the component may be stretched.

Dialog box section Stretch point

The position of the stretch point can be selected from the pull-down menu or defined by entering values (relative/polar).

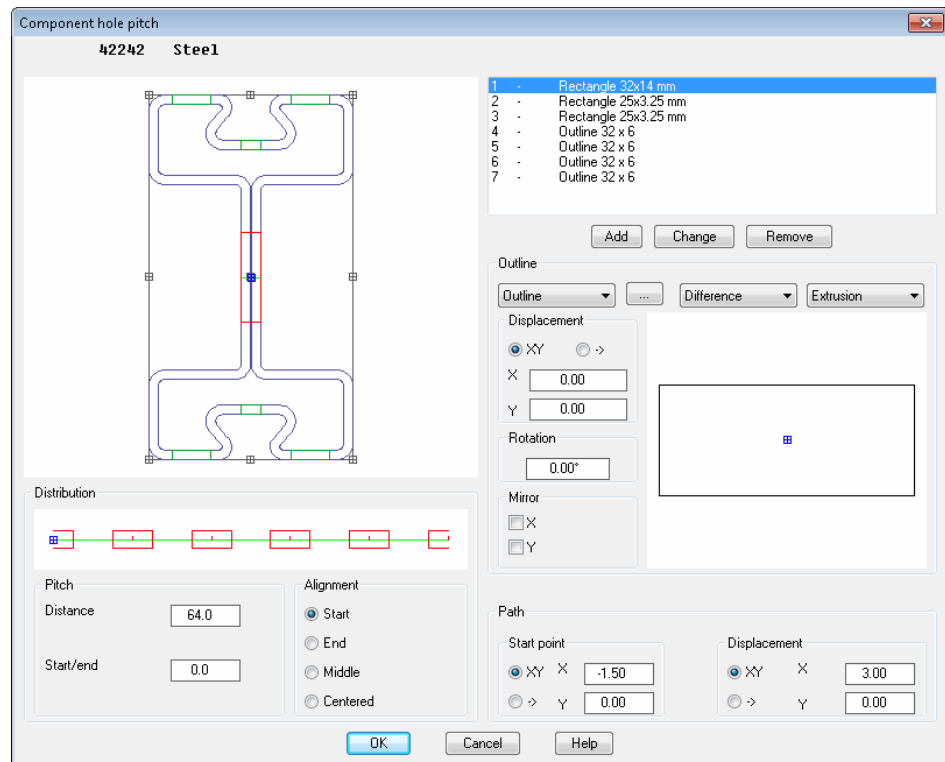
Dialog box section Reference point

The reference point specifies the point from which the scaling is to take effect. For example, the default setting Top center has the effect of anchoring this component at the fixed point and stretching it downwards in the Y direction. The required position of the stretch point can be selected from the pull-down menu.

1.2.5 Component hole pitch

Defines the hole pitch for components of bar assemblies. A hole pitch is regular editing process for a component. One example is an bracket which can have regular hole spacing.

Dialog box Component hole pitch



Dialog box section Graphics

All the processes used are displayed in the display box, whereby the active process is highlighted in color.



The processes produced with this function are not evaluated as a process during the evaluation with Bar list. If an evaluation is to take place, use the command **Arrangement manager** for this.

Dialog box section Processing

In this section you can see the processes assigned to the component. The entries in the list include, provided they are issued, a designator and an item number, but always their quantity. A minus sign after the quantity indicates that the process works as a difference volume on the component. A plus sign after

the quantity indicates that the process works as an addition volume on the component.

Add

With this function you add a process to the list, whereby a copy of the selected process is created. This new process can now be appropriately adapted to requirements.

Modify

With this function you overwrite the marked entry in the list with the settings of the current process in the store.

Remove

With this function you remove the marked process from the list.

Dialog box section Outline

In this section there are two ways of assigning outlines using pull-down menus.

Input

By pressing the button [...] the dialog box is temporarily closed and the following input request appears.

Command prompt

Select outline or [?]:

Use the mouse to select the objects whose outlines are to be assigned to the process. These can be circles or polyline outlines.

Specify base point or [?]:

Use the mouse to select the base point of the process. This input request does not appear if you have chosen a block when selecting an object, because it has a base point.

Outline

The button [...] gives you access to the Dialog box Outline in which you can edit the corresponding process.

A tick in the tick box Hole causes the process to act as a difference volume on the component.

Dialog box section Displacement

In this section you define the displacement of the base point for the process.

Positive and negative entries are possible. Entries with more than two decimal places are rounded. The entries can be made with orthogonal or polar values.



All changes are displayed in the graphics windows for a visual check, but they only become effective on clicking the button Change.

Dialog box section Rotation

In this section you define the rotation for the process. A positive entry causes a rotation in the mathematically positive sense about the base point of the process.

Dialog box section Mirror

In this section you define the mirroring for the process. X mirroring causes mirroring of the current component about the Y axis. The process is in this respect mirrored about its base point.

Y mirroring causes mirroring of the current process about the X axis. The process is in this respect mirrored about its base point.

Dialog box section Path**Start point**

In this section you define the position of the process on the component. Positive and negative entries are possible. Value entries with more than one decimal place are appropriately rounded off. The entries can be made with orthogonal or polar values.

Displacement

In this section you define the displacement for the process. The displacement causes a spatial thickness in the X or Y direction to be added to the process. Positive and negative entries are possible. Value entries with more than one decimal place are appropriately rounded off. The entries can be made with orthogonal or polar values.

Dialog box section Distribution

In this section you define the distribution of the assembly in the longitudinal direction of the subsequent bar.

The distribution is schematically displayed in the graphics window for visual control.

Dialog box section Array**Distance**

This entry defines the spacing between the processes.

Start/End

This value defines the distance from the start/end of the bar to the start of the first/last process.

Start

This setting causes the distribution of the process to start from the set reference point with the start distance.

End

This setting causes the distribution of the process to start from the set reference point with the end distance.

Middle

This setting causes the distribution of at least two processes to start from the middle of the construction line at the distance. It follows that with this setting an even number of processes is always produced.

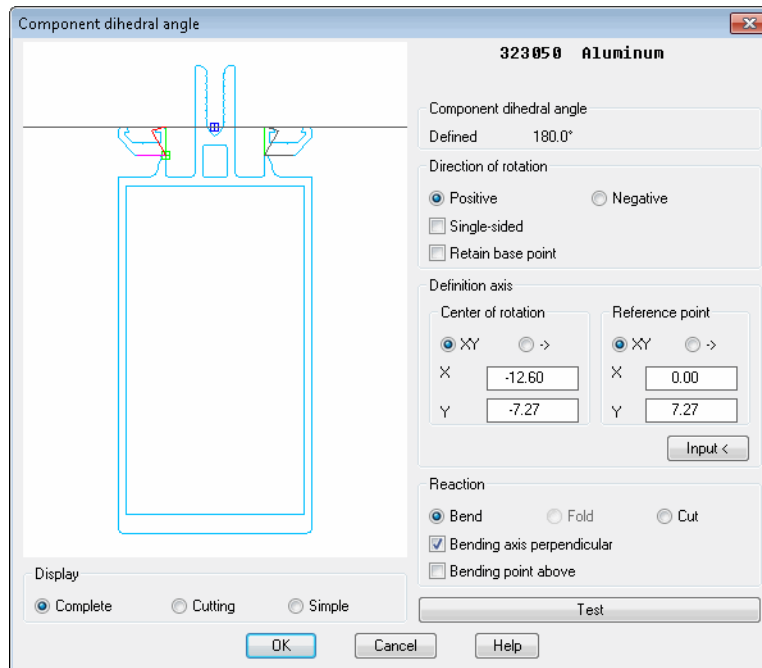
Centered

This setting centers an odd number of processes along the bar axis. This means that the first process is positioned in the center of the bar axis.

1.2.6 Component dihedral angle

Defines the behavior of angle-dependent components.

Dialog box Dihedral Angle Component



Dialog box section Graphics

In the display field angles are displayed symbolically which facilitate a visual control of the defined dihedral angles. The active definition can be seen on the base point of the definition axis.

Dialog box section Display

You can change the preview display with the option buttons Complete, Cutting and Simple:

- Full: Complete depiction of the assembly
- Cutting: Shows the cutting outlines or outer outlines
- Simple: Shows the enclosing rectangles

Dialog box section Info

Via the dialog box section for dihedral angle there is an info section in which, if it has been issued, the name of the component with its item number and the material are displayed.

Dialog box section Dihedral angle

Here the display of the dihedral angle defined on the component is made.



The specification of the dihedral angle and of the variable range on the component is a mandatory requirement for the execution of the function.

Dialog box section Direction of rotation

Here, the direction of rotation of the dihedral angle to be defined can be specified. Also here, the change can be visually checked in the display.



The direction of rotation depends on the defined variable range. This means that:

For a component which in the initial state has a defined dihedral angle of 180° and a variable range from min. 90° to max. 180°, the specification of a negative value causes a rotation in the mathematically negative direction of rotation.

Dialog box section Definition axis

In this section you define the points for the ranges affected by the dihedral angle.

Dialog box section Center of rotation

Here, the center of rotation of the definition axis can be specified relatively or with polar values. The dimensions entered here describe the initial point of the definition axis relative to the base point of the component. This point must be selected such that it lies outside of the component outline and the perpendicular to the bending plane passes through the bending point.



Practically, the required dimensions on the component should be accessed from the drawing prior to the definition.

Dialog box section Reference point

The reference point forms the bending axis starting from the point of rotation of the definition axis relative to the bending plane of the component. With many profile systems the bending plane lies at the level of the glass support without gasket.

The base point of the assembly is determined anew from the reference point depending on the dihedral angle of the assembly, so that the assembly is correctly positioned.

Here too the dimensions can be specified with relative or polar values. A symbolic angle is shown in the display for a visual check.

Dialog box section Reaction

Through the selection of the appropriate option button, you can determine the reaction of the function on the component.

Option Bend

This option has the effect that the section of the component cut by the definition axis is bent depending on the dihedral angle.

Option Fold

This option works similar to bending with the difference that it can only be used for ATHENA sheets. In addition, the central section of the sheet is stretched during folding in dependence of the dihedral angle.

Option Cut

This option causes sections of the component which would be overlapped by changes to the dihedral angle to be cut at the definition axis, e.g. with insulation.

Tick box Bending axis perpendicular

This tick box must be activated if the line between the center of rotation and the reference point does not cut the component outline, but rather the bending axis (the intersection line through the component outline) is located perpendicular to it. The bending axis is displayed as a symbol for a visual control.

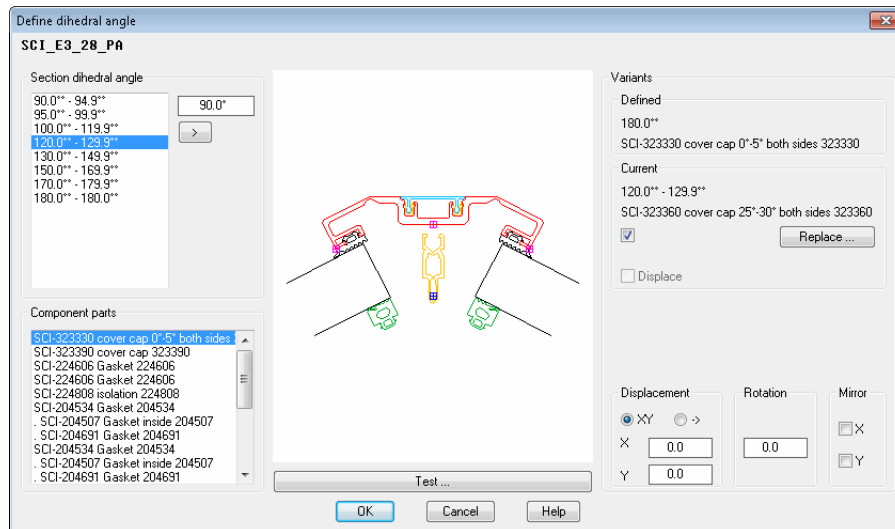
The effect of the definition can be checked with the button Test.

You terminate the function with OK, all entries are discarded with Cancel and ? calls the online help.

1.2.7 Define dihedral angle

Here you can define dihedral angle ranges and in dependence of them manipulate components (displace, rotate, mirror and replace).

Dialog box Define dihedral angle



Dialog box section Section dihedral angle

List

Displays the angular ranges already defined. Here, select a range to edit it.

Input field

Defines a new angular range.

[>]

Removes the marked angular range from the list.

Dialog box section Component parts

List

Shows the components in the assembly. Here, select a component to edit it.

Dialog box section Variants

Defined

Shows the defined dihedral angle in the assembly.

Current

Shows the current dihedral angle in the assembly.

Replace

Replaces the marked component. With the button you open the Dialog box For object selection where you can choose a component. You will find further information on this in the Chapter *Object selection* on page 138.

Move

Activates further dialog box sections where you can define the displacement of the component.

Displacement XY

Activates the Cartesian coordinate input for the component displacement.

X

Defines the absolute X distance between assembly base point and component base point.

Y

Defines the absolute Y distance between assembly base point and component base point.

Displacement ->

Activates the polar coordinate input for the component displacement.

--

Defines the absolute distance between assembly base point and component base point.

<

Defines the angle between assembly base point and component base point.

Rotation

Defines the rotation of the component.

Mirror X

Mirrors the component in the X direction.

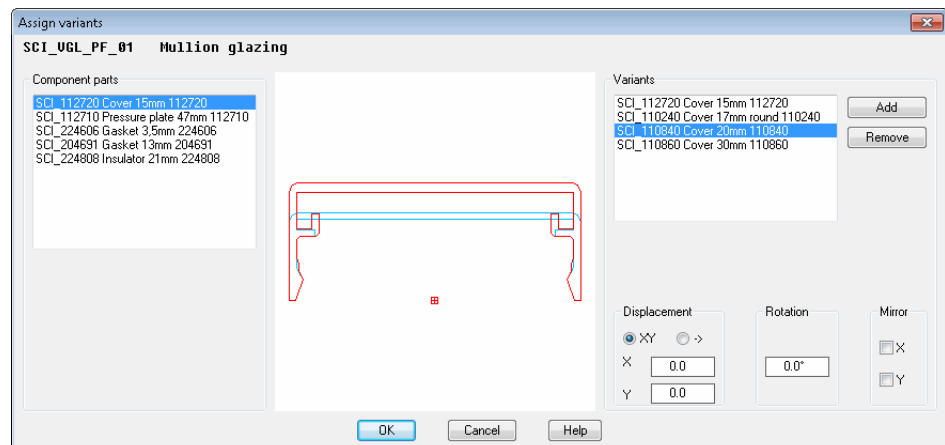
Mirror Y

Mirrors the component in the Y direction.

1.2.8 Variants

This dialog box is used for creating assemblies containing exchangeable components, e.g. reinforcing profiles.

Dialog box Assign variants



When you activate the command, the Component variants dialog box is opened. A description of the various sections of the dialog box follows.

Dialog box section Component parts

All the components referenced in the main dialog box are listed. With several components the one to be processed is selected here.

Dialog box section Variants

One or more variants can be assigned to the selected component with Add (refer also to *Object selection* on page 138). These can then be positioned using Displacement, Rotation and Mirror. If the variant assigned to the selected component can be omitted, the tick box Component may be omitted should be ticked under Components. The word Omitted is then added above the variant. After quitting the dialog box with OK, the correctness of the definition can be checked with the button Test in the main dialog box.

1.3 Infill manager



Ribbon: Tab ATHENA > Group Model > Infill Manager

Menu: Model > Manager > Infill manager

Toolbar: ATH Manager

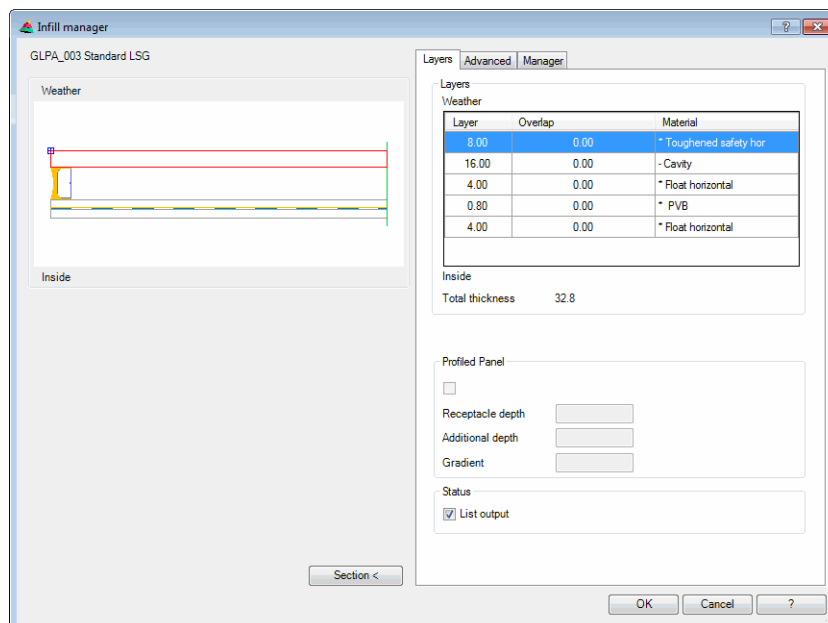
Command input: ath_panel_edit

This function is used for creating and editing infills (glazing or panels).

Use the command Apply infill to use saved infills in the drawing.

Dialog box Infill manager

The dialog box contains several register buttons with which you can change between various sections. The Manager section is the same for all objects. You will find a detailed description of the Manager section in the Chapter *Management of objects* on page 134.



Dialog box section Preview

Displays the preview of the actual infill. The active layer is marked in red.

Dialog box section Representation

Left

Creates the left side of an infill, the right side is interrupted.

Complete

Creates a complete infill without interruption.

Right

Creates the right side of an infill, the left side is interrupted.

Dialog box section Paste

Length <

Defines the overall length of the infill.

Inset <

Defines the inset of the infill.

Gasket thickness

Defines the thickness of the gasket.

With the buttons Length <, Inset < and Gasket thickness < you can access the appropriate values from the drawing. When you click one of the buttons, the dialog box is temporarily closed and you can select two points. The distance between the points is transferred into the corresponding input field. Alternatively, you can also write the value directly into the corresponding input field.

Mirror

Mirrors the infill about its own axis.

Explode

Releases the infill into its constituent parts. With a double click the exploded infill can no longer be edited as a whole, but instead only its layers.

Section

Opens the Dialog box Bar cross section, where you can define settings for the section of the edge seal. You will find further information in the section *Bar cross section* on page 153.

Dialog box section Output**Insert text**

Activate this tick box to label a layer.

Text height

Defines the text height of the label text.

Default

Restores the default text height (current figure height dimension).

Dimensioning

If you activate the tick box, the infill is dimensioned.

Settings ...

Opens the Dialog box Dimension settings, where you can define how the infill is to be dimensioned.

Register button Layers

Layers Advanced Manager

Layers

Weather

Layer	Overlap	Material
8.00	0.00	* Toughened safety hor
16.00	0.00	- Cavity
4.00	0.00	* Float horizontal
0.80	0.00	* PVB
4.00	0.00	* Float horizontal

Inside

Total thickness 32.8

Profiled Panel

☐

Receptacle depth

Additional depth

Gradient

Status

☒ List output

Dialog box section Layers

List

Displays the defined layers with their properties from outside (weather side) to the inside (room side). The dimensional properties, such as layer thickness and overlap left/right, can be changed directly in the cells. Further options are available in a context menu which you can activate with a right click.

Context menu options:

Add

Creates a new layer. ATHENA inserts the new layer below the marked one (if present). The properties of the marked layer are accepted. Once you have created a new infill and no layer exists yet, you must first write its thickness into the Layer field.

Remove

Removes the marked layer from the list.

[...]

Opens the Dialog box Layer, where you can modify the properties of the marked layer. You will find further information on this in the Chapter *Membrane* on page 172.

Material

Opens the Dialog box Material selection, where you can select a material for the layer. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Dialog box section Profiled panel

The tick box activates the function Profiled panel. It is only available when a foldable material has been selected.

Receptacle depth

Defines the distance from the sheet edge to the first fold.

Additional depth

Defines the thickness of the receptacle.

Slope

Defines the gradient of the receptacle.

Dialog box section Status**List output**

When you activate the tick box List output, the marked layer is output in parts lists. This is indicated in the display with an asterisk (*). The output occurs with the commands *Infill list* and *Infill diagram*.



The tick box only has an effect when you use the infill in a 3D construction.

Register button Expanded

Layer	Material
8.00	Toughened safety hor
16.00	Cavity
4.00	Float horizontal

Inside

Edit edges ☐

Chamfer ☐ Weather ☐ Inside ☐

Coating

Weather ☐

Inside ☐

Infill

☐

☐

Edge spacer

☒

Composite layer

☐

Fold

☐ ☐ Elevation side

List

Lists the existing layers from the weather side to the inside. Here, you can mark a layer to modify its properties.

Dialog box section Edit edges

This section is not available for a cavity. Activate the tick box when the layer is to have edge processing. Then select an edge process from the pick list and, where applicable, change the angle in the input field.

Activate the tick box chamfer and enter a value into the input field if the layer is to be chamfered. Two additional tick boxes are activated with which you can turn the chamfer at the top and bottom on or off separately.

Click the button [...] to create user-defined edge processes. To do this the Dialog box Edge processes is started.

Dialog box section Coating

This section is not available for a cavity. Activate the tick box for the weather side or inside if the layer is to receive a coating. Then select a coating from the pick list. Click the button [...] to create user-defined coatings. The dialog box Coatings

is started. The dialog box control is described in the section Dialog box Edge processes.

Dialog box section Infill

The section is not available for a cavity. Activate the tick box when the cavity is to have an infill. Then select an infill from the pick list. Click the button [...] to create user-defined infills. The dialog box Infills is started. The dialog box control is described in the section Dialog box Edge processes. Activate the tick box Insulation when the cavity is to be filled with insulation. Click the button Insulation to change its properties. You will find further information in the Chapter *Insulation* on page 177.

Dialog box section Edge spacer

Activate the tick box to define an edge spacer for the layer.
Select insulating glass to use a predefined edge spacer for insulating glass. Click [...] to choose an insulating glass edge spacer in the Dialog box Edge spacer.
Select Spacer to use a rectangular edge spacer. Click [...] to adapt the spacer properties. You will find further information in the Chapter *Spacer* on page 236.
Choose Profile to use a freely defined profile as the edge spacer. The profile definition occurs in the Dialog box Bar Assembly Manager. You will find further information in the Chapter *Bar Assembly Manager* on page 632.

Dialog box section Composite layer

Activate the tick box when you want to insert a composite layer and select a composite layer from the list. Click the button [...] to create user-defined composite layers. The dialog box Composite layers is started. The dialog box control is described in the section Dialog box Edge processes.

Dialog box section Fold

The tick box activates the folding capability of a sheet-metal layer. When you activate the tick box, the Dialog box For object selection is opened, where you can select a fold sequence.

Elevation side

Inverts the elevation side of the metal sheet. This function is not available when a sheet with folds is used in an external view (weather side or inside).

[...]

Opens the Dialog box For object selection, where you can load a saved fold sequence. See Object selection on page 138.



Fold sequences can be created and saved with the command **Sheet processing**. You will find further information on this in the Chapter *Sheet processing* (ab Seite 318 ff.), Section Tab Fold.

End of program

If you quit the dialog box with OK, the settings which have been made are saved.

Associated commands:

- Infill
- Apply infill

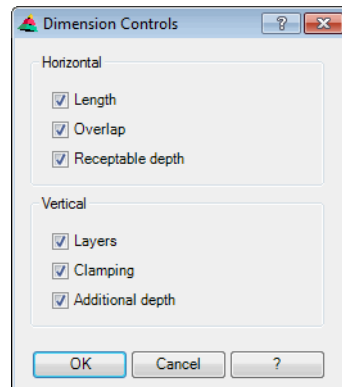
1.4 Infill manager sub-dialog boxes

This section describes sub-dialog boxes which are called exclusively from the Dialog box Infill manager.

- Dimension settings

1.4.1 Dimension settings

Dialog box Dimension settings



Dialog box section Horizontal

Length

Dimensions the overall length of the infill.

Overlap

Dimensions the overlap of a layer of the infill if one has been defined.

Profiled panel clamp

Dimensions the clamp depth for profiled panels.

Dialog box section Vertical

Layers

Dimensions the thickness of the individual layers.

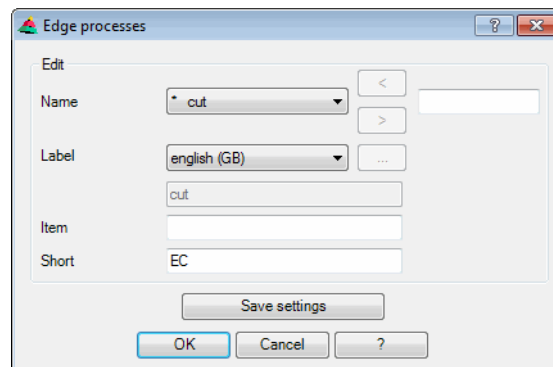
Clamping

Dimensions the clamping thickness of the infill.

Profiled panel height

Dimensions the height for profiled panels.

Dialog box Edge processes



Name

In the pick list all available edge processes are displayed. Enter a name into the

input field at the top right to create a new edge process and then click the [<] button. Click the Remove button to delete the marked process. Processes specified by the system are marked with an * and cannot be removed.

Label

Defines the language-dependent designation.

Item

Defines an item number.

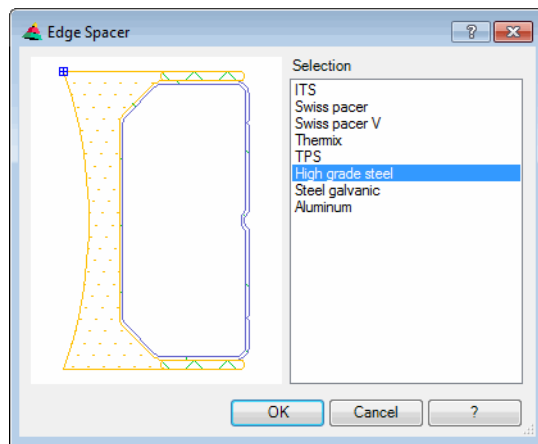
Short

Defines a short designation.

Save settings

Saves the settings in the file ath_obj_prop.dex.

Dialog box Edge spacer



Select the required edge spacer from the pick list.

With triple panes with two cavities 3-fold spacers are also offered (see illustration). If you use them, you must note the following: The 3-fold spacer must be assigned to the first cavity from outside and the middle layer must be defined with negative overlap (approx. half the layer thickness).

1.5 Wall manager

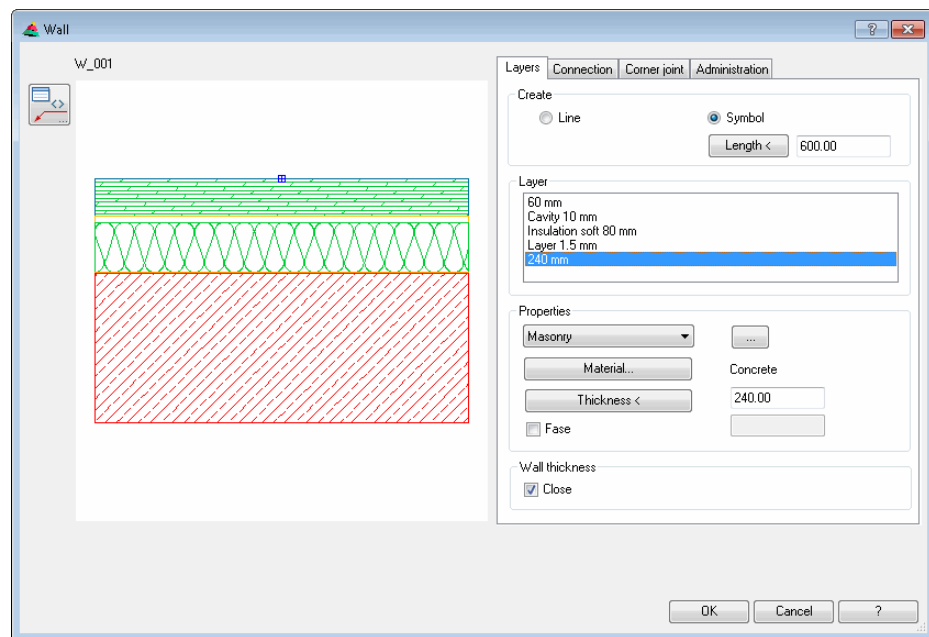


Ribbon: Tab ATHENA > Group > Model > Wall manager
Menu: Model > Manager > Wall manager
Toolbar: ATH Manager
Command input: ath_wall_edit

Defines a wall consisting of one or more layers.

Use the command Apply wall to use saved walls in the drawing.

Dialog box Wall manager



Display section

On the left side the dialog box contains a preview of the defined wall cross-section. On one hand it provides a visual check and on the other hand also a further function: Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.



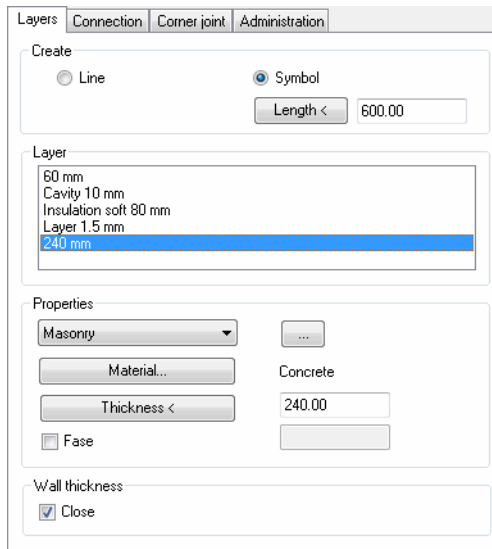
Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Operating section

On the right side of the dialog box there is the operating section with the tabs:

- Layers
- Connection
- Corner joint
- Manager

Tab Layers



Dialog box section Create

Line

Creates the wall as a series of lines which is determined by specifying points.

Symbol

Creates the wall as a segment with fixed length.

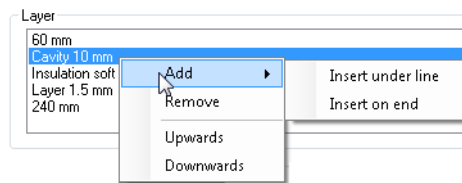
Length <

Defines the length of the wall section.

Dialog box section Layer

Lists the wall layers already defined. You can select wall layers here in order to change their properties. The selected layer is shown red in the preview.

With a right click in the list a context menu is displayed with various options. Context menu functions:



Add

Adds a new layer to the list. If layers are already present, a copy of the selected layer is created.

Insert under line

Inserts a new layer below the selected layer. This option only appears if at least one layer is already present.

Insert at end

Inserts a new layer at the end of the list. This option only appears if at least one layer is already present.

Remove

Removes the selected layer from the list.

Upwards

Moves the selected layer by one position upwards.

Downwards

Moves the selected layer by one position downwards.

Dialog box section Properties

Defines the properties of the marked layer.

You can select a layer type in the pull-down menu. The following layer types can be used:

- Wall layer
- Insulation
- Membrane
- Sheet metal (Sheet metal section)
- Cavity

[...]

Opens a further dialog box, depending on the layer type, where you can make further settings.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Thickness

Defines the thickness of the selected layer. You can also enter the thickness directly into the input field. If you click the button, you can access the thickness from the drawing. For this, the dialog box is temporarily closed and an input request follows.

Chamfer

Activates a circumferential chamfer for the selected layer. The chamfer margin is defined in the input field.

Dialog box section Wall thickness**Close**

Closes the lowermost layer of the wall structure.

Tab Connection

Connection	
Left	Right
<input type="radio"/> Open	<input type="radio"/> Open
<input checked="" type="radio"/> Closed	<input checked="" type="radio"/> Closed
<input type="radio"/> Fast inside	<input type="radio"/> Fast inside
<input type="radio"/> Fast outside	<input type="radio"/> Fast outside
Thickness: 60.00	Thickness: 60.00
Width: 60.00	Width: 60.00

Dialog box section Left/Right

Determines the type of connection to the right or left side of the wall.

Open

Opens the wall on the corresponding side, for example for an interrupted representation.

Closed

Closes the wall on the corresponding side.

Abutment inside

Shows a mounting on the inside of the wall with the entered thickness and width.

Abutment outside

Shows a mounting on the outside of the wall with the entered thickness and width.

Thickness

Defines the thickness of the abutment.

Width

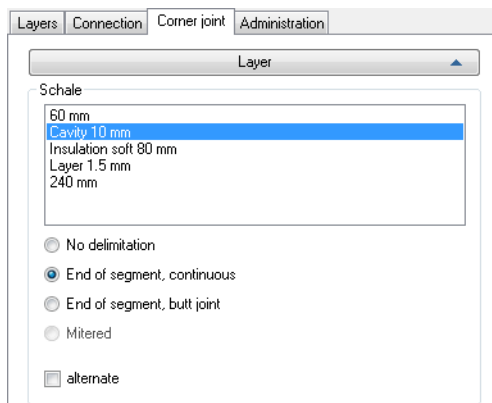
Defines the width of the abutment.

Tab Corner joint



The content of the tab is only active, when the option Line has been activated in the Tab Layers.

Drop-down menu Layer



Specifies how the individual wall layers are joined in the corners. Select the layer from the list for which you would like to specify the corner joint.

No delimitation

Creates a continuous wall layer. The ends are open and no joint is drawn.

End of segment, continuous

Creates a wall layer whose end runs on.

End of segment, butt joint

Creates a wall layer whose end joins the next segment.

Mitered

Creates a wall layers with mitered joints.

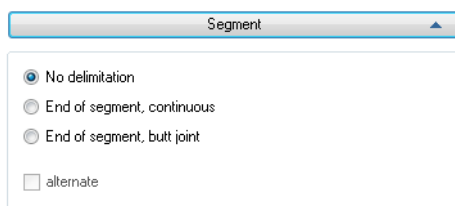
Alternate

Creates alternate continuous and joining segments.



This tick box works together with the options End of segment continuous or End of segment butt joint.

Drop-down menu Segment



Specifies how the wall segments are joined in the corners.

No delimitation

Creates continuous wall segments. The ends are open and no joint is drawn.

End of segment, continuous

Creates wall segments whose ends are continuous.

End of segment, butt joint

Creates wall segments whose ends each form butt joints with the next wall segment.

Mitered

Creates wall segments with mitered joints.

Alternate

Creates alternate continuous and joining wall segments.



This tick box works together with the options End of segment continuous or End of segment butt joint.

End of program

If you quit the dialog box with OK, the settings are saved.

Associated commands:

- Wall
- Apply wall

1.6 Frame Element Manager

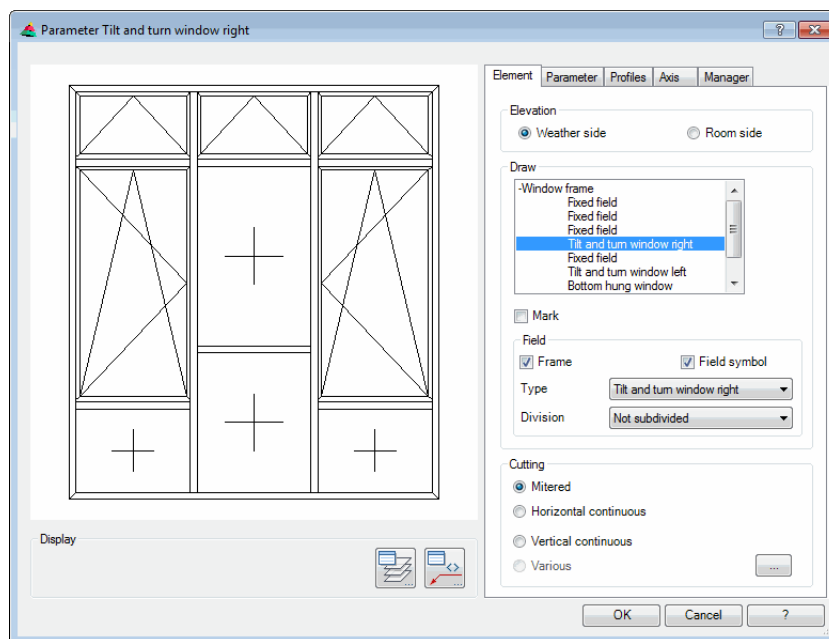


Ribbon:	Tab ATHENA > Group > Model
Menu:	Model > Manager > Frame Element Manager
Toolbar:	ATH Manager
Command input:	ath_elem_edit

Defines and changes frame elements (views of windows, doors). Frame elements can be saved as library objects.

Use the command Apply frame element to use saved frame elements in the drawing.

Dialog box Frame Element Manager



Display section

On the left side the dialog box contains a preview of the defined element elevation. Active elements are marked in color. The preview is used for the visual check; it changes when you change tabs.

Additional functions are activated by clicking in the preview with the mouse wheel. You will find further information on this in the section *Object preview* on page 102.



Opens the Dialog box Layer assignment. You will find further information on this in the Chapter *Layer assignment* on page 108.

Operating section

On the right side of the dialog box there is the operating section with the tabs:

- Element
- Parameters
- Profiles

- Axes
- Manager

Tab Element

The screenshot shows the 'Tab Element' dialog box with the 'Manager' tab active. The 'Elevation' section has two radio buttons: 'Weather side' (selected) and 'Room side'. The 'Draw' section contains a list box with the following items: 'Window frame', 'Fixed field', 'Fixed field', 'Fixed field', 'Tilt and turn window right' (highlighted), 'Fixed field', 'Tilt and turn window left', and 'Bottom hung window'. Below the list box is a 'Mark' checkbox. The 'Field' section has two checkboxes: 'Frame' (checked) and 'Symbol' (checked). Below these are two dropdown menus: 'Type' (set to 'Tilt and turn window right') and 'Division' (set to 'Not subdivided'). The 'Cutting' section has four radio buttons: 'Mitered' (selected), 'Horizontal continuous', 'Vertical continuous', and 'Various'. There is a small '...' button at the bottom right of the 'Cutting' section.

Dialog box section Elevation

Weather side

Creates the weather side or outer side of an element.

Room side

Creates the room side or inner side of an element.

Dialog box section Draw

List

Displays the available objects of an element. Here, select the object which you want to modify.



You can also select the object directly in the preview. Click the appropriate object with the left mouse key.

Mark

Displays the marked object in the preview colored or not.

Dialog box section Field

Frame

Switches the frame of the selected object on or off.

Symbol

Switches the tilt and turn symbol of the selected object on or off.

Type

Defines the window or door type for the selected object.

Division

Defines how the selected object is divided up.

Dialog box section Cutting

Mitered

Cuts the profiles of the selected object for mitering.

Horizontal continuous

Cuts all profiles of the selected object horizontally continuous.

Vertical continuous

Cuts all profiles of the selected object vertically continuous.

Various

Carries out the profile cuts differently. Click the button [...] to individually define the blanks for the profile cuttings.

[...]

Opens the Dialog box Cutting, where you can individually define the profile cuttings for the selected object.

Tab *Parameters*

The screenshot shows the 'Parameter' tab of a dialog box. It contains several input fields and a list. The 'Depth' field is set to 70.00 and the 'Offset' field is set to 15.00. Under 'Frame width', there is a 'Default' field set to 80.00 and a 'Various' checkbox. Below these are four more fields: 'Bottom' (80.00), 'Right' (80.00), 'Top' (80.00), and 'Left' (80.00). Under 'Overlap', there is a 'Default' field set to 40.00 and a 'Various' checkbox. Below these are four more fields: 'Bottom' (13.00), 'Right' (40.00), 'Top' (40.00), and 'Left' (40.00). At the bottom, there is a list box labeled 'Similar type' containing the following items: 'Tilt and turn window right', 'Side hung right hand', 'Side hung left hand', and 'Tilt and turn window left'. The first item is selected. An 'Adopt' button is located at the bottom right of the list box.

Dialog box section Frame width

Default

Defines the frame width for all profiles as default.

Various

Releases further input fields for the definition of various frame widths.

Bottom, Right, Top, Left

Defines the frame width for the profile at the relevant position.

Dialog box section Overlap

Default

Defines the overlap for all profiles as default.

Various

Releases further input fields for the definition of various overlaps.

Bottom, Right, Top, Left

Defines the overlap for the profile at the relevant position.



For irregular shapes, for which the positions of the profiles (bottom, right, top, left) cannot be clearly determined, these dimensional entries for frame width and overlap are ineffective.

Dialog box section Similar type

Displays a list with similar element types.

Here, you can select the element types for which you want to use the set parameters as default. Multiple selection is possible with a pressed CTRL or SHIFT key.

Adopt

Accepts the set parameters for the selected element types.



This default is saved for the duration of the drawing session.

Tab Profiles

Defines additional dimensions of the profiles and infills. These parameters are used when generating sections.

The Profiles tab includes the following drop-down menus:

- Frame
- Sash
- Infill

The preview shows a window and a door element (profiles and infills in each case) in the section. The elements of the active drop-down menu are shown in red. Dimensional changes can therefore be immediately tracked.

Drop-down menu Frame

Dialog box section Depth of rebate

to sash

Defines the rebate depth of the frame profile to the sash.

to sash bottom

Defines the rebate depth of the frame profile to the sash bottom.

to door sash bottom

Defines the rebate depth of the frame profile to the door sash bottom.

Frame rebate for glazing
Defines the glass rebate of the frame profile.

Muntin rebate for glazing
Defines the glass rebate for the muntin.

Dialog box section Facade clamping

Activates the clamping frame profile for the window frame. Additional input fields are released, where you can specify the additionally required dimensions. The bottom clamping frame must be activated separately.

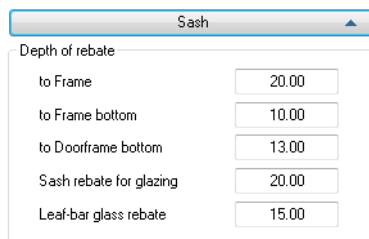
Bottom
Activates a clamping frame for the frame profile bottom.

Thickness
Defines the thickness of the clamping section.

Width
Defines the width of the clamping section.

Offset
Defines the distance from the clamping section to the front edge of the frame.

Drop-down menu Sash



Depth of rebate	
to Frame	20.00
to Frame bottom	10.00
to Doorframe bottom	13.00
Sash rebate for glazing	20.00
Leaf-bar glass rebate	15.00

Dialog box section Depth of rebate

to frame
Defines the rebate depth of the sash profile to the frame.

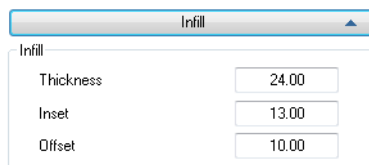
to frame bottom
Defines the rebate depth of the sash profile to the frame bottom.

to door frame bottom
Defines the rebate depth of the sash profile to the door frame bottom.

Sash rebate for glazing
Defines the glass rebate of the sash profile.

Glass rebate for sash muntin
Defines the glass rebate for the sash muntin.

Drop-down menu Infill



Infill	
Thickness	24.00
Inset	13.00
Offset	10.00

Dialog box section Infill

Thickness
Defines the thickness of the infill.

Inset

Defines the inset of the infill in the surrounding profiles.

Offset

Defines the distance of the front edge of the infill to the front edge of the surrounding profiles.

Tab Axes
Dialog box section Axis

The list displays the available axes.

For an element with three horizontal and three vertical elements the axes are listed as follows:

- 1, 2: Vertical axes from left to right.
- 101, 102, 103 and 201, 202, 203: Horizontal axes, in each case from left to right and from bottom to top.

From the list select the axis whose settings you want to change. The selected axis is shown red in the preview.



You can also select the axis directly in the preview. Click the appropriate axis with the left mouse key.

Colinear equal

Causes identical unit spacing for axes is used in alignment. Deactivate this tick box if you want to set different unit spacings.

Dialog box section Position**Through point**

Activates the axial displacement through a defined point. This tick box releases input fields where you can define axial displacement.

Equal clearance

Sets equal clearance dimensions for all axes.

Relative

Defines the ratio of the fields separated by the axis. For example 0.5 for equally

large fields (gives a central axial position) or 0.75 for a field ratio of 3/4 to 1/4. If you enter a relative value, the absolute distance is calculated automatically.

Distance

Defines the absolute distance of the axial displacement. When you change this value, the relative value is automatically calculated.

Displacement

Defines the axial displacement starting from the defined distance.



The starting point of the axial displacement is left for vertical axes and below for horizontal axes.

Dialog box section Parameter

All equal

Uses the entered dimensions for all profile axes.

Width

Specifies the width of the axis.

Depth

Specifies the depth of the axis.

Offset

Specifies the offset of the axes.

1.7 Frame element manager sub-dialog boxes

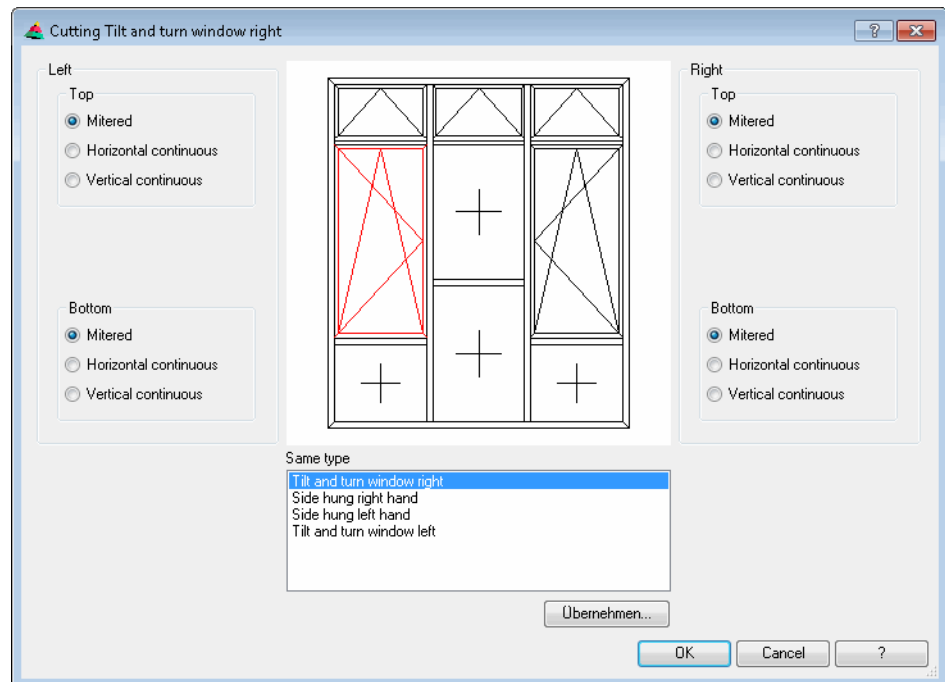
This section describes the sub-dialog boxes of the frame element manager.

- Cutting

1.7.1 Cutting

Manages the cuttings in the element corners for frame elements of the same type.

Dialog box Cutting



Dialog box section Left, Right, Top, Bottom

Mitered

Cuts the profile of the appropriate side for mitering.

Horizontal continuous

Cuts the profile of the appropriate side horizontally continuous.

Vertical continuous

Cuts the profile of the appropriate side vertically continuous.

Dialog box section Adopt

Displays a list with similar objects. Here, you can select the objects for which you want to use the parameters as default. Multiple selection is possible with a pressed CTRL or SHIFT key. The default is saved for the duration of the drawing session.

1.8 Assembly Manager

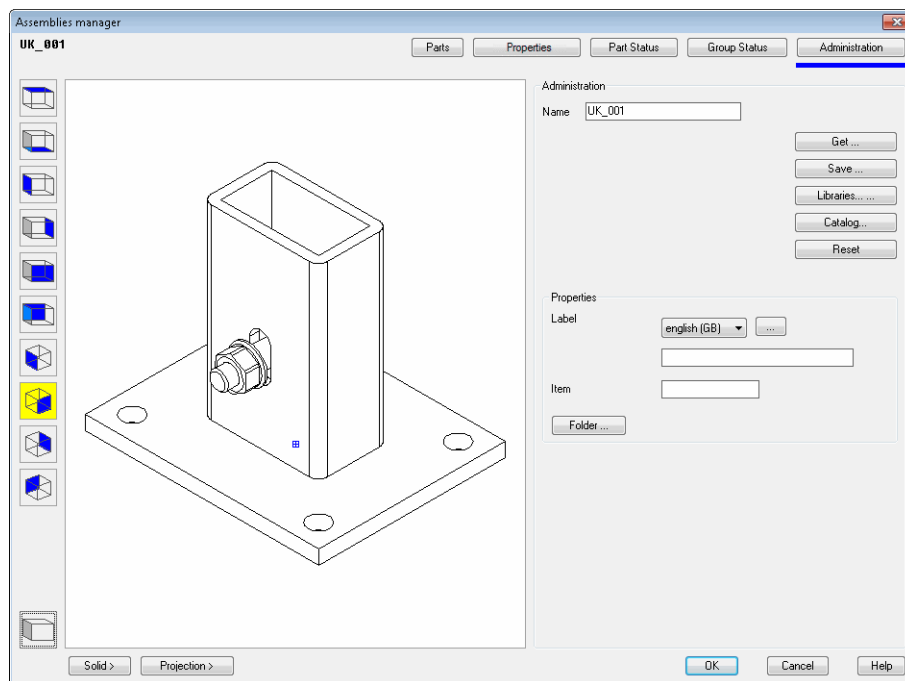


Ribbon:	Tab ATHENA > Group Model > Assembly Manager
Menu:	Model > Manager > Bar assembly manager
Toolbar:	ATH Manager
Command input:	ath_bgr_edit

With this function you can create, edit, evaluate and save 3D assemblies. 3D assemblies are mergers of solids and extracted volumes, which can be structured and which can be applied individually or via distribution rules to bar assemblies.

The dialog box facilitates the combination of extruded or rotated outlines, standard parts, drilled holes, sheet metal, 3D components and referenced 3D assemblies. New objects can be created during the definition. Any number of groupings with any nesting depth can be formed. This means that many elements can be logically combined to form a group or a single part and integrated into a larger structure.

Dialog box Assembly Manager



Display section

On the left side the dialog box contains a preview of the current assembly.

Changing the view

Using the buttons on the left side you can switch between various permanently defined views of the assembly. Apart from the wire frame display, a hidden display is possible



The current setting is highlighted in color.



Plan

Shows the complete assembly as viewed from above.



Side elevation from the left

Shows the assembly as a side elevation from the left.



Side elevation from the right

Shows the assembly as a side elevation from the right.



Front elevation

Shows the assembly as viewed from the front.



Rear elevation

Shows the assembly as viewed from the back.



ISO view SW

Shows the assembly as the isometric view from the southwest.



ISO view SE

Shows the assembly as the isometric view from the southeast.



ISO view NE

Shows the assembly as the isometric view from the northeast.



ISO view NW

Shows the assembly as the isometric view from the northwest.



Hidden view

Masks out the hidden edges of the assembly solid in the current view.

Solid >

Inserts the assembly into the drawing as a solid.

Projection >

Inserts a projection of the assembly into the drawing. The currently set viewing option applies to the 2D projection.

If you insert a solid or projection into the drawing, the dialog box is temporarily closed and the following input request appears:

Command prompt

Specify insertion point or [?]:

Use the mouse or enter coordinates to specify the insertion point. You call the help with the option ?.

Specify rotation angle or [?] <0>:

You specify the rotation angle about the Z axis or you confirm the default angle with a right click or ENTER. You call the help with the option ?.

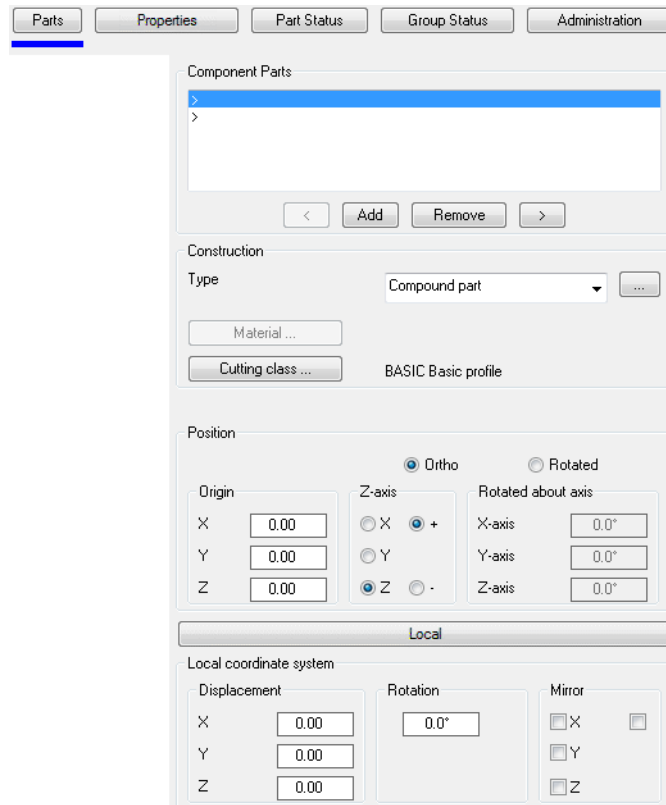
Operating section

On the right side of the dialog box there is the operating section with register buttons for the following sections:

- Register button Parts
- Register button Properties
- Register button Part status
- Register button Group status
- Administration

A description of the register buttons follows further below in this section. The uses of the Administration section include saving and loading assemblies. The section is identical as with other objects. You will find a detailed description of the functions of the manager in the Chapter *Management of objects* on page 134.

Register button Parts



Dialog box section Component parts

Component list

Shows all components of the current assembly. Here, you can select a component to modify its properties. The selected component is shown red in the preview.

<

Activates the previous component in the list. The active component is shown red in the preview.

Add

Adds a new component to the list. If the assembly already contains parts, a copy of the marked component is generated.

Remove

Removes the active component from the list.

>

Activates the next component in the list. The active component is shown red in the preview.

Dialog box section Construction**Type**

Specifies the content of a component of the group by describing a new component or referencing an existing one. The following component types are available:

- Extrusion
- Rotation
- Angled extrusion
- Extrusion path
- Standard Part
- Drilled hole
- Sheet metal
- Grid
- Slice plane
- Counted part
- Group
- Compound part

The selection of the component type is decided in the course of the description.

Further description of the component is provided by Register button Properties. You will find further information in the section *Component types*.

[...]

Changes the properties of the component. The appropriate dialog box is started depending on the selected component type. You will find further information in the section *Component types*.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Cutting class

Opens the Dialog box Cutting class, where you can assign a cutting class to the component.

With the assignment of a cutting class, operations such as Unify, Difference and Section are possible with elements. You will find further information in Sections *Cutting class* on page 124 and *Bar joint manager* on page 709.

Dialog box section Position

In this section the position and orientation of the component coordinate system are described. All details are determined relative to the group coordinate system.

The position of this coordinate system is determined relative to the reference system in the subsection Origin. The orientation of this coordinate system can be specified orthogonally or rotated to the reference system. The orthogonal specification means that all axes remain parallel to the reference system and just the axis definitions (+/-X, +/-Y, +/-Z) change. The rotated specification means: the coordinate system is rotated about the coordinate axes at its origin.

Dialog box section Local coordinate system

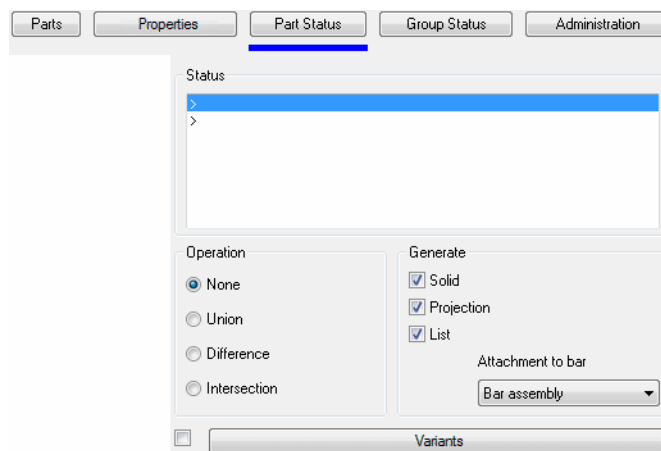
In this section the position and orientation of the component in its coordinate system are described. All details are described relative to the component coordinate system. Coordinate figures, rotation angle and mirroring are available.

Register button Properties

The component properties are very different for the various component types. For this reason the Properties section is dynamic and displays the properties of the relevant active component type.

You will find further information in the section *Component types* on page 691.

Register button Part status



Dialog box section Status

Component list

Shows all components of the current assembly. Here, you can select a component to modify its properties. The selected component is shown red in the preview.

Dialog box section Operation

A selectable operation defines the reaction to spatially adjacent components.

None

Specifies that no operation between the components occurs.

Union

Specifies that the component is unified with another component.

Difference

Specifies that the component acts as the difference (e.g. drilled hole) to other components.

Intersection

Specifies that the overlapping of the components forms the intersection.



For all operations matching of the cuttings of the operating components is to be ensured.

Dialog box section Generate

Controls the generally possible component output.

Solid

Specifies that output as a solid occurs.

Projection

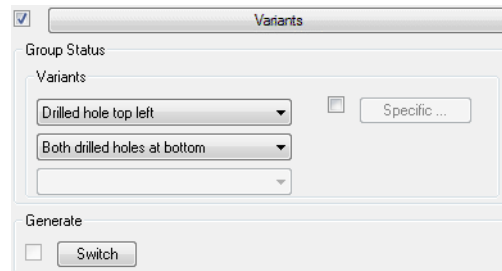
Specifies that output as a sectional view (projection) occurs.

List

Specifies that output as a calculation (list) occurs.

Attachment to bar

Defines the component attachment for a structured output.

Drop-down menu Variants**Dialog box section Group status****Variants**

Shows the variants of the assembly by selection of a criterion.

Specific

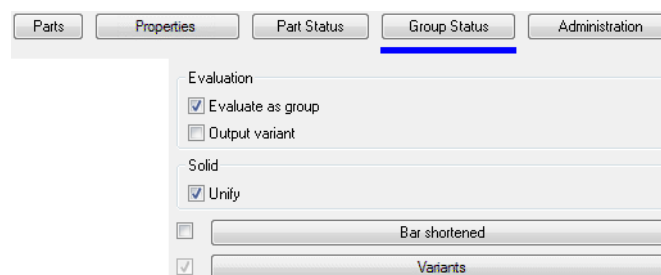
Shows the variants of the assembly by selection of a criterion. To do this, the Dialog box Button assignment is opened.

Dialog box section Generate**Switches**

Opens the Dialog box Button assignment, where you can assign a defined switch to the current component.



The corresponding switches and criteria for the assembly variants must first be defined under the Register button Group status. You will find further information in the Chapter *Assembly variants* on page 704.

Register button Group status**Dialog box section Evaluation**

These options apply to the structured evaluation of the 3D assembly, for example in the XML format.

Evaluate as group

Causes the evaluation of all individual elements of a group in the assembly.

Output variant

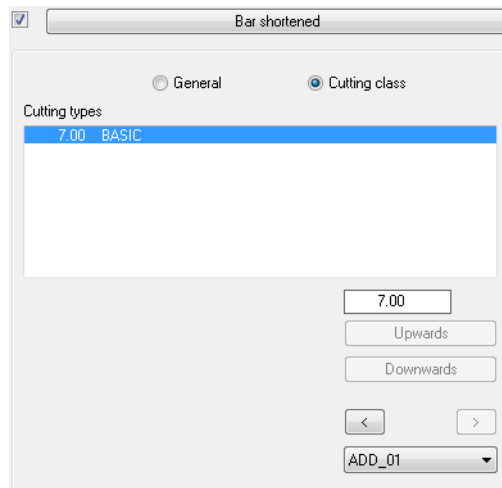
Gives the tag for the current variant. This is only effective when variants are defined.

Dialog box section Solid

Join

Executes all parts operations (Difference, Unify...) and treats the assembly as one part.

Drop-down menu Bar shortened



Here you can define the shortening of a bar if the assembly is assigned to such a component.

General

Causes shortening of all cutting classes (components) of the bar.



If entries for several components exist, they are added up. The following components are displaced by the corresponding amount.

Cutting class

Causes shortening of the cutting classes (components) of the bar.

The list automatically includes all elements of the 3D assembly. Each element can have a dedicated reduction assigned to it which has an effect on its follower.

Upwards

Moves the selected component by one position upwards.

Downwards

Moves the selected component by one position downwards.



The order of the components is significant if the definition sequence does not match the spatial arrangement in the 3D assembly.

These buttons are only active when the General option has been selected.

[<]

Supplements the cutting classes selected in the pick list in the list.

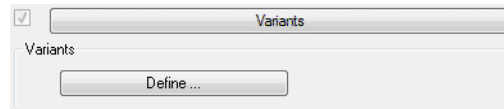
Removes the cutting class selected in the list.



With these options you have the possibility of bringing about a shortening of several cutting classes.

These buttons are only active when the Cutting class option has been selected.

Drop-down menu Variants

**Define**

Opens the Dialog box Variant definition, where you can define switches and criteria for assembly variants. You will find further information in the Chapter *Assembly variants* on page 704.

1.8.1 Component types

Register button Properties

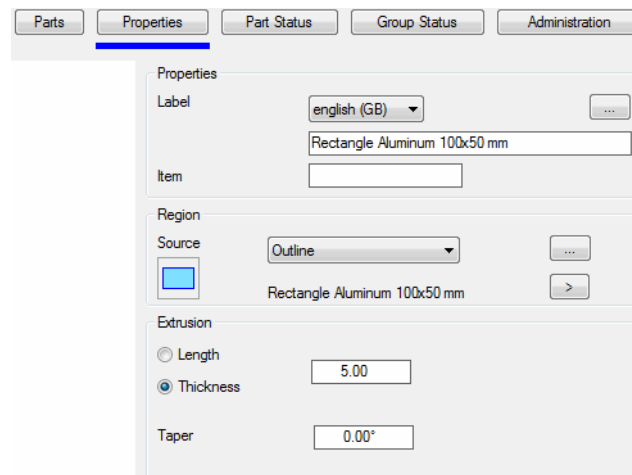
The register button Properties shows, in dependence of the selected component type, ways of describing a component. Describable constructions are:

- Extrusion
- Rotation
- Angled extrusion
- Extrusion path
- Standard Part
- Drilled hole
- Sheet metal
- Grid
- Slice plane
- Counted part
- Group
- Compound part

1.8.1.1 Extrusion

Specifies a component through the extrusion of a region.

Register button Properties of the Extrusion component type



Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels

for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.

Dialog box section Region

Source

Provides various ways of describing two-dimensional geometries.

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

Dialog box section Extrusion

Length

Defines the input as component length.

Thickness

Defines the input as component thickness.



The options Length and Thickness only have influence on the naming in the output.

Taper

Defines the taper angle of the component.

1.8.1.2 Rotation

Specifies a component through the rotation of a region about the X axis. The base point of the region determines the position of the axis of rotation.

Register button Properties of the Rotation component type

Properties				
Label		english (GB)	...	
Rectangle Aluminum 10x10 mm				
Item				
Region				
Source		Outline	...	
			Rectangle Aluminum 10x10 mm	>
Rotation				
Angle		360.0°		
Outlines				
Displacement		Rotation		Mirror
X	0.00	0.0°	<input type="checkbox"/> X <input type="checkbox"/> Y	
Y	0.00			

Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.

Dialog box section Region

Source

Provides various ways of describing two-dimensional geometries. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Standard Part
- Semi-finished product
- Gasket
- Spacer
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

Dialog box section Rotation

Angle

Determines by which angle the outline is rotated. The rotation occurs about the X axis.

Dialog box section Outlines

Displacement

Moves the outline by the entered distance in the X or Y direction.

Rotation

Rotates the outline by the entered angle.

Mirror

Mirrors the outline in the X direction or Y direction

1.8.1.3 Angled extrusion

Specifies a component via the bending of a region in the XY plane and its extrusion in the Z direction. Folding takes place about the Y axis which is determined by the position of the base point.



The bent component is created in that the output is centrally parted and bent by the specified angle. The edges of the region are then shortened or extended such that a closed outline is produced.

Register button Properties of the Bent extrusion component type

Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.

Dialog box section Region

Source

Provides various ways of describing two-dimensional geometries. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Standard Part
- Semi-finished product
- Gasket
- Spacer
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

Dialog box section Development

Angle

Defines the angle of the outline.

Inner radius

Defines the internal radius of the outline.

Outer radius

Defines the external radius of the outline.

Dialog box section Outlines

Displacement

Moves the outline by the entered distance in the X or Y direction.

Rotation

Rotates the outline by the entered angle.

Mirror

Mirrors the outline in the X direction or Y direction

1.8.1.4 Extrusion path

Specifies a component through the extrusion of a region along an extrusion path.

Register button Properties of the Extrusion path component type

Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.

Dialog box section Region

Source

Provides various ways of describing two-dimensional geometries. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Standard Part
- Semi-finished product
- Gasket
- Spacer
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

Dialog box section Outlines

Displacement

Moves the outline by the entered distance in the X or Y direction.

Rotation

Rotates the outline by the entered angle.

Mirror

Mirrors the outline in the X direction or Y direction

Dialog box section Extrusion path

Source

Provides various ways of describing two-dimensional geometries. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



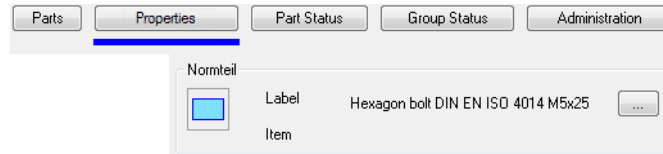
Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

1.8.1.5 Standard Part

Uses a joining element as a component. The Dialog box Standard Parts is displayed to select the joining element.

Register button Properties of the Standard part component type

[...]

Opens the Dialog box Standard Parts, where you can select a standard part. Only joining elements (no profiles) are displayed. You will find further information about standard parts in the chapter *Standard Part* on page 248.

1.8.1.6 Drilled hole

Uses a drilled hole as a component. The Dialog box Drilled hole is displayed to select the joining element.

Register button Properties of the Drilled hole component type

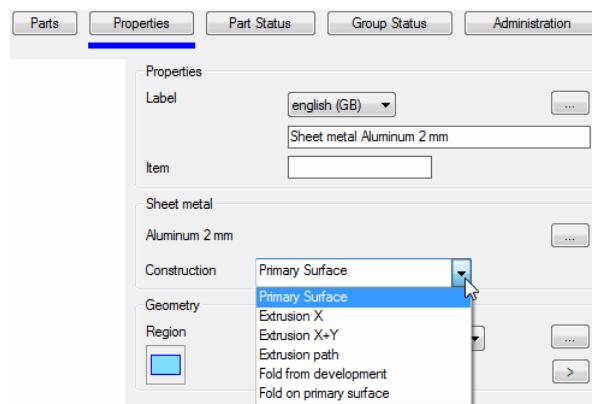
[...]

Opens the Dialog box Drilled hole, where you can select a drilled hole. You will find further information about drilled holes in the chapter *Drilled hole* on page 267.

1.8.1.7 Sheet metal

Describes a sheet using different types of construction. Depending on the selection of the construction method, the appropriate options for the definition of the geometry are displayed in the dialog box.

All the definition methods use as a basis the general sheet settings, in which the type of sheet, material, fold table, sheet thickness, bending radius and crimped fold distance are described.

Register button Properties of the Sheet metal component type

Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.

Dialog box section Sheet metal

[...]

Starts the Dialog box Sheet metal section, where you can define sheet metal settings. You will find further information on this in the Chapter *Sheet metal section* on page 304.

Construction

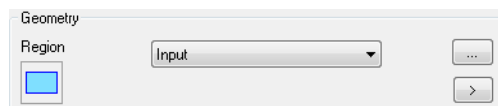
Creates the sheet metal part with the selected construction method. The following methods are available:

- Primary surface
- Extrusion X
- Extrusion X+Y
- Extrusion path
- Folding from development
- Folding a primary surface

Primary surface

Specifies an unfolded sheet by the extrusion of a region. The extrusion height is the sheet thickness set in the sheet properties.

Dialog box section Geometry



Region

Provides various ways of describing two-dimensional geometries. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Standard Part
- Semi-finished product
- Gasket
- Spacer
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

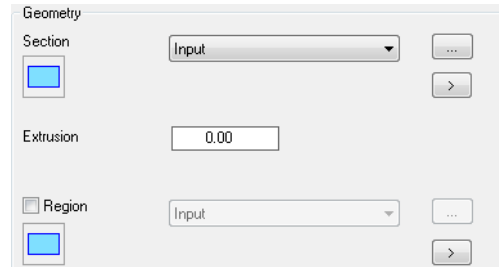
[>]

Inserts the outline as 2D projection into drawing.

Extrusion X

Specifies a sheet by the extrusion of a cross-section. Optionally a region can be specified which acts on the sheet from above as an intersection operation (punching).

Dialog box section Geometry

**Cross-section**

Defines the cross section of the sheet.

[...]

Imports a sheet metal section from the drawing.



Shows the view of the sheet metal section in the preview.

[>]

Inserts the sheet metal section into the drawing.

Extrusion

Extrudes the sheet by the entered length.

Region

Provides various ways of describing two-dimensional geometries. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Standard Part
- Semi-finished product
- Gasket
- Spacer
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

Extrusion X+Y

This definition specifies a sheet by the extrusion of a sheet cross-section each in the X and Y directions. Optionally a region can be specified which acts on the sheet from above as an intersection operation (punching).

Extrusion path

This definition specifies a sheet by the extrusion of a cross-section along a path.

Folding from development

This definition specifies a sheet by the folding of a region. The fold line is determined by the position in the plane. The fold angle must be specified. The sheet thickness is determined from the sheet data.

Folding a primary surface

This definition specifies a sheet via a selected primary surface to whose outer edges a described fold can be assigned. As standard no outer edge of the primary surface is selected. This is specified with a mouse click in the vicinity of an outer edge and highlighted in color.

1.8.1.8 Grid

This function specifies a component through the (parallel) extrusion of a multiple offset region (grid bar) within a delimitation surface. Additionally, the extrusion direction can be specified by an angle.

Register button Properties of the Grid component type

Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.

Dialog box section Region

Source

Provides various ways of describing two-dimensional geometries for grid bars. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Standard Part

- Semi-finished product
- Gasket
- Spacer
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

Displacement

Moves the outline by the entered distance in the X or Y direction.

Rotation

Rotates the outline by the entered angle.

Mirror

Mirrors the outline in the X direction or Y direction

Dialog box section Delimitation

Source

Provides various ways of describing two-dimensional geometries. The following sources are made available.

- Input (outline, which you have to import from the drawing)
- Standard Part
- Semi-finished product
- Gasket
- Spacer
- Outline

[...]

Starts a dialog box in dependence of the selected source in order to describe the outline properties.



Shows the outline view in the preview.

[>]

Inserts the outline as 2D projection into drawing.

Dialog box section Parameter

Angle

Rotates the grid bars about the entered angle within the delimitation.

Distance

Moves the grid bars by the entered distance within the delimitation.

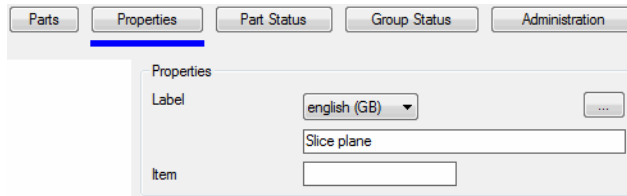
Dialog box section Alignment

Defines the starting point for the distribution of the grid bars within the delimitation surface.

1.8.1.9 Slice plane

This function specifies a plane for slicing elements. Here, a plane is described whose operative side slices or removes all components.

Register button Properties of the Slice plane component type



Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

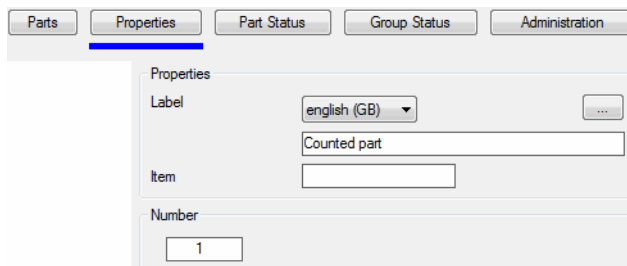
Item

Defines the item number of the current object. The item number is independent of the language.

1.8.1.10 Counted part

This function specifies a component as a non-graphical counted part. This may be necessary when elements, activities or comments should not or cannot be displayed, but must appear in the evaluation.

Register button Properties of the Counted part component type



Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.

Number

Defines the component quantity. The figure is restricted to integers.

1.8.1.11 Group

A group references an existing assembly.

All parts of the referenced assembly are incorporated into the existing assembly before operations such as Unify or Difference are executed.

Register button Properties of the Group component type
Dialog box section Properties**Label**

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.



A group is a referenced assembly. Its properties are taken over from the source and cannot be edited at this point.

Dialog box section Reference**[...]**

Starts a dialog box where you can select another assembly as reference for the current group.

Dialog box section Switches**Use as group**

Causes the group to appear as a part during the output.

Write code

Activates the output of the bit code for assemblies with variants.

1.8.1.12 Compound part

A compound part references an existing assembly.

In contrast to the group, with compound parts all operations such as Unify or Difference are executed within the part. Thus, only one part is incorporated into the higher-level assembly.

Register button Properties of the Compound part component type

Dialog box section Properties

Label

Defines a designation of the current object in the selected language.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the item number of the current object. The item number is independent of the language.



A group is a referenced assembly. Its properties are taken over from the source and cannot be edited at this point.

Dialog box section Reference

[...]

Starts a dialog box where you can select another assembly as reference for the current compound part.

Dialog box section Switches

Evaluate parts

Activates the evaluation of the individual parts of the compound part.

Write code

Activates the output of the bit code for assemblies with variants.

1.8.2 Assembly variants

Variants describe selectable combinations of **one** assembly. This is achieved through the change of visualization and effect of single components. Thus, assemblies can be created of which different variations can be used through the selection of self-defined parameters. Example: Plug plate with alternatively two or four drilled holes which can also be arranged differently.

The variation with assemblies is achieved through the display and suppression of components. This also affects existing operations such as Unify, Difference and Intersection.

Dialog box Variant definition

This is used for the definition of assembly variants. The dialog box contains the following register buttons:

- Switches
- Criterion
- Manager

Register button Switches



By ticking a tick box the associated input line is released. You can now write the designation of the switch into the input line.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

[<] [>]

Displays the next or the previous page. On each page there are eight lines for defining switches. You can define a maximum of 24 switches per group.

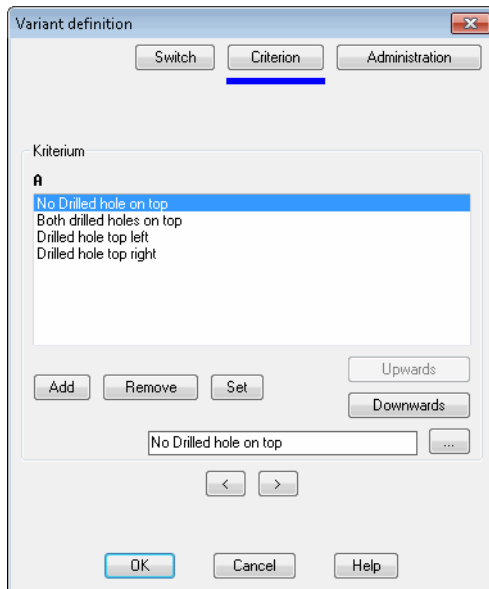
[<<] [>>]

Displays the next or the previous variant group. A maximum of three variant groups is possible.



You can define variants in different groups. The variants of different groups can be combined. This means that even more flexible assembly definitions are possible.

Register button Criterion



The list shows the criteria already established.

Add

Adds a new criterion to the list. By clicking the button the input line is released where you can enter a designation for the criterion.

Remove

Removes the marked criterion from the list.

Set

Defines the switch position for the respective criterion. For this, the Dialog box Button assignment is started where you can tick or clear the appropriate tick boxes for the criterion.

Upwards

Moves the selected criterion by one position upwards. With this function you can sort the display sequence of the criteria.

Downwards

Moves the selected criterion by one position downwards. With this function you can sort the display sequence of the criteria.

Label

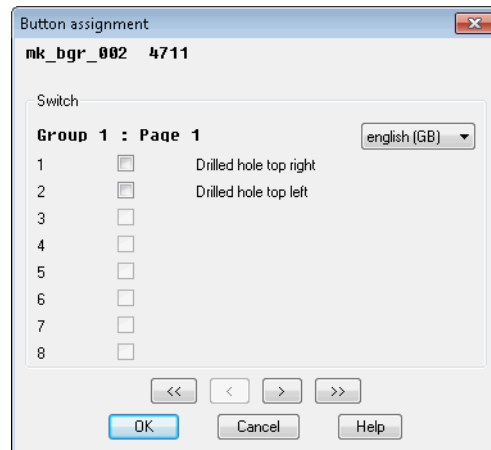
Defines the designation of the criterion. The designation can be specified in various languages.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

[<] [>]

Displays the next or the previous criterion group. A maximum of three criterion groups are possible.

Dialog box Button assignment

Defines the rules for the criterion. In the upper section of the dialog box the name of the active criterion is displayed.

Dialog box section Switches

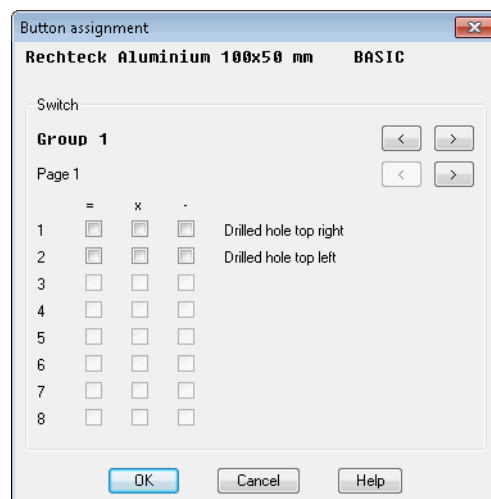
Defines the tick box settings for the criterion. By ticking or clearing you can define what is to occur during the selection of the appropriate variants (criterion). Example: For the criterion **Both drilled holes on top**, the tick boxes for the drilled hole at the top right **and** the drilled hole at the top left are ticked.

[<] [>]

Displays the next or the previous page. Eight tick boxes are located on each page.

[<<] [>>]

Displays the next or the previous variant group. A maximum of three variant groups is possible.

Dialog box Button assignment

Assigns a switch and a situation to the active component. The name of the active component is displayed in the dialog box at the top left.

Dialog box section Switches

[<] [>]

[<<] [>>]

Changes between the various groups and pages. On the left is shown in which

group and on which page of the relevant group you are located. Each group contains three pages each with eight switches.

By ticking the appropriate fields you can allocate logical operations to the switches. A differentiation is made between AND (=), OR (x) and XOR (-) operations:

- AND: The component is displayed when **all** switches are switched on.
- OR: The component is displayed when **at least one** switch is switched on.
- XOR: The component is displayed when **exactly one** switch is switched on.

1.9 Bar joint manager



Ribbon:	Tab ATHENA > Group Model > Bar joint manager
Menu:	Model > Manager > Bar joint manager
Toolbar:	ATH Manager
Command input:	ath_c3d_join_edit

You can create, edit and manage bar joints with this function.

A bar joint determines the joining of bars which meet one another. Cuttings and assemblies can be applied here.

The way in which bars come together differs. A differentiation is made between three types, longitudinal joint (I), corner joint (L) and center joint (T). This differentiation is necessary to specify the application and the possibilities associated with it.

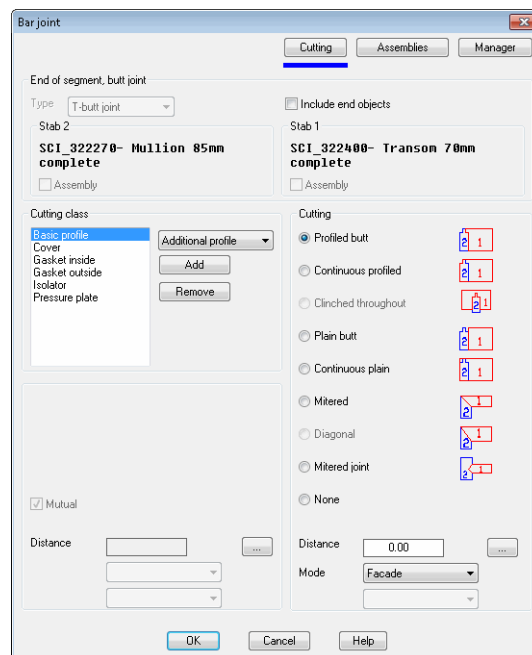
- the I-joint:
I-shaped joint of two bars.
- the L-joint:
L-shaped joint of two bars at the ends, irrespective of their included dihedral angle.
- the T-joint:
T-shaped joint of two bars (one bar runs through, the other butts against it).

Dialog box Bar joint

The dialog box contains the following register buttons:

- Cutting
- Bar assemblies
- Manager

Register button Cutting



Dialog box section Butt

Type

Defines the initial situation of the joint and thus its possibilities. When you call this dialog box via the function Assign bar joint, the type is automatically recognized.

Include end objects

Has the effect that the situation defined for the T-butt joint is also implemented for the L-butt joint.

The option Include end objects is released via the function call Determine bar joint.

Other options are shown for information and can be changed in the Assemblies section.

Dialog box section Cutting class

List

Displays the cutting classes of the selected bar assigned to the components. Here, mark one or more objects and select a cutting.

Selection menu

Shows the cutting types currently present in the document. You will find more about the topic under *Cutting* on page 89.

Add

Adds the selected cutting type to the list.

Remove

Removes the cutting type marked in the list.

Mutual

Also processes the boundary object. This tick box is often activated for the cuttings, Mitered and Diagonal. The following cuttings are used for the boundary object.

Butt-joining bar

Profiled butt
Continuously profiled
Notched throughout
Plain butt
Continuous plain
Mitered
Diagonal
Mitered joint

Boundary object

Continuously profiled
Profiled butt
Mutual not possible
Continuous plain
Plain butt
Mitered
Diagonal
Mutual not possible

The description for the other options (Distance, Mode, ...) can be found further below in this section.

Dialog box section Cutting

Here you assign a cutting to the individual components.

For example, it is possible to have a straight butt joint between the mullion and transom cover sections while having a profiled fitted joint between the transom and the mullion themselves.

Profiled butt

Cuts the marked component at the joint side of the boundary object and notches the bar end according to the cutting outline.

Continuously profiled

Cuts the marked component at the opposite side of the boundary object and notches the bar end according to the cutting outline.

Notched throughout

Notches the marked component according to the cutting outline of the boundary object.

Plain butt

Cuts the marked component on the joint side of the boundary object.

Continuous plain

Cuts the marked component on the opposite side of the boundary object.

Mitered

Cuts the marked component for mitering (bisecting the angle).

Mitered joint

Joins the joining component with a mitered joint to a through-running component.



This cutting option is only available with T-joints. With I and L-joints the option is grayed out.

Diagonal

Cuts the marked component diagonally.

None

Does not realize any cutting. If a cutting was defined for the component previously, it is removed.

Distance

Shortens the joining component by the entered distance. Enter a negative value to lengthen the component.

Mode

Defines the cutting mode for the joining component.

The following cutting modes can be selected:

- With the joint cuttings Plain butt, Continuous plain:
 - Basic - cuts the joining component to the enclosing rectangle of the cutting outline of the boundary object.
 - Axis - cuts the joining component to the axis of the boundary object.
- With the cutting Profiled butt:
 - Basic - cuts the joining component to the cutting outline of the boundary object. Protruding areas are cut off on the opposite side of the cutting outline.
 - Facade - cuts the joining component to the cutting outline of the boundary object. Protruding areas are cut off on the joint side of the cutting outline.

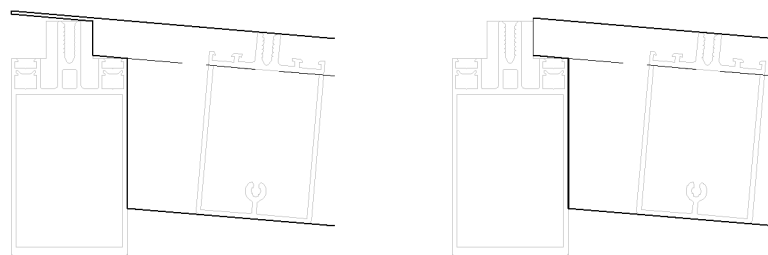


Fig. 1.1: Cutting modes: left basic, right facade

[...]

Makes further cutting settings available. To do this the Dialog box Cutting options is started.

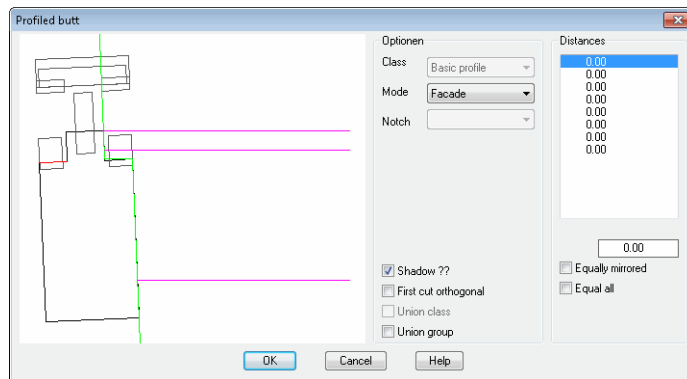
Selection menu

Specifies a component as a notch. If a component has been identified as a component in a bar assembly, you can select this process here. The process notches the boundary object with the intersection of the joining bar.



This is only possible with T-joints with the profiled types of cutting. Furthermore, a component must have been defined as process for the boundary object.

Dialog box Cutting options



On the left side of the dialog box the current cutting situation is graphically displayed. The active cutting outline of the boundary object can be seen in section (active cutting class black, all other cutting classes gray). The joining component of the active cutting class is displayed in magenta. The green line corresponds to the cutting line.

Dialog box section Options

Class

Displays the current cutting class. The cutting class cannot be changed at this point. This is only possible in the higher-level dialog box.

Mode

Defines the cutting mode for the active component.

Notch

This option is currently not available.

Shadow

Activates the angular shadow cutting method. This option is only available for profiled cuttings.

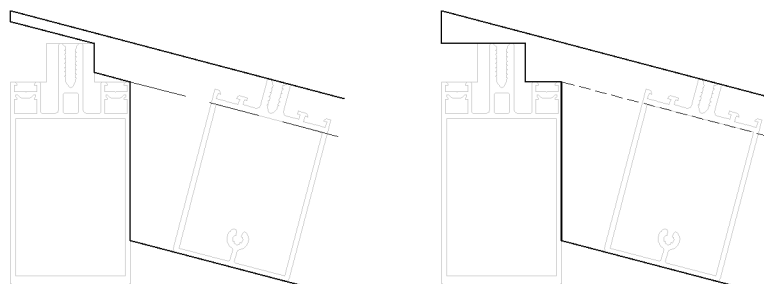


Fig. 1.2: Angular shadow: left on, right off

First cut orthogonal

Causes the first cut to be carried orthogonally (perpendicular to the joining component). Otherwise the first cut is carried out perpendicular to the boundary object.

Unify class

Unifies boundary objects with identical cutting classes so that they are used like a part for the cutting.

Unify group

Unifies all assemblies of the boundary object so that they are used like a part for the cutting.

Dialog box section Distances

Lists the distances for each side of the boundary object. The side belonging to the marked value is displayed in red in the preview.

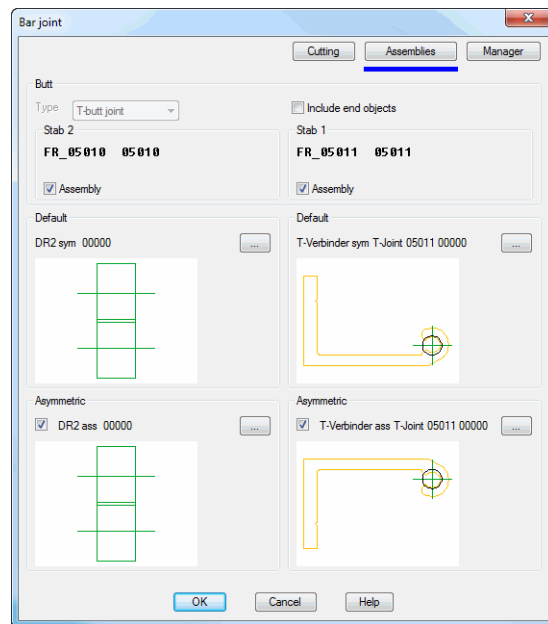
To shorten the joining bar on one side of the boundary object, you can mark the value in the list and modify the distance in the input field.

Equally mirrored

Accepts the entered distance for the opposite side of the boundary object.

All equal

Accepts the entered distance for all sides of the boundary object.

Register button Bar assemblies**Dialog box section Butt****Type**

Defines the initial situation of the joint and thus its possibilities. When you call this dialog box via the function Assign bar joint, the type is automatically recognized.

Assembly

Defines that an assembly is assigned to the bar for the joint.

You can select the assembly from the pull-down list or by clicking the button [...].

Dialog box section Standard

[...]

Opens the Dialog box For object selection where you can select a saved assembly and assign both ends of the bar for the bar joint. See Object selection on page 138.

Dialog box section Asymmetrical

Activate the tick box when you want to use a different assembly (e.g. a mirrored assembly) on the opposite side of the bar for the bar joint.

[...]

Opens the Dialog box For object selection where you can select a saved assembly and assign the opposite side of the bar for the bar joint. See Object selection on page 138.

1.10 Process manager



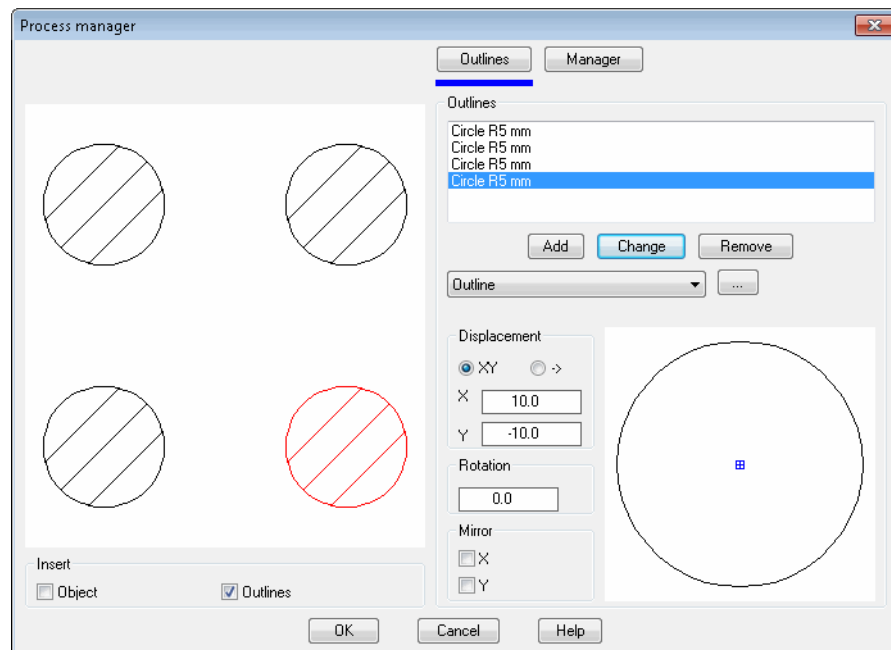
Ribbon:	Tab ATHENA > Group Model > Process manager
Menu:	Model > Manager > Process manager
Toolbar:	ATH Manager
Command input:	ath_hole

Using this function you can create, edit and save 2D processes.

A 2D process is a two-dimensional display of one or more operational steps. These may be drilled holes, cutting operations, milling, etc. which can be combined to form one working step. Saved processes can be used for the description of 3D objects.

Dialog box Process manager

The dialog box contains several register buttons with which you can change between various sections. The Manager section is the same for all objects. You will find a detailed description of the Manager section in the Chapter *Management of objects* on page 134.



Dialog box section Paste

Object

Activates the output as combined block which can react to Modify ATHENA or double click.

Outlines

Outputs the process as single polylines or objects.

The selection is confirmed with OK.

Register button Outlines

To describe a process the corresponding outlines are consecutively described, positioned, orientated and added to the group list. Settings or modifications only become visible with acceptance (Add or Change) into the list.

List of outlines

Shows the current outlines of the process. The selected process is shown red in the preview.

Add

Adds the selected outline to the list.

Modify

Replaces the selected outline with the current one.

Remove

Removes the marked outline from the list.

List

Defines the outline source. Outline, Drilled hole and entry are available.

[...]

Starts the Dialog box Outline or Dialog box Drilled hole depending on the selected source. You will find further information in Chapters *Outline* on page 221 and *Drilled hole* on page 267. With the entry of the outline source the input request follows:

Command prompt

Select objects:

You select one or more ATHENA drilled holes or closed polylines by picking or by a selection window.

Specify the base point for the assembly or [?]:

Determine the base point of the outlines to be imported for the positioning in the group by specifying coordinates or using a mouse click. You call the help with the option ?.

Dialog box section Displacement

XY

Activates Cartesian coordinates for the displacement.

X

Defines the displacement of the marked process in the X direction.

Y

Defines the displacement of the marked process in the Y direction.

->

Activates polar coordinates for the displacement.

-

Defines the displacement distance of the marked process.

<

Defines the displacement angle of the marked process.

Dialog box section Rotation

Rotates the marked process by the entered angle.

Dialog box section Mirror

X

Mirrors the marked process in the X direction.

Y
Mirrors the marked process in the Y direction.

1.11 Arrangement manager



Ribbon: Tab ATHENA > Group Model > Arrangement manager

Menu: Model > Manager > Arrangement manager

Toolbar: ATH Manager

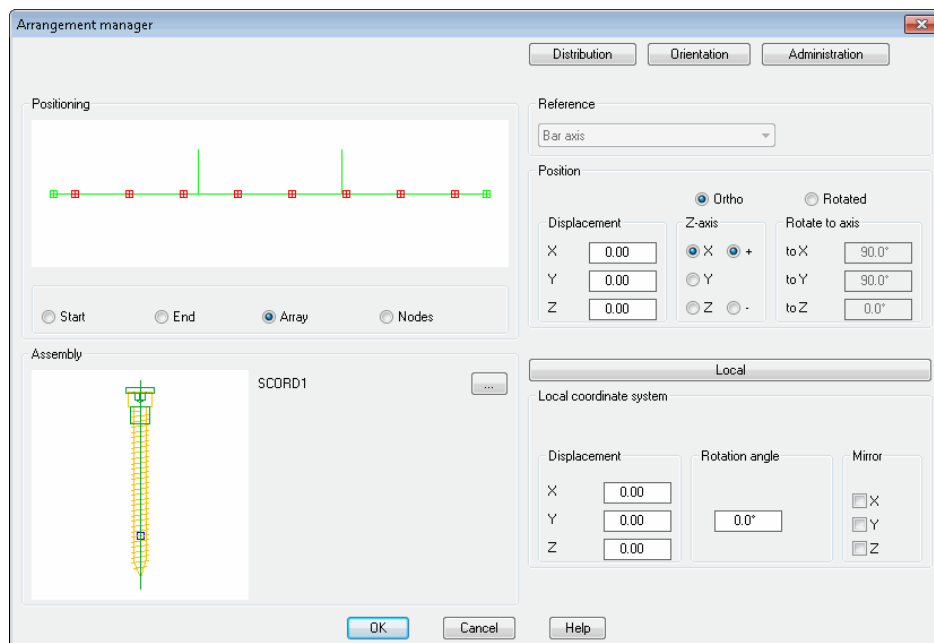
Command input: ath_raster_edit

With this function you can create, edit and manage bar arrangements for the assignment of assemblies to bars.

An arrangement defines the position, distribution and orientation of an assembly on a bar. Only one assembly is assigned to a bar arrangement. The arrangement is a rule which does not depend on the length of a bar, so the same arrangement can be used on different bars. The contained group is always positioned at a reference point or by a distribution on a section.

Dialog box Arrangement manager

The dialog box contains several register buttons with which you can change between various sections. The Manager section is the same for all objects. You will find a detailed description of the Manager section in the Chapter *Management of objects* on page 134.



Dialog box section Positioning

The preview displays the bar with its nodes and the current arrangement. When you click the button Orientation, a UCS symbol is also displayed in the preview.

Start

Positions the element at the starting point of the bar.

End

Positions the element at the end point of the bar.

Array

Positions multiple elements distributed over the length of the whole bar.

Nodes

Positions the element at each axis node on a bar.

Dialog box section Assembly

The preview shows the assembly aligned with the bar.



The viewing direction on the bar is towards the end of the bar for the positioning methods Start, Array and Nodes. With the End positioning mode the viewing direction is however from the end of the bar to the start of the bar.

[...]

Opens the Dialog box For object selection, where you can select a saved assembly to attach it to the bar appropriate to the arrangement. See Object selection on page 138.

Here, irrespective of the structural depth of their included elements, assemblies are treated as a component.

Register button Distribution
Dialog box section Array**Nominal distance**

Defines the basic spacing between the assemblies.

Minimum distance

Defines the minimum spacing. If undercut, the minimum spacing initiates an action for the nodes that are to be taken into account.

Start/End

Defines the distance to the start or end of the bar.

Node distance

Defines the distance to the nodes.

Number

Activates an input field where you can define the number of assemblies. After activation, enter the required number into the input field.

Distances

Activates the distances list. Here, you can define irregular distributed distances.

In each case enter the distance to the starting point in the input field below the list.

Remove

Deletes the marked distance from the distances list.

Dialog box section Alignment

Start

Aligns the components starting at the start of the bar.

End

Aligns the components starting at the end of the bar.

Middle

Aligns the components uniformly about the center of the bar. An even number of components is always produced.

Centered

Aligns a component at the center of the bar and distributes the other components uniformly on both sides. An odd number of components is always produced.

Segments equal

Produces a distribution with equal segments. The distances are calculated from the specified number.

Distance <= Nominal distance

Produces a distribution with fixed start and end distances. The distances are calculated from the specified start and end distances as well as from the nominal distance.

Start/end distance variable

Produces a distribution with variable start and end distances. The distances are calculated from the specified nominal distance.

Dialog box section Nodes

In this section you can specify the reaction to the existing nodes.

Ignore

Ignores existing nodes.

Move

Gives the distance to the nodes more priority; here, the nominal distance does not have to be maintained.

Delete

Removes all components which collide with the nodes due to the set distances.

Segments

Deals with the sections between the nodes as separate distributions.

Dialog box section Apply

Number

Defines the number of assemblies. If the specified number is greater than that computed according to the distribution rules, the entry is not taken into account.

Profile

Defines that the arrangement acts on the bars.

Infill

Defines that the arrangement also acts on infills which border the selected bar. Example: You want to fit a point support at each bar node and the pane is to receive mounting holes at the appropriate locations.

Dialog box section Bar

The bar parameters defined here only act on the preview. In this way you can set up a realistic representation of your situation.

Length

Defines the length of the bar.

Number of nodes

Defines the number of nodes for the bar.

Register button Orientation

Dialog box section Base coordinate system

In this section the position and orientation of the component coordinate system are described. All details are determined relative to the bar coordinate system, whose origin is the set reference point.

The orientation of this coordinate system can be specified orthogonally or rotated to the reference system.

The orthogonal specification means that: All axes remain parallel to the reference system and just the axis definitions (+/-X, +/-Y, +/-Z) change.

The rotated specification means that: The coordinate system is rotated about the coordinate axes at its origin.



Elements in a coordinate system positioned through rotation ($\neq 90^\circ$) are not displayed in the preview.

Dialog box section Local coordinate system

In this section the position and orientation of the component in its coordinate system are described. All details are described relative to the component coordinate system. Coordinate figures, rotation angle and mirroring are available.



The positioning of the component using one of the two descriptions is not sufficient in most cases. For more complex groupings it may be more helpful if a number of local coordinate systems are aligned using a basic coordinate system

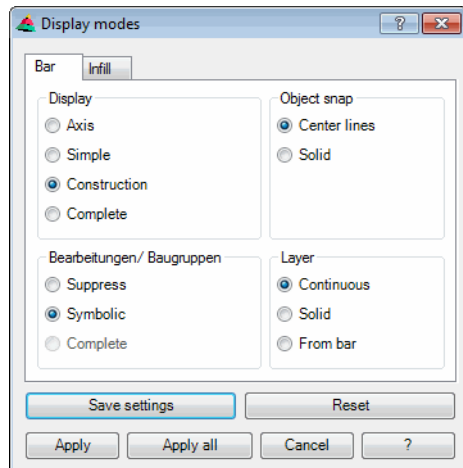
1.12 Display modes



Ribbon:	Tab ATHENA > Group Model > Display modes
Menu:	Model > Manager > Display modes
Toolbar:	ATH Manager
Command input:	ath_st3d_view

With this command you use a dialog box to define the type of display and the object snap settings for 3D bars and 3D infills.

Dialog box Display modes



The dialog box contains the following register buttons:

- Profile
- Infill

Register button Profile

Dialog box section Display

Axis

Displays the bars as axes. Centrally on the axis a red cone symbolizes the bar direction.



The display of the marking for the bar direction can be controlled in the Presettings.

Simple

Displays bars in simplified form.

The simplified form shows outlines with up to 16 sides. If the outline has more than 16 sides, the enclosing rectangle of the outline is shown. Internal outlines are generally masked out.

Construction

Shows bars in the construction mode. The construction mode largely corresponds to the full display, but is many times faster. However, processes and appended assemblies cannot be shown or only symbolically.

Complete

Displays bars in the full form. With this form you can also fully display processes and appended assemblies.



The display form affects performance: The more simply the bars are displayed, the better (faster) is the performance during 3D operations.

Dialog box section Processes/assemblies**Suppress**

Does not show processes and assemblies on the bar.

Symbolic

Shows processes and assemblies on the bar symbolically.

Complete

Shows processes and assemblies on the bar in the full form.

Dialog box section Object Snap**Center lines**

Causes the object snap to only respond at points on the center line of the bar.



If you keep the Ctrl key pressed while you snap points, this object snap setting is temporarily inverted. The OSnap then operates on all points on the solid of the bar.

Solid

Causes the object snap to respond to all points on the bar solid.



If you keep the Ctrl key pressed while you snap points, this object snap setting is temporarily inverted. The object snap then only responds at points on the center line of the bar.

Dialog box section Layer**Continuous**

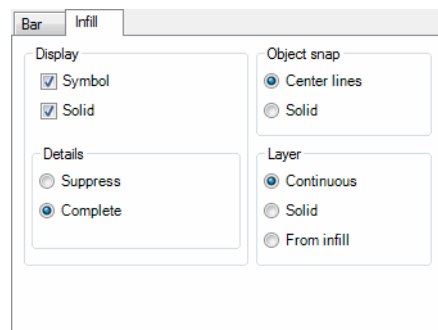
Shows the bar with the material layer for continuous lines.

Solid

Shows the bar with the material layer for solid bodies.

From bar

Shows the bar with the current layer.

Register button Infill**Dialog box section Display****Symbol**

Activates or deactivates the display of the infill as a symbol.

Solid
Activates or deactivates the display of the infill as a solid.



When the tick boxes Symbol and Solid are deactivated, only the border of the infill is displayed.

Dialog box section Details

Suppress
Shows each layer of the infill as an enclosing rectangle and masks out the cavity.

Complete
Shows each layer of the infill in full form.

Dialog box section Object Snap

Center lines
Causes the object snap to occur on the center lines (border) of the infill.

Solid
Causes the object snap to occur on the infill solid.

Dialog box section Layer

Continuous
Shows the infill with the material layer for continuous lines.

Solid
Shows the infill with the material layer for solids.

From infill
Shows the infill with the current layer.

End of program

Save settings
Saves the current settings as default.

Reset
Restores previously saved settings.

Apply
Uses the settings on objects which you must choose. For this, the dialog box is closed and the following input request appears:

Command prompt

Select objects:

Use the mouse to choose the objects whose type of display you wish to change. This input request is repeated until the command is terminated by pressing the Enter key.

X object(s) changed

Apply all
Uses the settings on all objects of the current drawing.

Notes

- Changing the type of display affects the working speed and the file size. The simpler the display mode, the quicker it is to work with the objects (e.g. with Fit) and the smaller the file size.
- No information relating to ATHENA 3D objects is lost. Only the way the objects are displayed is changed.
- Saving of the settings occurs in the file ath_obj_prop.dex.

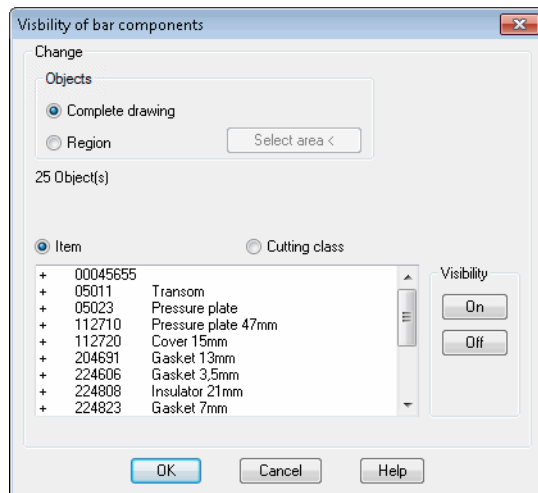
1.13 Visibility of bar components



Ribbon:	Tab ATHENA > Group Model > Visibility of bar components
Menu:	Model > Manage > Visibility of bar components
Toolbar:	ATH Manager
Command input:	ath_bar_bt_visible

Shows or suppresses bar components. This occurs alternatively in the whole drawing or in the selected area of the drawing.

Dialog box Visibility of bar components



Complete drawing

Changes the visibility of the bar components in the whole drawing.

Region

Changes the visibility of the bar components in a selected region of the drawing.

Select region <

Selects the drawing area in which the visibility of the bar components is to be changed. For this, the dialog box is temporarily closed and the following appears:

Command prompt

Select objects:

Select the objects of which the visibility is to be changed. The query is repeated until you terminate the object selection by pressing the Enter key.

Dialog box section Modify

Item

Lists the bar components according to item number and name.

Cutting class

Lists the bar components according to cutting classes.

List

Depending on the setting, this shows the components or the cutting classes of the drawing or of the selected region. Here, you choose the bar components which are to be displayed or suppressed.

Keep the Ctrl and Shift keys pressed to select several bar components.

Dialog box section Visibility

On

Switches on the visibility of the selected bar components. The marked bar components are identified in the list with a plus sign (+).

Off

Switches off the visibility of the selected bar components. The marked bar components are identified in the list with a minus sign (-).

End of program

The settings carried out are applied in the drawing with OK.

1.14 Model Inspection

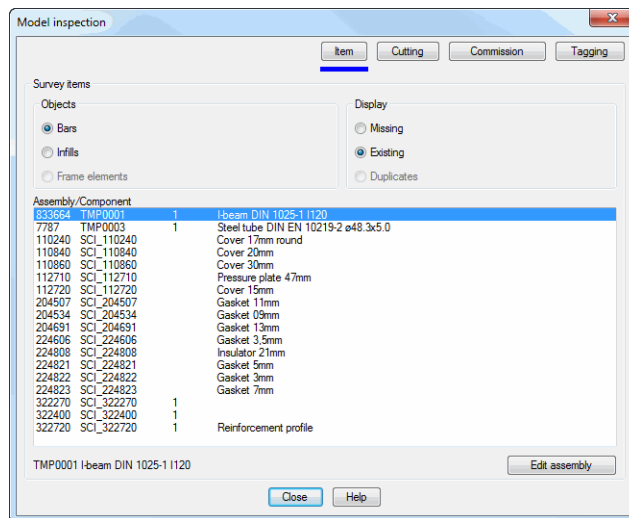


Ribbon: Tab ATHENA > Group > Model
Menu: Model > Manager > Model inspection
Toolbar: ATH Manager
Command input: ath_c3d_audit

Used for checking constructions, e.g. before generating production documentation.

When you call the command, a dialog box appears where information about axes, bars and frame elements, as well as their assignment to jobs is displayed. Optionally you can mark or edit objects.

Dialog box Model Inspection



The dialog box contains the following register buttons:

- Register button Item
- Register button Cutting
- Register button Job
- Register button Tag

Register button Item

This is used for checking the item numbers.

Dialog box section Objects

Bars

Lists the bar assemblies which are located in the drawing.

Infills

Lists the infills which are located in the drawing.

Frame elements

Lists the frame elements which are located in the drawing.



The display adjustment operates in conjunction with the display of the objects!

Dialog box section Display

Missing

Shows objects (bars, infills or frame elements) without item number. An item number is essential for the output.

Existing

Shows objects (bars, infills or frame elements) with item number.

Duplicates

Shows objects with item number which are multiply used. For example components which are referenced in many bar assemblies.

Assembly/component list

Lists the objects which are found according to the setting of object and elevation. The item numbers, assembly names, quantities and designations are displayed depending on the setting and object type.

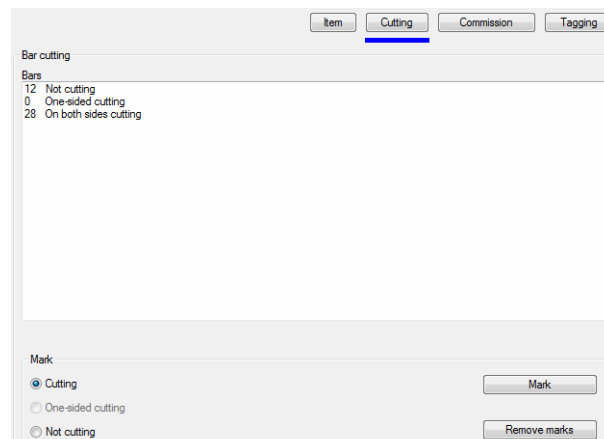
To be able to edit an object you can select it here and then click on the Edit assembly button.

Edit assembly

Closes the Model inspection dialog box and shows the selected object in the relevant dialog box (Bar Assembly Manager, Infill manager or Frame Element Manager) where you can edit it directly.

Register button Cutting

This is used for inspecting bar and cuttings.



Dialog box section List bars

Shows the number of bars which:

- Are uncut
- Are cut on one side
- Are cut on both sides

Dialog box section Mark

Cut

Marks all cut bars when you click on the Mark button.

Cut on one side

Marks all cut bars when you click on the Mark button.

Uncut

Marks all uncut bars when you click on the Mark button.

Mark

Closes the Dialog box Model Inspection and marks all bars according to the selection.



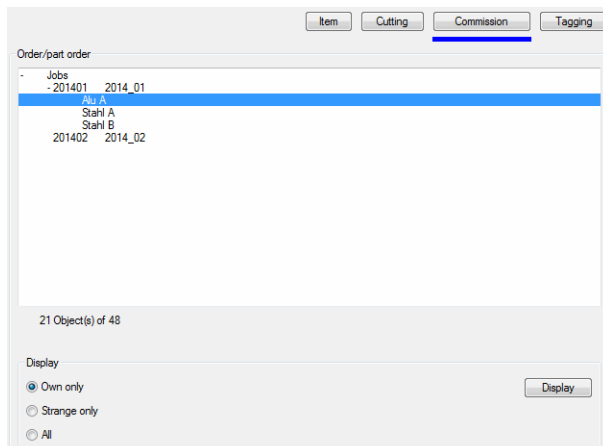
Uncut bars are marked in the center of the bar. Cut bars are in each case marked on the cut side.

Remove marks

Closes the Dialog box Model Inspection and removes all marks from the bars.

Register button Job

Used for checking jobs and job assignment of objects.



Dialog box section Job/job part

Shows the jobs and job parts present in the drawing. Once you have selected a job, the number of assigned objects is shown for information.

Dialog box section Display

Own only

Displays only objects of the selected job when you click on the Display button.

Only extraneous

Displays only objects which do not belong to the selected job when you click on the Display button.

All

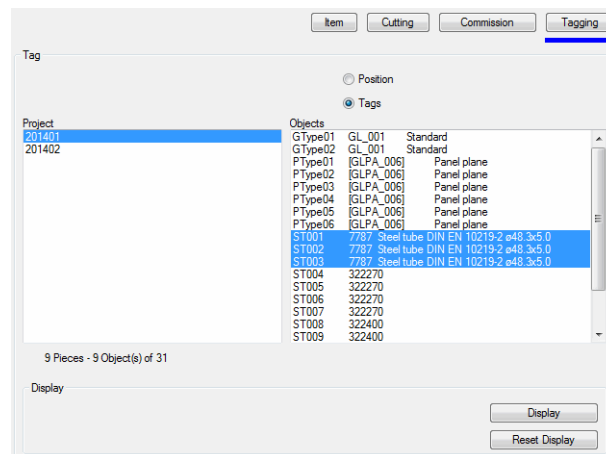
Displays all objects when you click on the Display button.

Display

Closes the Dialog box Model Inspection and marks all bars according to the selection.

Register button Tag

Used for checking positioned and tagged objects.



Dialog box section Tag

Positions

Lists all projects and positioned objects in the DWG.

Tags

Lists all projects and tagged objects in the DWG.

You can select single or several objects and display them in the drawing.

Dialog box section Display

Display

Closes the dialog box and displays all marked objects in the list.

Reset display

Closes the dialog box and displays all objects in the drawing.

End of program

When you click the Close button, the program is terminated.

1.15 Visualize axis model



Ribbon: Tab ATHENA > Group > Model
Menu: Model > Manager > Axis model
Toolbar: ATH Manager
Command input: ath_c3d_pos_vis

Draws regions in the sub-areas or an analyzed axis model (3D position) in order to visualize it.

Only plane areas are visualized. With this function you can therefore check whether all corner points of an area lie in the same plane.



The visualization (shading) of the areas is not visible in the wire-frame mode. Therefore a visual style with shading should be set before the command is run.

When you execute the command, the following appears:

Command prompt

Select 3D position or [?]:

Select an analyzed axis model to draw regions in the sub-areas.

Cancel visualization:

If you run the command again and select the same 3D position, the regions are removed.

2 Apply

Commands in this section:

- Auxiliary construction
- UCS object
- UCS/VIEW bar
- Apply bar assembly
- Assign bar assemblies
- Reassign bar assembly
- Axis Symbol
- Infill plane
- Apply infill
- Apply Assembly
- Apply wall
- Apply frame element
- Object to solid body
- ATHENA Extrusion
- Bar work
- Associative bar work
- Define nodes
- Apply processes to infill
- Assign processes to infill
- Apply arrangement
- Cutting
- Copy cutting
- Remove cutting
- Display associated bar nodes
- Slice Bar
- Analyze axis model
- Copy analyzed axis model
- Invert weather side
- Recalculate axis model
- Reset axis model
- Determine bar joint
- Determine infill
- Specify glazing

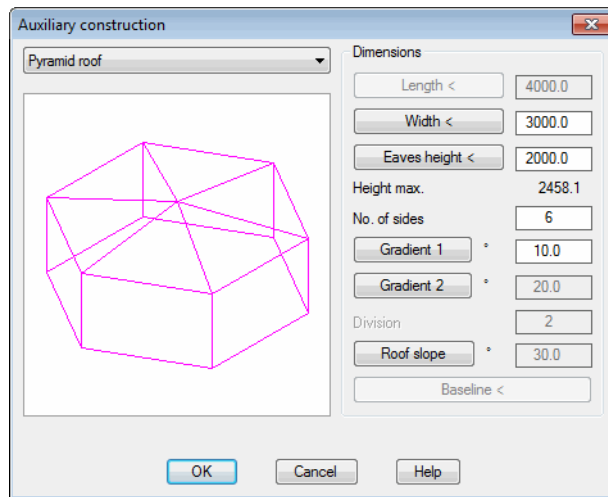
2.1 Auxiliary construction



Ribbon:	Tab ATHENA > Group Model > Construction aids
Menu:	Model > Apply > Construction aids
Toolbar:	ATH Apply
Command input:	ath_k3d_form

Routine for quickly inserting a three-dimensional auxiliary construction. You determine the type and dimensions of the auxiliary construction in a dialog box. The construction is inserted into the drawing on the layer Auxiliary line layer. Then you can assign profiles (profile groups) to the axes with the command [Apply bar assembly](#).

Dialog box Construction aids



In this dialog box you choose your construction type in the drop-down menu at the top left. After selection, the construction is displayed in a preview. The following constructions are available:

- lean-to roof
- gabled roof
- hipped roof
- compass roof
- pyramidal roof
- domed roof
- saw-toothed roof
- facade
- dormer lean-to roof
- dormer gabled roof
- dormer hipped roof
- dormer compass roof

Once you have selected a construction, enter the dimensions into the corresponding entry fields in the right half of the dialog box. Since the dimensions for the individual types of construction differ, not all of the entry fields are required. These are then masked out in gray.

The dimensions Length, Width and Eaves height can be both entered and displayed. To display a dimension, click the corresponding button with the mouse. The dialog box is then closed and you can specify a length by pointing to two points. Once you have entered the second point, the distance between these

points is accepted into the corresponding entry field.

The individual gradients can be specified as angles in ° or as slopes in %. Click the button of the relevant gradient to toggle between ° and %.

The button Baseline is only available in the facade type of construction. Click the button to define the baseline of the facade. You only obtain a preview of the facade when you have defined a baseline.

When you quit the dialog box with OK, the input request follows:

Command prompt

*All types of construction except **Facade***

Specify insertion point:

Use the mouse or enter co-ordinates to specify an insertion point for the construction.

Specify rotated angle<0>:

*Enter a rotation angle for the construction or confirm the default angle by entering **ENTER** or a **RIGHT CLICK**.*

Facade type of construction

The facade is directly inserted, because the baseline is known.

2.2 UCS object



Ribbon:	Tab ATHENA > Group Model > UCS objects
Menu:	Model > Apply > UCS objects
Toolbar:	ATH Apply
Command input:	ath_st3d_ucs

With this command you define a new co-ordinate system by selecting an ATHENA 3D object (bar or infill) or also an AutoCAD object (e.g. polyline). Then you obtain further options for aligning the UCS.

Command prompt

Select objects or [Origin/?]:

Use the mouse to select an object for the definition of a new coordinate system.

*Select the **Option Origin** to move the coordinate origin.*

You call the help with the option ?.

Select edge of plane or [Mirror/Rotate/Origin/Back/?]:

Select an edge to define the surface of the X-Y plane or choose an option.

*Choose the option **Undo** to repeat the previous input request.*

Select surface median or [Mirror/Rotate/Origin/Back/?]:

Select an object to determine the surface median. ATHENA determines the surface median from the surfaces given by the selected edges.

UCS [Mirror/Rotate/Origin/Undo/?]:

*Select **Option Mirror** to mirror the UCS (see the illustration Mirror UCS).*

*Select **Option Rotate** to rotate the UCS (see the illustration Rotate UCS).*

Option Mirror

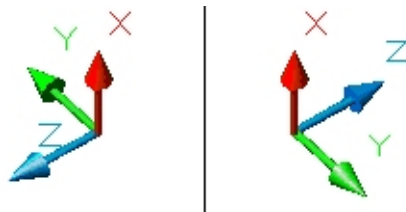


Fig. 2.1: Mirror UCS

Option Rotate

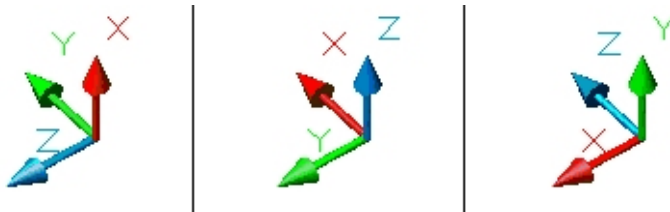


Fig. 2.2: Rotate UCS

Option Origin

Specify origin point or [?]:

Determine a point for moving the UCS origin. The direction of the axes is not changed by this.

2.3 UCS/VIEW bar



Ribbon:	Tab ATHENA > Group Model > UCS/View bar
Menu:	Model > Apply > UCS/View bar
Toolbar:	ATH Apply
Command input:	ath_bar_mtx

Sets the user coordinate system or the view by selecting a bar.

Command prompt

View+ucs

Select bar or [only UCS/?]

Select a bar to set the UCS and the view.

With the option only UCS only the UCS, but not the view is set.

You call the help with the option ?.

View [Cross section/Plan view/Side elevation] <cross section>:

With the option Cross section you set the view or the UCS to the cross section of the selected bar.

With the option Plan view you set the view or the UCS to the plan view (view from above) of the selected bar.

With the option side elevation you set the view or the UCS to the side elevation of the selected bar.

only UCS

Select bar or [View+ucs/?]:

Select a bar to set the UCS.

With the option View+ucs the UCS and the view are set.

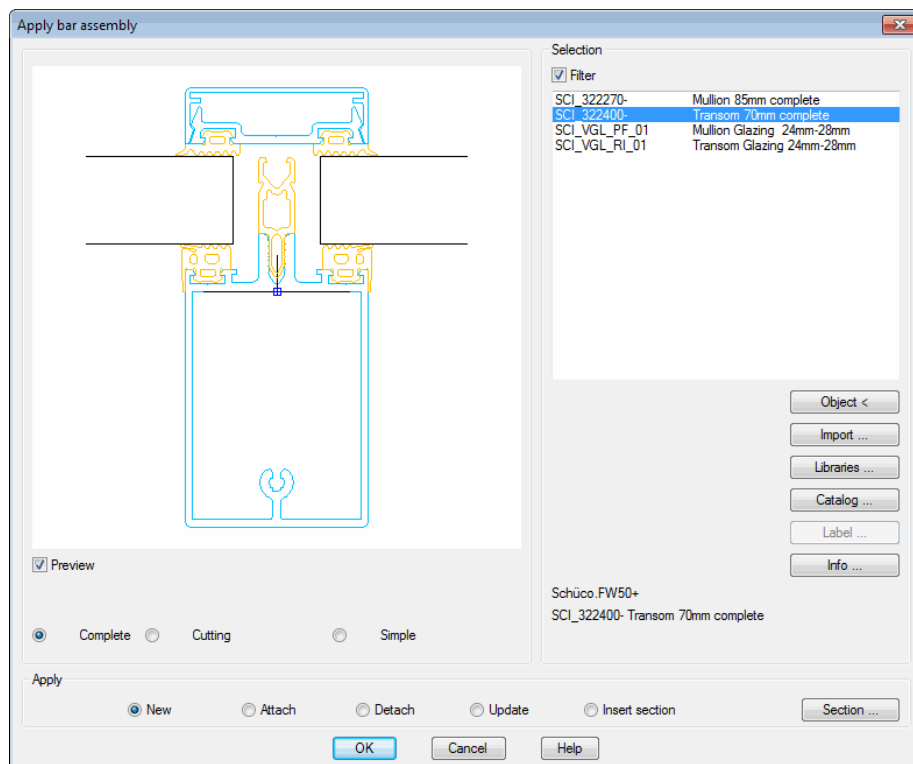
2.4 Apply bar assembly



Ribbon: Tab ATHENA > Group Model > Apply bar assembly
Menu: Model > Apply > Apply bar assembly
Toolbar: ATH Apply
Command input: ath_bar_use

With this routine you can apply bar assemblies in the drawing. A bar assembly consists of one or more components and can be used two dimensionally (as a section) or also three dimensionally (as an extrusion).

Dialog box Apply bar assembly



A description of the dialog box sections Selection and the graphical preview can be found in the Chapter *Object selection* on page 138.

Dialog box section Apply

You mark an option button in order to be able to realize the desired function.

The marked function is executed after clicking OK.



If you have selected a variable bar assembly (e.g. glazing), the Dialog box Parameters for bar assembly is started. Here you can carry out the desired settings for the bar assembly before you use it as a section or extrusion in the drawing.

New

Applies the selected assembly to a line or axis.

After clicking OK the dialog box is closed and the following occurs:

Command prompt*[Object Mode]**Select axis or [Points/Multiple/?]:**Select a system axis (line, polyline segment, circular arc, null bar or bar) with the mouse.**Select **Option Points** to specify the start and end points of the axis.**Use the **Option Multiple** to select several lines.**This option is recommended, for example, to set several glazings on carrier profiles (bars). The options for changing the orientation are not shown according to this option.**You call the help with the option ?.**If you apply them to an ATHENA axis containing nodes (refer also to Chapter Define nodes on page 766), the input request is extended as follows:**Axis [Complete/Segments/?] <Complete>:**You confirm the default <Complete> to create the assembly in the full length on the selected axis.**You select the option Segments to interrupt the assembly at the node points.**After selecting an axis you can determine the orientation of the bar.***Option Points***[Points mode]**Specify the axis start point or [Object/?]:**Use the mouse or enter co-ordinates to specify the axis start point.**Choose the option Objects when you want to select an axis.**Specify the axis end point or [Undo/Object/?]:**Use the mouse or enter co-ordinates to specify the axis end point.**You select the option Undo to start the point entry anew.**Choose the option Objects when you want to select an axis.***Option Multiple***[Multiple mode]**Select objects:**Select several system axes (lines, polyline segments, circular arcs, null bars or bars) with the mouse.**Press the Enter key to terminate the object selection.**After the axis or the second point has been chosen, you see:**Bar orientation [Mirror/ Rotate/ surfaceAxis/ surfaceMiddle/ rotationAngle/ Point/ ?]:**Choose an option to change the bar alignment.**Choose the option Mirror to mirror the bar.**Choose the option Rotate to rotate the bar in 90° steps.**Choose one of the following options.***Option surfaceAxis***Select axis or surface or [Points/?]:**Use the mouse to choose an axis or surface to align the bar in the same orientation.**Choose the option Points to specify an axis by entering two points.***Option surfaceMedian***Select first axis or surface or [Points/?]:**Use the mouse to choose the first axis or surface.*

Select second axis or plane or [Points/?]:

Use the mouse to choose the second axis or surface.



The bar is aligned along the bisecting angle of the defined surface. With this option you can for example align mullions in pyramid roofs.

Option rotateDegrees

Specify rotation angle or [?]:

Enter a rotation angle.

Option Point

Specify point on the weather side or [?]:

Use the mouse to specify a point on the weather side.

After terminating the command by pressing the Enter key, the input request appears again.

[Object Mode]

Select axis or [Points/?]:

Thus you can assign the bar assembly to further axes until you terminate the function by pressing the Enter key.

Attach

Applies the selected bar assembly to a one or more ATHENA axes (null bars). These arise either by executing Analyze axis model or by detaching bar assemblies.



With Attach you can only assign bar assemblies to those ATHENA axes which have not yet had any bar assemblies assigned to them or for which the bar assemblies have been detached.

After clicking OK the dialog box is closed and the following occurs:

Command prompt

Select objects:

You select single axes with the mouse or you select a range with crosses in order to assign this bar assembly to all axes having no assignment.

After execution of the function the output of the number of changed objects follows.

Detach

Deletes the assembly-specific properties of a bar. A null bar is produced by the detachment. A null bar only possesses information on position/orientation, dihedral angle and cutting.

After clicking OK the dialog box is closed and the following occurs:

Command prompt

Select objects:

You select the bars to be detached with the mouse or you cross a range to detach all axes assigned with the current bar assembly.



To detach a number of the same types of bars in one operational step, you select first the bar assembly to be detached from the pick list or use Object < to make it the current assembly.

Update

Implements changes to the assembly on the selected bars. You first select the bar assembly to be updated from the pick list or use Object < to make it the current assembly.



Only bars can be updated which match the current selection.

After clicking OK the dialog box is closed and the following occurs:

Command prompt*Select objects:*

You select the bars to be updated with the mouse or you select a range with crosses to update all axes assigned with the current bar assembly.

Insert section

Inserts the marked assembly as 2D section into the current UCS of the drawing.

After clicking OK the dialog box is closed and the following occurs:

Command prompt*Option Point**Specify insertion point or [Object]:*

Use the mouse or enter co-ordinates to specify the insertion point of the cross-section.

*Select the **Option Object** to attach one or more bar cross-sections to existing bar cross-sections.*

Specify rotated angle<0>:

Enter a rotation angle. Press the Enter key to accept the default angle of 0°.

*Option Object**Select bar cross-section or [Point/?]:*

Select an existing bar cross-section to attach the bar cross-section.

You should use this option, for example, to attach glazing to cross sections of existing carrier profiles.



The cross sections are inserted with their insertion point on the Glazing axis of the existing profile section. If none has been defined, the base point of the profile section is used.

Section ...

Opens the Dialog box Bar cross section, where you can change the properties of the 2D section. You will find further information in the Chapter *Bar cross section* on page 153.

Notes

- In order to change the orientation of a bar, you use the command **Modify ATHENA** or you can double click the bar.
- In order to change the properties of a 2D section, you use the command **Modify ATHENA** or you can double click the section.

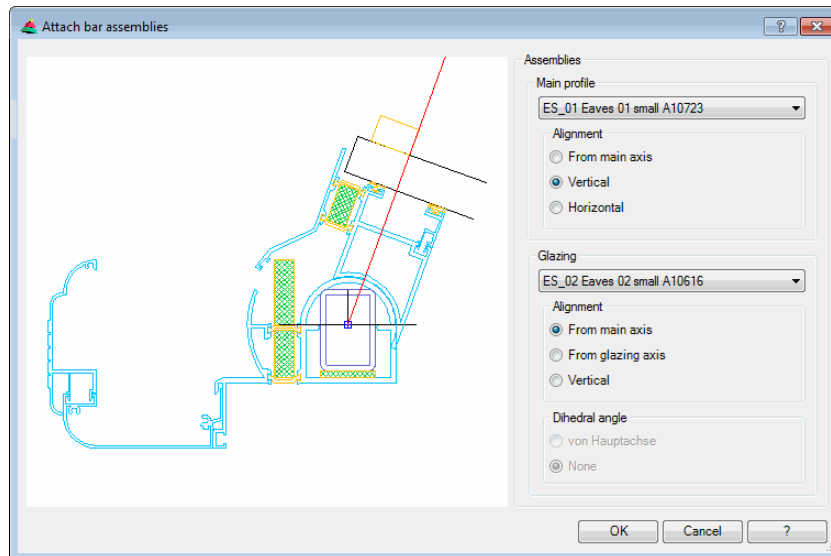
2.5 Assign bar assemblies



Ribbon: Tab ATHENA > Group Model > Assign bar assemblies
Menu: Model > Apply > Assign bar assemblies
Toolbar: ATH Apply
Command input: ath_c3d attach

Assigns two separate bar assemblies (e.g. carrier profile and glazing) to ATHENA axes (null bars). The bar assemblies can be aligned differently.

Dialog box Assign bar assemblies



Dialog box section Main profile

Determines the carrier profile of the construction.

Dialog box section Alignment

From main axis

Accepts the angle of the principal axis for the main profile.

Vertical

Aligns the main profile vertically.

Horizontal

Aligns the main profile horizontally.

Dialog box section Glazing

Determines the glazing of the construction.

Dialog box section Alignment

From main axis

Accepts the angle of the principal axis for the glazing.

Vertical

Aligns the glazing vertically.

Horizontal

Aligns the glazing horizontally.

Dialog box section Dihedral angle

From main axis

Accepts the dihedral angle of the principal axis.

None

Uses the defined angle of the assembly.



With bar assemblies without variable dihedral angles these options have no effect.

When you terminate the dialog box, the following appears:

Command prompt

Select bar or [?]:

Select a null bar to attach the profile combination. For assemblies with variants the Dialog box Parameters for bar assembly follows, where you can set the desired bar parameters.

This input request is repeated until you press the Enter key to terminate it.

2.6 Reassign bar assembly



Ribbon:	Tab ATHENA > Group Model > Reassign bar assembly
Menu:	Model > Apply > Reassign bar assembly
Toolbar:	ATH Apply
Command input:	ath_bar_exchange

With this command you can assign new bar assemblies to several identical bars.

When you start the command, the Dialog box For object selection is started. Here, you select the bar assembly, which you want to reassign. You will find further information about this dialog box section in the Chapter *Object selection* on page 138. When you terminate the dialog box with OK, the following input request appears:

Command prompt

Select bar or [?]:

Select a bar as default. Only bars of the same type are considered during the assignment of the bar assembly.

Select objects:

Select the bars, which you want to assign anew.

This input request is repeated until you conclude the object selection by pressing the Enter key.

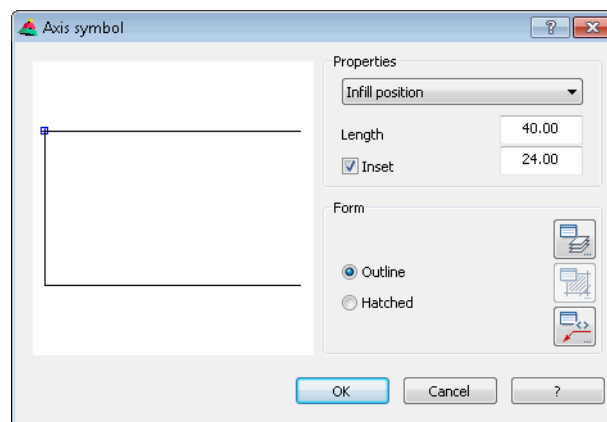
2.7 Axis Symbol



Ribbon:	Tab ATHENA > Group Model > Axis symbol
Menu:	Model > Apply > Axis symbol
Toolbar:	ATH Apply
Command input:	ath_axis

With this routine you generate qualified axis symbols. With these symbols the position and admissibility of glazing is defined. ATHENA detects, for example, when a bar assembly contains an axis symbol, how this assembly is glazed and where the glazing is positioned.

Dialog box Axis symbol



When you call the command, the Axis symbol dialog box is opened. A description of the various sections of the dialog box follows.

Dialog box section Properties

In the section Properties you select the type of symbol. The following are available for selection:

- Glazing axis
- Infill position
- Infill plane

The permissible input fields are released according to the selected symbol.



Fig. 2.3: Glazing axis

The glazing axis defines the position and alignment of glazing. This means that the base point of the symbol forms the insertion point for glazing. The orientation for glazing is defined by the alignment of the symbol. The length of the symbol can be freely set.



Fig. 2.4: *Infill position*

The infill position defines the position and alignment of an infill. This means that the base point of the symbol forms the insertion point for an infill. The orientation for an infill is defined by the alignment of the symbol. The length of the symbol can be freely set. Select the tick box Clamping to specify the clamping thickness of the infill.

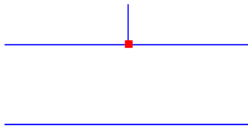


Fig. 2.5: *Infill plane*

The infill plane defines the position and alignment of an assembly, e.g. mounted prongs, on an infill. This means that the base point of the symbol forms the insertion point for the assembly. The orientation for the assembly is defined by the alignment of the symbol. The length of the symbol can be freely set. Select the tick box Clamping to specify the Clamping thickness of the infill.

Dialog box section Display

Here, you can influence the layer, hatch and label properties of the symbol with the Layer ..., Hatch ... and Label ... buttons. You will find further information on this in Chapters *Layer assignment* on page 108, *Hatch pattern assignment* on page 110 and *Label* on page 117.

If you click OK in the dialog box, the settings are saved for the duration of the drawing session and the input request follows:

Command prompt

Specify insertion point or [Object/?]:

Use the mouse or enter coordinates to specify the insertion point.

You use the option Object to generate the symbol on an object. If the object has a base point, the symbol is generated on it and the interrogation of the rotation angle is omitted, because it is accepted by the object.

You call the help with the option ?.

Specify rotation angle or [Object/?] <0>

Specify the rotation angle.

You use the option Object to align the symbol on an object.

Press the Enter key to accept the default angle.

2.8 Infill plane



Ribbon:	Tab ATHENA > Group Model > Infill plane
Menu:	Model > Apply > Infill plane
Toolbar:	ATH Apply
Command input:	ath_panel_plan

Creates an infill plane.

You can convert infill planes quickly into infills by assignment or use them as boundary objects for the Cutting of bars.

When you execute the command, the following appears:

Command prompt

Specify first point for plane or [UCS/?] <UCS>:

Define the first point of the infill plane.

With the option UCS the infill plane is created in the center of the XY plane of the current UCS.

You call the help with the option ?.

Specify second point for plane or [Undo/?]:

Define the first point of the infill plane.

With the option Undo you can reverse the last step.

Specify third point for plane or [Undo/?]:

Define the first point of the infill plane.

The infill plane is shown as a circle with a line. The line indicates the direction (weather side of the infill).

Notes

- With the command Cutting you can use an infill plane as a boundary object for a bar cutting. This has an associative effect, i.e. the bar cutting is adapted when you move or rotate the infill plane.



The direction of the infill plane has no effect on the cutting. The shorter end of the bar is always cut.

- With the command Apply infill you can attach saved glazing or panels to infill planes.

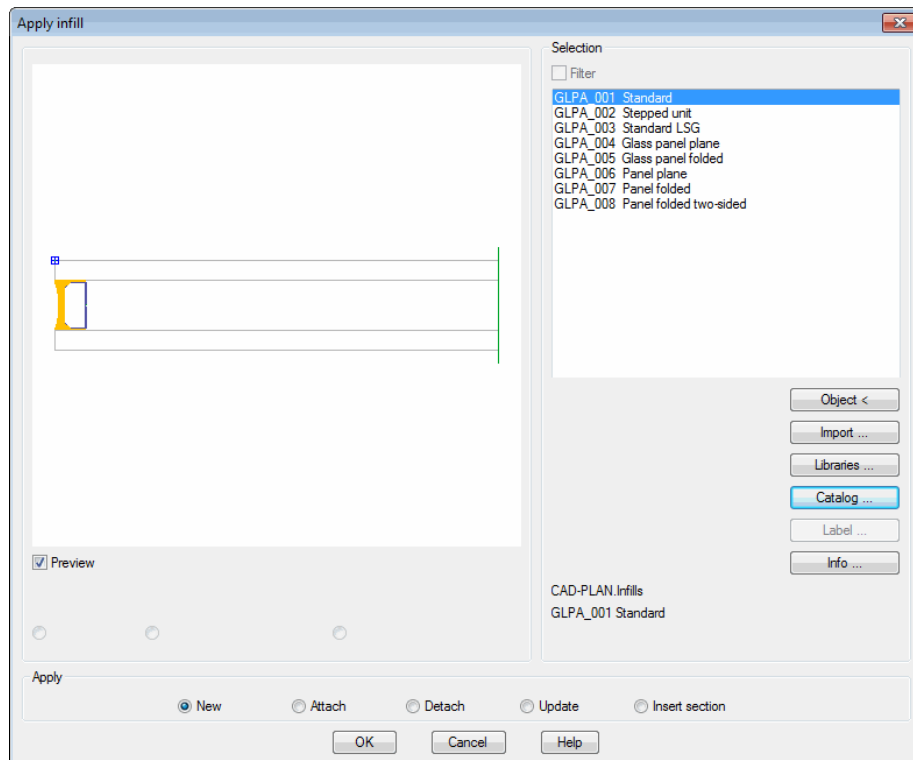
2.9 Apply infill



Ribbon: Tab ATHENA > Group Model > Apply infill
Menu: Model > Apply > Apply infill
Toolbar: ATH Apply
Command input: ath_panel_use

With this routine you can apply infills in the drawing. Infills are glazing or panels; they can consist of several layers and be applied two-dimensionally or three-dimensionally.

Dialog box Apply infill



A description of the dialog box sections Selection and the graphical preview can be found in the Chapter *Object selection* on page 138.

Dialog box section Apply

You mark a field in order to be able to realize the desired function. The marked function is executed after pressing OK.

New

Creates a new 3D infill by selection of enclosing objects.

After clicking OK the dialog box is closed and the following occurs:

Command prompt

Select boundary object or [?]:

Choose a bar as a boundary object for the infill.

You call the help with the option ?.

Select boundary object or [Undo/?]:

Choose another bar as a boundary object.

*Choose the option **UNDO** to repeat the last input request.*

Select boundary object or [Undo/?]:

You can select other boundary objects (at least three intersecting boundary objects are needed). This input request is repeated until the selection of the boundary objects is terminated by pressing the Enter key. Then the Apply infill dialog box is opened.

Attach

Applies the selected infill to a one or more infill symbols. For this, the dialog box is closed and an input request follows. Infill symbols arise through the detachment of infills.

Detach

Deletes the properties of the selected infill and only the infill symbol is retained. For this, the dialog box is closed and an input request follows.

Update

Updates the infill when changes have been made. For this, the dialog box is closed and an input request follows.



Infills can only be updated if they have been inserted into the drawing with the command Apply infill, because only these infills are linked to the saved library object.

Infills, which have been generated with the Infill command, are not referenced to saved objects and cannot therefore be updated.

Command prompt for Assign, Detach and Update

Select objects:

Use the mouse to choose the infills you wish to change.

Insert section

Inserts a section of the marked infill into the current drawing. For this, the Dialog box Infill parameters is opened, where you can change the infill settings. You will find further information in the Chapter *Infill parameters* on page 148.



The sections, which you insert with this command, are linked to the saved library object and can be updated.

After clicking OK the dialog box is closed and the following appears:

Command prompt

Option Point

Specify insertion point or [Object/?]:

Use the mouse or enter co-ordinates to specify the insertion point for the infill.

*Select the **Option Object** to append an infill to one or more existing bar cross-sections.*

You call the help with the option ?.

Specify rotated angle<0>:

Enter a rotation angle. Press the Enter key to accept the default angle of 0°.

These two input requests are repeated until you press the Enter key to terminate the command.

Option Object

Select first bar cross-section or [Point/?]:

Select an existing bar cross-section to append the infill.

Select second bar cross-section or [?]:

Select a further bar cross-section (flush with the one previously selected) to fit the infill between the two bar cross-sections.

If you press the Enter key at this point, the infill is positioned with interrupted representation on the first bar cross-section.

*Select the **Option Point** to append an infill to one or more existing bar cross-sections.*

These requests are repeated until you press the Enter key to terminate the command.



The infills are inserted on the Infill position of the previous bar cross-section. If none has been defined, an appropriate notice appears in the command line.

Notes

- With infills, which are inserted into the drawing as 3D objects, the weather side of the infill is marked by a symbol.
- The layers of the infill are drawn on material-dependent layers which you can manage in the Dialog box Material properties.
- To change the properties of an infill, use the command **Modify ATHENA** or double click on the infill with the mouse.
- If **conflict messages** are obtained on positioning infills with the aid of infill positions, then the infill positions are probably not in alignment. If the offset is so slight that the rubber seal can accommodate it, you can influence the accuracy in the Dialog box Presettings. You will find further information in the section *Presettings* on page 91.

Associated commands

- Infill
- Infill manager

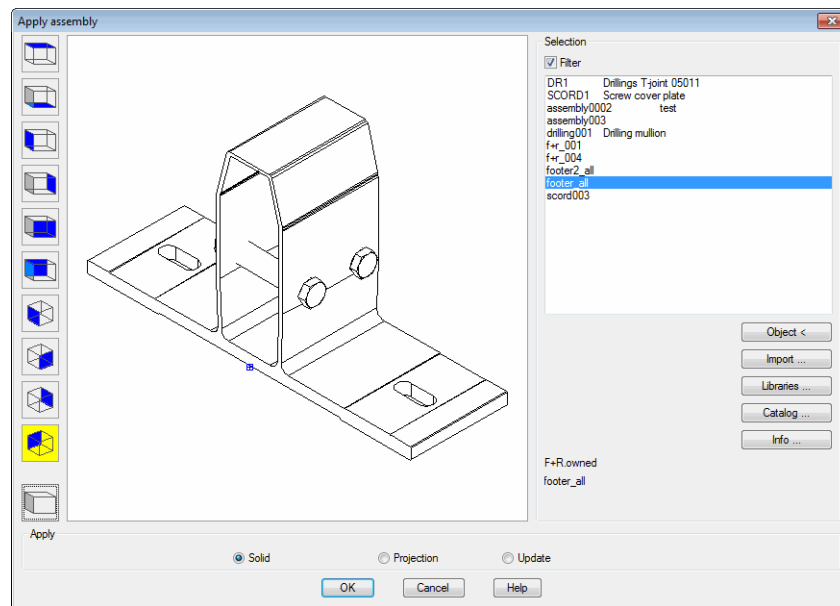
2.10 Apply Assembly



Ribbon: Tab ATHENA > Group Model > Apply assembly
Menu: Model > Apply > Apply assembly
Toolbar: ATH Apply
Command input: ath_bgr_use

With this function you can insert an assembly into the current drawing.

Dialog box Apply assembly



Display section

On the left side the dialog box contains a preview of the current assembly.

Changing the view

Using the buttons on the left side you can switch between various permanently defined views of the assembly. Apart from the wire frame display, a hidden display is possible



The current setting is highlighted in color.



Plan

Shows the complete assembly as viewed from above.



Side elevation from the left

Shows the assembly as a side elevation from the left.



Side elevation from the right

Shows the assembly as a side elevation from the right.



Front elevation

Shows the assembly as viewed from the front.



Rear elevation

Shows the assembly as viewed from the back.



ISO view SW

Shows the assembly as the isometric view from the southwest.



ISO view SE

Shows the assembly as the isometric view from the southeast.



ISO view NE

Shows the assembly as the isometric view from the northeast.



ISO view NW

Shows the assembly as the isometric view from the northwest.



Hidden view

Masks out the hidden edges of the assembly solid in the current view.

Dialog box section Apply

Solid

Inserts a 3D solid of the assembly into the drawing.

Projection

Inserts a 2D projection of the currently set preview of the assembly into the drawing.

Update

Updates assemblies which have been linked to bars by grid allocation or by joining bars.



Singly inserted assemblies cannot be updated. All assemblies of the selected bars are always updated. The assembly selection here has no effect.

Dialog box section Selection

You will find a detailed description of the functions of the dialog box in the Chapter *Object selection* on page 138.

Select an assembly from the list and click OK to insert the assembly into the current drawing. If you are using an assembly with variants, the Dialog box Assembly parameters appears, otherwise the following:

Command prompt for the options Solid and Projection

Specify insertion point or [?]:

Specify the insertion point of the assembly.

You call the help with the option ?.

Specify rotation angle or [?] <0>:

Specify the rotation angle of the assembly or adopt the default angle with a right click.

Command prompt for the option Update

Select objects

Select the bars, whose linked assemblies you want to update.

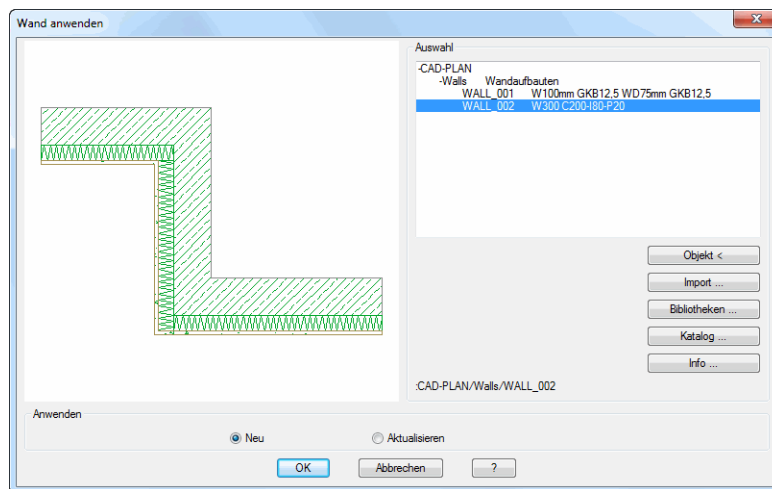
2.11 Apply wall



Ribbon: Tab ATHENA > Group Model >
Menu: Model > Apply > Apply wall
Toolbar: ATH Apply
Command input: ath_wall_use

With this command you can produce in the drawing cross sections of walls which have been previously saved with the Wall manager.

Dialog box Apply wall



A description of the dialog box sections Selection and the graphical preview can be found in the Chapter *Object selection* on page 138.

Dialog box section Apply

New

Creates a new wall. For this, the Dialog box Wall parameters is opened, where you can define further settings before the wall is inserted.

Update

Updates an existing wall cross section, the properties of which have been changed.



Walls can only be updated if they have been inserted into the drawing with the command Apply wall, because only these walls are linked to the saved library object.

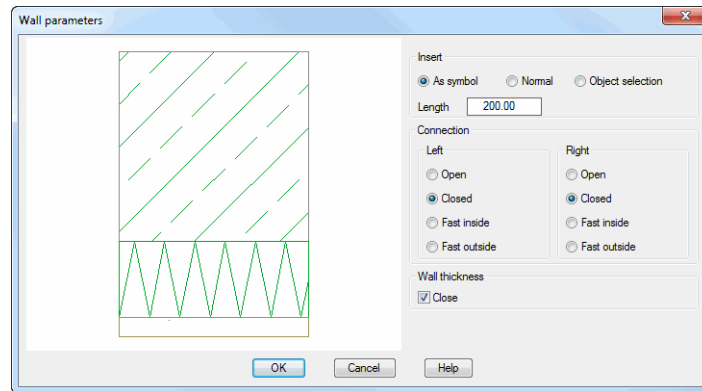
Walls, which have been generated with the Wall command, are not referenced to saved objects and cannot therefore be updated.

Command prompt

Select objects:

Select the walls which you want to update and then press the Enter key.

x object(s) modified

Dialog box Wall parameters**Dialog box section Paste****As symbol**

Creates the wall as a segment with fixed length. The dialog box is closed and an input request follows:

Length

Defines the length of the wall section when you insert the wall as a symbol.

Normal

Generates a wall as a series of lines. The dialog box is closed and an input request follows:

Object selection

Generates a wall by selecting a line object. The dialog box is closed and an input request follows:

Dialog box section Left/Right

Determines the type of connection to the right or left side of the wall.

Open

Opens the wall on the corresponding side, for example for an interrupted representation.

Closed

Closes the wall on the corresponding side.

Abutment inside

Shows a mounting on the inside of the wall with the entered thickness and width.

Abutment outside

Shows a mounting on the outside of the wall with the entered thickness and width.

Dialog box section Wall thickness**Close**

Closes the lowermost layer of the wall structure.

When you terminate the dialog box with OK, an input request follows depending on the input option selected.

Wall as symbol**Command prompt***Specify insertion point:*

Define the insertion point of the wall.

Specify rotated angle<0>:

Specify the rotation angle of the wall or press the Enter key to accept the default angle.

Generate wall with the Normal option

Command prompt

Specify starting point of wall or [?]:

Specify the starting point of the wall.

You call the help with the option ?.

Specify next point or [Arc point/Undo/Direction/Length/?]:

Specify the second point of the wall.

Use the option Undo to repeat the previous input request.

Specify side for wall thickness or [Central/Undo/?] <Central>:

Specify on which side of the polyline the wall structure is to be created.

With the option Central the wall structure is positioned on both sides of the polyline.

Specify next point or [Arc point/Undo/Direction/ANgle/Length/?]:

Specify the next point of the wall.

*With the **Option Arc point** you can specify an arc point. Further input requests follow.*

*With the **Option Direction** you can specify the direction for the next wall section. Further input requests follow.*

*With the **Option Angle** you can specify an angle. Further input requests follow.*

*With the **Option Length** you can specify the length for the wall section. Further input requests follow.*

This input request is repeated. Press the Enter key to terminate the command.

Specify side for wall thickness or [Central/Same/Undo/?] <Same>:

Specify on which side of the polyline the wall structure is to be created.

Press the Enter key to draw the wall on the side previously specified.

Option Arc point

Specify arc point or [Line/Undo/?]:

Specify an arc point.

*With the option **Line** you can draw straight wall sections.*

Specify endpoint of the arc or [Undo/Line/Arc point/?]:

Enter the end point of the arc or select an option.

Option Direction

Specify direction or [Length/Angle/Point/Undo/?]:

Specify the direction of the wall section or select an option.

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the wall section by pointing with the mouse or entering the length.

Option Angle

Specify angle or [Length/Direction/Point/Undo/?]:

Define the angle relative to the last wall section by pointing with the mouse or entering the angle or selecting an option.

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the wall section by pointing with the mouse or entering the length or selecting an option.

Option Length

Specify limb length or [Direction/Angle/Point/Undo/?]:

Define the length of the wall section by pointing with the mouse or entering the length or selecting an option.

Specify angle or [Length/Direction/Point/Undo/?]:

Define the angle relative to the last wall section by pointing with the mouse or entering the angle or selecting an option.

Generate wall with the Object selection option

Command prompt

Select Polyline or [?]:

Select a polyline to create the wall.

You call the help with the option ?.

Specify side for wall thickness or [Central/?] <Central>:

Specify on which side of the polyline the wall structure is to be created.

With the option Central the wall structure is positioned on both sides of the polyline.

Delete polyline [Yes/No/?] <Yes>:

Select the option Yes to delete the polylines..

The polyline is retained with the option No.



All walls, which you insert with this command, are linked to the saved library object and can be updated.

Associated commands:

- Wall manager
- Wall

2.12 Apply frame element

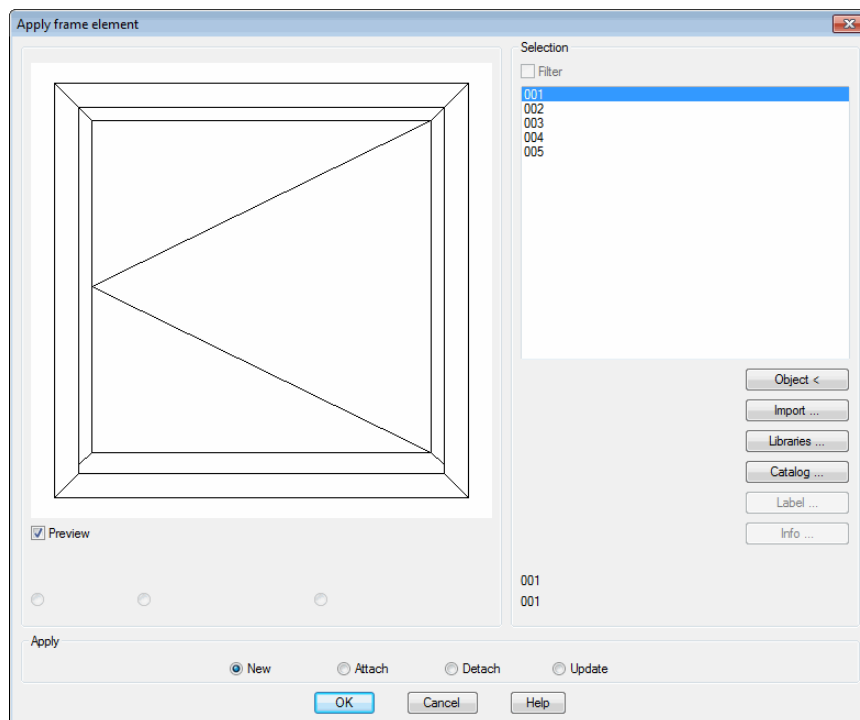


Ribbon: Tab ATHENA > Group > Model
Menu: Model > Apply > Apply frame element
Toolbar: ATH Apply
Command input: ath_elem_use

Uses frame elements (views of windows, doors) in the drawing.

Use the Frame Element Manager to define frame elements and to save them.

Dialog box Apply infill



A description of the dialog box sections Selection and the graphical preview can be found in the Chapter *Object selection* on page 138.

Dialog box section Apply

New

Creates a new frame element When you terminate the dialog box with OK, the following appears:

Command prompt

Option Area

Specify point in the area or [Corner points/Object selection/?]:

Enter a point within a closed area to specify the element outline.

*Select the **Option Corner points** to specify the corner points of the element.*

*Use the **Option Object selection** to select an object.*

You call the help with the option ?.

Option Corner points

Specify corner point or [Area/Object selection/?]:

Specify the first corner point of the element.

*Select the **Option Area** to specify a point in a closed area.*

Specify corner point or [Area/Object selection/Undo/?]:

Specify the second corner point of the element or select an option.

*Choose the option **Undo** to repeat the last input request.*

Specify corner point or [Area/Object selection/Undo/?]:

Specify the third corner point of the element or select an option.

Specify corner point or [Area/Object selection/Undo/Close/?] <Close>:

Enter the fourth corner point or select an option.

*Input **ENTER** for Close to close the outline.*

Option Object selection

Select object:

Select a closed polyline or lines which form an outline and the end points of which touch in order to define the element outline.

Attach

Applies the selected frame element to a one or more null elements. For this, the dialog box is closed and an input request follows.



Null elements (frame elements without assignment) arise when frame elements become detached.

Detach

Deletes the properties of the selected frame element and only the blank frame element is retained. For this, the dialog box is closed and an input request follows.

Update

Updates the frame element when changes have been made. For this, the dialog box is closed and an input request follows.



Infills can only be updated if they have been inserted into the drawing with the command Apply infill, because only these infills are linked to the saved library object.

Infills, which have been generated with the Infill command, are not referenced to saved objects and cannot therefore be updated.

Command prompt for Assign, Detach and Update

Select objects:

Use the mouse to select the objects to be changed.

2.13 Object to solid body



Ribbon:	Tab ATHENA > Group Model > Object to solid body
Menu:	Model > Apply > Object to solid body
Toolbar:	ATH Apply
Command input:	ath_obj_solid

With this command you can convert two-dimensional standard parts, semi-finished products, screwed joints and drilled holes as well as projections of them into three-dimensional ATHENA solids.

Command prompt

Select objects:

*Select the objects which you want to convert to ATHENA solids. The selected objects are immediately converted into ATHENA solids.
This input request is repeated until you press the Enter key.*

Notes

- Profile section cannot be converted, because they do not have any length information.
- Profile solids are "known" bar assemblies, the names of which consist of the TMP and an incremented number.
You will find further information on bar assemblies in the sections *Bar Assembly Manager* on page 632 and *Apply bar assembly* on page 738.
- Sliced or trimmed 2D projections retain their sliced planes or cutouts when the boundary objects are similarly converted. For this, all parts have to be chosen in a selection set.
- Drilled holes, screwed joints and joining elements can also still be modified as 3D objects with a double click.

2.14 ATHENA Extrusion



Ribbon:	Tab ATHENA > Group Model >
Menu:	Model > Apply >
Toolbar:	ATH Apply
Command input:	ath_extr

Extrudes 2D outlines (circles or polylines) and creates a 3D solid. This is primarily used as a process on a bar. Extrusions can be modified with grips and can be saved as library objects.

When you execute the command, the following appears:

Command prompt

Extrusion [Objects/Fetch/?] <Objects>:

Choose the option Objects to extrude outlines.

Choose the option Fetch to use an existing extrusion.

Select objects:

Select the outlines which you want to extrude. You can select many outlines (an outer outline and optionally a number of inner outlines).

State the height of the extrusion or [Taper angle/Material/?]:

Determine the height of the extrusion.

Option Taper angle

Specify taper angle or [?] <0>:

Determine the taper angle of the extrusion. Press the Enter key to accept the default angle of 0°.

After specifying the taper angle, the selected outlines are extruded.

Option Fetch

Starts the Dialog box For object selection. Here, you can insert extrusions already saved into the drawing. You will find further information on this in the Chapter *Object selection* on page 138.

Option Material

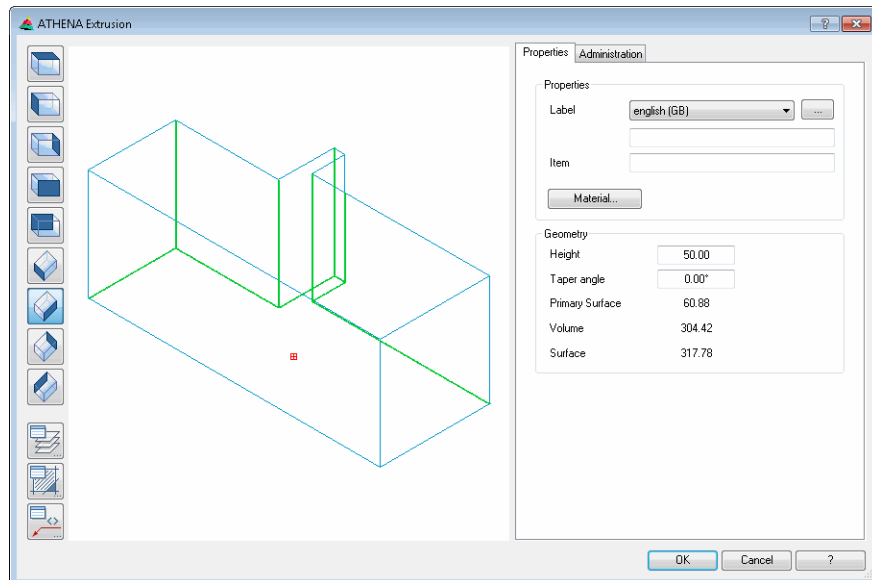
Opens the Dialog box Material selection, where you can optionally select a material for the extrusion. You will find further information on this in the Chapter *Material selection* on page 123.

Properties of extrusions

Extrusions can be used immediately without saving them, for example as a process (notch) on a bar.

You can save an extrusion or modify its properties. For this, double click the extrusion and go through the required options in the Dialog box ATHENA Extrusion.

Dialog box ATHENA Extrusion

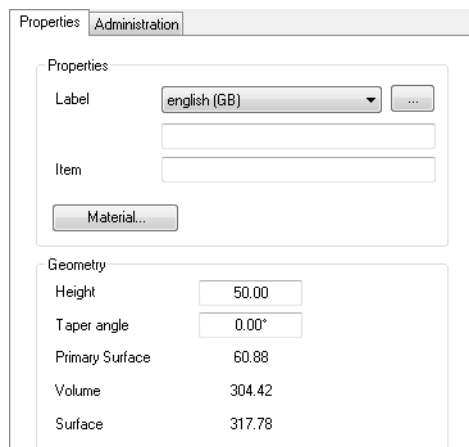


On the left side the dialog box includes a preview of the extrusion as well as various control elements and display options for modifying the preview. You will find further information in the sections *Object views* on page 102 and *Display options* on page 104.

On the right side of the dialog box there is the operating section with the tabs:

- Properties
- Manager

Tab Properties



Dialog box section Properties

Label

Defines the extrusion label. You can save the label in various languages. To do this, choose the required language from the list.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Item

Defines the extrusion item number.

Material

Opens the Dialog box Material selection, where you can select a material. You will find further information on materials in Chapters *Material* on page 81 and *Material selection* on page 123.

Dialog box section Geometry

Height

Defines the height of the extruded outlines.

Taper angle

Defines the taper angle of the extruded outlines.

As additional information the primary surface, volume and surface of the extruded outline are displayed.

2.15 Bar work



Ribbon:	Tab ATHENA > Group Model > Bar work
Menu:	Model > Apply > Bar work
Toolbar:	ATH Apply
Command input:	ath_bar_work

Sets manual processes on a bar.

A manual process can be an ATHENA Extrusion or a Drilled hole. The attachment of the process to the bar occurs in the drawing.

All bar components cut by the processing outline are processed. The cutting classes of the bar components are not taken into account!

When you execute the command, the following appears:

Command prompt

Select bar or [?]:

Select the bar which you want to process.

Add

Select process or [Add/Remove/?]:

Select a processing outline (extrusion or drilled hole) to attach it to the bar.

With the option Remove you can remove existing processes from the bar.

Remove

Select process or [Add/Remove/?]:

Select the process outlines which you want to remove. The process on the bar is removed without deleting the processing outline.



Processes are not coupled to the respective bar. Moving the processing outline or the bar leads to a change of position of the process on the bar! Similarly, the process on the bar is removed when you delete the processing outline.

2.16 Associative bar work



Ribbon:	Tab ATHENA > Group Model > Associative bar work
Menu:	Model > Apply > Associative bar work
Toolbar:	ATH Apply
Command input:	ath_bar_work1

Sets associative processes on a bar.

An associative process can be an ATHENA Extrusion or a Drilled hole. The attachment of the associative process occurs in the drawing. If required, you can edit the rules of arrangement.

When you execute the command, the following appears:

Command prompt

Select bar or [?]:

Select the bar which you want to process.

Select the process

Select objects:

Select the processing outlines to attach them associatively to the bar.

This input request is repeated until you terminate the command by pressing the Enter key.

The processing of mullions is carried out directly. In doing so, the selected processing outlines are combined to an assembly which processes the bar (drills, notches...). The original processing outlines are removed. You can process the assembly with the Assembly Manager.



The processing is saved on the bar. A change of position by moving the bar or the processing outline in the drawing environment is not possible.

With the command Apply arrangement you can change the position of the processes or expand them using rules. You will find further information in the section *Apply arrangement* on page 771.

2.17 Define nodes



Ribbon:	Tab ATHENA > Group Model > Define nodes
Menu:	Model > Apply > Define nodes
Toolbar:	ATH Apply
Command input:	ath_c3d_nodes

With this function you can manually add a node to a bar which meets a second one or you can delete the node. This may be necessary with bars which are to form a T-shaped joint in a construction.

Nodes can also be formed automatically with the command **Analyze axis model**.

Command prompt

Select bar or [?]:

You choose the bar on which nodes are to be formed due to bars coming together. You call the help with the option ?.

Select bar for nodes or [Remove/Add/?]:

Now you select one or more bars, one after the other, which are to form nodes on the initial bar. You terminate the command by pressing the Enter key.

2.18 Apply processes to infill



Ribbon: Tab ATHENA > Group Model > Apply processes to infill
Menu: Model > Apply > Apply processes to infill
Toolbar: ATH Apply
Command input: ath_panel_hole

With this routine you can attach processes to existing infills (glazing or panels).
 With multilayer infills these processes can be applied specifically to single layers.

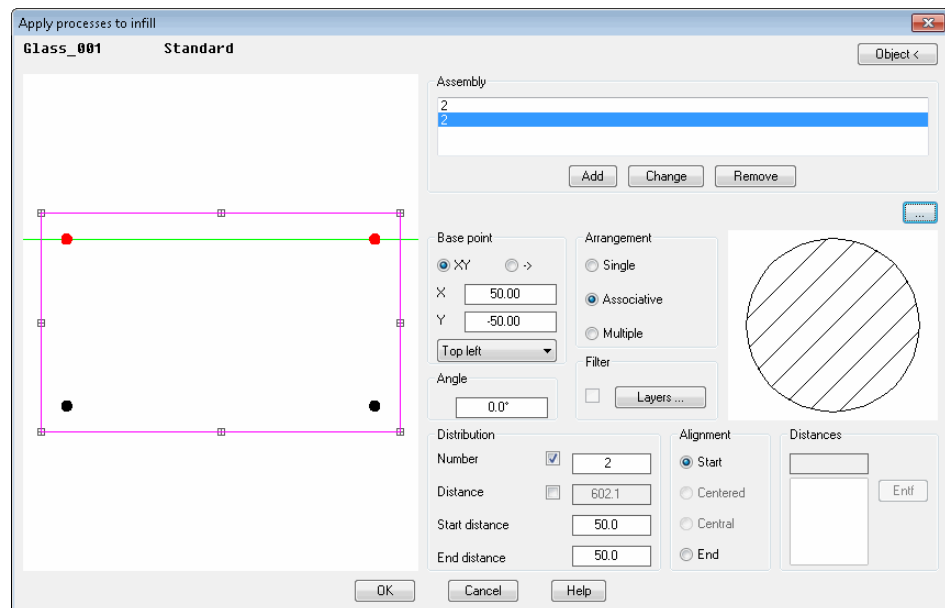
Command prompt

Select infill:

Select the infill to which you want to assign processes.

After the object selection a dialog box appears in which you can define the processes. Where the selected infill already possesses processes, these are displayed.

Dialog box Apply processes to infill



Dialog box section Graphics

In the display box all assemblies are shown which have been used with this function.

Dialog box section Assembly

The list shows the processes which are assigned to the current infill. You can select the processes in the list in order to change their properties.

Add

Adds the current processing definition to the list (selected sheet surface).

Modify

Replaces the marked entry in the list with the current processing definition.

Remove

Removes the selected process from the list.

[...]

Opens the Dialog box Process manager in which you can edit the processes. You will find further information in the Chapter *Process manager* on page 715.



The positioning of the assemblies always occurs along a construction line which is determined by the base point, angle and reference point. The intersection points of this line with the delimiting axes of the panel represent the computational basis for the distribution of the assemblies.

Dialog box section Base point

Specifies the base point used for the process. Enter the X and Y coordinates in the appropriate input fields. Since a process can also be applied many times in a straight line, a line is displayed in the preview. The coordinates refer to the reference point which you can select from the list. The changes only become effective when you click the button Modify.

Angle

Defines the angle of the process. The rotation occurs in the counterclockwise direction. The changes only become effective when you click the button Modify.

Elevation

The display provides a visual check of the current process. The insertion point of the process is identified in color.

Dialog box section Arrangement

Single

Arranges the processes once at the defined base point on the current surface.

Associative

Arranges processes regularly, according to the defined distribution, on a straight processing line on the current surface. With the associative arrangement additional dialog box sections are released where you can define the distribution rules.

Multiple

Arranges processes irregularly, according to the defined distances, on a straight processing line on the current surface. With the multiple arrangement an additional dialog box section is released where you can define the distances.

Dialog box section Filter

Layers

Opens the Dialog box Include layers, where you can define on which layers the process is to be applied. You will find further information on this in "Dialog box Include layers" auf Seite 770.

Dialog box section Distribution

Number

Defines the number of processes.

Distance

Defines the spacing between the processes.

Start distance

Defines the distance of the process from the start of the infill.

End distance

Defines the distance of the process from the end of the infill.

Dialog box section Alignment**Start**

Sets the starting point of the process series taking into account the starting distance at the start of the infill.

Centered

Centers an odd number of processes on the surface. The automatic system included prevents the specification of an even number of processes.

Central

Positions an even number of processes centrally on the infill. The automatic system included prevents the specification of an odd number of processes.

End

Sets the starting point of the process series taking into account the starting distance at the end of the infill.

Dialog box section Distances

Defines the spacing of the processes. Enter in each case the spacing of the processes in the input field and confirm them to transfer them into the list.

Remove

Removes the selected entry from the list.

OK terminates the dialog box and applies the processes to the selected infill.

Cancel closes the dialog box and discards the processes.

2.19 Assign processes to infill



Ribbon:	Tab ATHENA > Group Model > Assign processes to infill
Menu:	Model > Apply > Assign processes to infill
Toolbar:	ATH Apply
Command input:	ath_panel_hole1

With this routine you can apply outlines as processes to existing infills (glazing or panels). With multilayer infills these processes can be applied specifically to single layers.

To do this you draw an outline which is to be used as a process and place it on the infill.

When you call the command, the following input request appears:

Select infill:

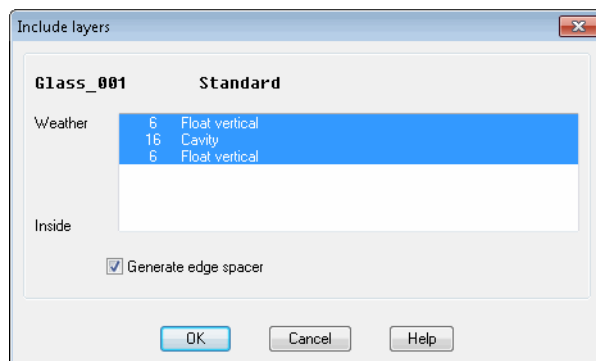
Use the mouse to choose the infill.

Select outline for process or [?]:

Select the outline you want to use as a process with the mouse.

After selecting the process outline, a dialog box opens where you can define the layers to be used.

Dialog box Include layers



Here you select the layer on which the process is to be applied. To select a number of layers, keep the shift key pressed when selecting. If the infill contains a layer with the material type Cavity, the tick box Generate edge spacer can be set. This has the effect of producing an edge spacer along the processing outline.

When you quit the dialog box with OK, the following input request appears:

Specify the infill reference point or [?]:

You select a reference point on the infill to which the process is to relate.

2.20 Apply arrangement



Ribbon:	Tab ATHENA > Group Model > Apply arrangement
Menu:	Model > Apply > Apply arrangement
Toolbar:	ATH Apply
Command input:	ath_bar_bgr

With this function you can assign assemblies to a bar via defined arrangements. Such an arrangement can be created directly or, if already present, assigned. More details on the subject of defining bar arrangements can be found in the Chapter *Arrangement manager* on page 718.

Any number of arrangements can be assigned to a bar. Select an appropriate bar for the assignment of bar arrangements.

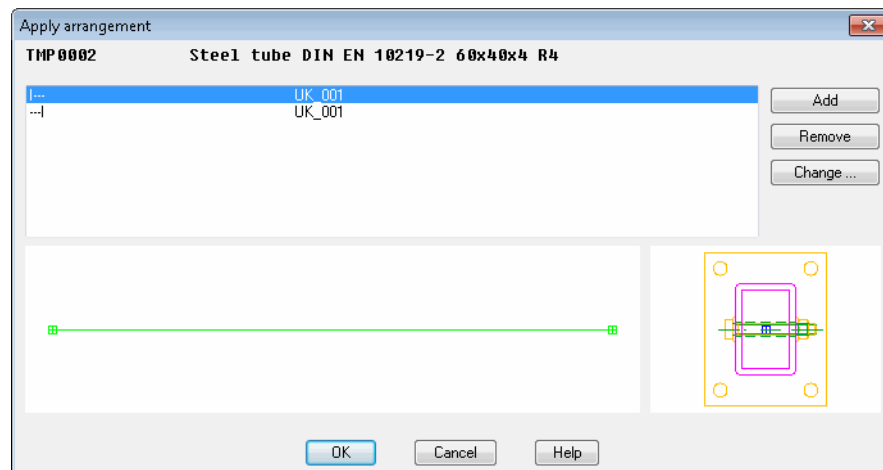
Command prompt

Select bar or [?]:

You select an existing bar from the current drawing by picking. You call the help with the option ?.

A dialog box with a list of any already assigned bar arrangements appears.

Dialog box Bar assemblies/processes



The list provides information about already assigned arrangements and displays their names and item numbers as well as the names of the assemblies. The two previews show schematically the distribution and orientation of the assembly.

Add

Opens the Dialog box Arrangement manager where you can describe an arrangement anew or select a saved one and import it. You will find further information in the Chapter *Arrangement manager* on page 718.

Remove

Deletes the active entry in the list.

Modify

Opens the Dialog box Arrangement manager, where you can edit the selected arrangement. You will find further information in the Chapter *Arrangement manager* on page 718.

2.21 Cutting



Ribbon: Tab ATHENA > Group Model > Cutting
Menu: Model > Apply > Cutting
Toolbar: ATH Apply
Command input: ath_st3d_stoss

With this command you can automatically create profile cuttings. Once you have selected an object to be trimmed, you can choose the boundary objects in a dialog box and assign the cuttings to the relevant component.

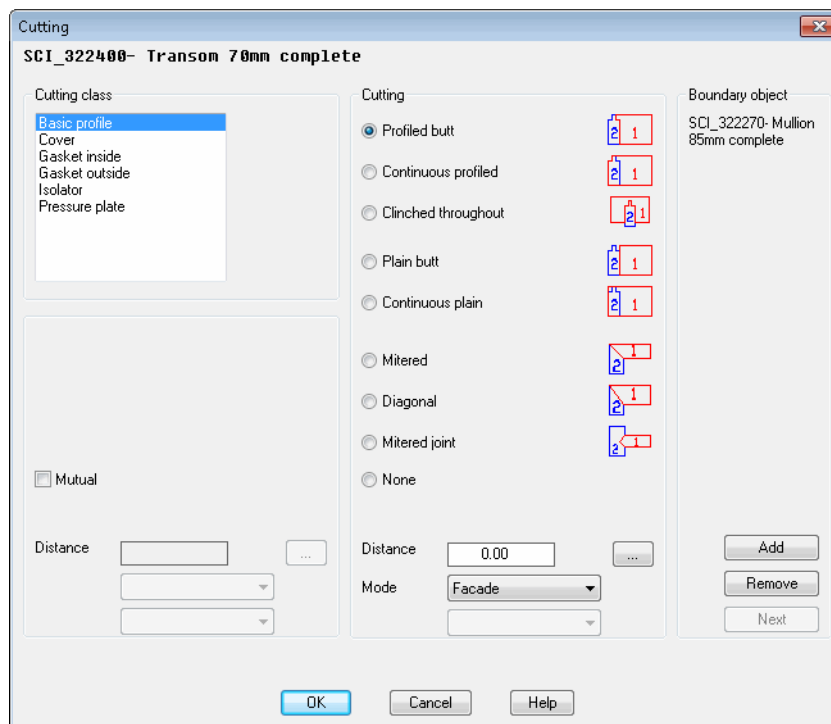
When you execute the command, the following appears:

Command prompt

Select bar for cutting:

Use the mouse to choose the bar you wish to cut. At this point you can only choose one object. After selecting the object, the Profile cutting dialog box appears.

Dialog box Profile cutting



Dialog box section Cutting class

List

Displays the cutting classes of the selected bar (assembly) assigned to the components. Here, mark one or more components and select a cutting.

Mutual

Causes the boundary object also to be trimmed. This option is often used for

diagonal cuts and miter cuts.

The following cuttings are used for the boundary object.

Butt-joining bar	Boundary object
Profiled butt	Continuously profiled
Continuously profiled	Profiled butt
Clinched throughout	Mutual not possible
Plain butt	Continuous plain
Continuous plain	Plain butt
Mitered	Mitered
Diagonal	Diagonal
Mitered joint	Mutual not possible

Distance

Changes the distance of the boundary object by the entered value. A positive value shortens the component, a negative value lengthens it.

Selection menu

Specifies a component as a clinch. If a component has been identified as a component in a bar assembly, you can select this process here. The process clinches the boundary object with the intersection of the joining bar.

This is only possible with L-joints with the profiled types of cutting. Furthermore, a component must have been defined as process for the boundary object.

Dialog box section Cutting

Determines the cutting of the marked components.

For example, it is possible to have a straight butt joint between the mullion and transom cover sections while having a profiled fitted joint between the transom and the mullion themselves.

Profiled butt

Cuts the marked component at the joint side of the boundary object and clinches the bar end according to the cutting outline.

Continuously profiled

Cuts the marked component at the opposite side of the boundary object and clinches the bar end according to the cutting outline.

Clinched throughout

Clinches the marked component according to the cutting outline of the boundary object.

Plain butt

Cuts the marked component on the joint side of the boundary object.

Continuous plain

Cuts the marked component on the opposite side of the boundary object.

Mitered

Cuts the marked component for mitering (bisecting the angle).

Diagonal

Cuts the marked component diagonally.

Mitered joint

Joins the joining component with a mitered joint to a through-running component.



This cutting option is only available with T-joints. With I and L-joints the option is grayed out.

None

Does not realize any cutting. If a cutting was defined for the component previously, it is removed.

Distance

Changes the distance of the boundary object by the entered value. A positive value shortens the component, a negative value lengthens it.

Mode

Defines the cutting mode for the joining component.

The following cutting modes can be selected:

- With the joint cuttings Plain butt, Continuous plain:
 - Basic - cuts the joining component to the enclosing rectangle of the cutting outline of the boundary object.
 - Axis - cuts the joining component to the axis of the boundary object.
- With the cutting Profiled butt:
 - Basic - cuts the joining component to the cutting outline of the boundary object. Protruding areas are cut off on the opposite side of the cutting outline.
 - Facade - cuts the joining component to the cutting outline of the boundary object. Protruding areas are cut off on the joint side of the cutting outline.

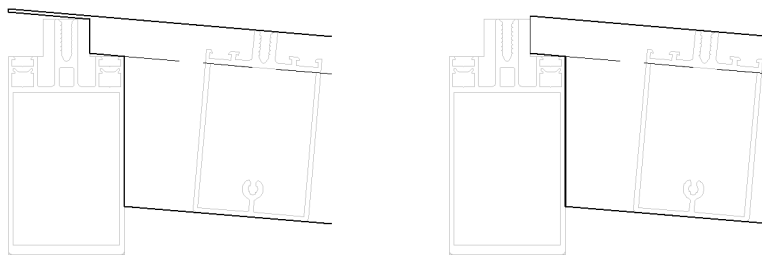


Fig. 2.6: Cutting modes: left basic, right facade

[...]

Makes further cutting settings available. To do this the Dialog box Cutting options is started.

Selection menu

Specifies a component as a clinch. If a component has been identified as a component in a bar assembly, you can select this process here. The process clinches the boundary object with the intersection of the joining bar.

This is only possible with T-joints with the profiled types of cutting. Furthermore, a component must have been defined as process for the boundary object.

Dialog box section Boundary object

In the Boundary object section you can add or remove objects to which the selected bar is to be trimmed using the Add and Remove buttons. The Next button allows you to change between the selected boundary objects. The name of the active boundary object is displayed above the buttons. If you have only selected one boundary object, the Next button is hidden in gray.

If you close the dialog box with OK, the profile cutting is implemented on the selected bar.

Notes

- You can use bars as well as infills as boundary objects.
- With this command you can also cut 2D projections from standard parts and semi-finished products.

2.22 Copy cutting



Ribbon:	Tab ATHENA > Group Model > Copy cutting
Menu:	Model > Apply > Copy cutting
Toolbar:	ATH Apply
Command input:	ath_st3d_st_copy

With this command you can transfer the cuttings from one profile to another.

Command prompt

Select bar as default or [?]:

Select the bar whose cutting you would like to copy.

You call the help with the option ?.

Select objects:

Select the destination objects to which the cutting is to be copied.

x object(s) modified

ATHENA shows the number of modified objects.

2.23 Remove cutting



Ribbon:	Tab ATHENA > Group Model > Remove cutting
Menu:	Model > Apply > Remove cutting
Toolbar:	ATH Apply
Command input:	ath_bar_cut_rem

With this command you can remove the cutting information from one side of a cut bar.

Command prompt

Select Polyline or [?]:

Select the bar on the cut side where the cutting is to be removed.

2.24 Display associated bar nodes



Ribbon:	Tab ATHENA > Group Model > Display associated bar nodes
Menu:	Model > Apply > Display associated bar nodes
Toolbar:	ATH Apply
Command input:	ath_c3d_nodes_orbit

With this command you activate the 3D orbit with the selected bar and its associated bars.

Command prompt

Select bar or [?]:

Select the bar which is to be displayed with its associated nodes in 3D orbit.

You will find further information about the 3D orbit in your AutoCAD documentation.

2.25 Slice Bar



Ribbon:	Tab ATHENA > Group Model > Slice bar
Menu:	Model > Apply > Slice bar
Toolbar:	ATH Apply
Command input:	ath_st3d_slice

With this command you can slice the components of a bar on a plane which you must define by specifying three points.

Command prompt

Select bar to slice:

With the mouse to click the bar you wish to slice.

Slice plane [Add/Shorten/eXit/?] <Add>:

*Choose the **Option Add** to create a new slice plane.*

*Select the **Option Shorten** to change the length of the bar assembly.*

*Choose the option **eXit** to terminate the command.*

*You call the help with the option **?**.*

Option Add

Specify first point of slice plane or [Undo/?]:

Determine the first point in the plane on which you would like to slice the bar.

Choose the option Undo to repeat the previous input request.

Specify the second point of slice plane or [Undo/?]:

Determine the second point in the plane on which you would like to slice the bar.

Specify the third point of slice plane or [Undo/?]:

Determine the third point in the plane on which you would like to slice the bar.

With this third point the cutting plane is defined. ATHENA now starts the Dialog box Cutting.

Option Shorten

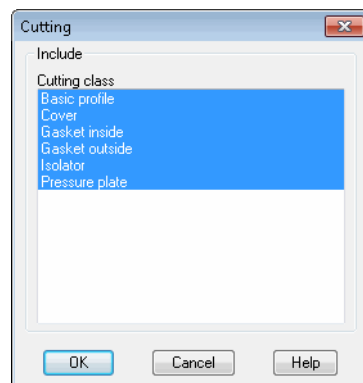
Specify end of bar or [Undo/?]:

With the mouse click the side of the bar assembly to be shortened.

Specify distance or [Undo/?]: <1>:

Specify the distance for the shortening (a negative value extends the bar assembly). ATHENA now starts the Dialog box Cutting.

Dialog box Cutting



In this dialog box you click the component parts of the bar which ATHENA is to slice. You can click a number of component parts by pressing the CTRL or SHIFT key (Windows standard).

Options, if you have already defined one or more slice planes for a bar.

Cutting XXX

ATHENA shows the name of the current cutting.

Slice plane [Add/Shorten/Trim/Remove/Next/Previous/eXit/?] <Add>:

*Choose the **Option Add** to create a new slice plane.*

*Select the option **Trim** to open the Dialog box Cutting in order to modify the sliced components.*

*Choose the option **Remove** to remove a slice plane.*

*Choose the option **Next** or **Previous** to show the next or previous slice plane.*

ATHENA identifies the current slice plane on the bar with a symbol as shown in the illustrationMarking the slice plane.)

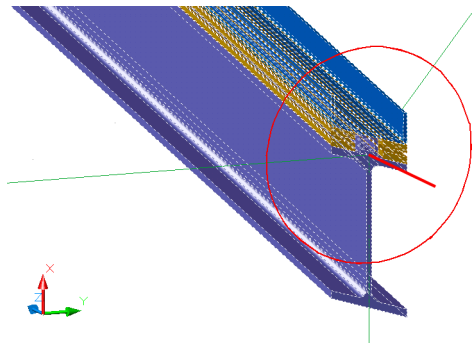


Fig. 2.7: Marking the slice plane

Notes

- Both 3D bars and also 2D projections can be sliced.
- The shorter side of the bar is always sliced.

2.26 Analyze axis model



Ribbon:	Tab ATHENA > Group Model > Analyze axis model
Menu:	Model > Apply > Analyze axis model
Toolbar:	ATH Apply
Command input:	ath_c3d_pos

With this function you can automatically analyze and preprocess an axis model. This preprocessing of an axis model simplifies and speeds up the attachment of profiles, glazing and infills, because the angles of the axes with respect to one another and the weather side of the construction are determined beforehand.

During preprocessing, the function converts all existing lines into ATHENA axes (null bars), forms corresponding node entries and supplements further information such as dihedral angle and weather side.

It is not mandatory to work with this function, but it is to be recommended.

Hinweis: Not every axis model can be preprocessed without problem. Cases in which axes can indiscriminately overlap cannot be unambiguously specified. Where applicable, the nodes must be set manually at such places using the command **Define nodes**. For the visual check after the analysis, closed fields are edged in color.

This function can be used repeatedly on already analyzed axis models in order to integrate other manually added elements.

Command prompt

Select objects:

You select the desired elements by single picking or pulling out a selection window. This input request is repeated until you press the Enter key to terminate the object selection.

Uncorrected surface situation

Remove surface

Select boundary object of the surface or [?]:

This message appears if an axis borders more than two surfaces. The relevant axis is highlighted in red.

Select an axis which borders the uncorrected surface in order to remove it.

You call the help with the option ?.

Confirm weather side [Invert/?]:

One side of the construction is identified by red lines. Press the Enter key to accept the identified side as the weather side. With the option Invert you use the opposite side as the weather side.



In order to be able to more easily recognize the suggested weather side, you should set an isometric perspective view.

Confirm plane [Remove/All/?]:

Press the Enter key to accept the plane highlighted in green.

The option Remove removes the plane from the construction.

With the option All you accept all planes.

This input request is repeated until all planes have been determined.

2.27 Copy analyzed axis model



Ribbon: Tab ATHENA > Group Model >
Menu: Model > Apply >
Toolbar: ATH Apply
Command input: ath_c3d_pos_copy

Copies an analyzed axis model (3D position).



Analyzed axis models must never be copied conventionally (e.g. copy, mirror, ...). The expanded data of the axis model are not completely copied and further operations will fail!

When you execute the command, the following appears:

Command prompt

Select 3D position or [?]:

Select an analyzed axis model to draw regions in the sub-areas.

You call the help with the option ?.

Specify base point:

Use the mouse or enter coordinates to specify the base point.

Specify second point:

Use the mouse or enter coordinates to specify the target point of the copy.

2.28 Invert weather side



Ribbon:	Tab ATHENA > Group Model > Invert weather side
Menu:	Model > Apply > Invert weather side
Toolbar:	ATH Apply
Command input:	ath_c3d_plan_inverse

Changes the weather side of an already analyzed axis model.

You can also invert the weather side once you have assigned profiles and infills to an analyzed model. This causes the alignment of the profiles and infills to be adapted.

Command prompt

Select bar or [?]:

Select any bar of an analyzed axis model.

Confirm weather side or [Invert/?]:

One side of the construction is identified by red lines. Press the Enter key to accept the identified side as the weather side. With the option Invert you use the opposite side as the weather side.



In order to be able to more easily recognize the suggested weather side, you should set an isometric perspective view.

2.29 Recalculate axis model



Ribbon: Tab ATHENA > Group Model >
Menu: Model > Apply >
Toolbar: ATH Apply
Command input: ath_c3d_pos_upd

Leads to a recalculation of an analyzed axis model (3D position) after geometrical changes.

Recalculation is not necessary when you have rotated or moved a complete axis model. However, if you delete individual axes or make other geometrical changes, recalculation is necessary.

When you execute the command, the following appears:

Command prompt

Select 3D position or [?]:

Select the geometrically modified axis model which is to be recalculated.

You call the help with the option ?.

2.30 Reset axis model



Ribbon:	Tab ATHENA > Group Model > Reset axis model
Menu:	Model > Apply > Reset axis model
Toolbar:	ATH Apply
Command input:	ath_c3d_pos_reset

With this command you convert analyzed axes and ATHENA 3D constructions into lines.



All bars are converted into lines. In this way they lose their cutting information! Infills, which are contained in the construction, are removed.

Command prompt

Select objects

Select the objects which are to be converted into lines.

2.31 Determine bar joint



Ribbon:	Tab ATHENA > Group Model > Determine bar joint
Menu:	Model > Apply > Determine bar joint
Toolbar:	ATH Apply
Command input:	ath_c3d_join

You can assign a bar joint to selected bars with this function.

This type of definition is independent of definitive bars and can be saved as a template and reused. The bar assemblies used for the definition of the joint are not saved with it. The assemblies which have been used in the joint are saved as a reference entry.

Command prompt

Select bar for cutting:

Select the bar on which changes are to be made due to meeting another bar.

Select boundary object:

Select the bar which is to delimit the bar to be changed.

For the selection or new definition of a bar joint this function uses the Dialog box Bar joint (refer to *Bar joint manager* on page 709).

Here you define the bar joint for the constituent parts of the bars (cutting classes). When you terminate the dialog box with OK, the following appears:

Command prompt

Select objects:

Select the other bars in order to attach this bar joint just defined. This input request is repeated until you terminate the object selection by pressing the Enter key.

With diagonal joints or miters, which are not realized reciprocally (boundary object is not processed), the following input request also appears:

Select first bar or [?]:

Select a bar at the end on which the process is to be carried out. The boundary object is not processed! This input request is repeated until all previously selected bars have been processed.

Notes

Before bars can be selected for the specification of a cutting, they must have at least one common node. This can be manually specified by Define nodes or automatically by Analyze axis model.

2.32 Determine infill



Ribbon:	Tab ATHENA > Group Model > Determine infill
Menu:	Model > Apply > Determine infill
Toolbar:	ATH Apply
Command input:	ath_c3d_panel

With this function you can assign infills to the surfaces of a facade construction. In order to be able to use this function a position must first be specified with the function Analyze axis model.

The orientation of the assigned glazing depends on the orientation defined in the position.

Command prompt

Select position or [?]:

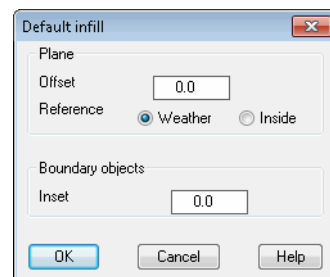
Select a position.

You call the help with the option ?.

After the selection of a correct position the Dialog box Apply infill appears where the desired infill can be selected.

Once the appropriate infill has been selected, an interrogation is made for positioning.

Dialog box Default infill



The offset positions the infill, depending on the set infill reference, in the weather or room direction. Here, a positive value specifies the displacement in the weather direction and a negative value the displacement in the room direction.

The reference defines the dimensions for the planar offset from the weather or the room side of the infill. The position of the base point is not significant here.

The inset specifies the same spacing to the adjacent objects on all sides of the infill. Here, a positive value indicates an enlargement over the adjacent objects and a negative value a reduction.

Select boundary object or [?]:

You select a boundary object by picking or drawing out a selection window in the drawing. You repeat the step until the assignment of the infill is definitive and confirm it with ENTER. You call the help with the option ?.

2.33 Specify glazing



Ribbon:	Tab ATHENA > Group Model > Specify glazing
Menu:	Model > Apply > Specify glazing
Toolbar:	ATH Apply
Command input:	ath_c3d_glazing

With this function you can assign glazing to the axes of a 3D facade construction.

The principal difference to Apply bar assembly is the analysis of the selected objects according to already existing glazing. If elements with already assigned glazing are detected, they are not changed.

The orientation and the possible dihedral angle of the assigned glazing depends on the orientation defined in the position.

Command prompt

Select position or [?]:

You select a created position by picking in the drawing.

You call the help with the option ?.

After the selection of the position the Dialog box For object selection appears in which glazing can be selected from various sources. After confirming the selection you assign the bar to the facade grid as follows:

Select objects:

Through picking or by drawing out a selection window in the drawing, you select one or more elements from which the glazing is to be assigned. You repeat the step until all the desired elements are selected and confirm them with ENTER.

The orientation of the glazing is predetermined by the position.

3 Analyze

Commands in this section:

- Project Manager
- Positions
- Assign tags
- Position model
- Bar list
- Infill list
- List of frame elements
- List of facade elevations
- Assembly list
- Bar diagram
- Infill diagram
- Export CNC
- Export NC-X
- Export IFC
- Export ERP
- Isolate bar
- Generating a section from 3D
- Shear object

3.1 Project Manager



Ribbon: Tab ATHENA > Group > Model
Menu: Model > Analyze
Toolbar: ATH Analyze
Command input: ath_build_admin

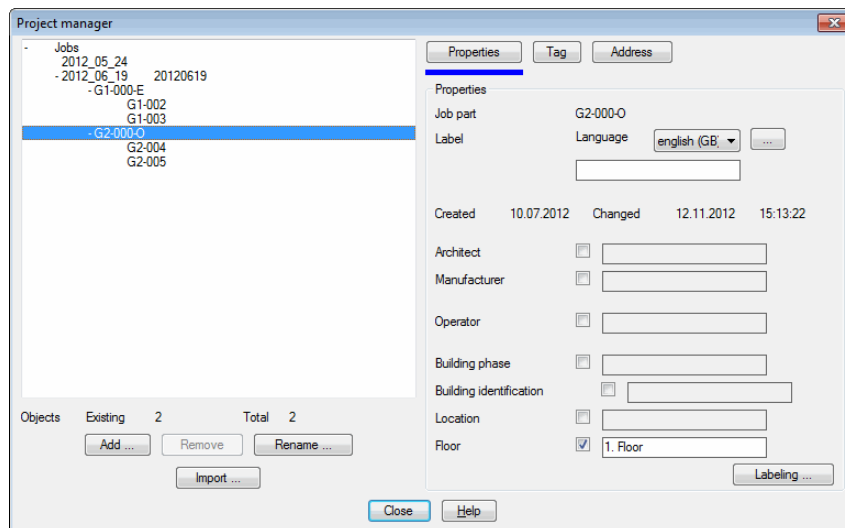
Defines jobs and job parts. These can be nested as required and are therefore individually adaptable to your project structure.



Projects and project data are generally only saved within drawings, but can be imported from other drawings.

Jobs are entities. Bars, infills, frame elements and facade elevations are assigned to a job.

Dialog box Project manager



On the left side of the dialog box a tree structure shows you the defined jobs (first level of the tree structure) and the job parts (subordinate levels in the tree structure). Here, you can select a job/job part to adapt its properties or to supplement it with further jobs/job parts in the structure.

Job parts (subordinate job levels) can also be superimposed or masked out by clicking the relevant job/job part.

Once you have assigned the job data to your objects, they are displayed below the tree structure.

Object existing

Shows the number of objects assigned to the selected branch.

Total

Shows the number of objects assigned to the selected branch and the sub-branches.

Add ...

Adds a job part to the selected job structure. For this, the Dialog box Add is opened, where you can specify the job data.



A job name must be issued. The name of a job/job part within a project must be unique. For example, if the job name is 001, no job part with the name 001 is admissible.

You will find further information on this in the section *Add job* on page 793.

Remove

Deletes the selected job/job part from the list.

Rename ...

Changes the name of the selected job/job part. For this, the Dialog box Rename is opened, where you can change the job data. You will find further information on this in the section *Rename job* on page 793.

Import ...

Imports a complete project with all jobs and job parts from another drawing. You will find further information on this in the section *Import job* on page 794.

On the right side of the dialog box there are the register buttons:

- Properties
- Tag
- Address

Register button Properties

Job/job part

Shows the name of the selected job/job part. The name is independent of the language and cannot be changed at this point.

Label

Defines the job/job part label. You can save the label in various languages. To do this, choose the required language from the list.

[...]

Opens the Dialog box Designation, where you can conveniently edit the labels for various languages. You will find further information in the Chapter *Designation* on page 122.

Under the label you can see the offer designation and the dates on which the job was created and modified.

Further job data

In the following input fields you can define further job data, such as architect, operator, building phase, etc. These data are normally stated in the job and are transferred to the job parts.

If you want to use job data, which is different to the superordinate data, for a job part, you can activate the tick box in front of the relevant input field. The relevant input field is enabled and the corresponding property can be changed. The change is transferred again to the subordinate job part if present.

Labeling

Opens the Dialog box Label. You will find further information on this in the Chapter *Label* on page 117.

Register button Tag

Defines the default values of tags for bars, infills and frame elements. They are incremented when you execute same-part recognition with the command Assign tags.



Since tags are automatically incremented, the default values defined here may only consist of letters and numbers. Special characters (also umlauts) or space characters are not permissible and will be automatically removed.

You will find further information on this in the section *Assign tags* on page 801.

Register button Address

Defines the address data for the project.

These data are normally stated in the job and are transferred to the job parts. If you want to use other address data for a job part, you can activate the tick box. The entry fields are then enabled and the address data can be changed.

End of program

Click the Close button to close the dialog box.

3.2 Project manager sub-dialog box

This section describes the sub-dialog boxes of the project manager.

- Add job
- Rename job
- Import job

3.2.1 Add job

Adds a job or job part to a project.

Dialog box Add

Job (job part)

Defines the name of the job or job part. The job name is a mandatory field.



The name of a job/job part within a project must be unique. For example, if the job name is 001, no job part with the name 001 is admissible.

Offer

Defines the name of the offer.

Default

When you activate the tick box, you can select an existing job as default to copy its properties.

Job parts

When you activate this tick box, the properties of the job parts are also used for the new job.

Dialog box section Tag

Defines the default values of tags for bars, infills and frame elements. They are incremented when you execute same-part recognition with the command Assign tags.



The default values for tags may only consist of letters and numbers and must not contain any special characters or spaces.

Tags can only be defined for the job (first level of the tree structure).

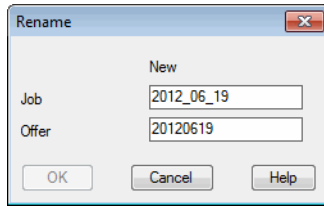
You will find further information on this in the section *Assign tags* on page 801.

Close the dialog box with OK to create the job/job part.

3.2.2 Rename job

Renames a job or job part.

Dialog box Rename



Job (job part)

The current name of the job or job part is displayed and can be overwritten to change it.

Offer

The current name of the offer is displayed and can be overwritten to change it.

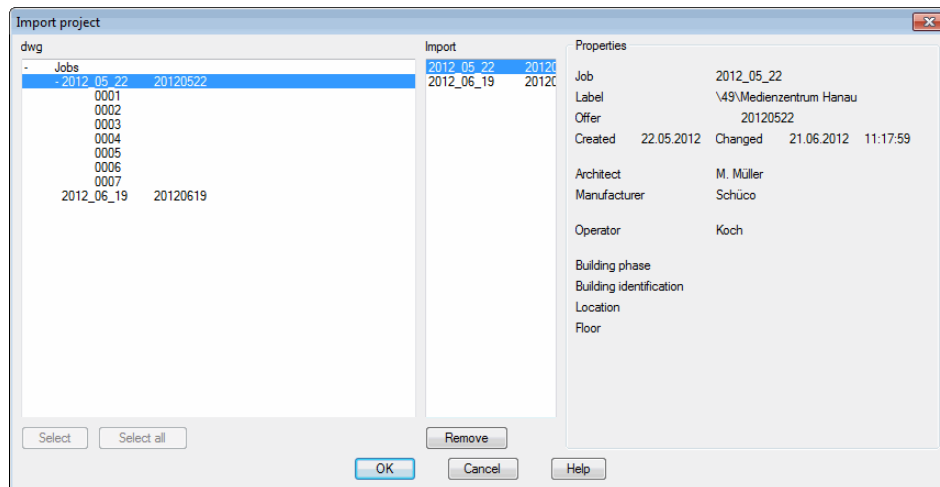
Close the dialog box with OK to implement the change of name.

3.2.3 Import job

Imports one or more jobs from another drawing (*.dwg, *.dwt, *.dxf) into the current drawing.

Once you have selected a drawing file in the standard dialog box for file selection, the following Dialog box Project import is displayed:

Dialog box Project import



On the left side of the dialog box a tree structure shows you the defined jobs (first level of the tree structure) and the job parts (subordinate levels in the tree structure) of the selected drawing.

Job parts (subordinate job levels) can also be superimposed or masked out by clicking the relevant job/job part.

Import

Lists the jobs to be imported.

Properties

Shows the properties of the selected job/job part.

Select

Chooses the selected job and transfers it to the import column.

Select all

Transfers all jobs to the import column.

Remove

Removes the selected job from the import column.

When you terminate the dialog box with OK, the project data of all jobs shown in the import column are copied into the current drawing.

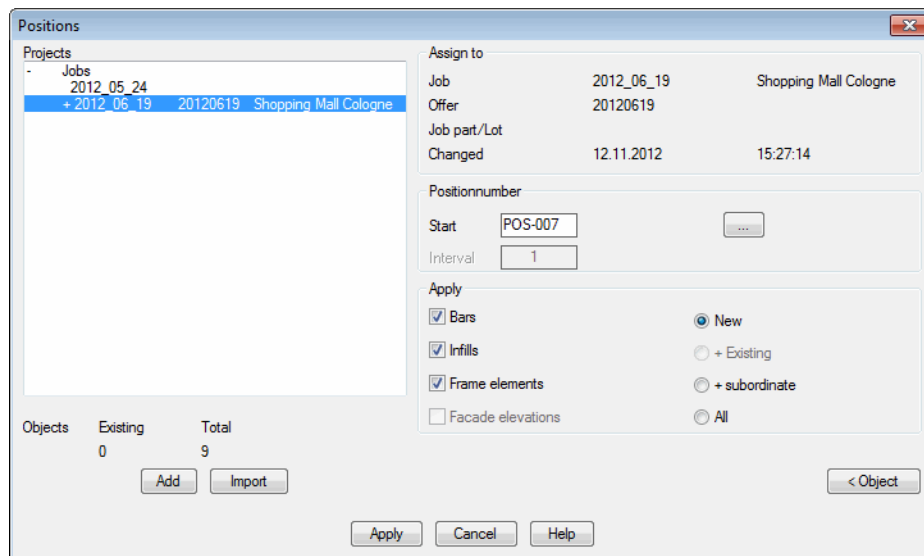
3.3 Positions



Ribbon: Tab ATHENA > Group Model > Positions
Menu: Model > Analyze > Positions
Toolbar: ATH Analyze
Command input: ath_st3d_renumber

Allocates bars, infills, frame elements to a job and issues position numbers.

Dialog box Positions



On the left side of the dialog box a tree structure shows you the defined jobs (first level of the tree structure) and the job parts (subordinate levels in the tree structure). Here, you can select a job/job part.

Job parts (subordinate job levels) can also be superimposed or masked out by clicking the relevant job/job part.

Once you have assigned the job data to your objects, they are displayed below the tree structure.

Object existing

Shows the number of objects assigned to the selected branch.

Total

Shows the number of objects assigned to the selected branch and the sub-branches.

Add ...

Adds a job part to the selected job structure. For this, the Dialog box Add is opened, where you can specify the job data.



A job name must be issued. The name of a job/job part within a project must be unique. For example, if the job name is 001, no job part with the name 001 is admissible.

You will find further information on this in the section *Add job* on page 793.

Import ...

Imports a complete project with all jobs and job parts from another drawing. You will find further information on this in the section *Import job* on page 794.

Dialog box section Assign to

Shows the data of the selected job/job part.

Dialog box section Position number

Start

Defines the number (designation) of the first position. You can enter figures and also letters. Special characters are not permissible.

The position designation is incremented with the interval 1.



The last character of a number or letter is decisive for incrementing the position number.

Examples of position numbering:

- With "POS-001": POS-001, POS-002, POS-003, etc.
- For "Part A": Part A, Part B, Part C, etc.

[...]

Opens the Dialog box Overview, in which the positions are displayed, which have already been assigned to the selected job/job part. Here, you can select a position to continue with the positioning on the next free number.

Dialog box section Apply

Bars

Assigns the found bars to the selected job/job part.

Infills

Assigns the found infills to the selected job/job part.

Frame elements

Assigns the found frame elements to the selected job/job part.

New

Only new parts (those which have not been assigned to any job/job part) are assigned to the selected job/job part.

+ Existing

New parts and parts of the selected job (without job parts) are assigned to the selected job/job part.

+Subordinate

New parts and parts of the selected job (incl. job parts) are assigned to the selected job/job part.

All

Assigns all found objects to the selected job/job part. Also those which have been assigned to a job/job part.

< Object

Displays job data of a selected object in the dialog box. For this, the dialog box is temporarily closed and an input request appears for the object selection.

End of program

Click the Apply button to close the dialog box. Then use the following procedure:

Command prompt*Select objects:*

Use the mouse to choose the 3D objects which you would like to position. Press the Enter key to confirm the object selection and to assign the job, job part and the position number to the selected parts.

Notes

You **must** assign parts to a job/job part:

- When you want to execute same-part recognition with the command Assign tags.
- When you want to create parts lists of the corresponding parts with the commands Bar list, Infill list or List of frame elements.

3.4 Positions sub-dialog boxes

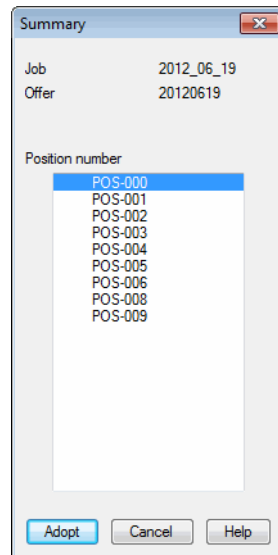
This section describes the sub-dialog boxes of the Positions command.

- Overview

3.4.1 Overview

Shows an overview of the position numbers of the selected job already issued.

Dialog box Overview



In the upper section of the dialog box the data of the selected job are displayed.

The list displays the positions which have already been assigned to the job. Positions labeled with an asterisk (*) are not directly assigned to the selected job, but instead to another job part.

Select a position from the list to continue the positioning with the next free number. In the example this would be the position number POS-007.

Adopt

Adopts the selection of the position and terminates the dialog box.

Cancel

Terminates the dialog box without adopting the selection.

3.5 Detach position

Ribbon:	Not present
Menu:	Not present
Toolbar:	Not present
Command input:	ath_build_detach

Removes the job assignments and position numbers assigned with the command Positions from bars, infills and frame elements.

Command prompt

Select objects:

Select the objects (bars, infills or frame elements) for which you would like to remove the job assignment.

This input request is repeated until you terminate the object selection by pressing the Enter key.

xx object(s) changed

3.6 Assign tags



Ribbon: Tab ATHENA > Group Model > Positions
Menu: Model > Analyze > Assign tags
Toolbar: ATH Analyze
Command input: ath_tagging

Assigns tags to bars, infills and frame elements of a job. Tags are identifiers which are assigned to the parts.

Same-part recognition takes place. Identical parts therefore receive the same identifier so that they can be brought together in the parts list during the analysis.

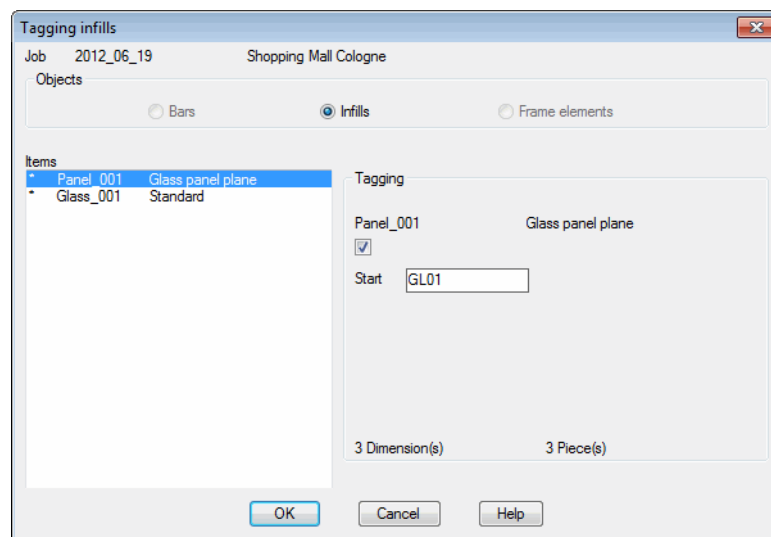


To assign tags the corresponding parts must have been allocated to a job with the command Positions.

Identical parts receive the same tag, starting with the largest part. When tags are again assigned, for example when the parts list or size of some parts have changed, all the old tags are removed and reassigned. Consequently, the possibility exists where some parts receive different tags after the reassignment.

When you run the command, the Dialog box Projects is opened, where you can select a job or job part. When you close the dialog box with OK after selecting the job, you can assign the tags in the following dialog box.

Dialog box Assign tags



In the top section of the dialog box the job is displayed which was selected in the preceding Dialog box Projects.

Dialog box section Objects

Bars

Shows all bars of the drawing in the item list. Assigns tags to the selected bars and executes same-part recognition when the dialog box is closed with OK.

Infills

Shows all infills of the drawing in the item list. Assigns tags to the selected infills and executes same-part recognition when the dialog box is closed with OK.

Frame elements

Shows all frame elements of the drawing in the item list. Assigns tags to the selected frame elements and executes same-part recognition when the dialog box is closed with OK.



Always, only the set objects (bars **or** infills **or** frame elements) are processed.

Dialog box section Item

Lists the parts contained in the drawing. Only the parts are displayed which have been switched on in the Dialog box section Objects: If you have switched on bars, all bars are displayed, etc.

Here, you select a part (for example the mullion) to define the rules for same-part recognition.

Dialog box section Same-part recognition

When you activate the tick box, a tag is assigned to the corresponding parts (for example all mullions).

Start

Shows the tag assigned to the first (largest) part.

Further below are shown how many parts and how many dimensions of the part are present in the drawing.

End of program

OK

Closes the dialog box and assigns the tags to the parts.

Cancel

This closes the dialog box without assigning tags.

3.7 Position model



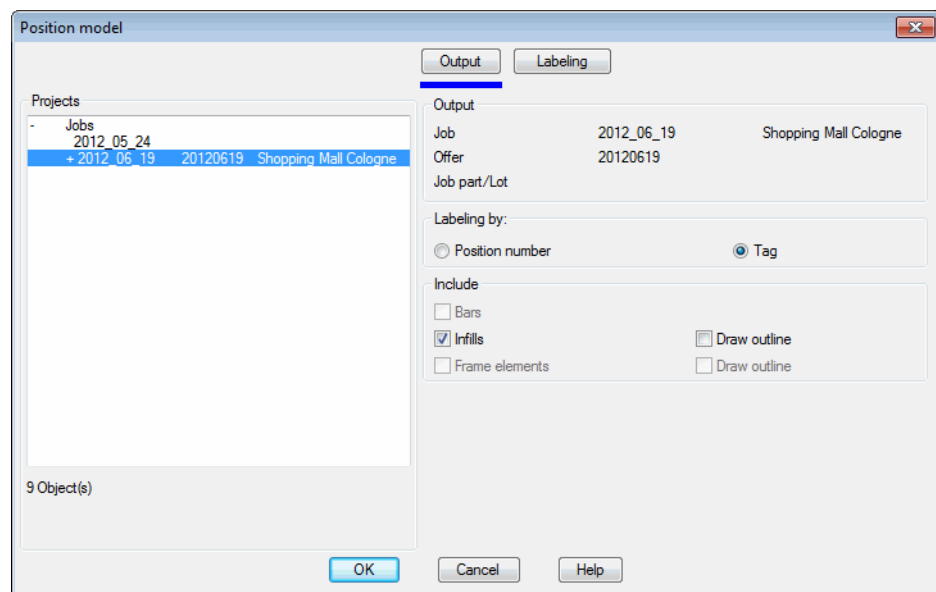
Ribbon: Tab ATHENA > Group Model > Position model
Menu: Model > Analyze > Position model
Toolbar: ATH Analyze
Command input: ath_c3d-scheme

Creates an axis model of a construction and labels the axes and areas with position numbers or tags.



To create a position model, the parts must have been allocated to a job with the command Positions.

Dialog box Position model

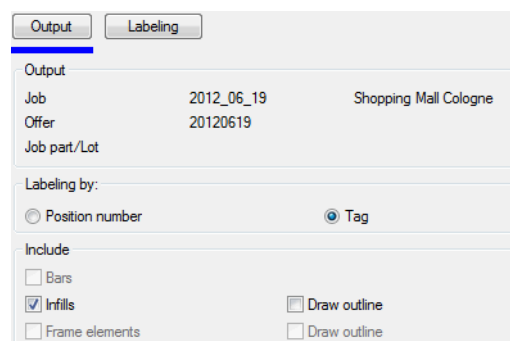


On the left side of the dialog box a tree structure shows you the defined jobs (first level of the tree structure) and the job parts (subordinate levels in the tree structure). Here, you can select a job/job part.

Job parts (subordinate job levels) can also be superimposed or masked out by clicking the relevant job/job part.

Below the job list you can see the number of objects assigned to the job.

Register button Output



Dialog box section Output

Shows the data of the selected job/job part.

Dialog box section Labeling by

Position numbers

Labels the axes and fields with position numbers.

Tag

Labels the axes and fields with tags.

Dialog box section Include

Bars

Labels bars with position numbers or tags according to the setting.

Infills

Labels infills with position numbers or tags according to the setting.

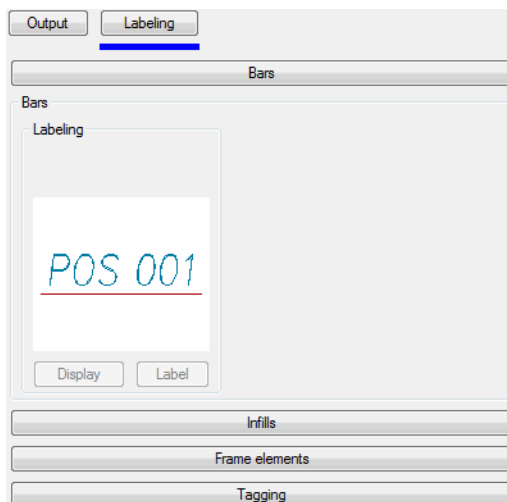
Draw outline

Inserts an outline of the infill or frame element.



This tick box should be activated if only infills or frame elements are included in the position model, because otherwise the positions are poorly visible.

Register button Labeling



Controls the display and labeling properties of the individual elements.

Drop-down menu Bars, infills and frame elements

In the Labeling section of the dialog box a preview of the label text and of the symbol used is displayed.

Display

Opens the Dialog box Leader, where you can adapt the display of the label symbol.. You will find further information in the Chapter *Leader* on page 419.

Labeling

Opens the Dialog box Label, where you can define which label texts are used. You will find further information in the Chapter *Label* on page 117.

Drop-down menu Tag

In the Labeling section of the dialog box a preview of the label text and of the symbol used is displayed.

Display

Opens the Dialog box Leader, where you can adapt the display of the label symbol for the tag. You will find further information in the Chapter *Leader* on page 419.

Monoline

Always labels tags as one line.

End of program

When you close the dialog box with OK, the following appears:

Command prompt

Specify base point or [?]<0,0,0>:

You define the base point of the axis model. By pressing the Enter key you adopt the specified base point (origin of the WCS).

Specify insertion point:

Specify the insertion point of the position model.

Specify rotated angle<0>:

Specify the rotation angle of the position model. Press the Enter key to accept the default angle.

3.8 Bar list



Ribbon: Tab ATHENA > Group Model > Bar list
Menu: Model > Analyze > Bar list
Toolbar: ATH Analyze
Command input: ath_st3d_bar_list

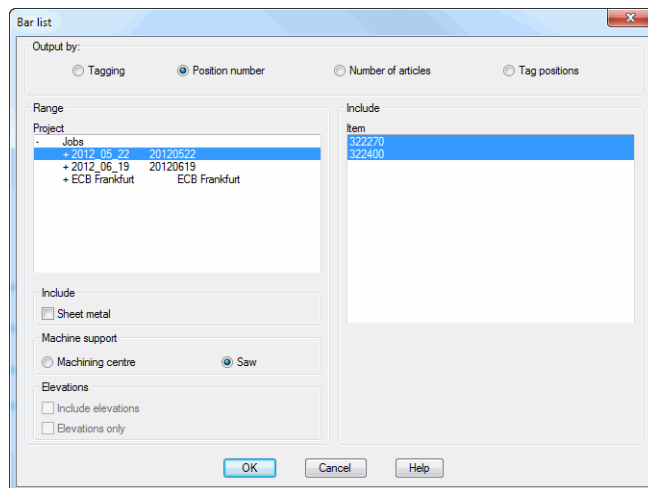
This command creates a parts list for the bars, and writes it to the Windows clipboard. It is then available for use in other programs, for example Excel.



To create a parts list, the parts must have an item number and have been allocated to a job with the command Positions.

In the Dialog box Bar list you can define what is to be written in the parts list.

Dialog box Bar list



Dialog box section Type

Tag

Creates a parts list in which parts with the same tags are brought together. A prerequisite for this is that you have previously assigned tags. See *Assign tags* on page 801.

Position number

Creates a parts list of the individual positions. The same parts are not brought together.

Number of articles

Creates a parts list in which the selected items are brought together. For example all mullions.

Tag positions

Creates a list of tags with the positions contained in them. A prerequisite for this is that you have previously assigned tags. See *Assign tags* on page 801.

Dialog box section Region

Sheet metal

Writes components which contain metal sheet to the parts list.

Machining center

Uses the machine support, which was set for the CNC machining center (CMC), for calculating the cutting angle.

Saw

Uses the machine support, which was set for the saw, for calculating the cutting angle.

Include groups

Also writes bars to the parts list which have been defined in a facade elevation.

Only groups

Writes only bars to the parts list which have been defined in a facade elevation.

Dialog box section Analyze**Item**

Lists bars of the selected job. Here you can select which parts are to be included in the parts list. By holding down Shift or Control (Windows standard) a multiple selection is possible.

End of program

After terminating the dialog box by clicking on OK, the bar list is written to the clipboard. Once this has occurred, a message appears which you have to confirm.

Notes

You can find a template for a bar list in Excel format in the folder ATHENA\SAMPLE\LIST.

Hinweis: This function produces parts lists of bars. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.9 Infill list



Ribbon: Tab ATHENA > Group Model > Infill list
Menu: Model > Analyze > Infill list
Toolbar: ATH Analyze
Command input: ath_st3d_panel_list

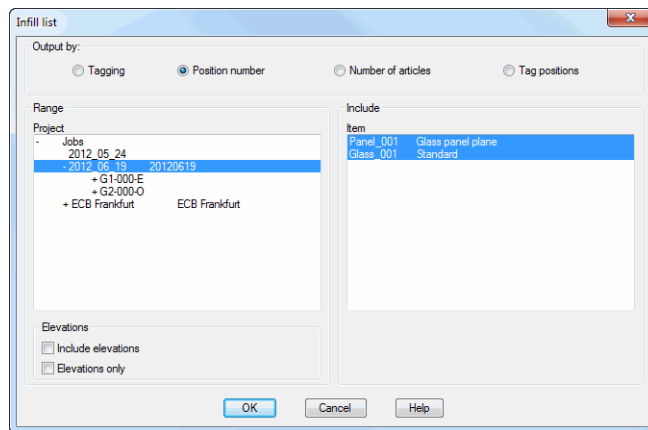
This command creates a parts list for the infills, and writes it to the Windows clipboard. It is then available for use in other programs, for example Excel.



To create a parts list, the parts must have an item number and have been allocated to a job with the command Positions.

In the Dialog box Infill list you can define what is to be written in the parts list.

Dialog box Infill list



The dialog box elements are largely identical to those of the dialog box Bar list. You will find the description in the chapter *Bar list* on page 806.

End of program

After terminating the dialog box with OK, the infill list is written to the clipboard. Once this has occurred, a message appears which you have to confirm.

Notes

The individual layers of the infills are analyzed if the option Analyze all layers is activated in Dialog box Apply infill.

You can find a template for an infill list in Excel format in the folder ATHENA\SAMPLE\LIST.

Hinweis: This function produces lists of infills. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.10 List of frame elements



Ribbon: Tab ATHENA > Group Model > Frame element list
Menu: Model > Analyze > Frame element list
Toolbar: ATH Analyze
Command input: ath_elem_list

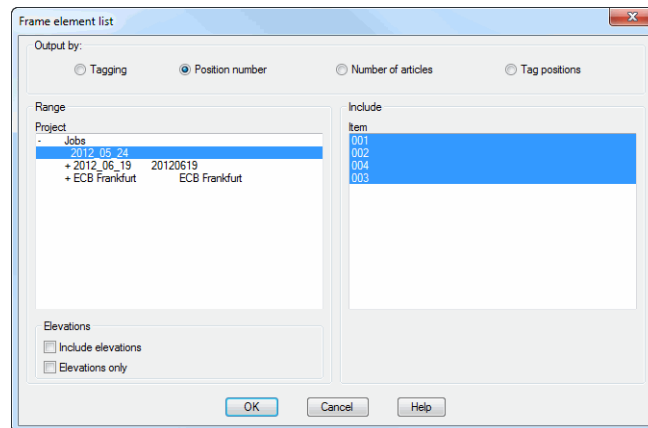
This command creates a parts list for the frame elements, and writes it to the Windows clipboard. It is then available for use in other programs, for example Excel.



To create a parts list, the parts must have an item number and have been allocated to a job with the command Positions.

In the Dialog box List of frame elements you can define what is to be written in the parts list.

Dialog box List of frame elements



The dialog box elements are largely identical to those of the dialog box Bar list. You will find the description in the chapter *Bar list* on page 806.

End of program

After terminating the dialog box with OK, the frame element list is written to the clipboard. Once this has occurred, a message appears which you have to confirm.

Notes

You can find a template for a list in Excel format in the folder ATHENA\SAMPLE\LIST.

Hinweis: This function produces lists of infills. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.11 List of facade elevations



Ribbon: Tab ATHENA > Group Model > Facade elevation list
Menu: Model > Analyze > Facade elevation list
Toolbar: ATH Analyze
Command input: ath_front_list

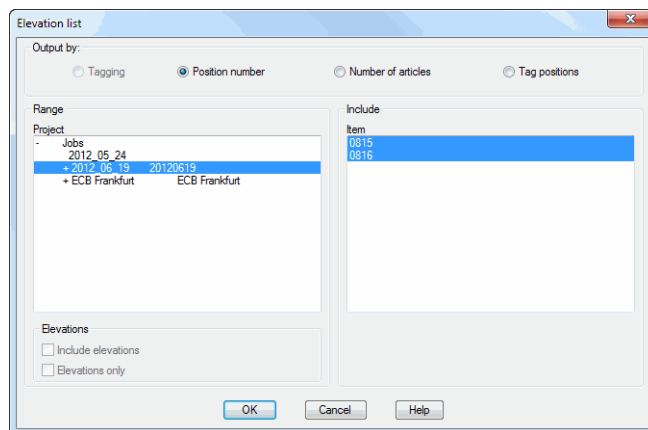
This command creates a parts list for the facade elevations, and writes it to the Windows clipboard. It is then available for use in other programs, for example Excel.



To create a parts list, the facade elevations must have been created with the command Facade elevation and allocated to a job with the command Facade elevation+.

In the Dialog box List of frame elements you can define what is to be written in the parts list.

Dialog box List of facade elevations+



The dialog box elements are largely identical to those of the dialog box Bar list. You will find the description in the chapter *Bar list* on page 806.

End of program

After terminating the dialog box with OK, the facade elevation list is written to the clipboard. Once this has occurred, a message appears which you have to confirm.

Notes

You can find a template for a list in Excel format in the folder ATHENA\SAMPLE\LIST.

Hinweis: This function produces lists of infills. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.12 Assembly list



Ribbon:	Tab ATHENA > Group Model > Assembly list
Menu:	Model > Analyze > Assembly list
Toolbar:	ATH Analyze
Command input:	ath_st3d_bgr_list

This command creates a parts list for the assemblies, and writes it to the Windows clipboard. It is then available for use in other programs, for example Excel.

The job, job part, assignment to the bar, assembly position number and assembly name are specified.

You can find a template for an assembly list in Excel format in the folder ATHENA\SAMPLE\LIST\partlist.xls.

When you run the command, all assemblies are written to the clipboard. A message with the number of assemblies appears as confirmation.

Notes

Job, job part and position number are only output if they have previously been provided with the appropriate information with the command Positions.

Hinweis: This function produces lists of assemblies. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.13 Bar diagram



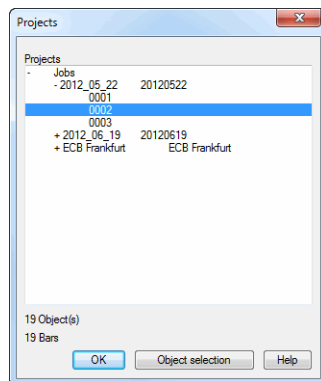
Ribbon:	Tab ATHENA > Group Model > Bar diagram
Menu:	Model > Analyze > Bar diagram
Toolbar:	ATH Analyze
Command input:	ath_st3d_bar_draw

Produces diagrams (derivations) of bars. You can generate diagrams of selected bars or of whole jobs.

The diagrams can be inserted into the current drawing or saved as separate drawings (the latter must occur in relation to the job).

When you start the command, the following appears:

Dialog box Projects



Projects

Shows a list of the jobs and sub-jobs (job parts) contained in the drawing in a tree structure. Jobs which include sub-jobs are labeled with +. Opened jobs are identified with -. If you want to generate diagrams related to jobs, you can select a job or sub-job here.

OK

Generates bar diagrams according to the job (see *Bar diagram according to job* on page 812). The Dialog box Projects is terminated and the Dialog box Bar diagram (job-related) follows, where you can carry out further settings for the job-related bar diagram.

Object selection

Generates bar diagrams according to object selection (see *Diagram according to object selection* on page 818). The Dialog box Projects is terminated and then follows:

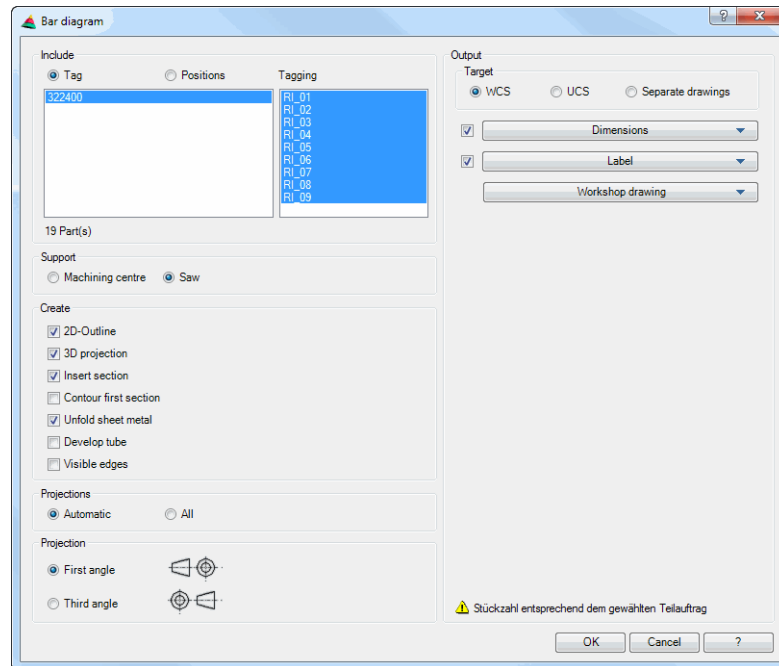
Command prompt

Select objects:

Choose the bars from which you would like to produce diagrams. Once you have terminated the object selection by pressing the Enter key, the Dialog box Bar diagram (selected object) appears.

3.13.1 Bar diagram according to job

Defines properties for the job-related bar diagram.

Dialog box Bar diagram (job-related)**Dialog box section Analyze****Tagging**

Generates bar diagrams according to type. During labeling the quantity of the relevant type is specified.

Positions

Generates bar diagrams according to position number.

List

Shows the existing bars. Here you can select one or several bars for the diagram. If you have selected the Tagging option, the tags (types) are listed. If you have selected the Positions option, the position numbers of the bars are listed.

Dialog box section Support**Machining center**

Uses the machine support, which was set for the CNC machining center (CMC), for calculating the diagrams.

Saw

Uses the machine support, which was set for the saw, for calculating the diagrams.

Dialog box section Create**2D outline**

Creates 2D outlines. These are calculated from the viewed sides of the cutting outlines of the selected bar components.

3D projection

Creates projections. These are derived from the solids of the selected bar components.



For 3D projections the full solids of the bar components are always used. The current Display modes of the bars are not taken into account!

Insert section

Inserts sections of the selected bar components. The sides of the views are numbered.

Contour first section

Supplements the views with an outline representing the first saw cut.

Unfold sheet metal

Develops sheets when the source of the sheet outline is a sheet metal section.

Develop tube

Develops tubes (only round cross-sections, no square or rectangular hollow profiles). The source of the tube outline must be a Standard Part or Semi-finished product.

Visible edges

Creates 2D outlines with visible edges.

Dialog box section Projections

Automatic

Produces an automatically calculated number of views. The number of views depends on the type and complexity of the cuttings and processes on the bar.

All

Produces all views.



You should activate this option when the automatic views are not enough for some reason.

Dialog box section Projection

First angle

Produces diagrams according to Projection Method 1 to DIN ISO 5456, formerly designated Method E (European) or First Angle Projection.

Third angle

Produces diagrams according to Projection Method 3 to DIN ISO 128-30, formerly designated Method A (American) or Third Angle Projection.

Dialog box section Target

WCS

Produces diagrams in the World Coordinate System (X/Y plane) of the current drawing.

UCS

Produces diagrams in the User Coordinate System (X/Y plane) of the current drawing.



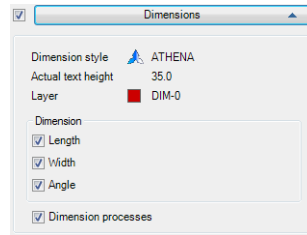
When you use the options WCS or UCS and terminate the dialog box with OK, you can insert the selected bar components consecutively into the drawing.

Unit drawing

Produces the diagrams in separate drawings for each bar.

You can carry out settings for the file (name, storage location, etc.) in the Drop-down menu Workshop drawing.

Drop-down menu Dimensioning



The switch to the left of the drop-down menu activates dimensioning. Important dimension settings (dimension style, actual text height and dimension layer) are displayed for information.

Dialog box section Dimension

Length

Dimensions the length of the bar.

Width

Dimensions the width of the bar.

Angle

Dimensions the trim angle of the bar.

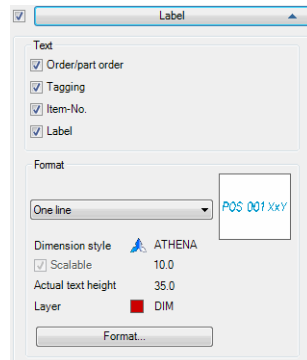
Dimension processes

Dimensions processes on bars.



Per process one dimension chain is generated, beginning at the start of the bar. Identical processes, which are aligned, are dimensioned with one dimension chain. Dimensioning occurs according to the center of gravity of the process. The process outline is not dimensioned. Drilled holes and slotted holes are labeled.

Drop-down menu Labeling



The switch to the left of the drop-down menu activates the diagram labeling.

Dialog box section Text

Job/job part

States the job designations in the label text.

Position

States the position number in the label text.

Item

States the item number in the label text.

Label

States the component designation in the label text.

Dialog box section Format

The list contains various label symbols. You can either select a symbol directly from the list or open the Dialog box Symbol selection by clicking the symbol preview and selecting a symbol there.

Dimension style

Displays the current dimension style for information.

Scalable

Indicates whether the current dimension style is of the Label type.

Actual text height

Displays the actual text height for information.



The effective text height is calculated from the text height of the dimension style and the label scale or the scaling of the dimension.

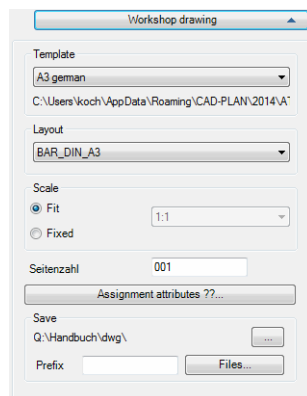
Layer

Displays the current layer for information.

Format

Opens the Dialog box Leader, where you can adapt the symbol settings. You will find further information in the Chapter *Leader* on page 419.

Drop-down menu Workshop drawing



Dialog box section Template

Displays the template drawings. You can choose the required template from the list.

The complete path for the template drawing is shown below the pick list.



Templates for diagrams can be adapted. You can create and use your own template files. In order to be able to use templates for bar diagrams, you must keep to the following conventions:

- File name - ath_work_xxx, whereby xxx can be any character string.
- The template must be saved in the folder for centralized data or local data; further information can be found in the section *Folder structure for ATHENA and CAD-PLAN applications*: on page 74.

Dialog box section Layout

Shows available layouts of the selected template. You can choose the required layout from the list.

Dialog box section Scale

Fit

Scales the viewport scale such that the complete bar diagram is visible.

Fixed

Activates the selection menu for scales where you can define a fixed scale for the viewport scale.

Number of pages

Specifies the page number for the first workshop drawing. The page number can be written in the caption via attribute assignment and is incremented.

Assign attributes

Assigns fixed texts (e.g. job number, editor, file name, etc.) to the attributes of the single drawings. To do this the Dialog box Assignment of attributes is started.

Dialog box section Save

Specifies the storage location for the workshop drawings. The current overlap is displayed.

[...]

Starts the dialog box Browse for folder. Here you can select the folder in which you want to save the workshop drawings.

Prefix

Defines the the file prefix for the workshop drawings. The file name comprises: the prefix, job designation and part number from the positioning. If no positioning has been carried out, the file name is generated automatically.

Files

Starts the Files dialog box and gives information about the available drawings (*.dwg) of the set folder.

End of program

If you quit the dialog box with OK, the bar diagrams are generated. If they generate workshop drawings, they are automatically saved. If you insert the bar diagrams into the WCS or UCS of the drawing, then follows:

Command prompt*Specify insertion point:*

Use the mouse or enter co-ordinates to specify an insertion point for the cutting outline.

Specify rotated angle<0>:

Determine the rotation angle of the cutting outline.

Press the Enter key to accept the default angle.

These two queries are repeated until the cutting outlines of all selected bar components have been inserted into the drawing.

Hinweis: This function generates production drawings (diagrams) of bars, optionally with additional details. All the data produced are to be checked by the user. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.13.2 Diagram according to object selection

Defines properties for the bar diagram according to object selection.

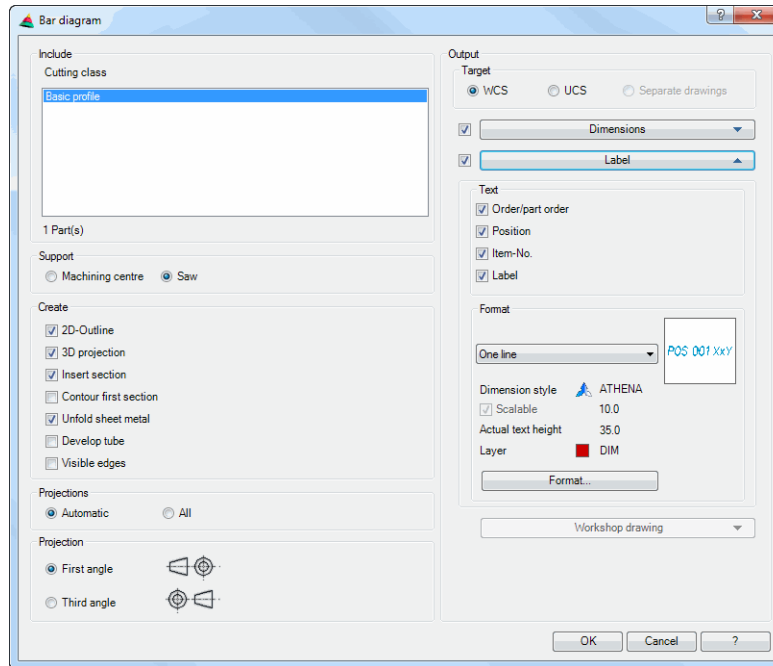


The output of workshop drawings is not possible here.

With workshop drawings job information and quantities are written into the captions via attribute assignment, which here would lead to incorrect details.

If you want to generate workshop drawings, a job-related analysis must take place.

Dialog box Bar diagram (selected object)



Dialog box section Analyze

Cutting class

Lists the cutting classes of the selected bars. Here, choose the bars from which you would like to produce diagrams.

All other dialog box elements are identical with those described in Section *Bar diagram according to job* (ab Seite 812 ff.).

End of program

If you quit the dialog box with OK, the bar diagrams are generated. If they generate workshop drawings, they are automatically saved. If you insert the bar diagrams into the WCS or UCS of the drawing, the same input request follows as described in Section *Bar diagram according to job*.

Hinweis: This function generates production drawings (diagrams) of bars, optionally with additional details. All the data produced are to be checked by the user. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.14 Infill diagram



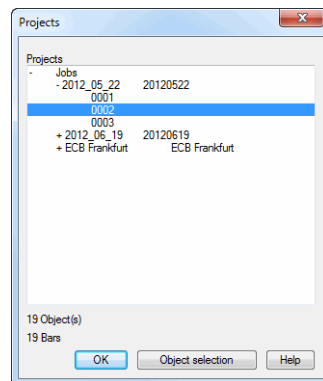
Ribbon:	Tab ATHENA > Group Model > Infill diagram
Menu:	Model > Analyze > Infill diagram
Toolbar:	ATH Analyze
Command input:	ath_st3d_panel_ext

Produces diagrams (derivations) of infills. You can generate diagrams of selected infills or of infills of a whole job.

The diagrams can be inserted into the current drawing or saved as separate drawings (the latter must occur in relation to the job).

When you start the command, the following appears:

Dialog box Projects



Projects

Shows a list of the jobs and sub-jobs (job parts) contained in the drawing in a tree structure. Jobs which include sub-jobs are labeled with +. Opened jobs are identified with -. If you want to generate diagrams related to jobs, you can select a job or sub-job here.

OK

Generates infill diagrams according to the job (see *Infill diagram according to job* on page 819). The Dialog box Projects is terminated and the Dialog box Bar diagram (job-related) follows, where you can carry out further settings for the job-related bar diagram.

Object selection

Generates bar diagrams according to object selection (see *Infill diagram according to job* on page 819). The Dialog box Projects is terminated and then follows:

Command prompt

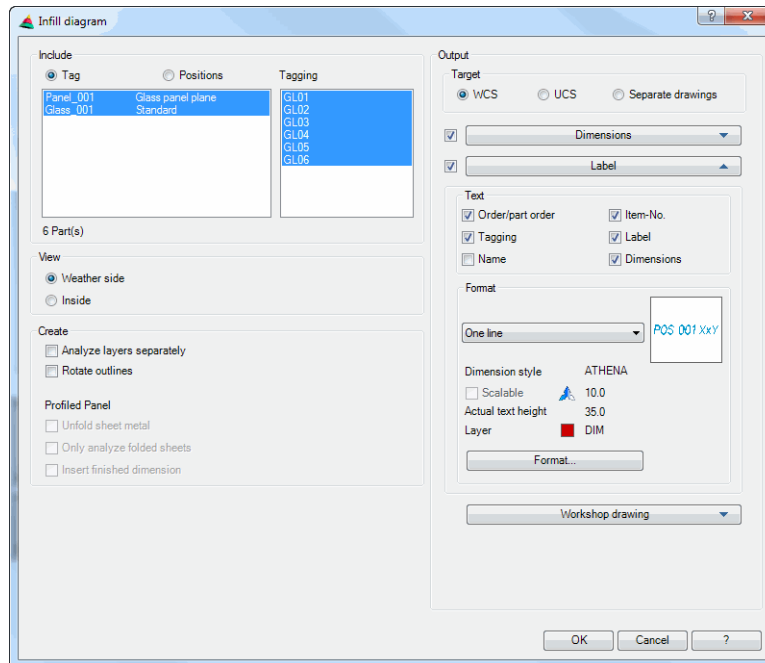
Select objects:

Use the mouse to choose the infills whose outlines you wish to insert. Once you have confirmed your choice of object, the Dialog box Infill diagram (job-related) is opened.

3.14.1 Infill diagram according to job

Defines properties for the job-related infill diagram.

Dialog box Infill diagram (job-related)



Dialog box section Elevation

Weather side

Creates a diagram from the weather side viewing direction.

Room side

Creates a diagram from the internal viewing direction.

Dialog box section Create

Analyze layers separately

Generates separate drawings for each layer of the infill (except cavities). This option is a good idea for sheet metal panels, for example.

Rotate outlines

Rotates the infill outline if possible so that at least one side is aligned parallel to the X axis of the UCS.

Dialog box section Profiled panel

Unfold sheet metal

Generates developments of folded sheet metal.

Only analyze folded sheets

Only generates developments of folded sheet metal.

Insert finished dimension

Draws the outline of the finished sheet metal in the development.



With the above methods inserted sheet developments can be imported into the Sheet processing where they can be processed further. You will find further information on this in the section *Sheet processing* on page 318.

Dialog box section Target

WCS

Produces diagrams in the World Coordinate System (X/Y plane) of the current drawing.

UCS

Produces diagrams in the User Coordinate System (X/Y plane) of the current drawing.

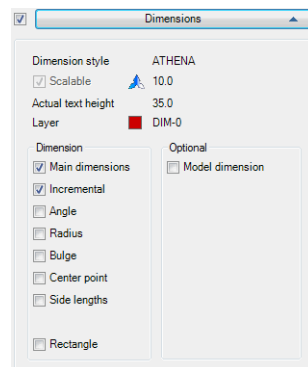


When you use the options WCS or UCS and terminate the dialog box with OK, you can insert the selected infills consecutively into the drawing.

Unit drawing

Produces the diagrams in separate drawings for each infill or layer.

You can carry out settings for the file (name, storage location, etc.) in the Drop-down menu Workshop drawing.

Drop-down menu Dimensioning

The switch to the left of the drop-down menu activates dimensioning. Important dimension settings (dimension style, actual text height and dimension layer) are displayed for information.

Dialog box section Dimension**Maximum dimension**

Dimensions the largest linear frame of the enclosing outline rectangle.

Connected dimension

Generates linear dimension chains for all dimension points of the outline.

Angle

Dimensions all angles of the outline which are not 90°.

Radius

Dimensions all radii of the outline.

Receptacle

Dimensions the receptacle for arc-shaped segments.

Center point

Dimensions the center for arc-shaped segments.

Edge length

Dimensions all outline edges with aligned dimensions.

Rectangle

Also dimensions rectangular outlines (otherwise only model slices).

Dialog box section Optional**Model dimension**

Uses special dimension methods for model slices. You will find further information on this in the section *Dimensioning options* on page 128.

Drop-down menu Labeling

The options available in the label drop-down menu are identical to those in Dialog box Bar diagram (selected object), see "Drop-down menu Labeling" on page 815.

Drop-down menu Workshop drawing

The options available in the workshop-drawing drop-down menu are identical to those in Dialog box Bar diagram (selected object), see "Drop-down menu Workshop drawing" on page 816.

End of program

If you quit the dialog box with OK, the infill diagrams are generated. If they generate workshop drawings, they are automatically saved. If you insert the infill diagrams into the WCS or UCS of the drawing, then follows:

Command prompt

x infill(s) found

Specify insertion point:

*Use the mouse or enter co-ordinates to specify the insertion point for the infill.
This input request is repeated until all outlines have been inserted.*

Notes

- If you have activated the option Analyze all layers in the Dialog box Apply infill, the outlines of all layers for these infills are generated.
- The outlines are inserted on the material layer of the corresponding infills. You will find further information about material layers in the chapter *Material* on page 81.

3.14.2 Diagram according to object selection

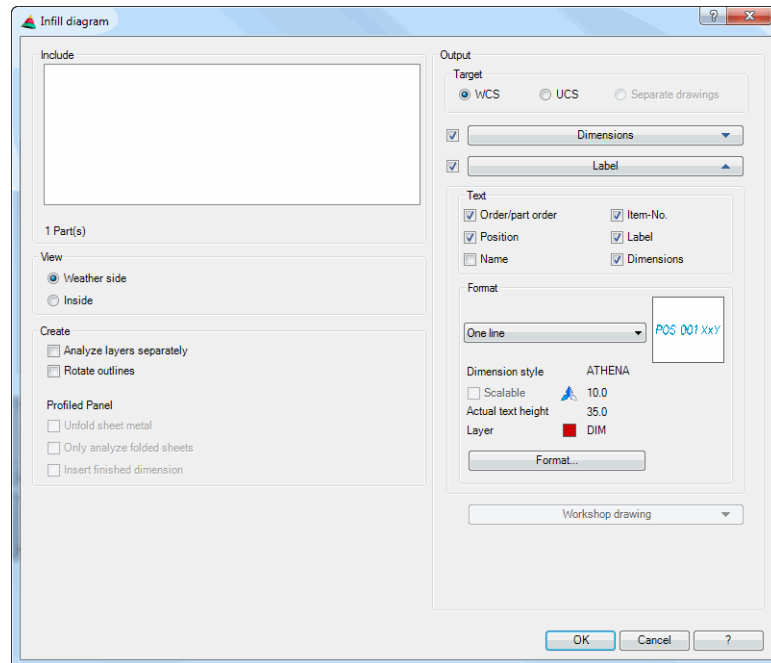
Defines properties for the infill diagram according to object selection.



The output of workshop drawings is not possible here.

With workshop drawings job information and quantities are written into the captions via attribute assignment, here would lead to incorrect details.

If you want to generate workshop drawings, a job-related analysis must take place.

Dialog box Infill diagram (selected object)**Dialog box section Analyze**

The list shows the Selected glazing and Panels. Here, choose the infills from which you would like to produce diagrams.

All other dialog box elements are identical with those described in Section *Infill diagram according to job* (ab Seite 819 ff.).

End of program

If you quit the dialog box with OK, the infill diagrams are generated. If they generate workshop drawings, they are automatically saved. If you insert the infill diagrams into the WCS or UCS of the drawing, the same input request follows as described in Section *Infill diagram according to job*.

Hinweis: This function generates production drawings (diagrams) of infills, optionally with additional details. All the data produced are to be checked by the user. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.15 Diagram sub-dialog boxes

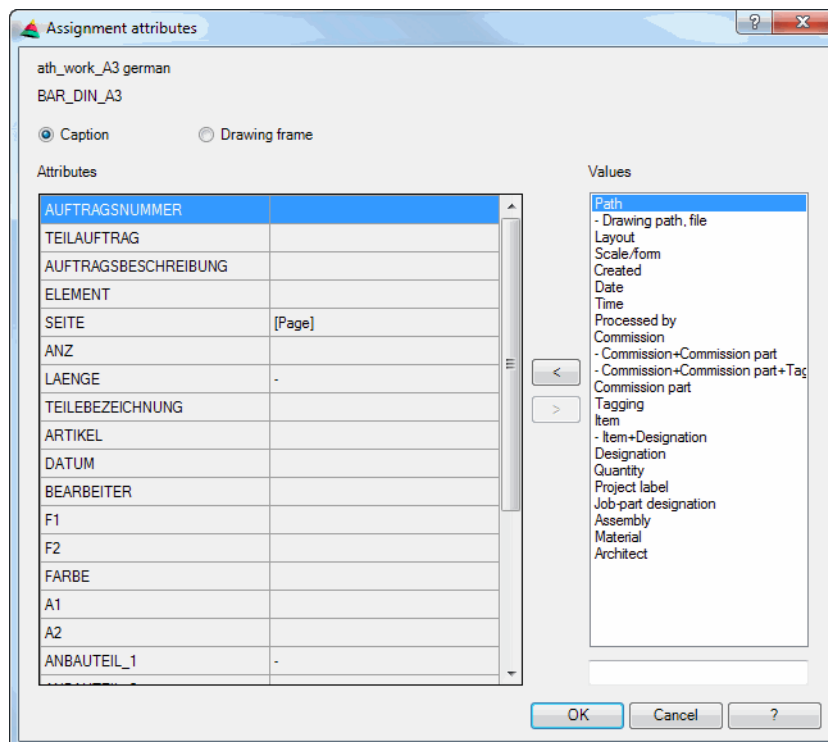
This section describes the sub-dialog boxes of the commands Bar diagram and Infill diagram.

- Assignment of attributes

3.15.1 Assignment of attributes

In this dialog box you can define how the attributes of the caption and drawing frame of the workshop drawing are to be completed. You can assign the attributes with fixed values from a list (e.g. job number, file name, etc.) or freely defined texts (e.g. Mullion general map, A. Müller, etc.).

Dialog box Assignment of attributes



Caption

Lists the caption attributes.

Drawing frame

Lists the drawing frame attributes.



The attributes are read out from the caption or the text box of the template. Captions and text boxes must be defined in the template drawing as blocks with attributes. For the block definition certain stipulations must be followed. You will find further information in Chapters *Insert drawing frame* on page 552 and *Complete caption* on page 555.

Attributes

Shows the attributes which are defined in the caption or drawing frame. Here, mark the attribute to which you want to assign a value.

Values

Shows the available values which you can assign to the attributes. Here you mark the value which you want to assign to the attribute.

Below the value list there is an input field in which you can write any texts to assign them to the attributes.

<

Assigns the marked value or free text to the marked attribute.

>

Removes the value or text which has been assigned to the attribute.

3.16 Export CNC



Ribbon: Tab ATHENA > Group Model > Export CNC
Menu: Model > Analyze >
Toolbar: ATH Analyze
Command input: ath_bar_job_export

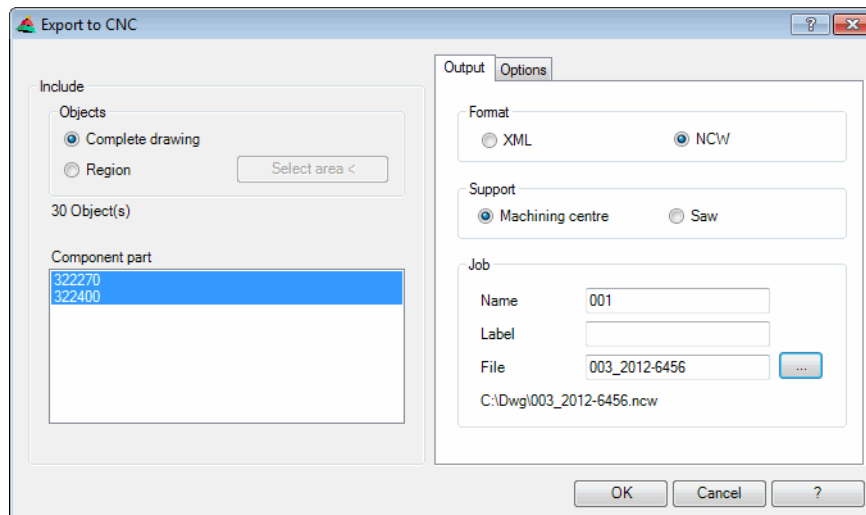
Exports the geometric information of bars in an NCW file or XML file.

These can be converted with suitable NC software and then transferred to a CNC machining center for production.



In the current as-shipped state the first saw cut of the two ends of the bar is contained in the export file. Processes (clinchs, holes, etc.) on the ends of the bar and within the bar are not contained in the export file.

Dialog box Export CNC



Register button Output

Dialog box section Analyze

Complete drawing

Writes the CNC data of all bars of the drawing to the export file.

Region

Exports the bar CNC data of the selected drawing region.

Select region <

Selects the bars of which the data is to be exported. For this, the dialog box is temporarily closed and an input request follows for the object selection.

Component part

Lists all analyzable bars in the selected drawing region.



In order to be able to analyze bars, they must first be numbered and positioned with the Positions command. Refer also to *Positions* on page 796.

Dialog box section Output

XML format

Exports bars in the XML format. An *.xml file is written.



XML files contain geometrical information of the bars for production (e.g. length and trim angle). Machine-specific information (for tools, clamping, etc.) must be converted with a suitable conversion program.

XML files can be used for bar optimization, for example.

CNC format

Exports bars in the CNC format. An *.xml file is written.



NCW files contain geometrical information of the bars for production (e.g. length and trim angle). Machine-specific information (for tools, clamping, etc.) must be converted with a suitable conversion program.

NCW files can be opened with an NCW viewer and viewed graphically.

Dialog box section Support

In the bar definition it must be stated how the bar is to be mounted on the machining center. Here you can specify which definition is to be used for the export.

Machining center

Uses the CMC machine mounting for the export.

Saw

Uses the saw machine mounting for the export.

Register button Options

Dialog box section Unit

Determines whether the dimensions for the geometry are stated in inches or millimeters.

Dialog box section Safety margin

Outside

Specifies the external safety margin.

Inside

Specifies the internal safety margin.



With an incorrectly set internal safety margin the inner chamber of the tool may be damaged.

Additional run

Specifies the number of additional runs.

End of program

When you terminate the program with OK, the geometrical data of the selected bars are exported in a file. A final message provides information about the number of written data records.

Hinweis: This function generates data for producing bars. The data are to be checked by the user. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.17 Export NC-X



Ribbon: Tab ATHENA > Group > Analyze
Menu: Model > Analyze > Export NC-X
Toolbar: ATH Analyze
Command input: ath_sat_export

Exports bars of a project into the NC-X format. Here an *.ncw file is written. The *.ncw file is a text file (can be opened in the Editor) and contains information needed for the NC production, such as job data and bars, as well as their cut-outs and processes.

By means of converter, the data contained in the *.ncw file can then be complemented by machine-relevant data (tools, kinematic information, etc.) and converted by post-processor into the format readable by the appropriate production machine.

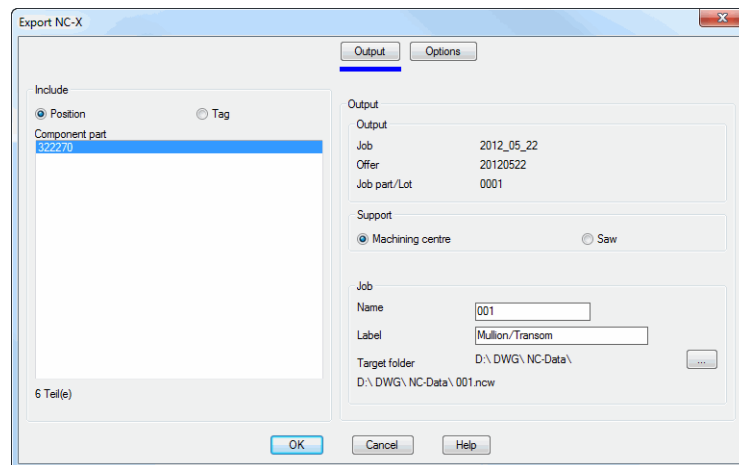


To produce an *.ncw file you require the chargeable 3DI2NCW plug-in from the company, elusoft GmbH.

Both the converter and the post-processor are chargeable products. They are required for the further use of the *.ncwfile, but not for the NC-X export!

When you execute the command, the Dialog box Projects is displayed. Here, you select the job containing the bars which you want to export into the NCW format and then close the dialog box with OK. The Dialog box Export CNC follows:

Dialog box Export CNC



Dialog box section Analyze

Position

Shows the positions of the job in the component list.

Tag

Shows the labeled types in the component list.

Component list

Shows the available components or types of job. Here, select the profiles which you want to export into the NC-X format. The sum of the selected profiles is displayed below the list.

Register button Output

Dialog box section Output

Shows the data of the selected job.

Dialog box section Support

Machining center

Uses the support defined in the Bar Assembly Manager for the machining center in order to orientate the selected profiles in the NCW output.

Saw

Uses the support defined in the Bar Assembly Manager for the saw in order to orientate the selected profiles in the NC-X output.

Dialog box section Job

Name

Defines the name of the *.ncw file.

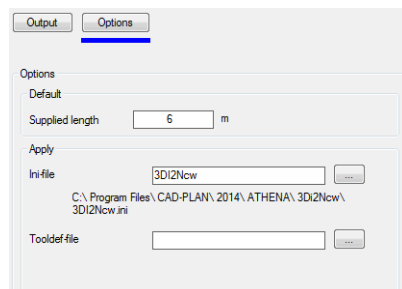
Designation

Defines a designation which, in addition to the job data, is written in the information section within the *.ncw file.

Target folder

Defines the folder in which the *.ncw file is saved. When you click the button [...], the dialog box Find folder is displayed.

Register button Options



Dialog box section Default

Supplied length

Defines the length in which the profiles are supplied. The supplied length is entered for each bar in the *.ncw file.

Dialog box section Apply

Ini file

Defines an ini file in which you can make fine adjustments for the NC-X output. Information on ini files can be found in the appropriate documentation from elusoft GmbH.

Tooldef file

Defines a Tooldef file in which you can define tools for processes. Information on Tooldef files can be found in the appropriate documentation from elusoft GmbH.

The file selection occurs in each case by means of an opening dialog box which is displayed by clicking the button [...].

End of program

If you terminate the dialog box with OK, the *.ncw file is saved in the specified folder. Cancel closes the dialog box and the settings are discarded.

The *.ncw file can be viewed in the NCW Viewer. You can download the NCW Viewer free of charge in the download section of <http://www.nc-x.com>.

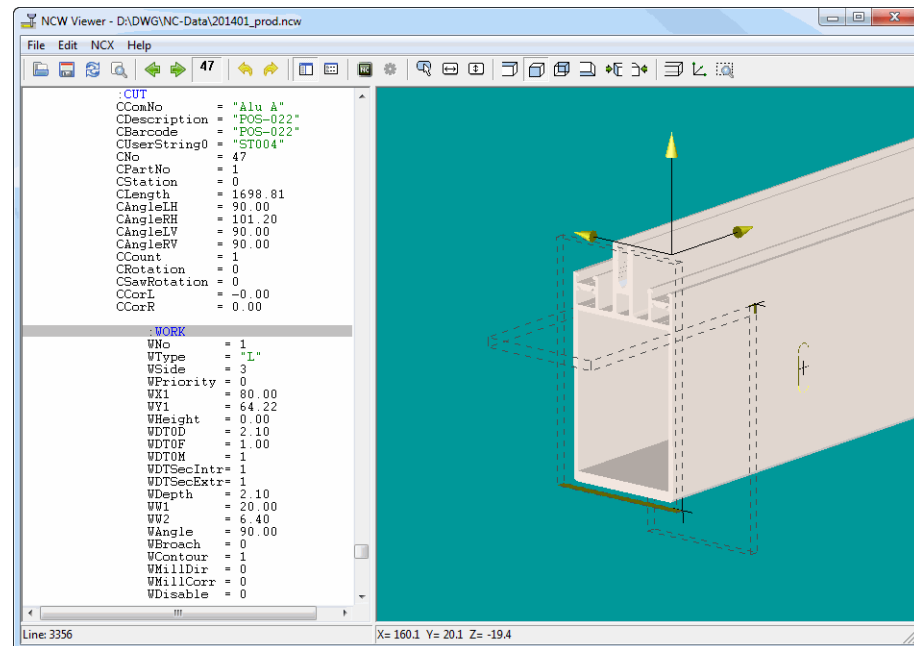


Fig. 3.1: Example: Bar of the *.ncw file in the viewer.

Hinweis: All the data produced are to be checked by the user. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.18 Export IFC



Ribbon: Tab ATHENA > Group > Analyze
Menu: Model > Analyze > Export IFC
Toolbar: ATH Analyze
Command input: ath_ifc_export

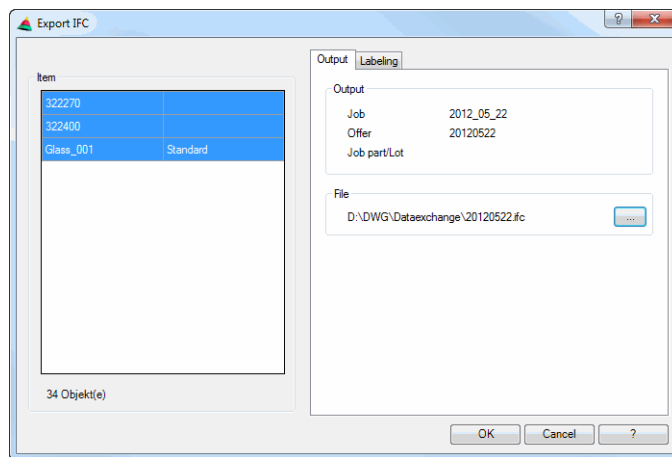
Exports a 3D model in the IFC file format.



The IFC (Industry Foundation Classes) format is an international standard for normal objects in construction and is used for the data interchange with BIM software.

When you run the command, the Dialog box Projects is opened, where you can select a job or job part. When you close the dialog box with OK after selecting the job, you can make other changes in the following dialog box to be able to carry out the IFC export.

Dialog box Export IFC



Dialog box section Item

Shows all items which are assigned to the selected job. Here, you select the item which you want to export.

Tab Output

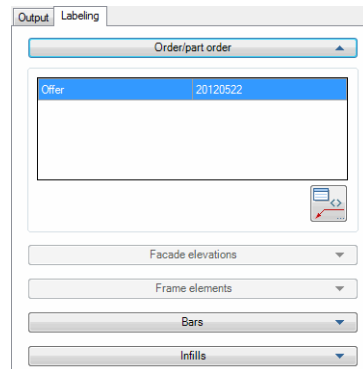
In the Output section the data of the selected job are displayed for checking.

In the File section the storage path of the IFC file is shown.

[...]

Changes the storage path and the name of the IFC file. A Save dialog box is displayed for this.

Tab Labeling



The label settings of the relevant objects are displayed in the individual drop-down menus.

Label button

Opens the Dialog box Label. Here you can adapt the settings for the labels. You will find further information on this in the Chapter *Label* on page 117.

End of program

If you close the dialog box with OK, the IFC file is saved in the specified path. Cancel closes the dialog box and no IFC file is saved.

Hinweis: All the data produced are to be checked by the user. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.19 Export ERP



Ribbon:	Tab ATHENA > Group Model > Export ERP
Menu:	Model > Analyze >
Toolbar:	ATH Analyze
Command input:	ath_export_erp

Outputs 3D models (bars, assemblies and infills) to an XML file. This file can be read in with an ERP system (ERPlus).

When you start the command, the following appears:

Command prompt

Select objects:

Select the objects to be exported. This input request is repeated until you terminate the object selection by pressing the Enter key. Then a dialog box appears where you have to specify the storage location and the name of the XML file.

Notes

- The data in the XML file can be used in ERP systems and includes all the relevant data (e.g. quantities) for the cost calculation, warehousing, etc.
- The XML file contains no geometrical information.
- Bars and infills must first be numbered and positioned with the Positions command. Refer also to *Positions* on page 796.



If you use ERPlus from T.A.Project, you can also transfer the item number of standard parts you are using in the drawing.

Hinweis: This function generates data for an ERP system. The data are to be checked by the user. CAD-PLAN GmbH can in no way be held liable for the results of this function and any errors and losses arising from it.

3.20 Isolate bar



Ribbon:	Tab ATHENA > Group Model > Separate bar
Menu:	Model > Analyze > Separate bar
Toolbar:	ATH Analyze
Command input:	ath_st3d_solid

With this routine you can release one or more bars from a construction. The bars with their cutting properties are converted into solids and inserted in the current co-ordinate system.



The bars are always separated in the full display irrespective of the set Display modes. It is therefore not necessary to change the display mode beforehand.

Command prompt

Select 3D bar or [?]:

Select the bar to be isolated.

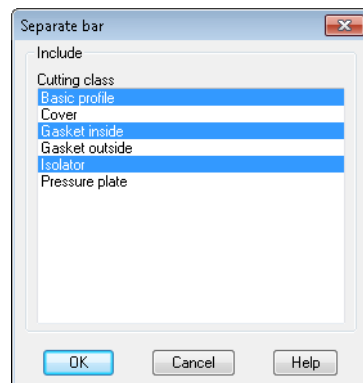
You call the help with the option ?.

Select additional bar or [?]:

Select a further bar to be separated. This input request is repeated until you terminate the object selection by pressing the Enter key or giving a right click.

After the object selection the Dialog box Separate bar is opened where you can choose the components which are to be released from the construction.

Dialog box Separate bar



Choose the components which you want to separated and click OK to insert them into the current drawing. The dialog box is closed and the following input request appears:

Command prompt

Specify insertion point:

Define the insertion point of the bar.

Specify rotated angle<0>:

Define the rotation angle of the bar.

Press the Enter key to accept the default angle.

3.21 Generating a section from 3D



Ribbon:	Tab ATHENA > Group Model > Generate section from 3D
Menu:	Model > Analyze > Generate section from 3D
Toolbar:	ATH Analyze
Command input:	ath_c3d_cross_sect

With this function you can generate a section from a 3D construction. To do this a suitable plane has to be specified.

When you start the command, the following appears:

Command prompt

Select objects:

Select the objects which are to be used for producing the section.

Select the first axis for the plane or [Points/Settings/?]:

Select the first axis of the plane. The end points of the selected axis define the first two points of the cut plane. ATHENA bar assemblies or lines can be used as axes.

*Use the **Option Points** to define the cut plane by specifying three points.*

*You start the Dialog box Settings for section generation with the **Option Settings**.*

You call the help with the option ?.

Select the second axis for the plane or [Ybar/Xbar/Points/Undo/?]:

Select the second axis of the plane.

Use the option Ybar to use the Y direction of the previously selected bar assembly for the plane definition.

Use the option Xbar to use the X direction of the previously selected bar assembly for the plane definition.

Use the option Undo to repeat the last input request.

Specify insertion point or [?]:

Define the insertion point of the section.

Option Points

Specify first point for plane or [Objects/?]:

Define the first point of the cut plane.

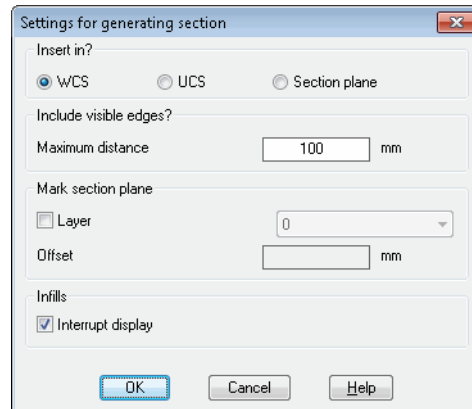
Specify second point for plane or [Objects/Undo/?]:

Define the second point of the cut plane.

Specify third point for plane or [Objects/Undo/View/?]: <View>:

Specify the third point of the cut plane.

The View option determines the third point of the plane in the current view. If you set the view appropriately before generating the section (e.g. perpendicular onto the bars to be cut), you save entering a point.

*Option Settings***Dialog box Settings for section generation****Dialog box section Insert in****WCS**

Inserts the generated section in the XY plane of the World Coordinate System.

UCS

Inserts the generated section in the XY plane of the User Coordinate System.

Section plane

Inserts the generated section in the specified sectional plane.

Dialog box section Include visible edges**Maximum distance**

Specifies the maximum distance to the sectional plane in which visible edges of the solid of unsectioned bars and infills are displayed in section.

E.g.: You are generating a horizontal section through two mullions. If a transom is located less than 100 mm below the sectional plane, its visible edges are shown in section.

Dialog box section Mark section plane**Layer**

The tick box activates the sectional plane marking. In the pull-down menu you can select a layer for marking.

Enlarge by

Specifies the enlargement of the sectional plane marking.



The sectional plane is identified by a region. The size of the region corresponds to the enclosing rectangle of the sectioned objects plus the enlargement.

Dialog box section Infills**Interrupt display**

Generates infills with interrupted display (right and left section instead of complete infill).

Notes

- Draw, where applicable in advance, two lines defining a sectional plane. You can select them via object selection.
- Note that object projections on a plane located obliquely in space may be displayed distorted.
- Default settings for section generation can also be made in the ATHENA options. See Presettings on page 91.

3.22 Shear object



Ribbon: Tab ATHENA > Group Model > Specify glazing
Menu: Model > Analyze >
Toolbar: ATH Analyze
Command input: ath_obj_shear

Displays a cross section cut at specified saw and oblique angles in the drawing.

With this you can, for example, show the cross section of a transom which meets a mullion diagonally (miter cut) (e.g. to draw the outline of the double-miter cut on the mullion).

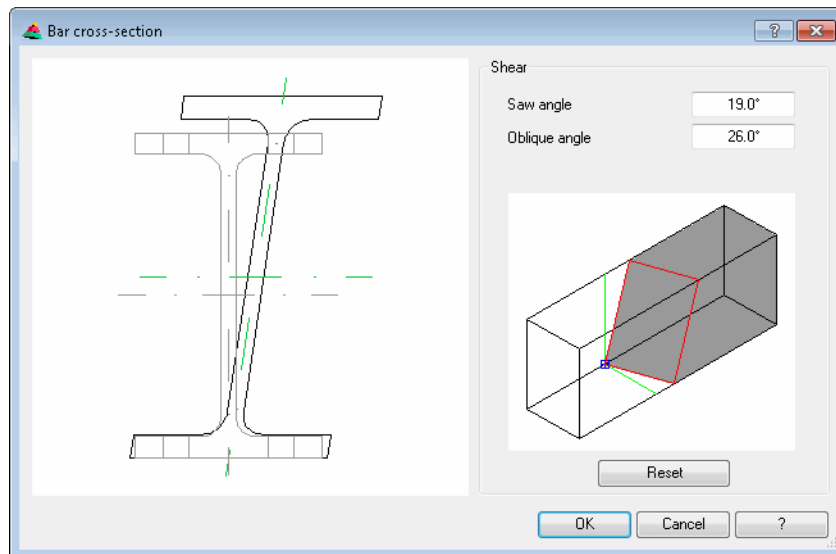
When you start the command, the following appears:

Command prompt

Select object:

Select the section which you want to display as sheared. Directly after the object selection the Shear dialog box is started.

Dialog box Shear



On the left side of the dialog box you can see the preview of the bar in the normal and the sheared representations. You can specify the angle on the right side.

Dialog box section Shear

Saw angle

Defines the setting angle of the saw.

Oblique angle

Defines the oblique angle of the saw.

Reset

Sets the defined angles to 0°.

End of program

When you terminate the dialog box with OK, the selected object is displayed sheared.

4 Definition of terms

Component (bar)

An element within a bar assembly consists of a cross-section geometry which is located in a bar assembly application as a sectional representation or solid.

Assembly

An assembly is made up of one or more component parts with permanent dimensions. The components can have an effect (e.g. difference or unification) within the assembly and also externally (on bars and infills).

Center of rotation

Point about which outlines are rotated, folded, bent or cut.

Bounding box

A bounding box can be assigned to a bar component when neither the enclosing rectangle of the outline nor the cutting outline are suitable for the cutting.

In contrast to the cutting outline the bounding box can be smaller than the enclosing rectangle of the component outline. If a bounding box is specified, it has priority for the cutting before all other outlines.

Infill plane

An infill plane is an infill with thickness 0 (zero) which does not contain any layer information. It is shown as a circle with a line. The line indicates the direction (weather side of the infill).

Infill planes can be converted into infills (glazing or panels) by assignment. Furthermore, they can be used as boundary objects for bars to slice them.

Criterion

One or more switches can be combined to form a criterion.

E.g.: The switch Drilled hole at the top right and Drilled hole at the top left are combined to form the criterion Drilled holes at the top.

Null bar

Qualified axes in space without further geometrical information which contain an alignment and form nodes with other null bars.

Null bars arise during the analysis of axes (refer to *Analyze axis model* on page 781) or when the assembly-specific properties of a bar are deleted due to detachment.

Reference

Link to an existing bar assembly (with a single component) for further use. This means that complicated and structured object definitions built up on single, saved objects are possible. Objects for referencing may be assemblies and bar assemblies.

Switches

Switches are logical denotations which can be assigned to components. Switches can themselves be defined and can include a multilingual designation.

E.g.: The switch Drilled hole at the top right is assigned to the top right drilled hole in a floor panel.

Bar

Extrusion of one or more material outlines (bar assembly) along a bar axis. The bar axis can be linear or arc shaped.

A bar which is not assigned to any assembly is called a null bar.

Bar assembly

Grouping of individual bar cross sections which are compiled using rules and, controlled with parameters, can be used as sections and extrusions.

Variants (assemblies)

Variants describe combinations of an assembly selectable due the change in the visualization and effect of individual components.

E.g.: A floor panel contains four drilled holes (one in each corner). The individual holes are defined as variants and can be superimposed or masked out as required.

Glazing

A glazing is a special, optionally parameterizable bar assembly with referenced components. It is used for automated application on subconstructions (bar assemblies with glazing axes), whereby position and orientation are determined / adopted.

Cutting

This is the type of the applied cutting operation. With bar assemblies the following operations are currently available:

- Profiled butt
- Continuously profiled
- Clinched throughout
- Plain butt
- Continuous plain
- Mitered
- Diagonal
- Mitered joint
- None

With assemblies these are the operations

- Union
- Difference
- Intersection

Cutting class

In order to be able to correctly control the cutting or processing of bars a cutting class (basic profile, cover section, etc.) is assigned to each component in a bar assembly.

Only components with the same cutting class can carry out an operation with one another.

Many components can belong to the same cutting class in a bar assembly.

Cutting outline

Closed outline, which determines, how the profile of a joining part is to be cut. The enclosing rectangle of the cutting outline determines other cutting types (plain, mitered, etc.).

Secondly, the cutting outline is used for calculating the bar diagrams. For this

reason the external dimensions of the cutting outline should never be larger or smaller than the enclosing rectangle of the component outline.

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