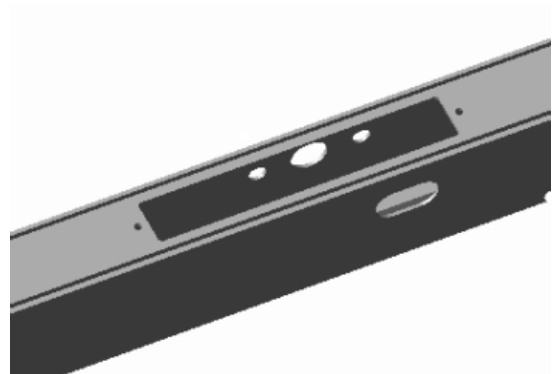


Processes from a 3D solid model 3DI2NCW Feature Recognition

Automatic Feature Recognition in metal construction

by Peter Fürle



Processes from a 3D solid model

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Preface

We're glad you are interested in a product from elusoft GmbH.

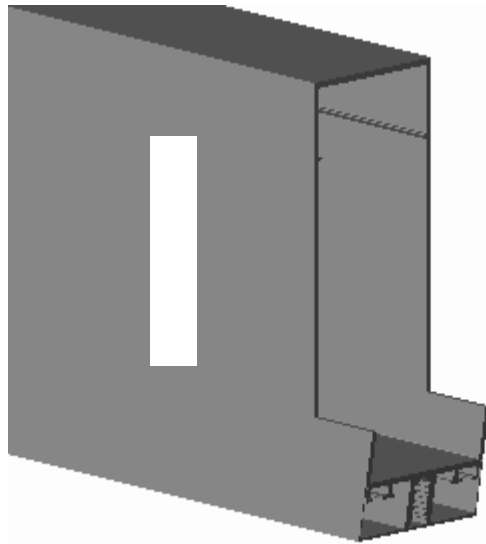
This manual will familiarise you with the feature recognition system and its characteristics. The technical specifications and figures in this manual are subject to change. No claims may be derived from them.

We reserve the right to make improvements without changing this manual.

A software maintenance contract will ensure that you have up-to-date software and license transfer.

Introduction

Part



1 Introduction

Operating manual version 1.0.4

The application

We have developed a new application named 3DI2ncw. 3DI stands for 3D Import, 2 stands for "to", and NCW describes the file format of the NC-X.

The application offers a AFR (automated feature recognition) and detects machining of 3D solids. The application has no user interface, can currently process STEP, SAT, or X_T files, and detects Drillings, circular pockets, elongated Drillings, grooves, rectangular pockets, counterbores, saw cuts, notches and free milling paths.

After these machining processes, with all features, such as position in the coordinate system, dimensions, and alignment with each other, are detected, 3DI2NCW.exe exports the profile section as a DXF file and the machining processes as a NCW file in NC-X Format. More on the NC-X format can be found at www.nc-x.com/doku

In 2008 a customer invested a lot of time and money to equip their company entirely for the design of 3D solids. Unfortunately, the data generated there could not be used to supply the 3-axis machining centre with the necessary information.

The programmer continued to create the programs based on 2D drawings, which first must be generated from the 3D solid in an additional step. This effort was not justified; the parts were actually only simply machined from above.

The 3DI2NCW converter was developed for such uses: Automatic recognition of parameterised or generated machining processes and the direct conversion into a machine readable format. The application is not suitable to recognise machining processes that are a combination of several machining steps.

1.1 Use

AFR

Automatic Feature Recognition

The goal of feature recognition is to develop an algorithm that can recognise processing features from the surfaces and volume information of a 3D volume, and convert them to processing information.

Step2NCW was the original working title. With the support of other formats like SAT and Parasolid, it was necessary to change the name. There are still some documents that mention Step2NCW. These describe an earlier version of 3Di2NCW.

Processes in aluminium profiles that are already available in a 3D model can now be programmed automatically. The feature detector by elusoft GmbH allows this "miracle at the press of a button". Time-consuming, error-prone data entry by hand during the programming of processes like drill Drillings, circular pockets, grooves, slots, rectangular features, notches and saw cuts can often be eliminated. The application is considerably faster than manual programming.

The transfer of data from an existing 3D volume has significant advantages.

- ✓ Our application requires no additional information added to the volume during the design process.
- ✓ Our application relieves the burden of the programming department and reduces programming costs.
- ✓ NC programs are done in a short time, relieve programmers from routine tasks and bypass the bottleneck in the process sequence from design to machine.
- ✓ Programmers are relieved of routine tasks and can concentrate on challenges which further increase their capabilities.
- ✓ As a result, feature recognition delivers an NC-X file whose implementation into tool paths starts on existing products whose functions have already proven themselves hundred of times.

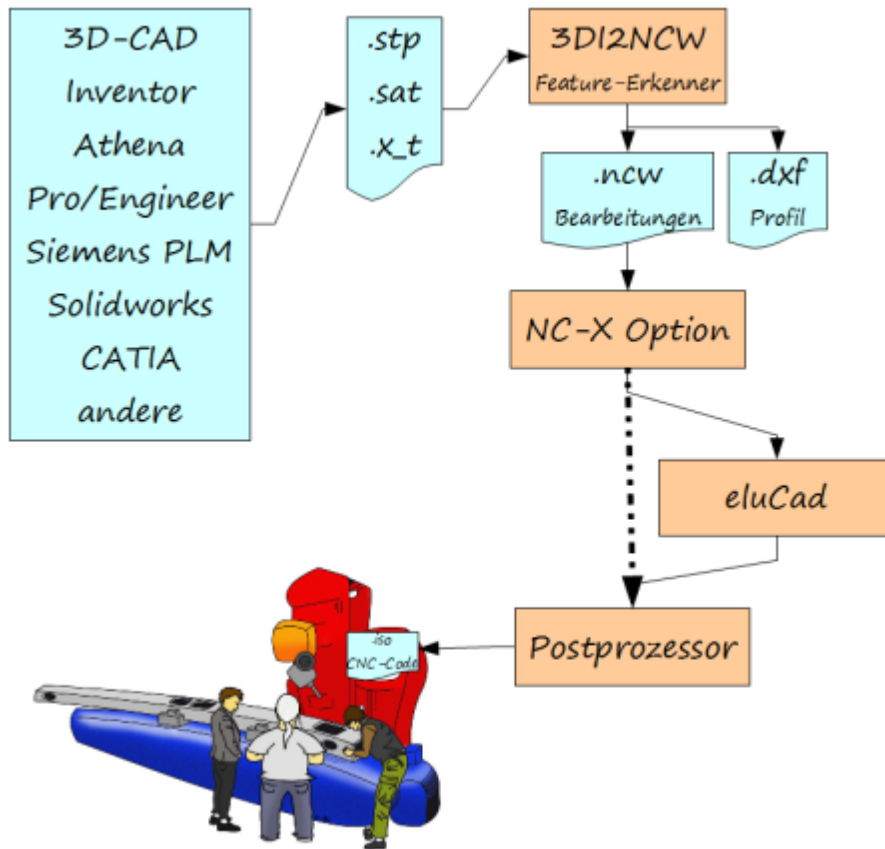
AFR (automatic feature recognition)

- ✓ Improves collaboration and increases the degree of process automation
- ✓ Increases productivity with minimal manual intervention
- ✓ Enables the detection of processable elements
- ✓ Enables the further use of 3D data in the production process

The search for machining processes is limited to 2.5D features, such as Drillings, grooves, pockets and saw cuts, which are typical for a machining process on profile processing centres. The detection of saw cuts as cuts on the profile start and end, as well as the detection of notches, can be enabled optionally.

1.2 Process

The data flow can be shown as follows:



Data flow during acceptance and detection of processes

A solid model created in a 3D CAD system is processed by the feature detector. The result is the NCW file with a description of the processes in geometric form, the profile cross section as a DXF for example and a non-listed report as a text file.

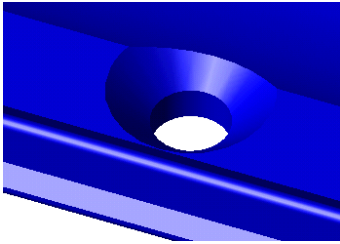
Using NC-X.exe or the NCW option in eluCad, the process technology can be added to the geometrical data.

A path-optimised program with assigned tools, technology data, safety distances and clamping situations is then available.

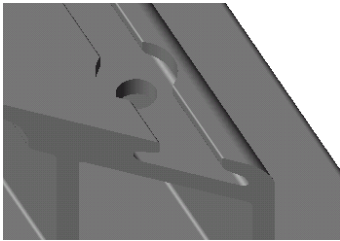
Post-processors are used to convert this to the format of the target machine.

1.3 Scope of services

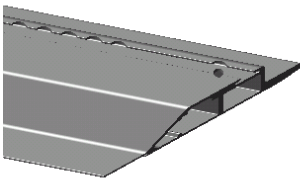
Examples of the functionality of feature recognition:



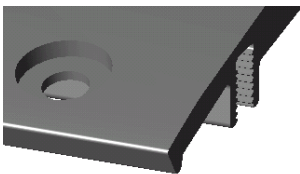
Drillings and countersinks,
even if incomplete
and on slanted surfaces



Drillings and circular pockets combined, or only partly present
Both circular segments are recognized as related,
since both have the same centre and are accessible from the
same direction



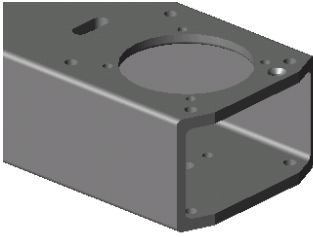
Saw cuts on the profile end,
also with two angles



Drillings with flat countersink at arbitrary angles

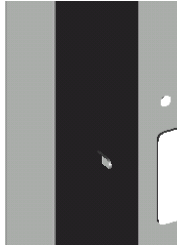


Circular pockets, drillings, slots, with 4th axis,
as is suitable for SBZ140, SBZ150 or SBZ151



Drillings on several sides with several standard machining processes.

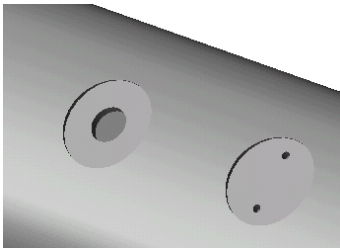
The chamfers are generated on the profile ends by saw cuts



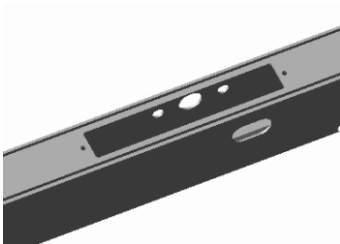
Recesses are added if possible, to make rectangular pockets.



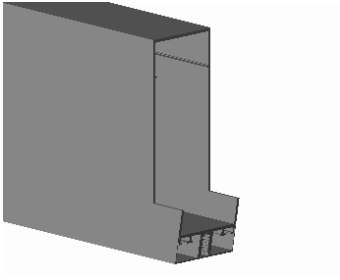
Recessed drillings have a depth table for technologically perfect approach positions



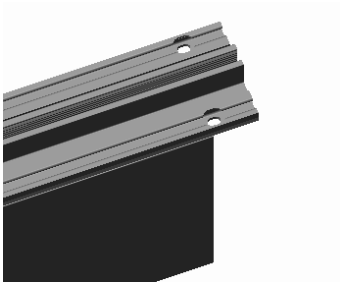
Circular pockets for the spot-facing of surfaces are assigned the attribute for countersinking the pocket.



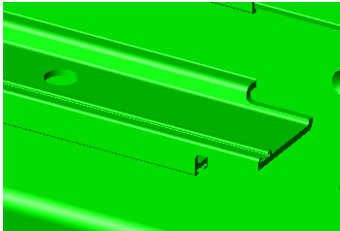
Multisided processes necessary, such as for "door-style"



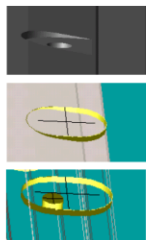
Notches and end cuts on profiles, including polygonal notches



Notches and processes typical for facades

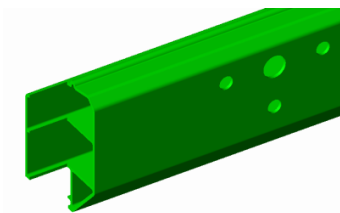


Ground features are filled out as standard templates such as slots or rectangular features



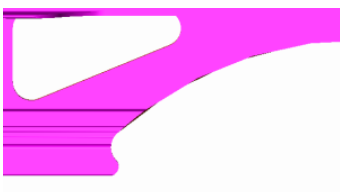
Feature completion:

A slot that is only partly present and located in a slanted surface is detected, extended, and the hole inside it is also detected.



Notches:

Notches, even polygonal ones, are detected and optionally converted to saw cuts



Free milling:

Inner and outer contours that do not correspond to the default processes are detected and implemented as free milling paths



free milling:

Simple outer contours such as radii or chamfers are implemented as free milling paths of a single element

1.4 User-Interface

3D import with automatic feature recognition has no user interface and works as a so-called command-line application.

This conscious decision drastically increases working speeds. If reprocessing is necessary, existing products like eluCad, NC-luX, or the PUMA System® can be used.

The feature recognition system can be integrated into other products using its command-line control. This integration has already been carried out for eluCad, PUMA System®, and NC-luX.

1.4.1 command line

Command line calls are used to start the feature recognition system:

```
*****
**          NC-X Numeric Control 3D to NCW Interface Ver: 1.0.0.17          **
**          Copyright (C) 2009 EluSoft GmbH                               **
*****
Usage: 3DI2Ncw.exe i=<Source-3D-File> o=<Target-NCW-File> [c=<ini-file>]
```

3DI2Ncw.exe i=example.stp o=example.ncw c=my_3D.ini

Generates the following output in a text window:

```
Reading 3D file "example.stp":
... Step File Reading : example.stp
... STEP File Read ...
Elapsed time: 0 Hour(s) 0 Minute(s) 0.01 Second(s)
CPU user time: 0 seconds
CPU kernel time: 0 seconds
*** Error on Record 139 (on 3341 -> 4 % in File) *** Ident #72
Complex Type incorrect : SOLID_ANGLE_UNIT / SI_UNIT ...
*** Error on Record 519 (on 3341 -> 15 % in File) *** Ident #355
Complex Type incorrect : NAMED_UNIT / LENGTH_UNIT ...
... Step File loaded ...
Elapsed time: 0 Hour(s) 0 Minute(s) 0.054 Second(s)
CPU user time: 0.0156001 seconds
CPU kernel time: 0 seconds
3341 records (entities,sub-lists,scopes), 9768 parameters
... Parameters prepared ... Elapsed time: 0 Hour(s) 0 Minute(s) 0.081 Second(s)
CPU user time: 0.0156001 seconds
CPU kernel time: 0 seconds
Report : 3 unknown entities
... Objets analysed ...
Elapsed time: 0 Hour(s) 0 Minute(s) 0.17 Second(s)
CPU user time: 0.0312002 seconds
CPU kernel time: 0 seconds
STEP Loading done : 2261 Entities
** Model Complete Check List **
Check:1 -- Global Check
Complex Type incorrect : SOLID_ANGLE_UNIT / SI_UNIT ...
Complex Type incorrect : NAMED_UNIT / LENGTH_UNIT ...

*****
***** Transferring one Entity *****
***** N0 in file : 170 Ident : #177 *****
***** Type : PRODUCT_DEFINITION *****
*****
-- Actor : Transfer Ent.n0 170 Type StepBasic_ProductDefinition
Cc1ToTopoDS : Length Unit = 1 Tolerance CASCADE = 0.0001
-- Actor : Transfer Ent.n0 150 Type StepShape_ManifoldSolidBrep
-- Actor : Transfer Ent.n0 171 Type StepGeom_Axis2Placement3d
```

Then the analysis starts:

```
Checking profile alignment...done
Creating profile section...done.
Analyzing profile...done.
Recognizing features...done.

Writing NCW file "example.ncw"...done.
Writing DXF file "example.dxf"...done.

Export information:
  6 Circles
  0 Debursrs
  0 Slots
  0 Rectangles
  2 Sawcuts
  0 Notches
  0 Planes

RunTime statistics:

Start conversion           :      0ms
Reading 3D file            :     911ms
Reading config file        :       3ms
Checking profile alignment :       1ms
Analyzing profile          :     93ms
Recognizing features       :    30ms
Writing NCW                :       5ms
Writing DXF                :       3ms

Total time                 :       1s

Conversion succesfully completed.
```

The model is oriented parallel to the X axis

The profile cross-section is calculated

Features are searched for a templates assigned

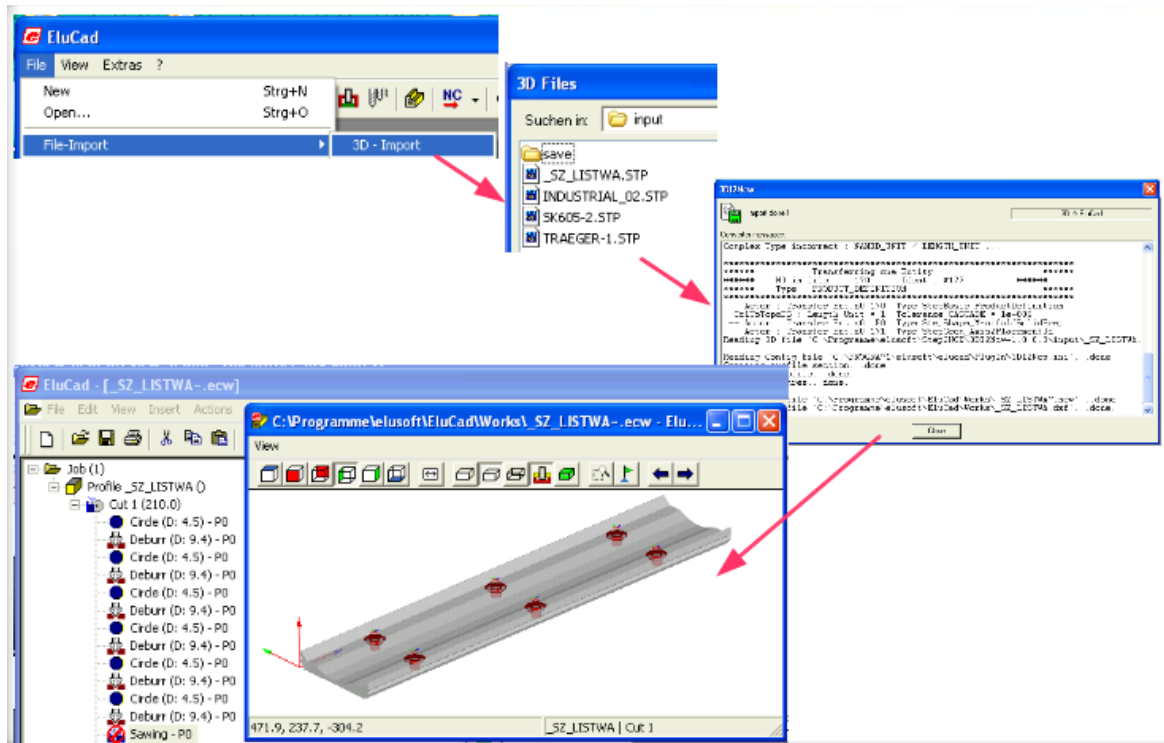
The output file is written in the NC-X format as a .ncw file

The profile cross section is written as a .dxf file

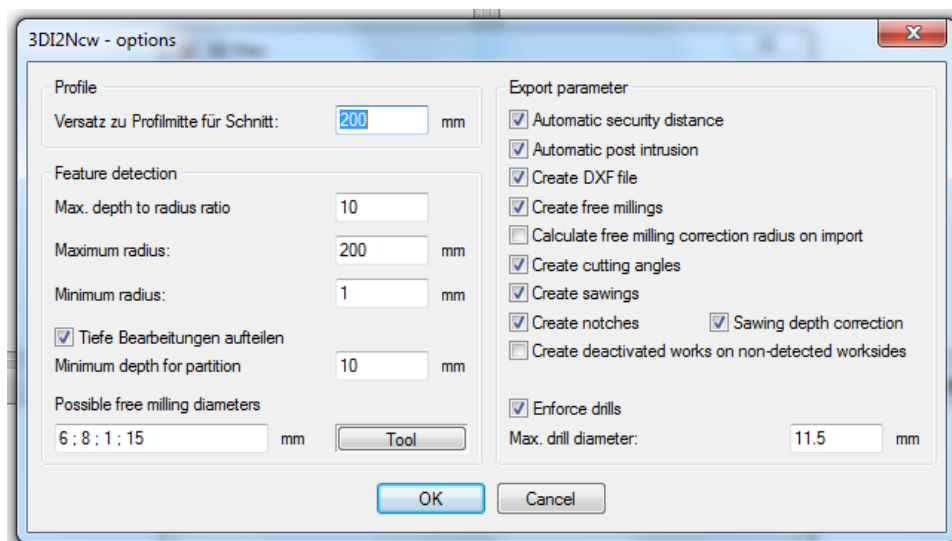
Finally, there is a report on the features recognized and the times needed.

1.4.2 eluCad PlugIn

One convenient option is integration of the recogniser into eluCad or NC-luX. If the corresponding approval is available and the 3DI2ncw is available in the plug-in directory, the import can take place through eluCad with graphical support



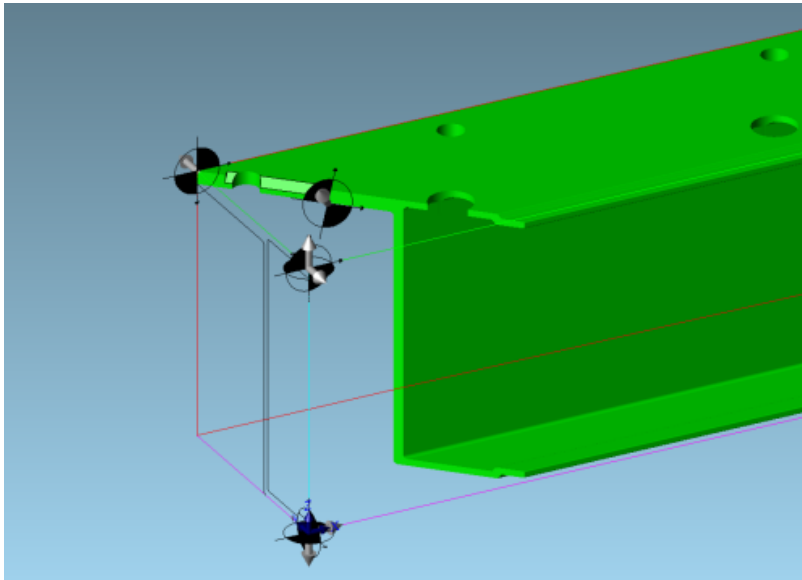
The 3D plugin provides an interface for the settings for feature recognition:



1.4.3 PUMA-System

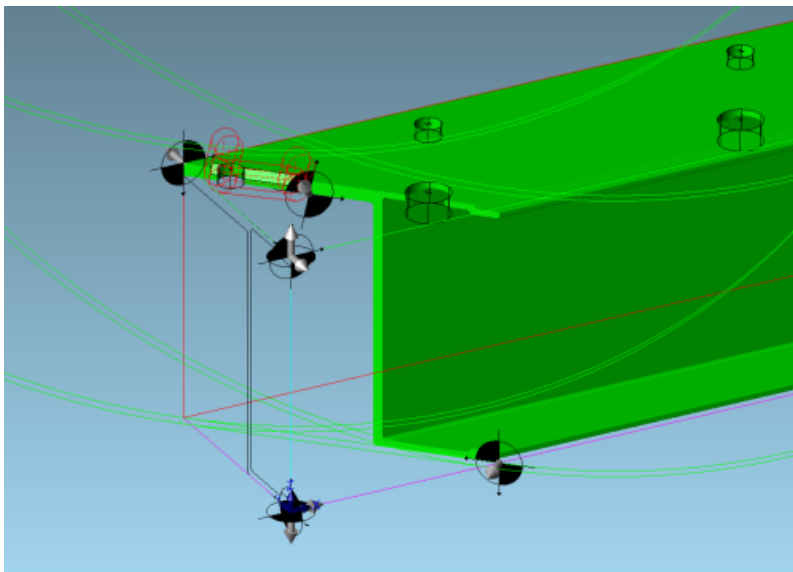
The advantage of using the PUMA system® with a separate 3D ACIS core is that models can be oriented in advance if the orientation of the solid model, e.g.: means that a great number of processing steps must be performed from beneath.

The extended processing options in the PUMA System® make it simple to add undetected milling contours as free milling paths directly on the solid model afterwards.



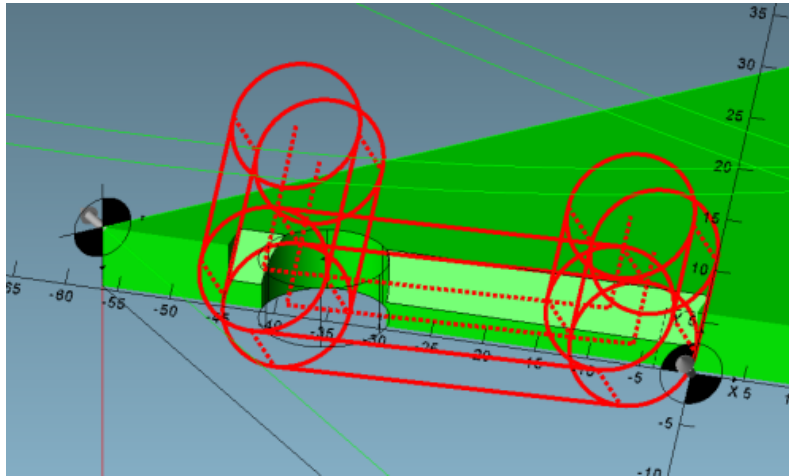
Before transfer to the 3DI2ncw plugin

and after transfer



Further processing in a compatible CAM system.

The chamfer on the left cut edge was added afterwards to the fully recognized features using the milling path function of the PUMA System®.



1.5 Getting help

You can get help by sending email to support@elusoft.de. Describe your situation and your goals briefly.

If you previously send a solid model to 3D@elusoft.de as a STEP file, our support staff can already have an idea of the result of the conversion. There will then also be a conversion report available.

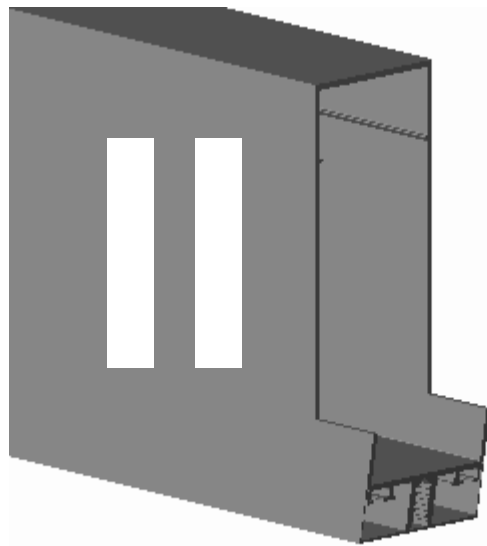
1.6 Terms

Collection of acronyms:

AFR	automatic feature recognition
IFR	interactive feature recognition
3DI2NCW	3D Import to NC Work
3D	solid model
NCW	text file with processing steps for NC-X
NC-X	converter that implements processing steps www.nc-x.com
DXF	cut section als drawing exchange file
STEP	STandard for the Exchange of Product model data
WSH	Windows Scripting Host
Feature	Features from which a processing step can be made.

Quick start

Part



2 Quick start

The following sections "Preparation" and "Calling the command line" will help you get a quick start.

2.1 Preparation

Our application automatically analyzes the solid and identifies machining patterns (standard machining processes). The basis is an AP203 Class 6-compliant STEP file provided by the customer.

Alternatively, .sat files generated from the ACIS kernel or .x_t files generated by the Parasolid kernel can also be used.

Our goal:

Every unnecessary step during data exchange between design and production can lead to frustrating errors in data transmission. We have tried to create compatibility.

License:

Feature Recognition is copy-protected and needs a Codemeter Hardlock. Please install this first, if you don't already have one of our installations (NC-X, eluCad, NC-luX).



2.2 calling the command line

```
*****
**      NC-X Numeric Control 3D to NCW Interface Ver: 1.0.0.17      **
**      Copyright (C) 2009 EluSoft GmbH                          **
*****
Usage: 3DI2NCW.exe i=<Source-3D-File> o=<Target-NCW-File> [c=<ini-file>]
```

The command `3DI2NCW.exe i=input.x_t o=output.ncw c=doors.ini` converts a Parasolid model into an NCW file. The doors.ini file is used for detailed settings.

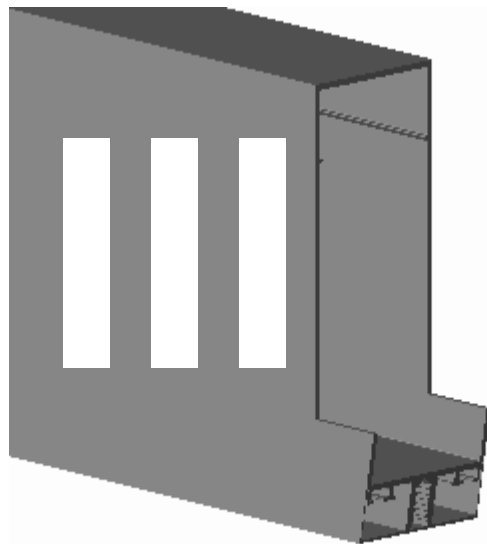
The command `3DI2NCW.exe i=input.sat o=output.ncw c=meinefirma.ini` converts an SAT model with an ACIS kernel to an NCW file. The mycompany.ini file is used for detailed settings.

The command `3DI2NCW.exe i=input.step o=output.ncw` converts a STEP model into an NCW file. By default, detailed settings are taken from the 3DI2NCW.ini file.

You can find more details about the command line [here](#).

Funtionality

Part



3 Functionality

Which data formats can be used, the scope of functionality, and how processing steps are assigned to the standard templates.

3.1 Data formats

STEP files, SStandard for the Exchange of Product model data

STEP is an interface standard initiated by the ISO (ISO 10303) that goes beyond pure geometrical data exchange (such as with DXF or IGES). STEP is intended to be able to represent all product data that occur during the lifetime of the product, and transmit them between different CAx systems. This includes all information about a product - from its initial development to retirement. Data are added during design, calculation, production, assembly, quality assurance, maintenance, and during operation. STEP's product model concept is based on the generation of an overall model that breaks down into partial models with specific, delineated information content, that are connected together and thus are related. STEP thus includes multiple standards within itself that are defined using a special language and consist of so-called parts, application protocols, and conformance classes. All types (wire, surface, solid models) of CAD data models can be integrated into the geometrical description. The AP 203 Class 2 application protocol describes wire and surface models, Class 4 defines surface models with topology, and Class 6 handles solid models. AP 214 is specifically designed for the automobile industry and can be used, for example, to transmit structural information (colours, levels, groups). For assemblies, STEP makes it possible to transmit information about the hierarchical structure. The ProSTEP organisation was founded to support the implementation of the STEP standard in current market products and services.

.x_t

Parasolid file, 3D CAD kernel format used by e.g. Unigraphics, SolidEdge, Solidworks, ANSYS, etc. Parasolid is a 3D CAD solid modeling kernel provided by Unigraphics Solution. Parasolid provides a library of C++ functions. Parasolid is not only the basis for systems by Unigraphics Solutions (SolidEdge and UNIGRAPHICS), but also 60 other CAD systems, finite element analysis packages, and NC programs by other providers.

.sat

ACIS file, 3D CAD kernel format used by e.g. ACIS, is also a volume kernel that has become a quasi-standard for CAD programs. ACIS is written in the object-oriented programming language C++ and is characterised by an open system architecture. In ACIS, wire and surface models can also be managed and manipulated. As of ACIS version 6, the function of "tolerant modeling" has been implemented. This permits imported models with inexact geometrical descriptions to be processed correctly.

3.2 What is recognized

Processing steps are anything that doesn't belong to the raw part.

The following introduction may be helpful:

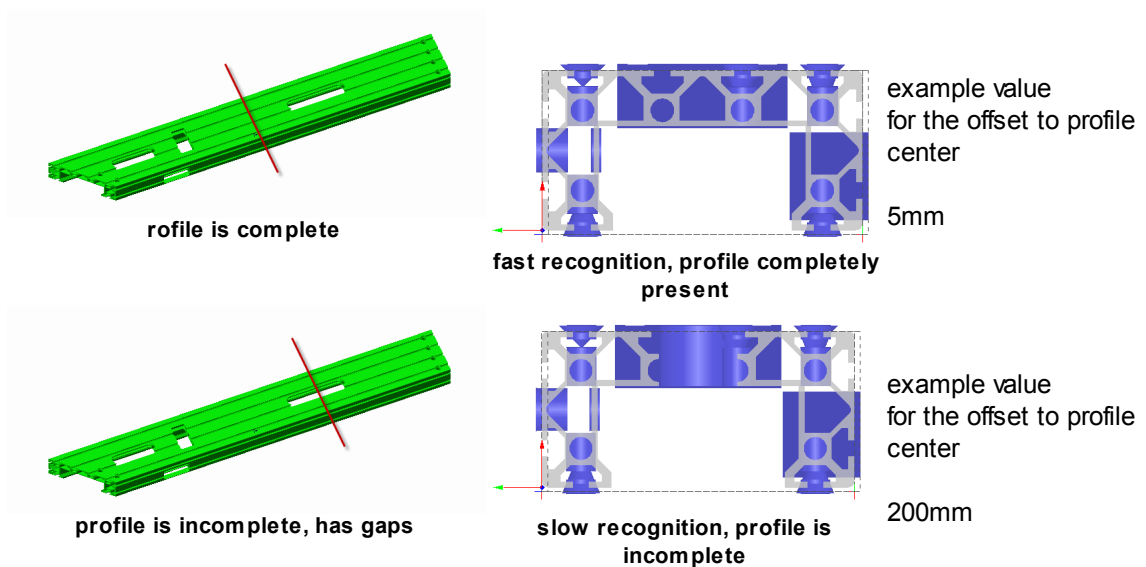
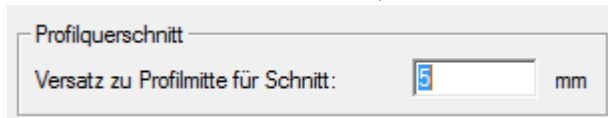
A profile cross section is determined from the model.

The profile cross section has the length of the model; differences between the newly generated model and the original model must be fabricated.

An attempt is made to assign these elements to a default template. If this is impossible, an attempt is made to generate a milling path, or to carry out the process as a saw cut / notch.

Important:

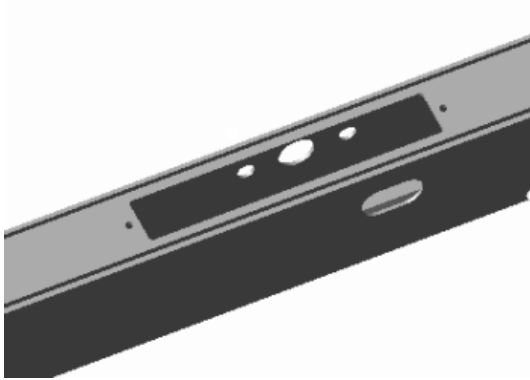
Until release with a new function, the orientation of the profile cross section is still important:



3.2.1 Standard processings

Standard templates cover most metalworking processes. These are

- Drillings
- circular pockets
- countersinks
- slots
- rectangles

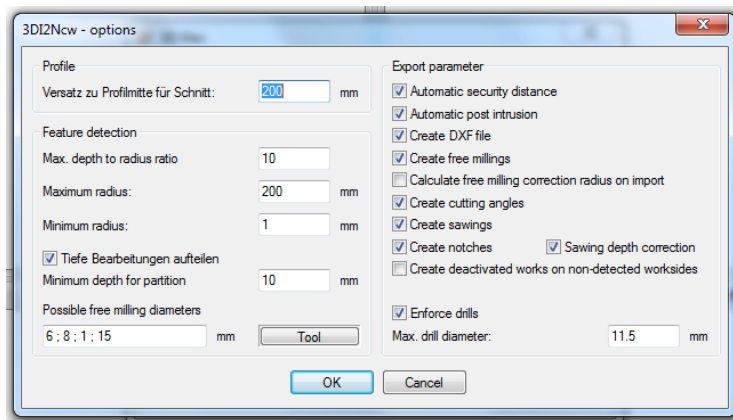


Rectangular features without radii in the corners cannot be fabricated and are not output. A service description with more illustrations can be found in the Introduction.

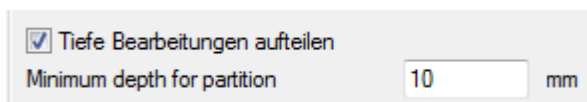
3.2.1.1 settings

For standard processes, options can be set to influence the process.

The options can be customized directly in the .ini files specified, or in the case of eluCad or NC-luX, in the Options dialog.

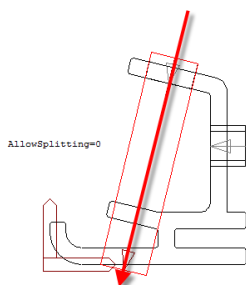


The Options dialog

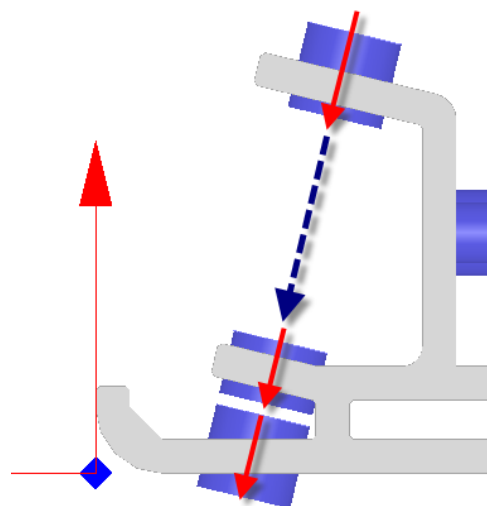


Options setup for depth tables has a direct influence on the direction from which the tool approaches.

If deep processing is not split up, a depth table is created and the processing is carried out from one side.

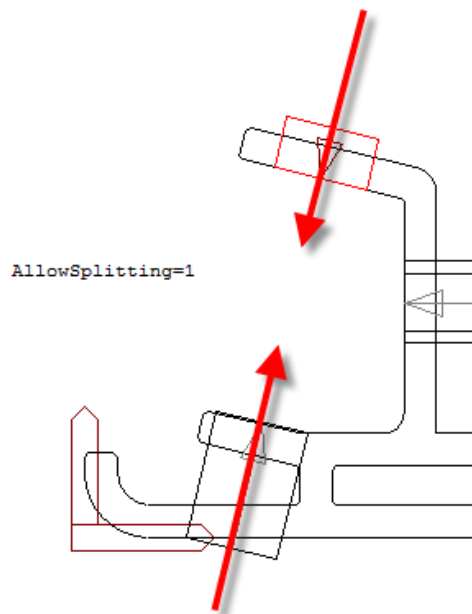


processing from one direction



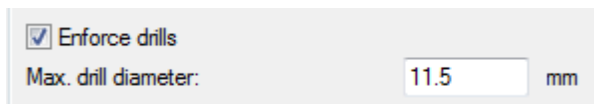
showing the depthtable

If the option is set to split up deep processing, a depth table is also created if necessary, but the tool approaches from both directions. This permits the use of shorter tools.



Processing split up in 2 directions

Drillings and circular pockets.

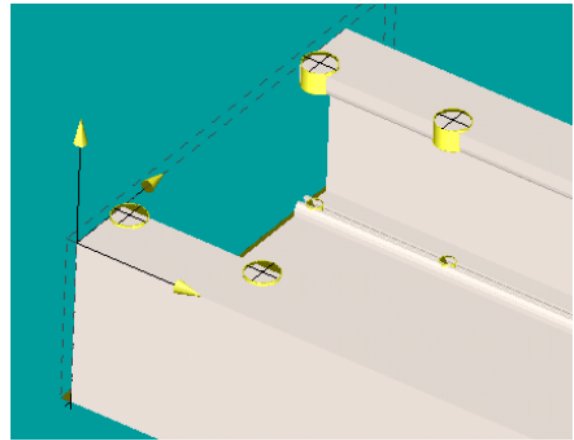
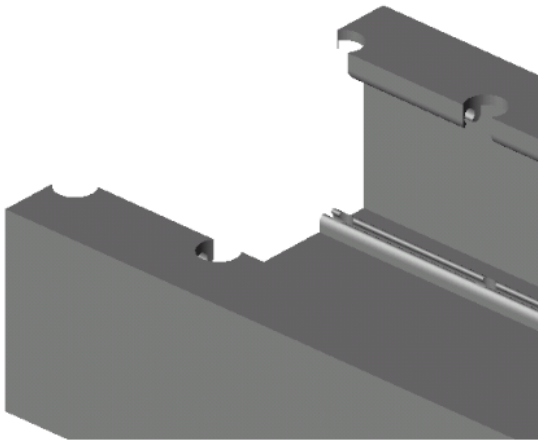


`MaxDiaDrillHoles=11.5`

Up to a diameter of 11.5 mm, an attempt is made to output closed circles as drillings. However, the circles may not start or end in any cut or partly on a web, in order to prevent the drill from skipping.

`ForceDrilling=1`

If `ForceDrilling=1`, closed circles up to the diameter `MaxDiaDrillHoles` will also be output as drillings even on unfavourable profile geometries.

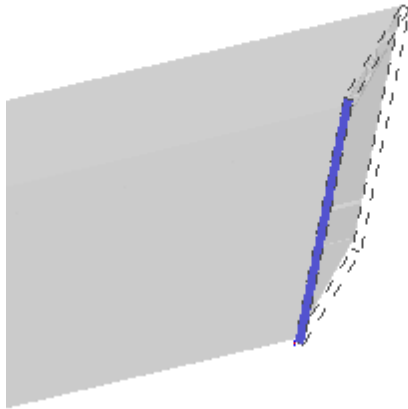


Unfavourable located, unclosed holes will always be converted to circular pockets

3.2.2 Cut left / right

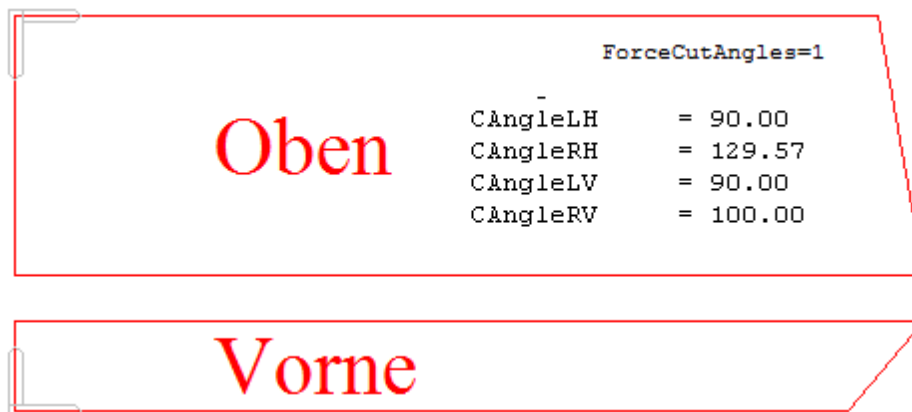
Sections, saw cuts on the start of the profile, are recognized and can be output. Sections are output as saw cuts if the corresponding option is set in the .ini file:

```
GenerateSawCuts=1
```



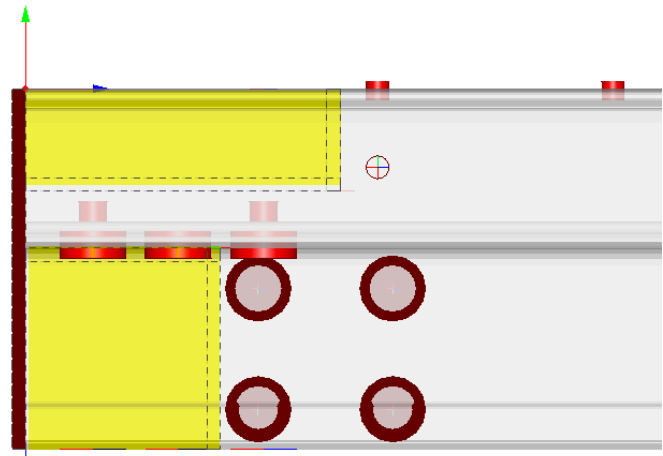
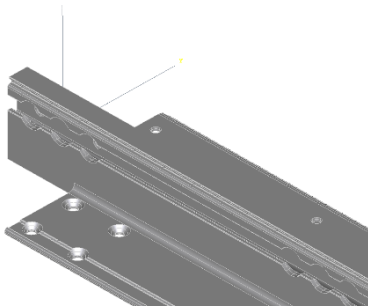
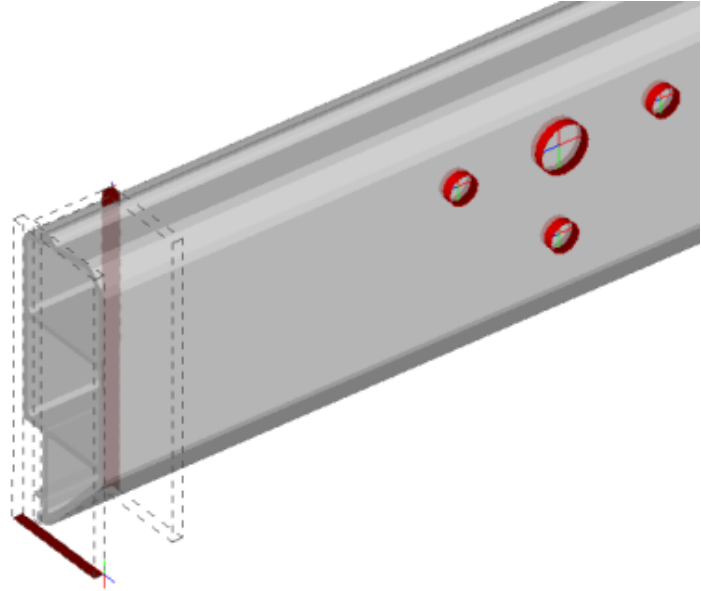
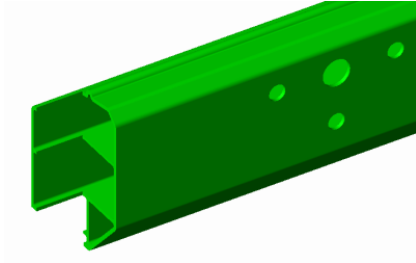
Sections are output in the parts definition as cut angles to the left/right with pivots/tilts, if the corresponding option is set in the .ini file:

```
ForceCutAngles=1
```

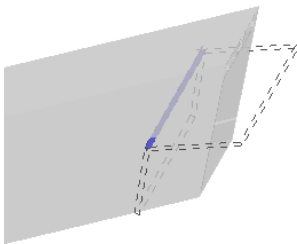


3.2.3 Notches

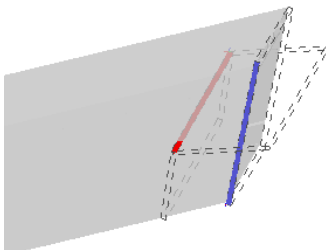
Saw cuts and notches are recognized by the feature recogniser. Notches are implemented in saw cuts if the access direction is specified and the saw blade will not collide with other model edges.



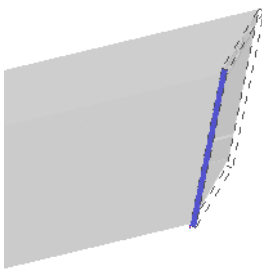
Saw cuts and notches will be output if the corresponding option is set in the .ini file:
GenerateSawCuts = 1 permits saw cuts in general.
GenerateNotches = 1 approves and outputs notches



GenerateSawCuts=0
GenerateNotches=1



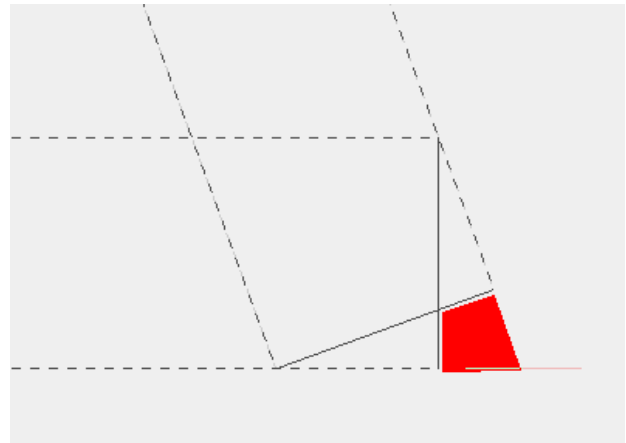
GenerateSawCuts=1
GenerateNotches=1



GenerateSawCuts=1
GenerateNotches=0

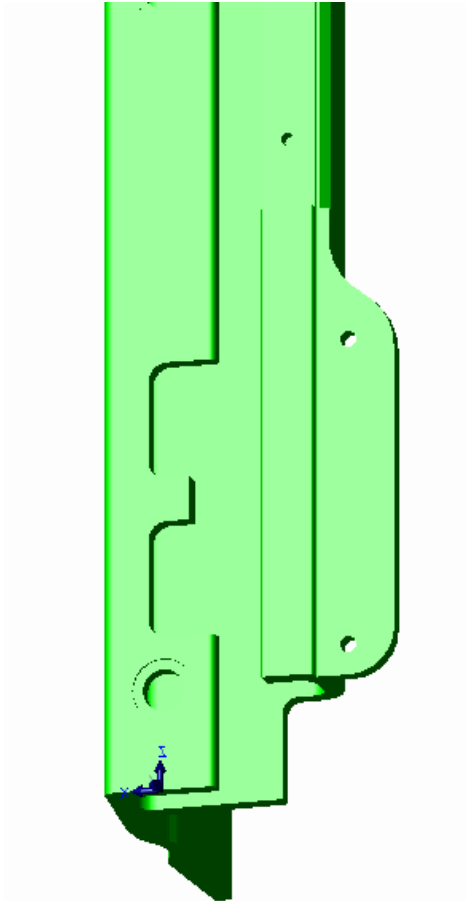
NotchCorr=1

NotchCorr=1 corrects the notch depth of notches with an angle of less than 90°, so the contour line will not be disrupted.

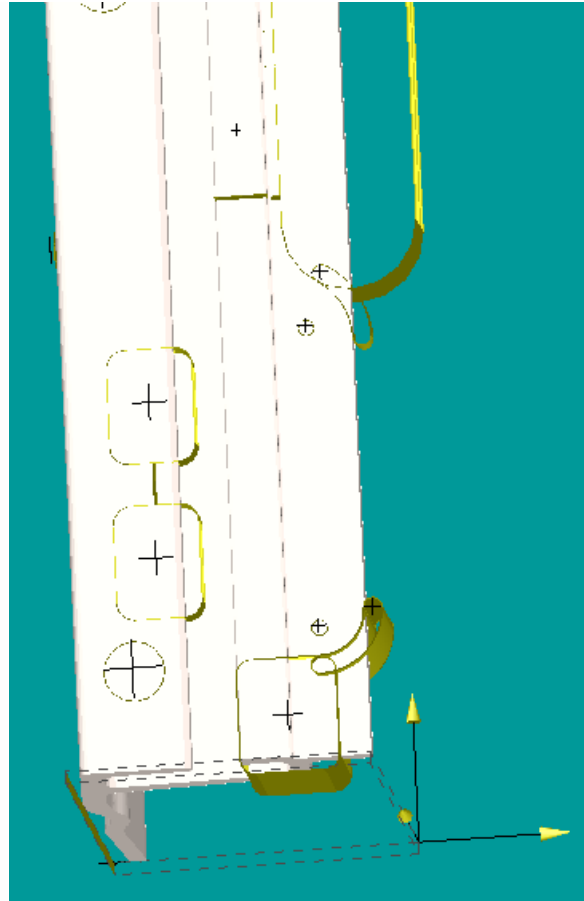


3.2.4 Milling paths

Up to ten tools can be entered in the INI file for free milling paths. The largest possible tool will be selected from this list, due to contour radii. The side dependency must be noted for angle head machines.

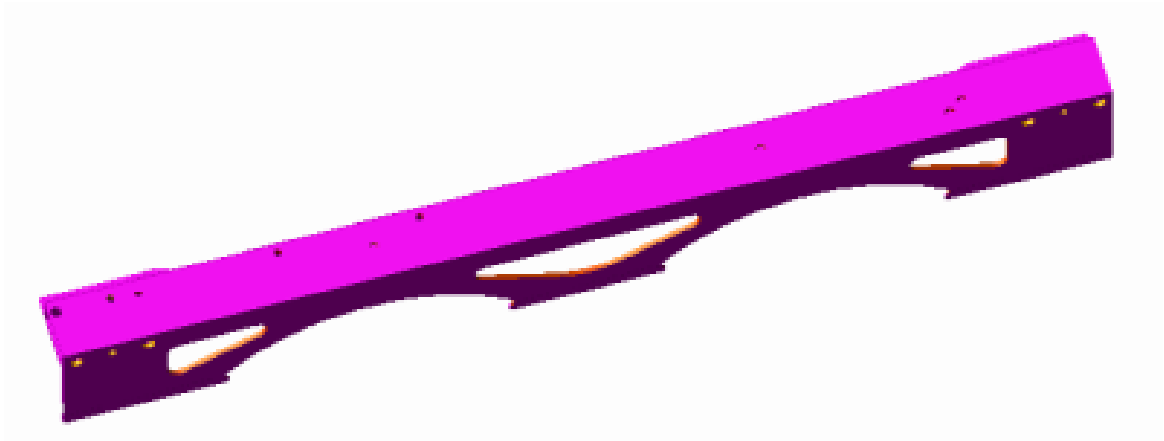


Milling path recognition combined with standard elements

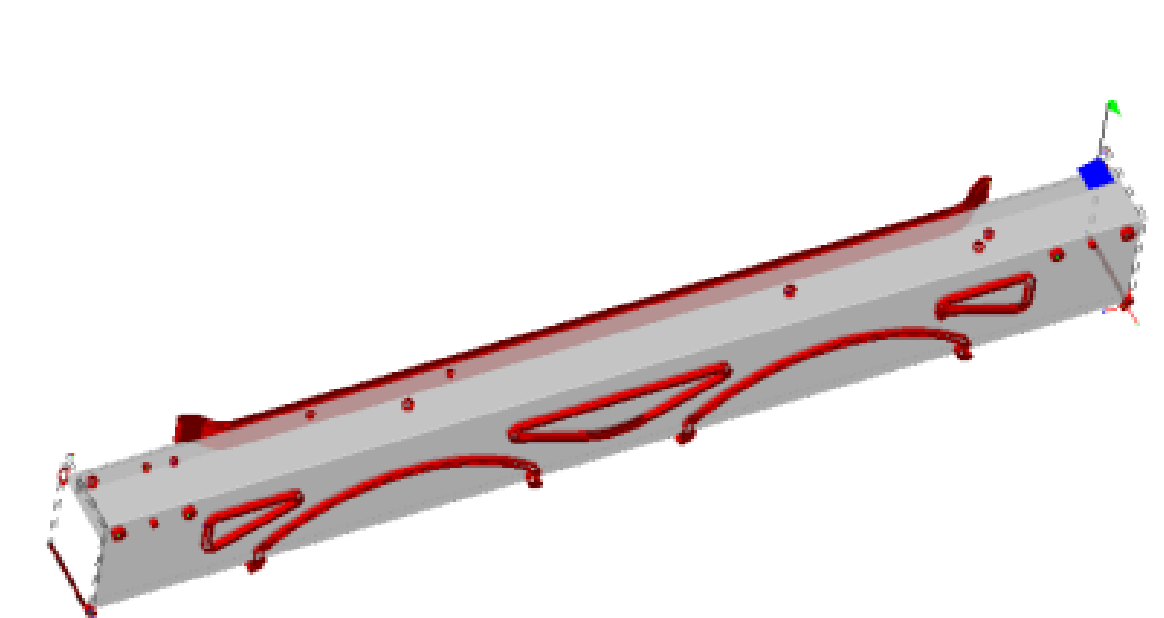


Milling path recognition combined with standard elements

The 3D import can work with inner and outer contours

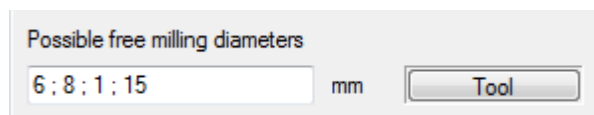


This model is implemented:



Processing after feature recognition

The milling tool that milling path processing will preferably be implemented is specified in the .ini file or in the dialog:

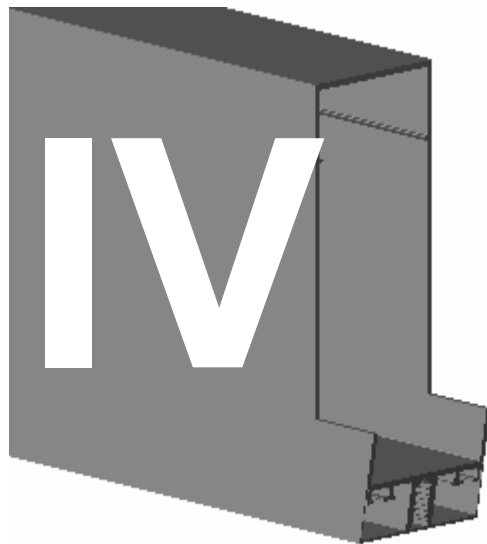


Diameter preset or fixed tool

If programming software with a compatible tool catalogue is available, a tool can be explicitly specified.

Admin info

Part



4 Admin info

This section provides information about the detailed settings for the INI file and an example of scripting using the Windows Scripting Host (WSH).

Our application automatically analyzes the solid model in different formats and identifies machining patterns (standard machining processes).

Our goal:

Every unnecessary step during data exchange between design and production can lead to frustrating errors in data transmission. We have tried to create compatibility.

4.1 Setup

You will need

- A license with 3DI release
- The installation files, consisting of 3DI2NCW.exe with the associated TK* DLLs.
- 3D solid model
- a NCW-Viewer (of www.nc-x.com)
- An NC-X converter for further processing of the .ncw files with the processing templates
- Post-processors for the target machine

4.1.1 The .ini file

The feature recogniser can be influenced by the user.

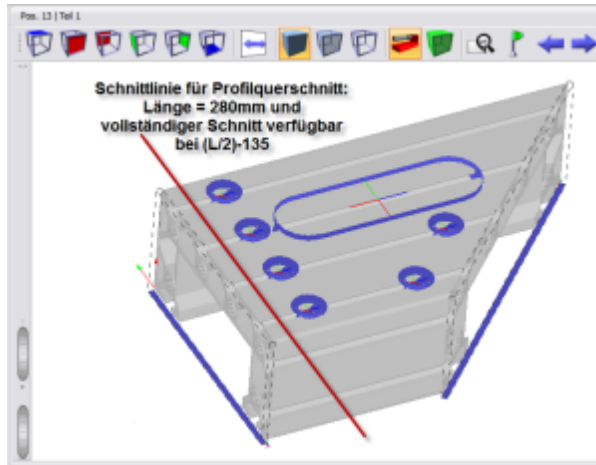
The idea is that project-specific or target machine-specific detailed settings can be set up.

To do this, the .ini file is modified, a purely ASCII text file that always starts with

```
[Settings]
```

```
SectionOffset=-5
```

gives the position at which the full profile cross section can be found.



The offset is positive or negative and runs from the centre of the part in the indicated direction to the indicated distance.

```
RatioDepthRadius=10
```

Circular pockets or Drillings with a depth greater than 10 times the diameter are not taken into consideration.

```
MaxRadius=200
```

```
MinRadius=1
```

Drillings and circular pockets can only have a diameter between 1 and 200 mm.

```
AllowSplitting=0
```

Deep processing may be split into processing from two sides. This is a practical setting for machines that process side front and back with an angle head. If Drillings go horizontally completely through an 8mm wide profile, they are split into processing from front and back. This permits the short tools of an angle head to work well.

```
MinSplittedDepth=10
```

If the splitting of processing is permitted, penetrating Drillings, elongated Drillings and rectangular features will be output from one side to the centre of the bounding box. However: If this means that the rest of a processing step would be required from the opposite side that is e.g. less than 10 mm deep, this processing step will not be split.

MaxDiaDrillHoles=11.5

Up to a diameter of 11.5 mm, an attempt is made to output closed circles as drill Drillings. However, the circles may not start or end in any cut or partly on a web, in order to prevent the drill from skipping

ForceDrilling=1

If ForceDrilling=1, closed circles up to the diameter MaxDiaDrillHoles will also be output as drilled Drillings even on unfavourable profile geometries.

GenerateFreeMilling=1

If true=1, milling paths will be generated; for false=0, no milling paths will be generated.

RadiusCorrFreeMilling=0

The converter calculates the radius contour if RadiusCorrFreeMilling=1, and outputs the corrected milling path in the NCW file with direction "Centre". For RadiusCorrFreeMilling=0, the contour detected will be output without correction, with direction "Left" or "Right". The correction can then be calculated e.g. in eluCad or NC-luX depending on the tool selected there.

ToolDiaFreeMilling_1=8

ToolDiaFreeMilling_2=6

ToolDiaFreeMilling_3=15

Up to 10 Tools ToolDiaFreeMilling_1 ToolDiaFreeMilling_10 can be supported.

There is no preferred order for selection; the selection will in future be made based on the radii in the milling path - the largest possible diameter will then be selected.

GenerateDXF=1

Should the profile cross section be output as .dxf?

This makes viewing the generated .ncw files in the NCW Viewer clearer.

GenerateSawCuts=1

GenerateNotches=1

Should saw cuts and notches be output?

If you have a machine without a sawing option, you can suppress the output by specifying =0

GeneratePlanes=0

Surfaces that cannot be assigned any process are output as deactivated surfaces, and the user can use dialog-oriented programming systems like eluCad or NC-luX to specify post-processing more easily.

OScale=1

The output is provided as a metric file 1 = 1 mm. If 25.4 is specified, a file in inches is assumed.

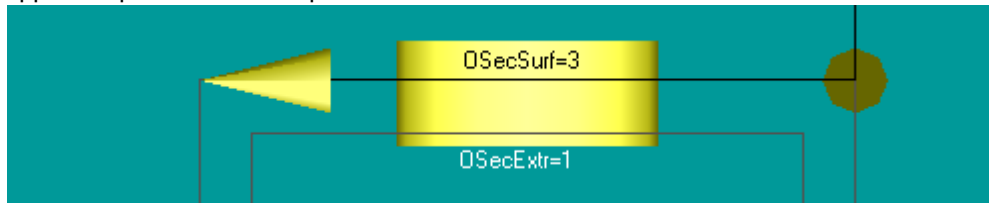
OSecSurf=3

OSecIntr=1

OSecExtr=1

These parameters control the values of the safety distances that are added to the actual calculated

approach positions and depths.



`WDTSecIntr=1`

`WDTSecExtr=0`

How should safety distances be handled?

=1 outputs them, =0 outputs no additional depths (follow-up).

`NotchCorr=1`

Notches with opening angles of less than 90° are corrected in the notch depth, so no contour violation occurs due to the thickness of the saw blade.

`ForceCutAngles=1`

Should the starting and ending angles be output in the part description?.

`Precision=2`

Describes the number of decimal places in the NCW file generated

`SawCutExtr=2`

Additional depth for saw cuts, so that sections can be cut cleanly.

Many parameters and their meanings depend on the NC-X code and are described in more detail at www.nc-x.com/doku.

4.2 Scripting with WSH

A solid model seldom occurs in isolation. To transfer hundreds of solid models, manual entry of the call parameters is not suitable. A script for this purpose created with built-in Windows tools is more practical for the transfer of all the files in a directory:



The input directory on the left, and output directory on the right.
The process runs until every file in the input directory has been processed.

4.2.1 Example-Script

```
' starts the step-converter with all files of a folder
' please check folders in your application
' -----
sSourcefolder = "D:\Program Files\elusoft\Step2NCW\3DI2Ncw-1.0.0.0\input\"
sOutpath      = "E:\Step2NCW\output\"

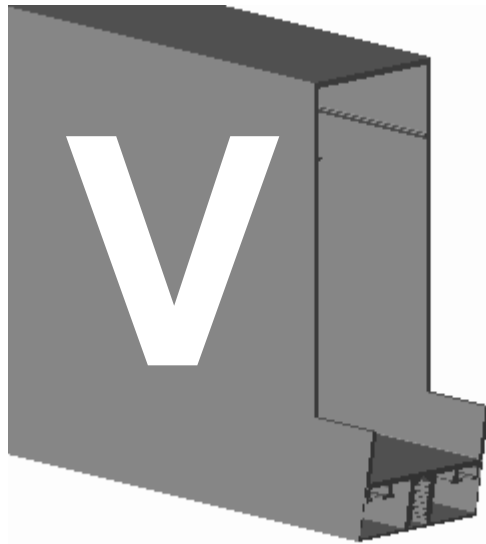
Dim fso, wshshell, hochkomma, filecounter
Dim RegPath, target, outpath
Dim ncwfile, reportfile

hochkomma = chr(34)
const app_path = "D:\Program Files\elusoft\Step2NCW\3DI2Ncw\" 'path to application
const app_name = "3DI2Ncw.exe" 'name of application

Set wshshell = createobject("wscript.shell")
Set oFSO = CreateObject("Scripting.FileSystemObject")
Set oFolder = oFSO.GetFolder(sSourcefolder)
' -----
filecounter = 0
For Each oFile In oFolder.Files
    filecounter = filecounter + 1
    filename = Right(oFile, Len(oFile) - Len(sSourcefolder))
    ncwfile = sOutpath & Left(filename, Len(filename) - 4) & ".ncw "
    wshshell.run(hochkomma & app_path & app_name & hochkomma & " i=" & hochkomma &
        oFile & hochkomma & " o=" & hochkomma & ncwfile & hochkomma )
    s = 0
    i = 0
    'give some time to convert
    do while s < 5000
        s = s + 1
        do while i < 5000
            i = i + 1
            Set objWMIService = GetObject("winmgmts:" _
                & "{impersonationLevel=impersonate}!\\" & "." & "\root\cimv2")
            Set colProcesses = objWMIService.ExecQuery _
                ("Select * from Win32_Process Where Name = '" & app_name & "'")
            If colProcesses.Count = 0 Then
                s = 5000
                i = 5000
            Else 'instance is already running, wait again
                wscript.sleep 2000
            End If
        loop
    loop
' -----
    ' start NCWViewer showing the result
    wshshell.run("E:\NcwView\NcwView.exe /LEFT " & ncwfile)
'
Next ' file from folder
' -----
WScript.echo "finished AFR of " & filecounter & " step files !"
```

Capacities

Part



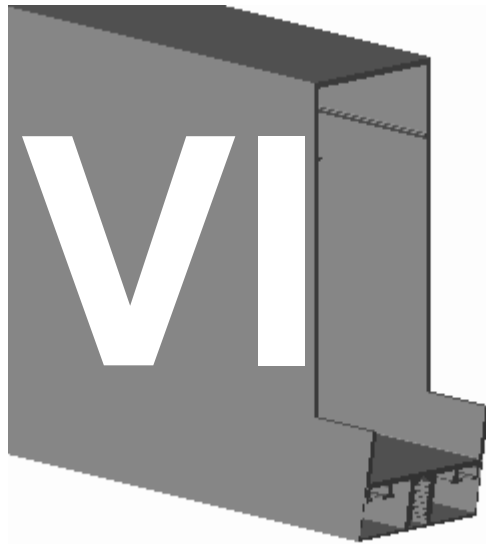
5 Capacities

The search for machining processes is limited to 2.5D features, such as drillings, grooves, pockets, saw cuts and free milling paths which are typical for a machining process on 3-axis machines. A 4th and 5th axis are supported, however not for simultaneous movements.

- * We process an AP203 Class 6 Step file (solid model)
- * The parts must be specified as single parts parallel to one axis of the coordinate system
- * Bent or hydroformed profiles cannot be machined by this feature recognition.
- * Processing must be possible (no rectangular features without a corner radius)

Tools

Part



6 Tools

a free viewer for step-files:

Download IDA-STEP from <http://www.ida-step.net/download>, where this info is copied from.

IDA-STEP v4 is installed in a two step process; you first install the basic framework on your PC and then the installation is completed or later updated out of the running application through the Internet. With a good Internet connection this should not take you less than 15 minutes.

First step: Download and execute the basic framework installer

- * IDA-STEP for Windows (32-bit) (37.18 MB)
- * IDA-STEP for Windows (64-bit) (37.57 MB)
- * IDA-STEP for Linux (32-bit) (21.2 MB)
- * IDA-STEP for Linux (64-bit) (21.3 MB)

and follow the instructions on the screen.

Note: For Linux you first have to extract the IDA-STEP archive into a folder before you can start the executable from there.

Second step: After the initial installation start IDA-STEP. The "Register, Update and Install" dialoge will show up.

Select either "Free license" or "Enter license ID" if you purchased one. IDA-STEP then connects to the Internet and displays available components to download (for the free license this is "Viewer: Basic"). Select all of the offered components for installation and follow the given instructions. At the end IDA-STEP restarts and the installation is finished.

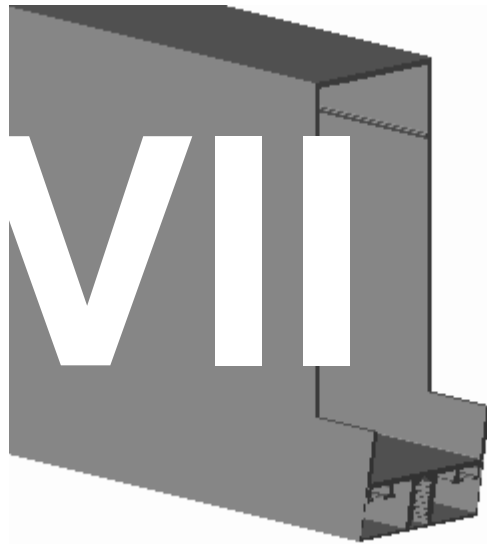
View detailed installation instructions for IDA-STEP v4 [here](#).

Free IDA-STEP Viewer Basic

When processing the second installation step with the "Free license" option, a free one year single computer "IDA-STEP Viewer Basic" license is automatically granted to you. You can repeat this process at any time (e.g. after one year) and on as many computers as you like. You can use the granted license for any purpose, including commercial, private and academic use.

Update history

Part



7 Update history

23.07.2010

- * 3DI: More tolerance, opening angle of 181° will still be recognized as a slot

18.06.2010

- * 3DI: New setting for follow-up for recognized separation cuts

08.06.2010

- * 3DI: New import options for free forms

23.03.2010

- * 3DI: New options "Max. diameter for drillings" and "Force drilling" built into the Options dialog.

23.02.2010

- * 3DI: The 3DI Options dialog now also writes the current settings for the dimensional system, the number of decimal places, and the current depth table safety distances to 3DI2Ncw.ini.

01.02.2010

- * 3DI: File selection added to the import file dialog for SAT and Parasolid XT.
- * 3DI: Code input for 3DI removed; in future 3DI will always be visible after dongle release.
- * 3DI: Option "ForceCutAngles" added for "Create cut angle" (even with cut loss) in the Options dialog.

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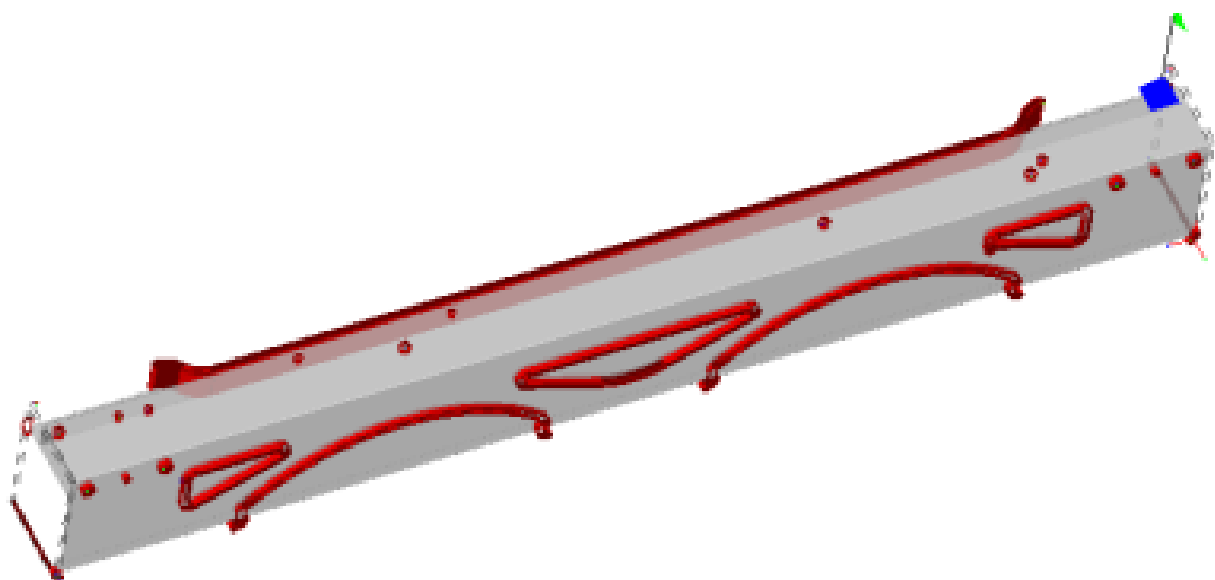
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www.elusoft.com & www.nc-x.com