Cimatron®

WHAT'S NEW

Cimatron 15 Official Release

3D SYSTEMS

Contents

Introduction 1

1 Version Highlights 2

1.1 Cooling Channel Design and Manufacturing 2

1.2 Conformal Cooling Design 2

1.2.1 A new conformal application 2

1.2.2 New analysis tools were added 3

1.3 Conformal Cooling Standalone seat for non-Cimatron users 3

1.4 Traditional Cooling Design 4

1.4.1 New cooling channels distance analysis 4

1.4.2 Cooling labeling 4

1.5 Additional Mold Design Capabilities 5

1.5.1 Mold Correction - Warpage Compensation 5

1.5.2 New Direction Analysis 5

1.6 New Mill/Turn Application 6

1.7 5 Axis Machining 7

1.7.1 3+2 Axis Roughing Automation 7

1.7.2 New Geodesic Procedure 8

1.7.3 New Deburring Procedure 9

1.8 Plate Machining and 2.5 Axis Milling 10

1.8.1 New Slot Procedure 10

1.8.2 New Corner Plunging Procedure 11

1.8.3 Helical Milling of Closed Profile 11

1.9 Programing Automation 12

1.9.1 3+2 Axis Remachine Automation 12

1.10 Easier Programing 13

1.10.1 Shank and Holder Dialog 13

1.11 Faster Machining 13

1.11.1 New Circle Segment Cutters 13

1.11.2 Creating UCS considering Circle Segment Cutters 14

1.12 On Machine Inspection – Read back Report 16

1.13 Automated Drill 17

1.13.1 Automated Drill - Automated Names of Groups 17

2 Mold Design 18

2.1 Conformal Cooling Improvements 18

2.1.1 Conformal Cooling design Improvements 18
2.1.2 Automatic Conformal Cooling Curve 20
2.1.3 Conformal Cooling Standalone seat for non-Cimatron users 20
2.1.4 Cooling Distance Map 21
2.1.5 Cooling labels 22
2.2 Mold Correction - Warpage Compensation 23
2.3 New Deformation Correction toolbar 25
2.4 Find best orientation for opening direction 26
2.5 Quick Split - Find best extraction orientation for Sliders and Lifters 27
2.6 Transfer colors and sets in cut active 28
2.7 Hole Analysis Tools Improvements 28
2.7.1 Working without the Pane commands 28
2.7.2 Hole Safety Distance – Sort by Distance 28
2.7.3 Hole Safety Distance – Separate analysis for active faces 29
2.7.4 Hole Safety Distance – Improved rendering 29
2.8 Collect text features in a folder 30
2.9 Cooling Items with Channel Improvements 30
2.9.1 Round length downwards 30
2.9.2 Go to higher level assembly in Add Cooling Item with Channel 30
2.10 Implement Add Diameter as Part Size for cooling parts (especially Baffles) 30
2.11 Ejector Pocket improvements 31
2.11.1 Use catalog diameter for ejector pocket 31
2.12 Measure minimum distance in ejector pocket 31
2.13 Correct Numbering of Sleeves in Table of Ejectors 32
2.14 Insert Design Improvements 33
2.14.1 Chamfer for Wedges 33
2.15 Create Runner part in the active sub-assembly 33
3 Electrode Design 34
3.1 Find electrode extraction direction 34
3.2 Allow to define Rotation Angle using direction arrow 34
3.3 Electrode – Add rotation angle to cylindrical blanks 35
3.4 Allow Select All / Manual Selection for burning faces in EDM Setup 35
3.5 Add Pictures in EDM Excel Report 37
3.6 EDM Setup - Output from Cimatron's EDM Setup to GF Machining Solutions' Multi-Process Preparation (MPP) software 38
3.7 EDM Setup – Interface with GF Machining Solutions' MPP 39

4 Die Design 40
4.1 Springback 40
4.2 Match Points on Skin 41
4.3 Add Correction Factor for FTI (forming) Forces 41
5 Part Design 42
5.1 Rib 42
5.2 Offset Wire Along Faces 42
5.3 Volume Inside Shell 43
5.4 Direct Modeling and Extend Object 43
5.4.1 Offset From Reference 43
5.4.2 Merge Adjacent Faces 44
5.5 Surface Design Improvements 45
5.5.1 Blend - Single/Multi Faces 45
5.5.2 Skin – Virtual Guides: 45
5.5.3 Loft – Smooth/Free Faces Connection: 46
5.6 Conflict Solver 46
5.7 Mesh Design 47
5.7.1 Composite on Mesh 47
5.7.2 Unite Mesh – New Options 47
5.7.3 Smooth Mesh 48
5.8 Minor Wireframe Design Improvements 49
5.8.1 Simplify wireframe 49
5.8.2 Multiple Wires 49
5.8.3 Skeleton Curve 49
5.8.4 Plane 49
5.9 Analysis 50
5.9.1 Measuring - Faster Calculation of Mass properties 50
5.9.2 Interference Check on Mesh 50
5.10 Sketcher 51
5.10.1 Slot Design 51
5.10.2 Improve associativity of offset operations 51
6 Assembly Design 52
6.1 BOM Enhancements 52
6.1.1 Add grouping parameters in purchase list 52
6.1.2 Add a picture of each part in the Excel report on the table 53

6.1.3 Allow to save report with assembly or in designated folder 54
6.1.4 Allow to select more than one folder or SUB-ASSEMBLY 54
6.1.5 Allow ignore in BOM for sub-assemblies 54
6.1.6 BOM File Name Generator 55
6.2 Project Number 55
6.3 New Add Duplicate Options 56
6.3.1 Drag & Drop Add 56
6.4 New Add This Component Options 56
6.4.1 Add as a Duplicate Copy 56
6.4.2 Delete Original Component 57
6.5 Hide / Show Multiple objects 58
6.6 Nested Add – Add on Cone Face 58
6.7 Restore Activate Main Assembly as part of the RMB 59
6.8 Cut Improvements 60
6.8.1 Hide and Show Cutting Objects 60
6.8.2 Cut Manager: Select All / Clear Selection - changed to a checkbox 60
6.9 Catalog Parts 61
6.9.1 Show catalog columns as file attributes 61
6.9.2 Catalog picture in Add Mold/Die Component 61
6.9.3 Cutting object with or without “transfer color" 62
6.9.4 Catalog Updates 63
6.9.4.1 New Functionality 64
7 Drafting 66
7.1 Predefined Styles for Symbols 66
7.2 Section line as sketcher 67
7.3 Clipped View 68
7.4 Multi View – Bounding Box Dimensions 69
7.5 Implement multi view options in a single view 70
7.6 Define depth of origin point using another view 71
7.7 More Characters in Geo Tol and Geo Datum 71
7.8 Start Center lines automatic ordinate at line end 71
7.9 Insert Picture – support PNG 71
8 General Enhancements 72
8.1 Data Management 72
8.1.1 Browser Improvements 72

8.1.2 Save part images when saving an assembly 73
8.1.3 Reconnect icons in a separate column to make them bigger and clearer 74
8.1.4 Do not open the Reconnect dialog 75
8.1.5 Concurrency – deal with duplicate names 75
8.2 Data Interface 76
8.2.1 Import and Export using the Cimatron Explorer 76
8.2.2 Imported file created as temporary file 77
8.2.3 Notify the user when DI operations fail 78
8.2.4 Decimate Mesh when triangle count is over 79
8.2.5 Read in pictures from DWG and DXF 79
8.2.6 File Attributes 79
8.2.7 Add control over turning black edges to white when writing to DWG/DXF 80
8.3 Graphics 81
8.3.1 Dynamic Section 81
8.3.2 Highlight style 82
8.3.3 Snap to Main View 82
8.3.4 Display Jagged Edges 83
8.3.5 Filter Improvements 83
8.3.5.1 Select a single filter 83
8.3.5.2 Point by Intersection 83
9 NC 84
9.1 Turning Application 84
9.1.1 Overview 84
9.1.2 Workflow 84
9.1.2.1 NC Setup and Configuration 85
9.1.2.2 Turning Cutters 86
9.1.2.3 Stock Definition and Update 87
9.1.2.4 Geometry and Contour Creation 87
9.1.2.5 Turning Procedures 88
9.1.2.6 Review and Output Toolpath 89
9.1.2.7 License: Turning Module and Standalone Turning 90
9.1.3 Custom Toolbar: Lathe Views 90
9.2 5 Axis Machining 91
9.2.1 3+2 Axis Roughing Automation 91
9.2.2 New Geodesic Procedure 92

9.2.3 New Deburring Procedure 93
9.2.4 Enhanced Multi Blade Basic procedure 94
9.2.5 5 Axis Tilting, more control 94
9.3 Plate Machining and 2.5 Axis Milling 94
9.3.1 New Slot Procedure 94
9.3.2 New Corner Plunging Procedure 95
9.3.3 Helical Milling of Closed Profile 96
9.3.4 New Safe Profile Procedure 96
9.3.5 Pocket Manager Enhancements 96
9.3.6 Plate Machining Pro seat 96
9.4 Programing Automation 97
9.4.1 3+2 Axis Remachine Automation 97
9.4.2 Add Spark Gap to Remachine Segments 97
9.4.3 Select Z Limits by Criteria 97
9.4.4 Max-Part-Z – new parameters 97
9.4.5 Closed Profile Start Point – better user control 98
9.4.6 Finish by Layers New Options 98
9.5 Easier Programing 99
9.5.1 Shank and Holder Dialog 99
9.5.2 Use scallop in Finish procedures 100
9.5.3 Improved Cutter display within the Cutter Table 101
9.5.4 Conic tool - better definition 101
9.6 Faster Machining 102
9.6.1 Rough Efficiency with AFC 102
9.6.2 New Circle Segment Cutters 102
9.6.3 Facing Efficiency 103
9.6.4 Rough Efficiency 103
9.6.5 Rough Smoothness 103
9.6.6 Fast Motions more Control 103
9.7 Faster Calculation 104
9.7.1 Rough Performance 104
9.7.2 Stock Update Performance 104
9.8 Enhanced User Control 105
9.8.1 On Machine Inspection – Read back Report 105
9.8.2 On Machine inspection – Highlight Probes 105

9.8.3 Stock and Part information in the NC Report 106
9.8.4 Utility Procedure 107
9.8.5 More Coolant and Multi Coolant 108
9.9 Surface Quality 109
9.9.1 New Rerough Strategy in Cleanup Procedure 109
9.9.2 Finish - 3D Cutter Compensations 109
9.9.3 Finish by 4 Limit Angles 110
9.9.4 Optimized Variable Down Step for Finish by Layers 111
9.9.5 Finish - Short Connections for coincident layers 112
9.9.6 Finish - Avoid sharp approach and retract 112
9.9.7 Cleanup - Round Corners 113
9.9.8 Chamfer - Opposite tangential approach 113
9.10 Automated Drill 113
9.10.1 Automated Drill - Manually assigned sequences are not lost 113
9.10.2 Automated Drill - Automated Names of Groups 114
9.10.3 Automated Drill - Add Sequential numbers for operations 115
9.10.4 Automated Drill – Split by Cutter 115
9.10.5 Cutter Compensation for Automated Drill Helical Profile 116
9.11 Simulator 117
9.11.1 Simulator - Saving deviation table, colors and safety 117
9.11.2 Simulator - Deviation on initial stock 117
9.11.3 Simulator – Display the current feed rate in the Motion List table 117
9.11.4 Simulator - Performance and movie smoothness improvements 118
9.11.5 Prevent false errors on imaginary chamfer 118
9.11.6 Support Shaped Cutters 118

Introduction

Cimatron 15 offers major benefits and more user value to manufacturing shops across their entire range of operations. From Tool design through NC programming, this latest version helps to ensure that our customers are more productive than ever, in their competitive marketplace.

1 Version Highlights

1.1 Cooling Channel Design and Manufacturing

The Cooling application was dramatically improved in this version, while we continue to support mixed/hybrid manufacturing environments with subtractive and additive solutions. A mix of both traditional and conformal cooling capabilities enables efficient design, resulting in shorter injection cycle time and better parts quality due to reduced warpage.

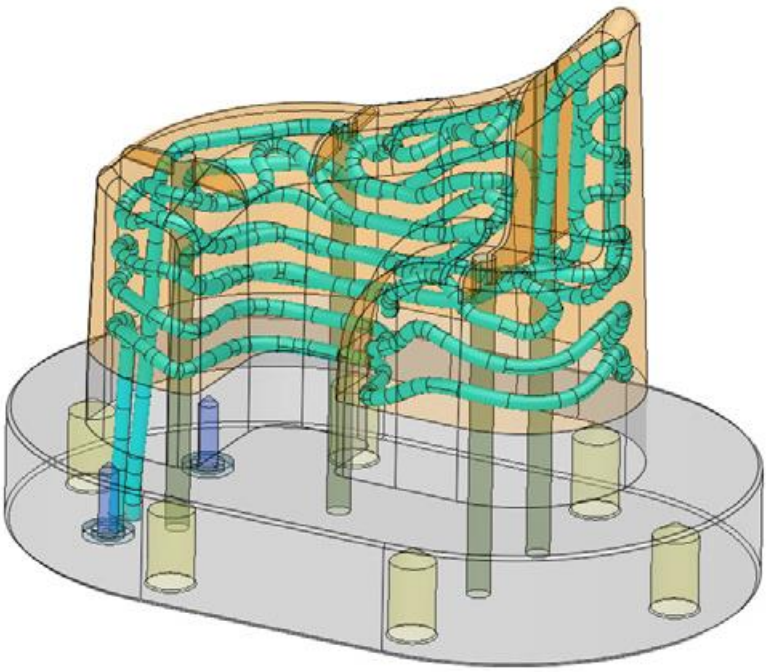
1.2 Conformal Cooling Design

1.2.1 A new conformal application

The advantages of using conformal cooling are clear in relevant cases but the design of conformal cooling channels used to required expertise and knowhow. With the new release, any mold maker will be able to generate conformal cooling channels with fast and easy to use automated tools. Hours of design work can turn into minutes.

Users can control and modify the automated results, or create their own channel design much easier.

For additional information, see page 18.



Automatically generated Conformal Cooling channel

1.2.2 New analysis tools were added

Two new analysis tools are available for conformal cooling ensuring the quality of the design in terms of successful 3D printing, cooling efficiency and safety of the cooling system throughout its life time (ensuring there are no breaches and leakage)

- Overhang analysis – to ensure no internal areas within the cooling channels will fail during printing.
- Distance analysis and optimization – identify areas where the channels may be too close or too far to/from the walls of the insert and adjust them automatically.

For additional information, see page 19.

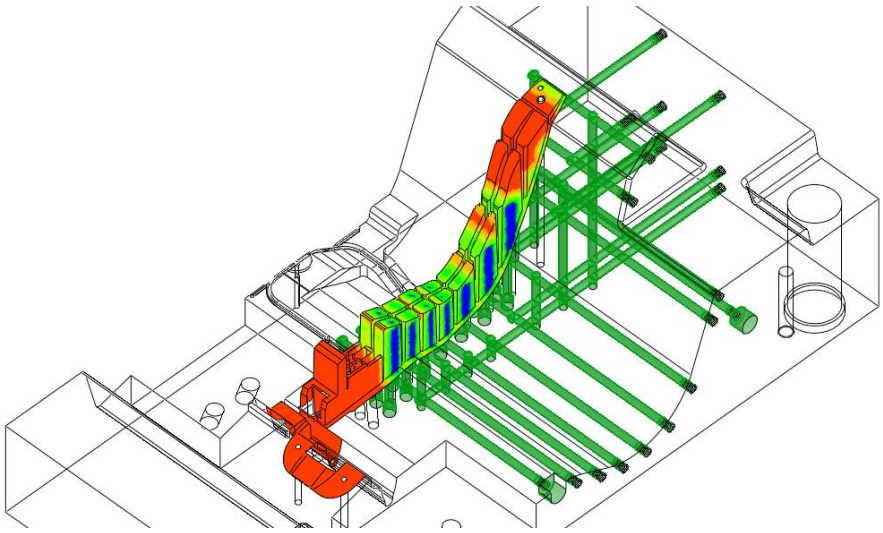
1.3 Conformal Cooling Standalone seat for non-Cimatron users

A standalone seat for conformal cooling design is offered to mold makers using other software for mold design or conformal for cooling printing bureaus, with all the tools necessary to complete the design.

1.4 Traditional Cooling Design

1.4.1 New cooling channels distance analysis

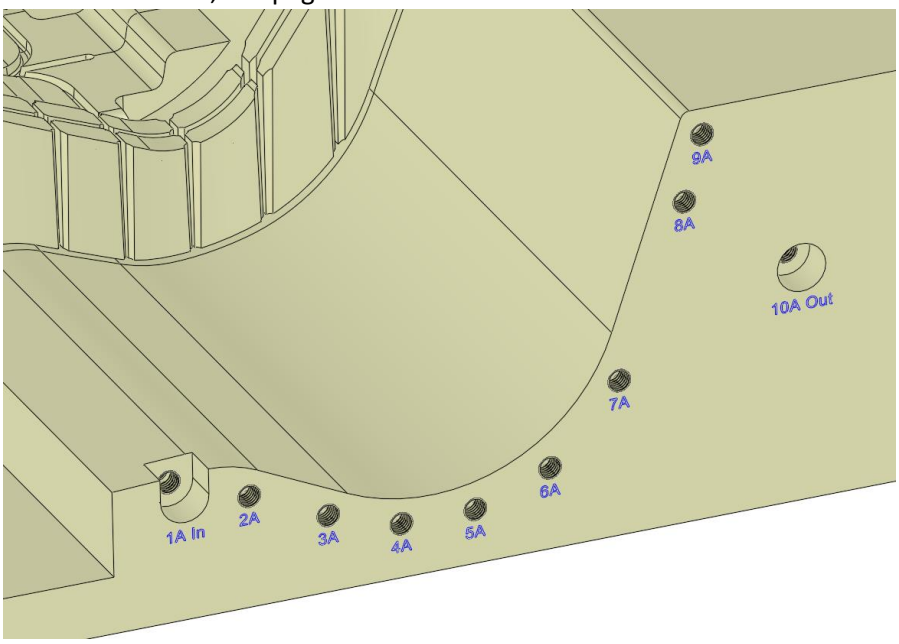
Extremely fast analysis for a quick glance at cooling efficiency based on the distance between the cooled faces and cooling channels.
For additional information, see page 21.



A color map showing the distances between the cooling channel and the faces of the mold that form the shape of the plastic part

1.4.2 Cooling labeling

Enables easily adding labels next to each channel, save time and avoid mistakes.
For additional information, see page 22.



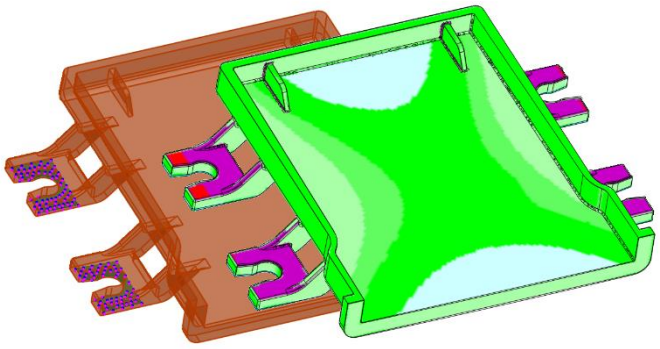
Cooling labels attached to each cooling holes easily and quickly with relevant suffixes

1.5 Additional Mold Design Capabilities

1.5.1 Mold Correction - Warpage Compensation

A new functionality enables getting the actual molded part model (either by molding the part and scanning it, or from a simulation system), and changing the design of the mold to compensate for warpage so that the final molded part fits the required dimensions and tolerances.

For additional information, see page 23.

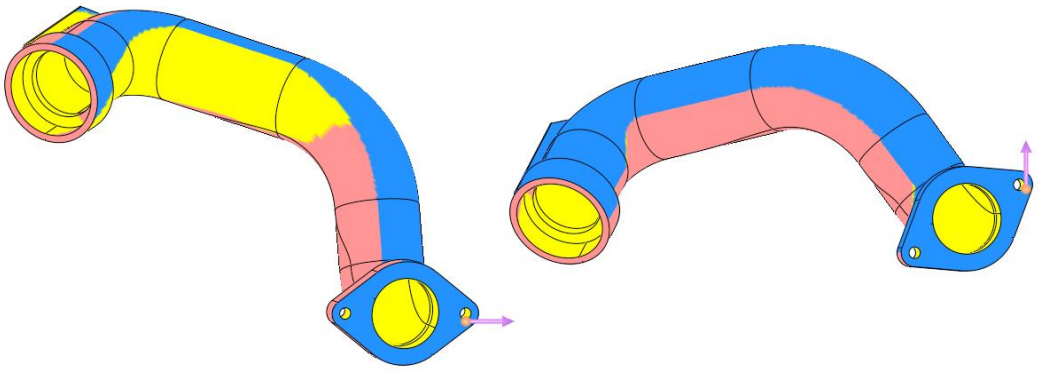


Deviation between molded part and part model recognized and compensated for

1.5.2 New Direction Analysis

The system can now provide a fast analysis to automatically find the best orientation for the mold's opening direction (a crucial initial step of mold design) with the least amount of undercuts. This is especially useful for parts with an ambiguous or hard to find parting line (for example – automotive pipes).

For additional information, see page 26.



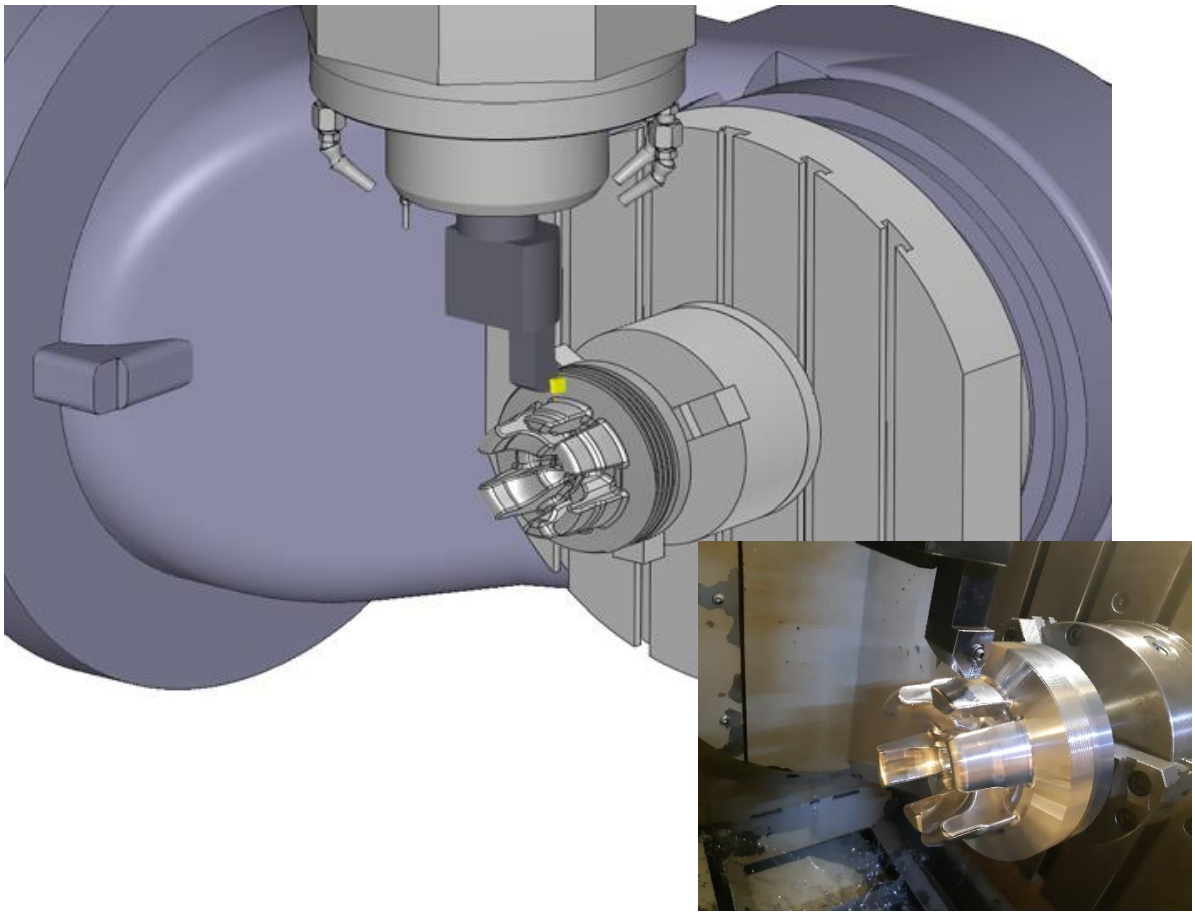
Blue and red areas can be extracted with no undercut as core and cavity. Yellow areas are undercuts. On the left the part is in its neutral orientation and opening direction. On the right it is after automatic optimization of orientation and opening direction.

1.6 New Mill/Turn Application

New Mill/Turn capabilities were added to Cimatron for faster machining and better surface quality for tool makers that want to use the familiar and safe environment of Cimatron to program their Mill/Turn and Turning machines. Turning with Cimatron supports full functionality for roughing, high-performance roughing (using VoluTurn), contouring, threading, center drilling, tapping and boring.

The application is fully integrated within the Cimatron CAM environment allowing easy programming, machine simulation and post processing of the entire NC process – Milling, Drilling and Turning.

For additional information, see page 84.



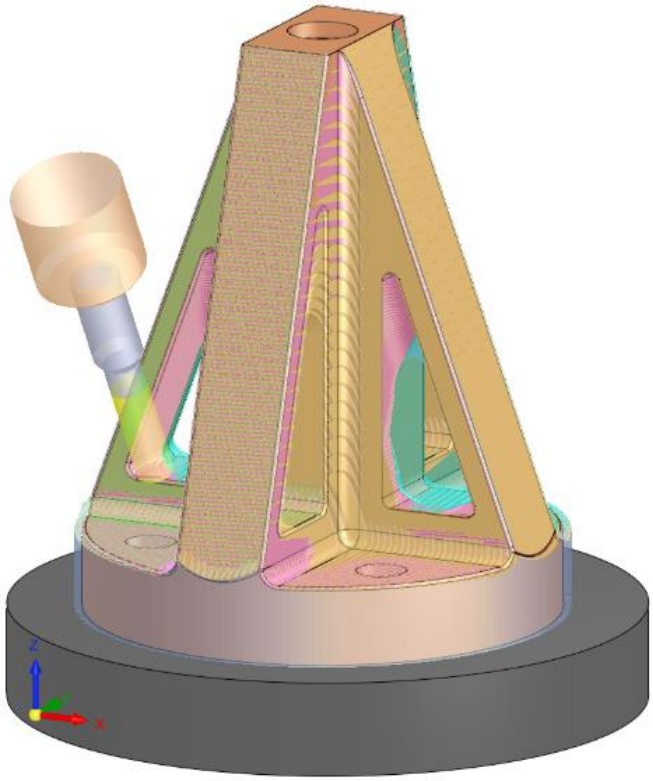
A new Mill/Turn application is fully integrated within the Cimatron CAM environment

1.7 5 Axis Machining

1.7.1 3+2 Axis Roughing Automation

Optimized Roughing for multi directions can now be programmed and calculated entirely and automatically by the system, for maximum material removal in minimal machining time, from optimal selected directions. Users can now let the system calculate complex part roughing for them.

For additional information, see page 91.

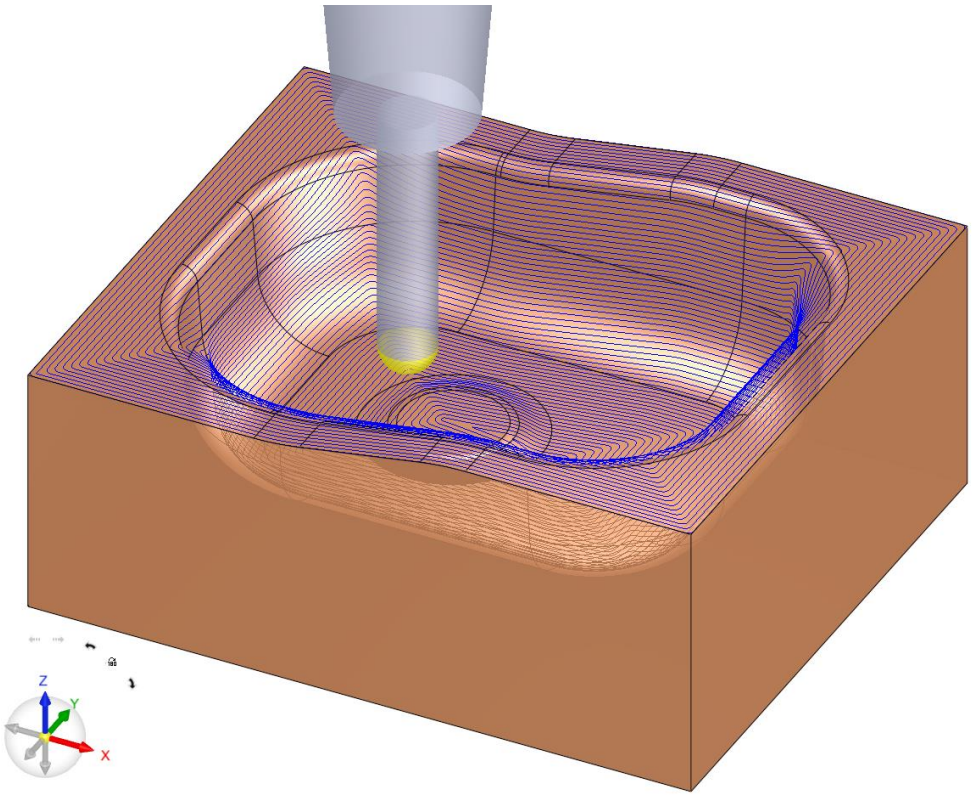


A Fixture rough cut, using automatic Rough 3 +2 axis set of operations. The stock is colored by the orientation. The directions are defined automatically by the system

1.7.2 New Geodesic Procedure

A new high quality toolpath calculates an even 3D step over complex parts. It includes support for guide curves, undercuts and considers the holder to prevent collisions.

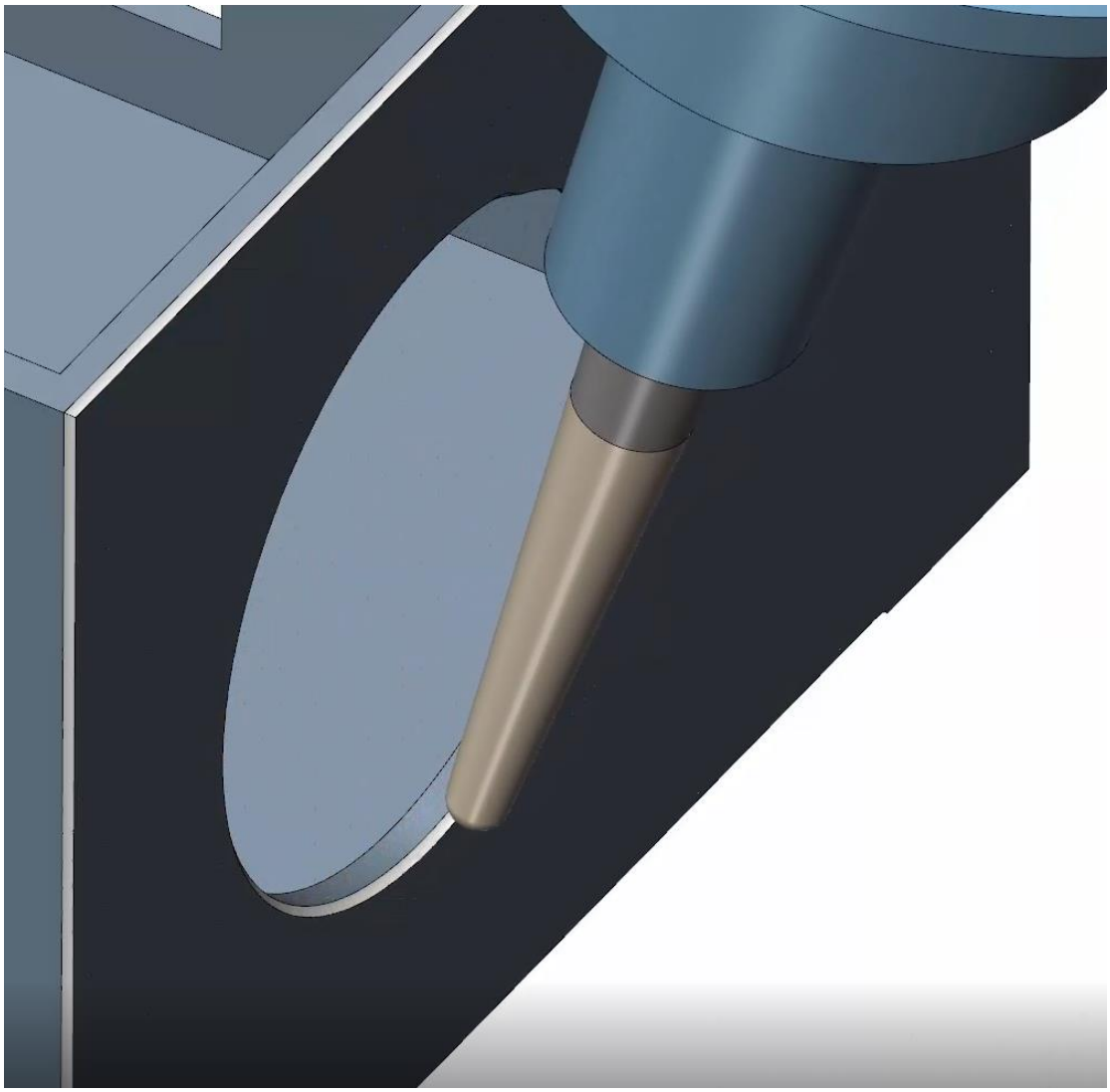
Users can use it for 3 axis, 4 axis and 5 axis toolpaths, typically used for parts that require a high quality surface finish, such as mold and die cores and/or cases where a continuous toolpath start-to-end without re-entries, is required.
For additional information, see page 92.



A start-to-end 3D Step Geodesic 3 axis toolpath on a steel cavity, with no re-entries, to achieve best surface quality without approach retract marks.

1.7.3 New Deburring Procedure

Automatically program deburring operations using ball tools, to remove sharp edges. The toolpath considers the machine axis limits and holder to create a collision-free toolpath. Users can use it for 3 axis, 4 axis and 5 axis toolpaths on jigs and fixtures, for example.
For additional information, see page 93.



1.8 Plate Machining and 2.5 Axis Milling

While plate machining is considered to be a rather simple 2.5 axis set of operations, it consists of many procedures with numerous parameters that may require time-consuming programming. New and enhanced capabilities were added to version 15, in order to automate and shorten the programming process and achieve safe and efficient toolpaths.

These include:

- Manufacturing feature recognition
- Pocket, Slot and Hole manager
- Pocket template
- Automated Drill including drilling sequences
- Rough pocket
- Corner Plunging – New. See page 95.
- Collision free profile
- Helical profile

.

1.8.1 New Slot Procedure

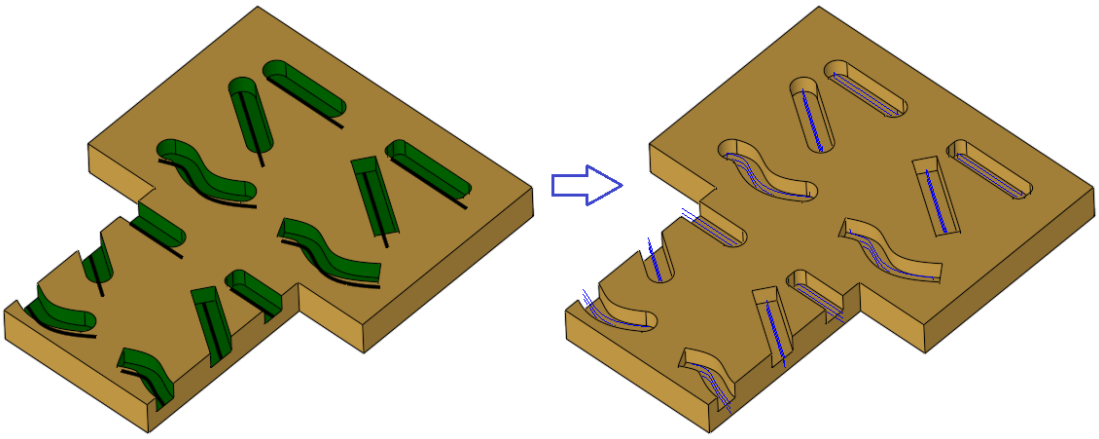
A new procedure that slots the centerline (skeleton) of a slot-shaped pocket.

Milling a slot may take several stages: Slotting, Roughing, Finishing the walls, Finishing the floor. The new procedure deals with the Slotting stage.

The slotting procedure will:

- Recognize open and closed slot ends.
- Support Simple Slotting and Trochoidal milling.
- Support plunging (Helical or Ramping).

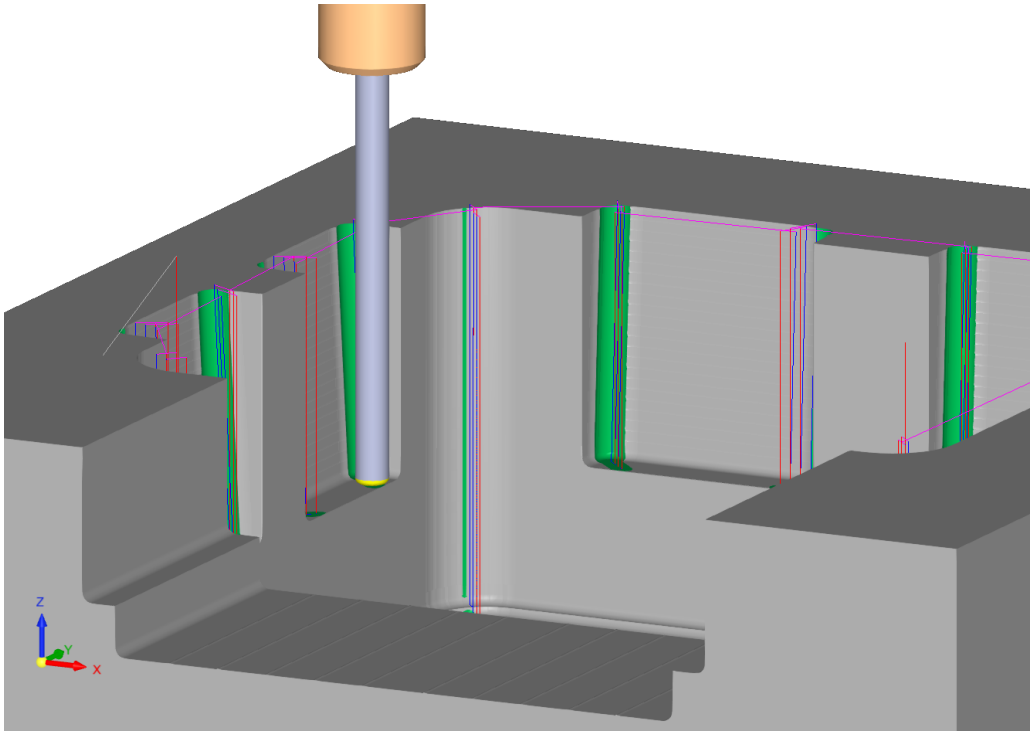
For additional information, see page 94.



Slotting milling on slots identified by the Pocket Manager

1.8.2 New Corner Plunging Procedure

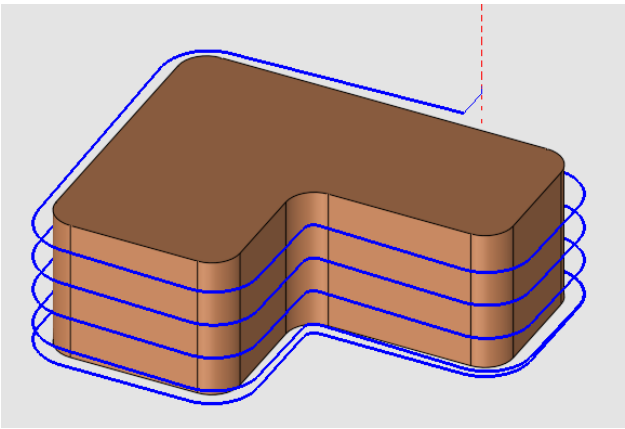
This new procedure creates plunging motions with a plunge cutter at rounded corners of pockets. It can replace the Cleanup procedure for pocket corners and save machining time. For additional information, see page 95.



An efficient re-rouging at corners is achieved with the new "Corner Plunging" procedure, where the stock is removed by down cutting operations, including an option for forward steps

1.8.3 Helical Milling of Closed Profile

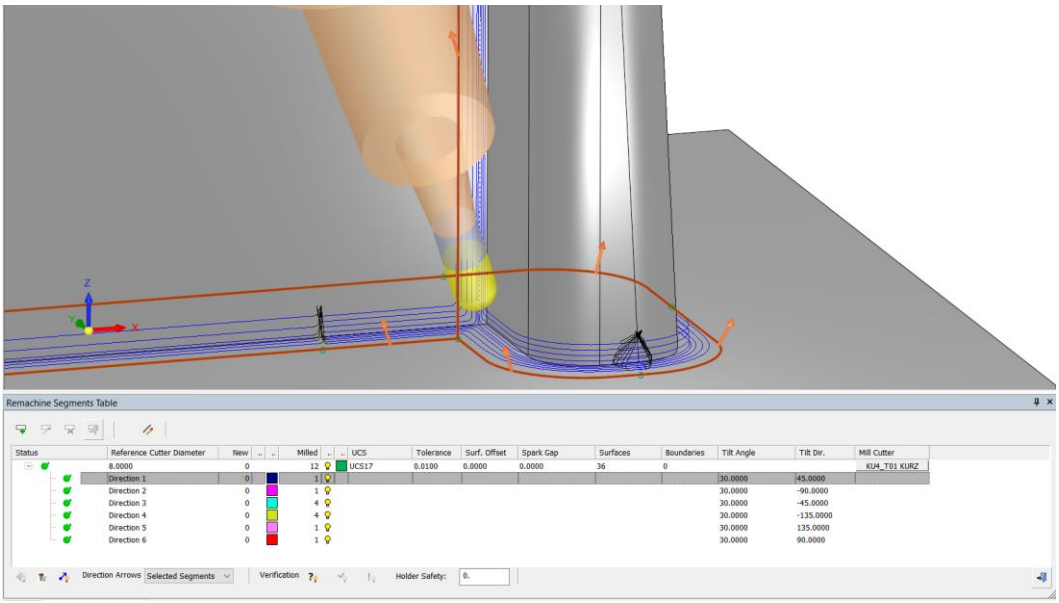
The new Helical Closed Profile allows milling the entire part in a single pass, without approaching and retracting each layer, creating better surface quality and faster machining. For additional information, see page 96.



1.9 Programing Automation

1.9.1 3+2 Axis Remachine Automation

With a click of a button, the system calculates optimized multi-directions cleanup motions, taking into account the tool and holder shape to protect from potential collisions.
For additional information, see page 97.



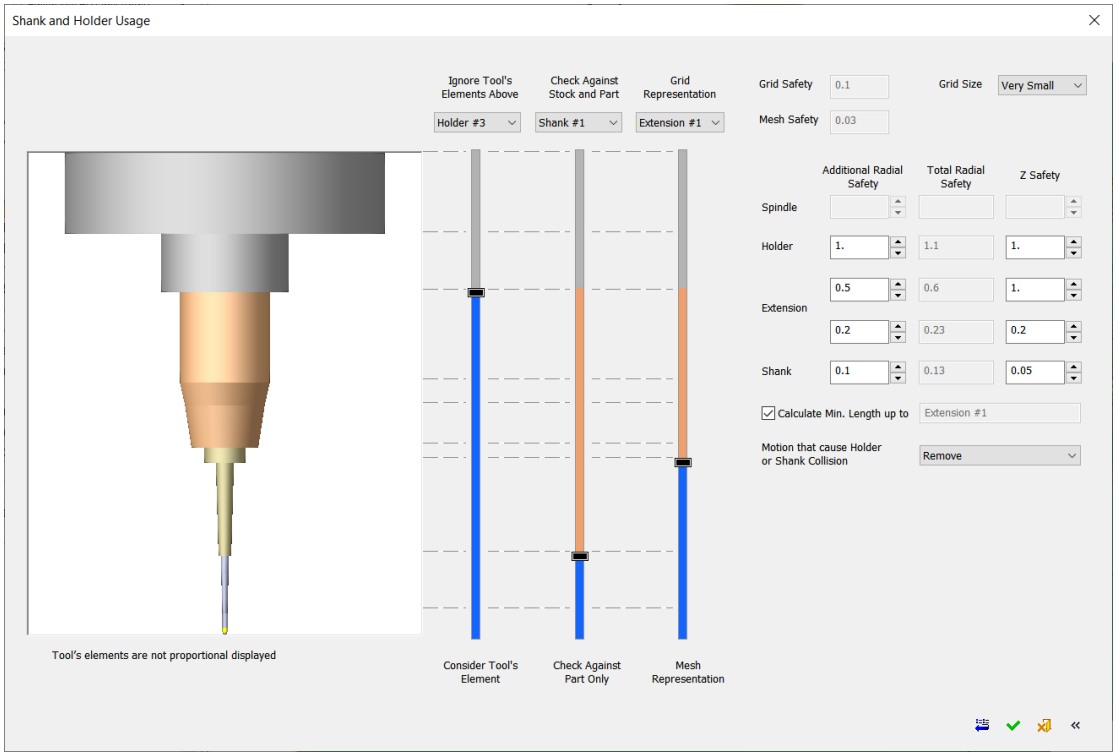
Zoomed-in picture of a cleanup corner, machined at an inclined 3 + 2 axis orientation. The orientation was calculated automatically by the system, as part of the guided cleanup auto tilt function.

1.10 Easier Programing

1.10.1 Shank and Holder Dialog

The insight and control over the shank and holder's safety and calculation parameters were greatly improved with a new dynamic dialog. Users can now control which elements will be considered/ignored and at which level of accuracy. The ability to better control and understand the influence and behavior of the different parameters, will enable the user to generate the desired toolpath.

For additional information, see page 99.



A new shank and holder dialog enables better user control and full transparent information over the different cutter assembly components. These include shank, extension, and holder stages, for considering or ignoring, for safety and for minimal clear length with complete interactive feedback

1.11 Faster Machining

1.11.1 New Circle Segment Cutters

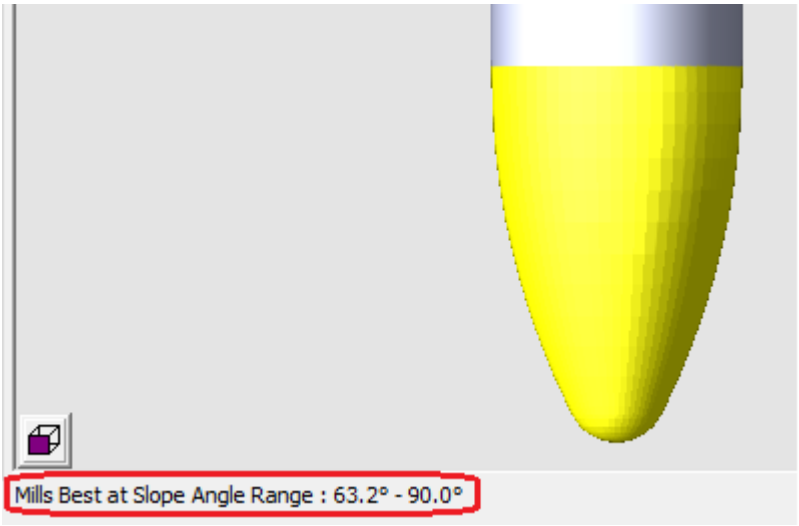
Barrel, lens, oval and now 3 radii segments cutting tools can be utilized by Cimatron toolpath strategies. Using any of these cutters will shorten the machining time for a given scallop while achieving even better surface quality.

For additional information, see page 102.

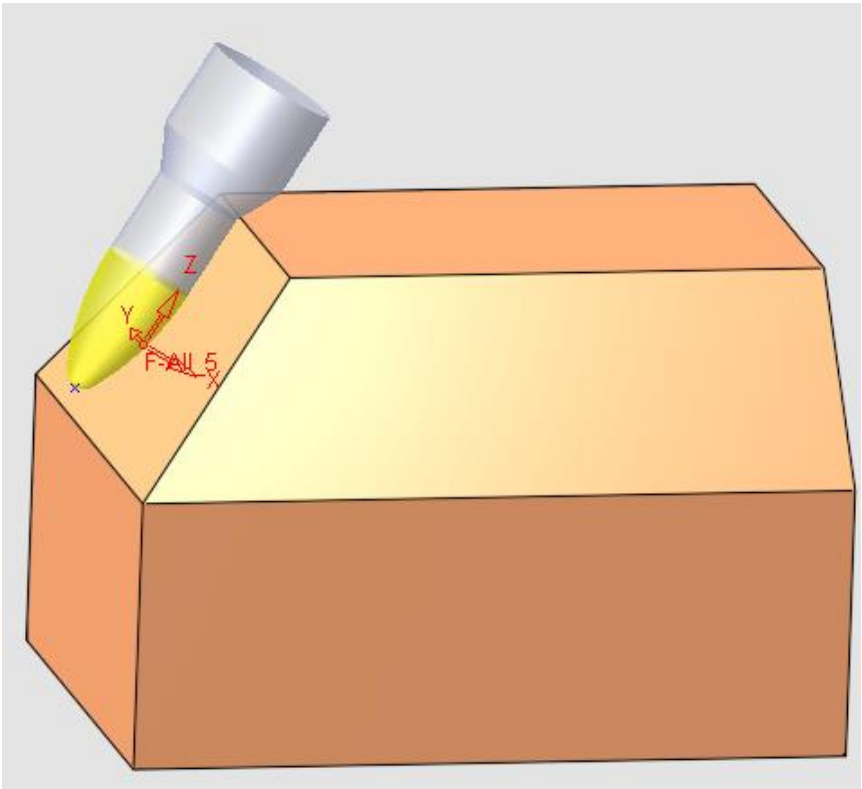
1.11.2 Creating UCS considering Circle Segment Cutters

Circle Segment cutters mill best when the largest radii of the cutter is tangent to the surface it mills.

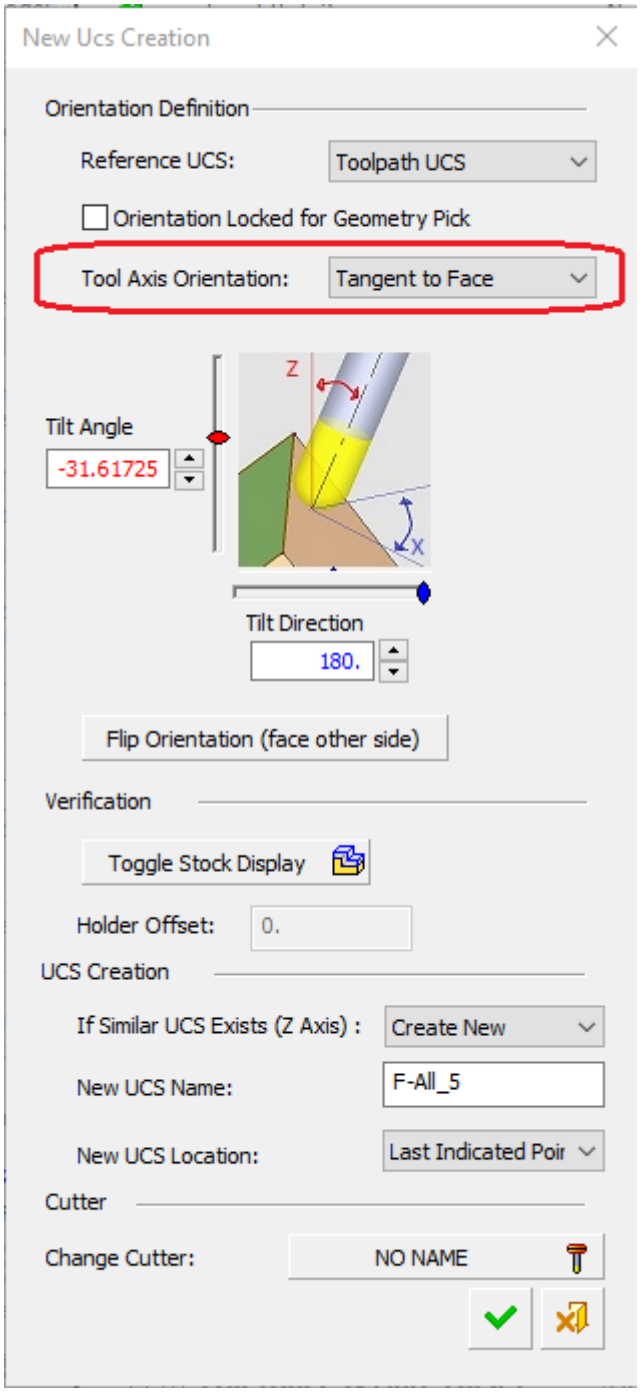
To help users know which faces can best be milled by a specific cutter, the angle between the cutter axis and the normal of the face it mills is shown in the cutter table.



When a user wants to mill a face with a circle segment cutter, a UCS can be created that makes the cutter tangent to the face at a proper angle.



A new option, “Tangent to Face”, has been added to “Create UCS” (within an NC procedure) and to UCS “By NC Tool”. This option creates an optimized UCS for the selected face and cutter.

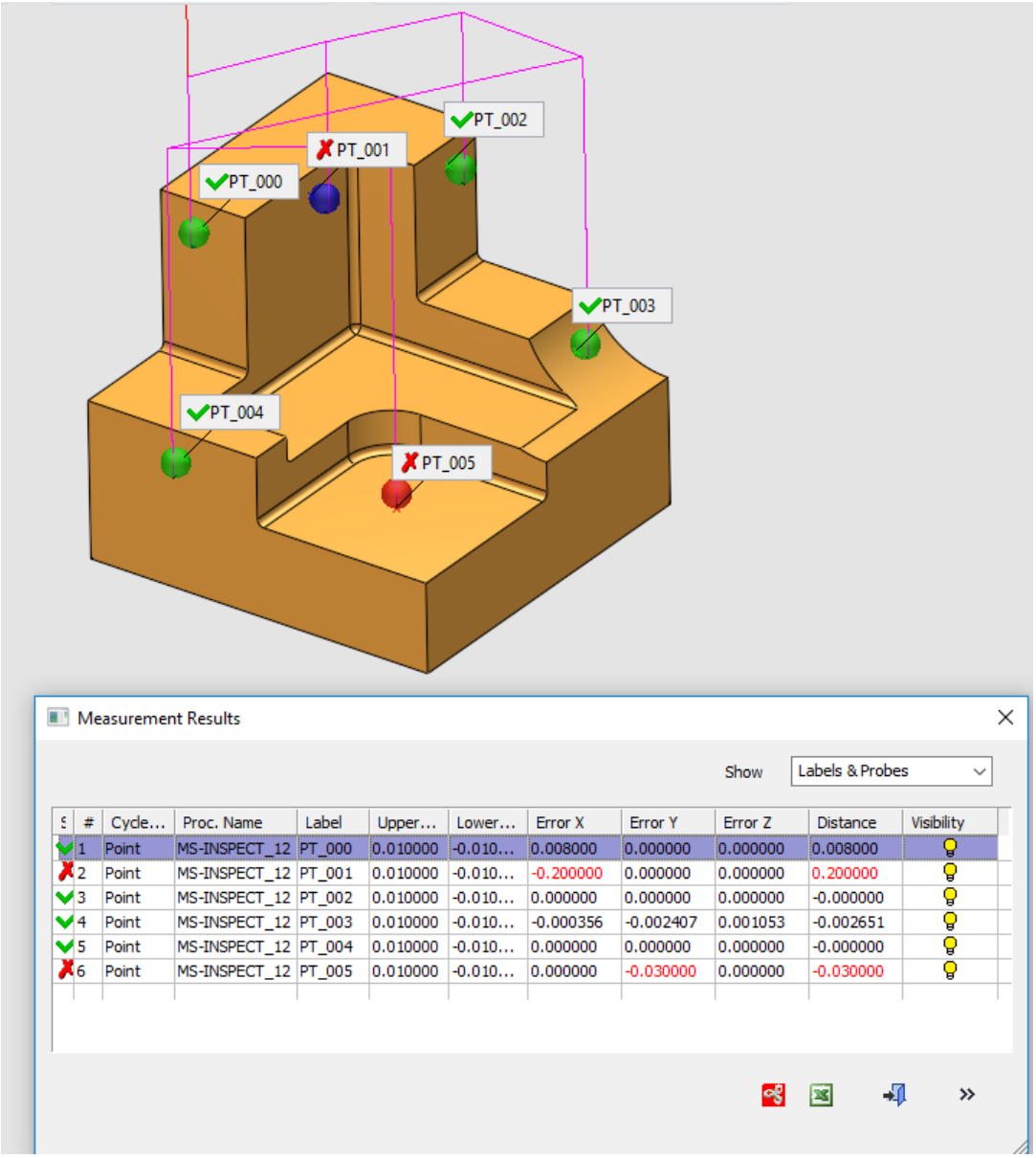


Note: In the case of regular milling cutters, such as Flat, Ball or Bullnose, the UCS is positioned so that the cutter is parallel (swarfing) to the selected face.

1.12 On Machine Inspection – Read back Report

A new option in Cimatron creates a Report for On Machine Inspection (OMI) procedures, by reading back the measurement results from the machine controller.

For additional information, see page 105.



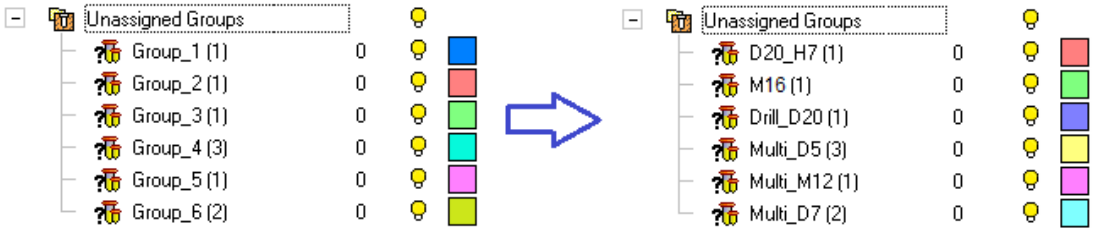
S	#	Cycle...	Proc. Name	Label	Upper...	Lower...	Error X	Error Y	Error Z	Distance	Visibility
✓	1	Point	MS-INSPECT_12	PT_000	0.010000	-0.010...	0.008000	0.000000	0.000000	0.008000	
✗	2	Point	MS-INSPECT_12	PT_001	0.010000	-0.010...	-0.200000	0.000000	0.000000	0.200000	
✓	3	Point	MS-INSPECT_12	PT_002	0.010000	-0.010...	0.000000	0.000000	0.000000	-0.000000	
✓	4	Point	MS-INSPECT_12	PT_003	0.010000	-0.010...	-0.000356	-0.002407	0.001053	-0.002651	
✓	5	Point	MS-INSPECT_12	PT_004	0.010000	-0.010...	0.000000	0.000000	0.000000	-0.000000	
✗	6	Point	MS-INSPECT_12	PT_005	0.010000	-0.010...	0.000000	-0.030000	0.000000	-0.030000	

1.13 Automated Drill

1.13.1 Automated Drill - Automated Names of Groups

Holes groups get significant names that helps users to understand the meaning of each group.

For additional information, see page 114.



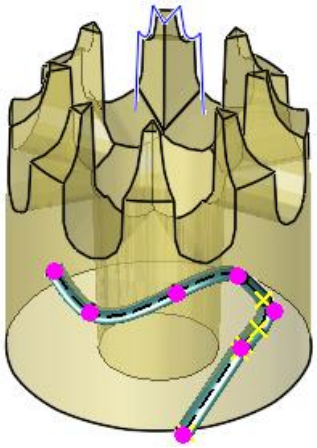
2 Mold Design

2.1 Conformal Cooling Improvements

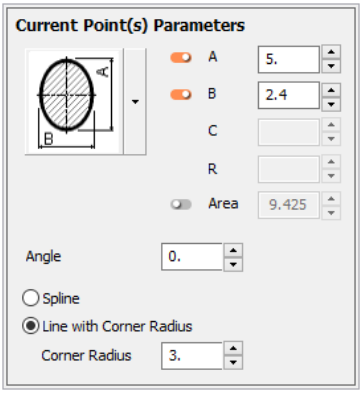
The conformal cooling application has been enhanced dramatically to allow for much faster and easier design of the cooling channel along with automation allowing anyone to generate a conformal cooling path. These improvements consists of enhancements to the conformal cooling tool itself, allowing for much smoother control of the channel path and the different sections along it, and a new automatic tool to generate the conformal cooling path based on the geometry of the part.

2.1.1 Conformal Cooling design Improvements

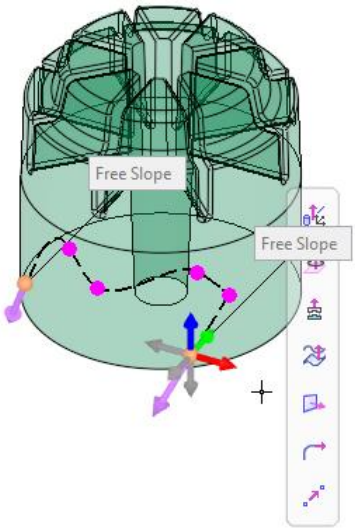
A number of improvements have been made to the interaction of the Conformal Cooling Design function allowing for easier control and faster results.



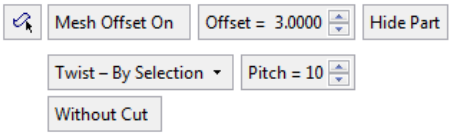
- In order to ensure faster previewing of the channel, by default, the preview of the created channel is not shown. The curve and sections are visible at all times. To see the actual channel the Preview button should be clicked. Automatic preview may also be used.
- The point section dialog is now available for edit at all times on the final stage of the feature guide without the need to open the dialog separately. Some of the controls relevant for the next point were also moved to the new dialog.
    - In addition the dialog now controls the currently selected or created point(s) eliminating the need to first edit parameters and then create the new point.
    - Settings on the dialog are kept when changing to a different shape. This means section area can be maintained when switching to a different section shape
    - The dialog may control several sections at once for multiple editing to enhance easy of use, speed, and uniformity throughout the channel. Right clicking selected points allows editing their parameters, removing them, reselecting position (for a single point) and changing their position by dragging.
        - Note that points may also be removed by selecting them and clicking the DEL key.
    - Points may be added to any point on the curve simply by clicking it.



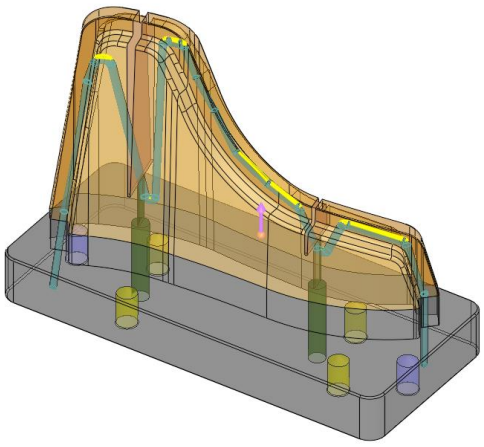
- Contours may now be selected as input for the channel spine. This means you can design your channel with any wireframe tool and then use it within the conformal cooling design tool. This includes of course using the contours resulting from the new automatic conformal cooling curve.



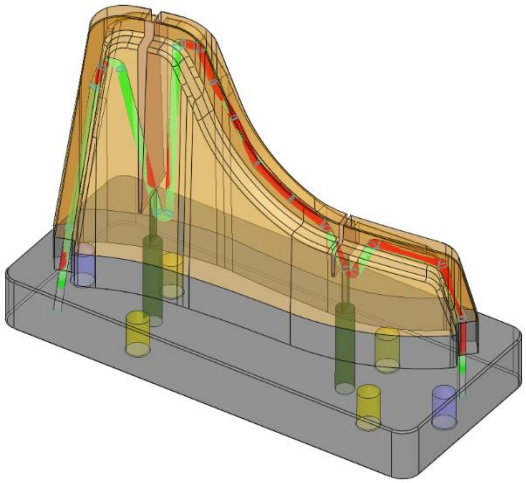
- Sections are automatically oriented in the right direction for printing so that they form a minimal overhang area (if any).
- A new optional stage allows controlling the slopes at the ends of the cooling channel curve.
- You may switch the progression direction of the conformal cooling curve, so that if you need to add a point at the start of the curve – you simply switch the direction of the start and end points and define the new point.
- If previous conformal cooling curves are present, there is an easy access to hide and show them through a dedicated button.
- A new option controls twisting of the conformal cooling channel. It has the options Twist – By Selection / Twist – Entire Curve / No Twist. Twist by Selection allows you to right click any point and define it as start/end point of twist. The Twist will start from that point until it reaches another point that is marked as ON, and then will stop. This takes into account the direction of the curve, and if the direction is changed – so does the behavior. While in preview mode the twisted section will be colored Cyan. Twisting the entire curve marks the entire curve as twisted. While in either of these two modes, the Pitch of the twist can be controlled.



- When defining a mesh offset, an option has been added to hide and show the part geometry for easier selection on offset faces.
- You may define a printing direction other than Z to adjust all printing related calculation to the orientation in which the part will be printed. This includes, for example, the orientation of sections and the new overhang analysis.
- Two new analysis tools are available within the tool as optional stages ensuring the quality of the design in terms of successful 3D printing, cooling efficiency and safety of the cooling system throughout its lifetime (ensuring there are no breaches and leakage).
    - Overhang analysis – to ensure no internal areas within the cooling channels will fail during printing. Based on the overhang angle and minimum allowed width with no support, associated with your printer, material, etc. – the system will create a preview of the channel and look for areas where an overhang occurs and supports are required to ensure proper printing (marked in yellow curves). Naturally, supports cannot be created within the cooling channels and so if such areas are found – they should be handled by rotating the section or making it narrower at the top.



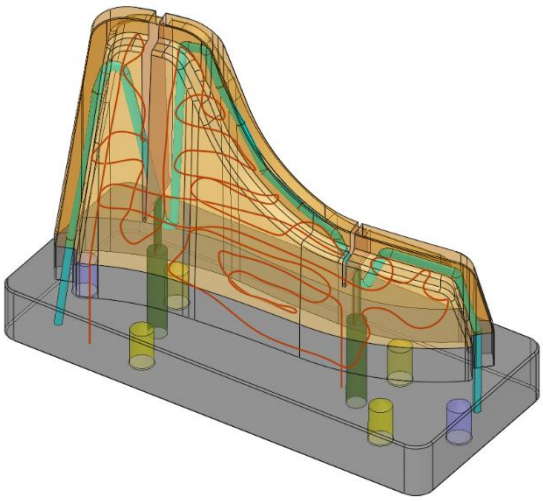
- Distance analysis and optimization – identify areas where the channels may be too close to, or too far from, the walls of the insert and adjust them automatically. The system creates a preview of the channel and finds for each point the closest face on the part. Based on an optimal distance, it will create a color map on the cooling channel indicating areas that are too close or too far to\from the part. Another option within the same stage allows optimizing the distance by automatically moving the spine points to adjust points that are too close or too far away. Note that not all issues can be fixed automatically and you may need to adjust the spine points manually.



2.1.2 Automatic Conformal Cooling Curve

A new tool now allows for the automatic generation of conformal cooling curves by analyzing the part's geometry. This task is made much simpler by this tool and makes it approachable for novice users.

Active faces that require cooling are selected and the diameter of the cooling channels is defined along with required distances from walls and from other cooling lines. The system will then calculate the optimal path of the conformal cooling channel and create a curve to be used as an input for the conformal cooling design tool.



An optional stage allows selecting additional cooling channels (even if they were not yet cut with the part) to make sure the cooling channel stays further away from them.

It is recommended to use a diameter representing the bounding circle of the sections that will be used for the conformal cooling channel itself.

2.1.3 Conformal Cooling Standalone seat for non-Cimatron users

A standalone seat for conformal cooling design is offered to mold makers using other software for mold design or conformal for cooling printing bureaus, with all the tools necessary to complete the design.

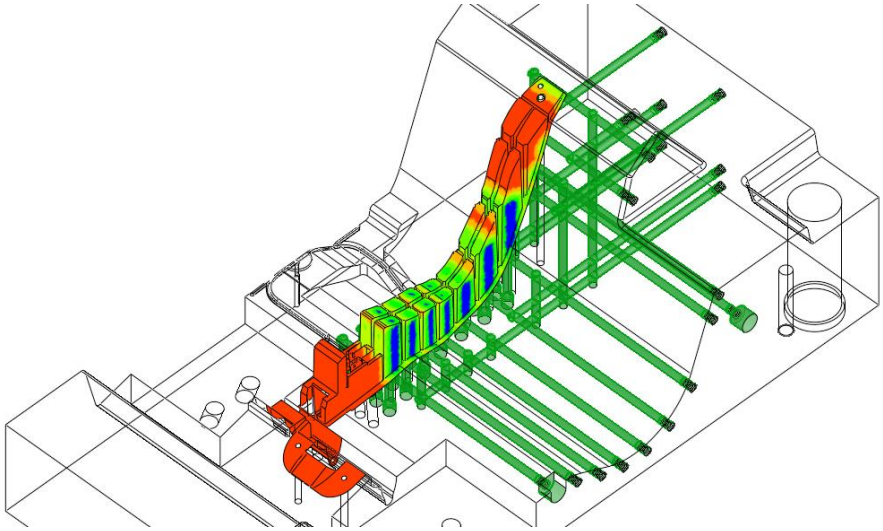
2.1.4 Cooling Distance Map

A new analysis tools offers extremely fast analysis for a quick glance at cooling efficiency based on the distance between the cooled faces and cooling channels.

Select cooling lines and then the active faces (faces that touch the plastic), then run the analysis to get a color map of distances between the active faces and the closest cooling line to assess cooling efficiency.

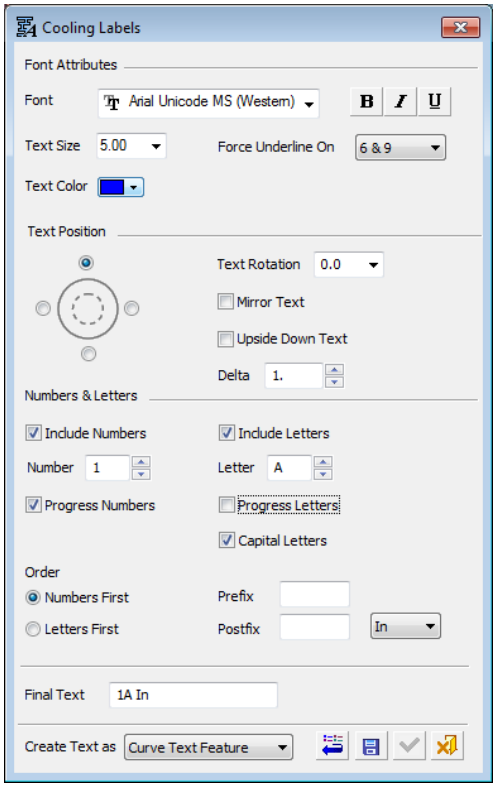
In the first stage of the tool, the system will automatically select all faces marked as cooling lines (faces are recognized as cooling lines if they were created with a cooling tool, or the Analyze Cooling Circuit tools were used to recognized them). You may add or subtract faces from this selection. In the second stage, active faces (resulting from QuickSplit) are selected automatically by clicking the Select Active faces button. You may also select faces by color.

In the final stage you may tweak the color map by changing color ranges, and may also export the results to a 3D PDF report.



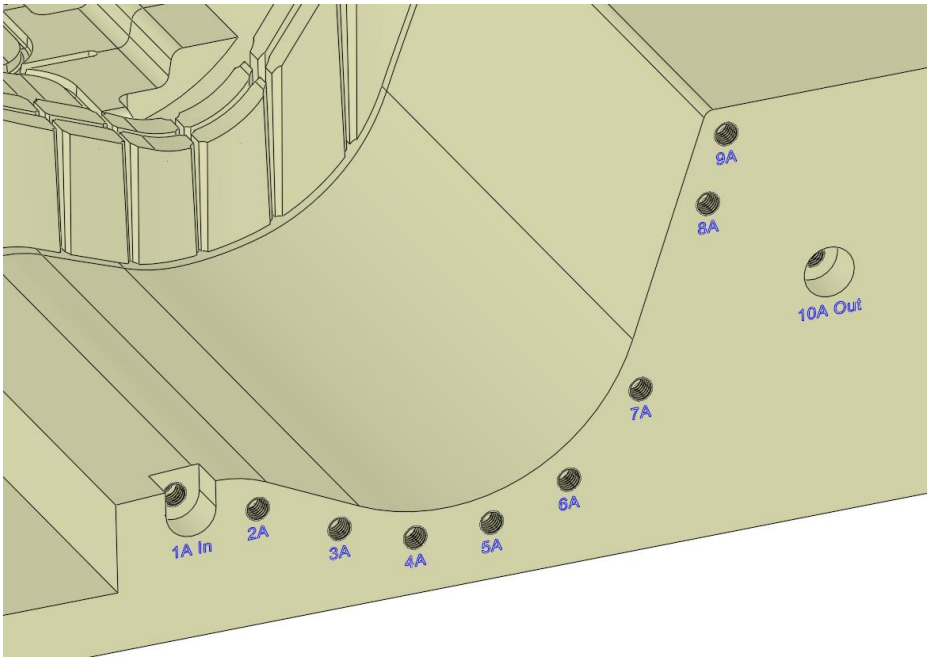
A color map showing the distances between the cooling channel and the faces of the mold that form the shape of the plastic part

2.1.5 Cooling labels



A new tool enables easily adding labels next to each channel, to save time and avoid mistakes.
A dialog, similar to that available in the Ejector Table or Baffle Table tool, allows to set the appearance of the text as well as its position, including an option to set the delta between the selected hole and the text.
Note that the default positioning of text has been dramatically improved so that the mirroring and rotation options are not required as often as before.
The generation of the text itself is done by a combination of letters and numbers that can each be included or excluded and progress automatically from one item to the next.
Prefix and suffices may be added. Standard suffixes are In and Out and the text for those can be controlled through the preference.
Another option in the In/Out dropdown is Alternate In/Out which changes the postfix from In to Out and vice versa. Using it will assign the In prefix to the first selected hole and then Out to the next one, and then In again. This allows selecting in and out holes of a single cooling line consecutively with great ease.
The text creation is done by selecting one cooling hole curve after another, where the text progresses as defined in the dialog.
As in the previously mentioned tools, the text can be created as a text feature or as PMI.

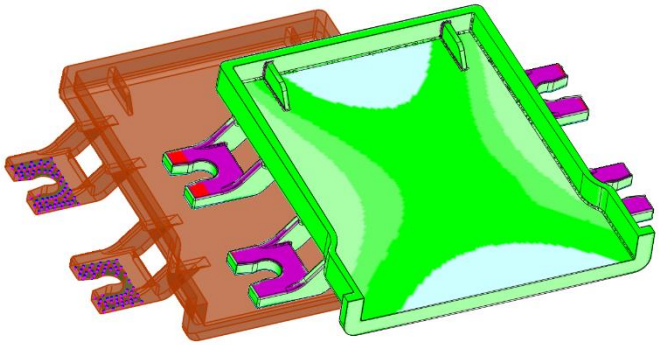
Note: The Cooling Labels tool can only be used when an assembly is active, since it can be used on several parts and creates an assembly feature.



Cooling labels attached to each cooling hole easily and quickly with relevant suffixes

2.2 Mold Correction - Warpage Compensation

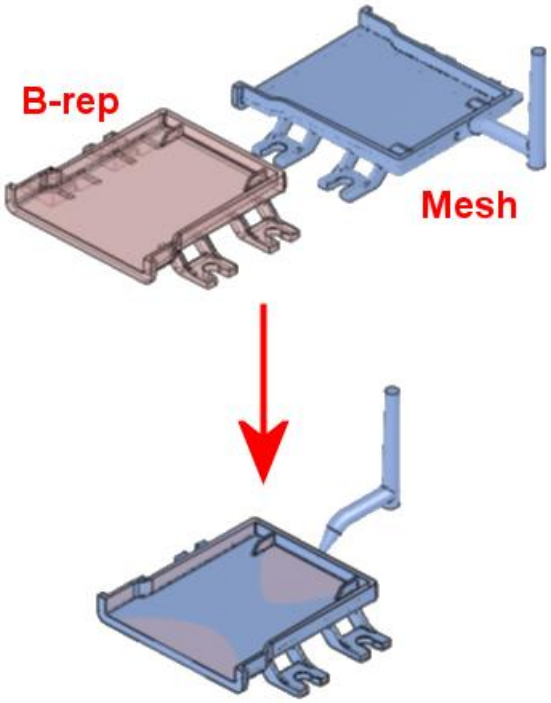
A new functionality enables getting the actual molded part model (either by molding the part and scanning it, or from a simulation system), and changing the design of the mold to compensate for warpage so that the final molded part fits the required dimensions and tolerances.



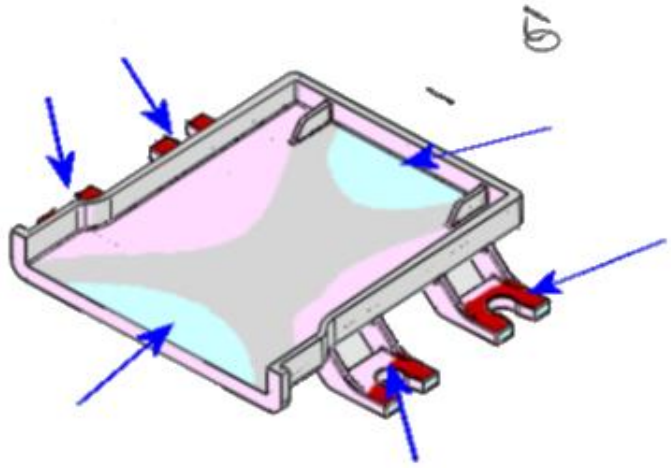
Deviation between molded part and part model recognized and compensated for

For example, to compensate a B-rep object for warpage against a mesh object, which may be a result from either scanned data or an FEA analysis:

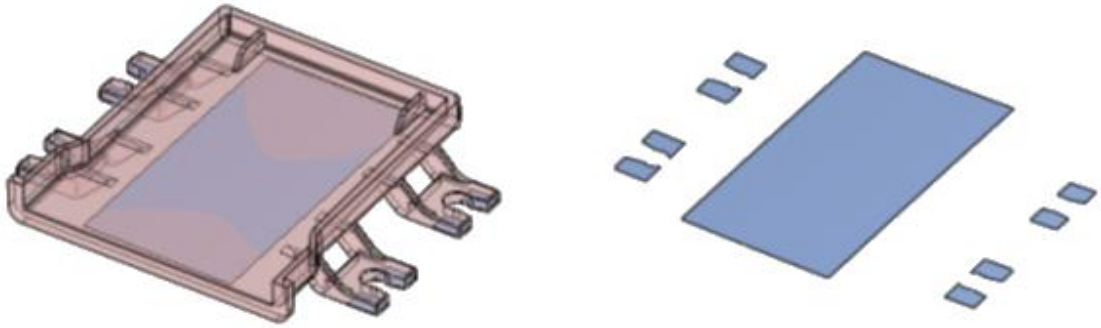
The first stage is to align the mesh object to the B-rep.



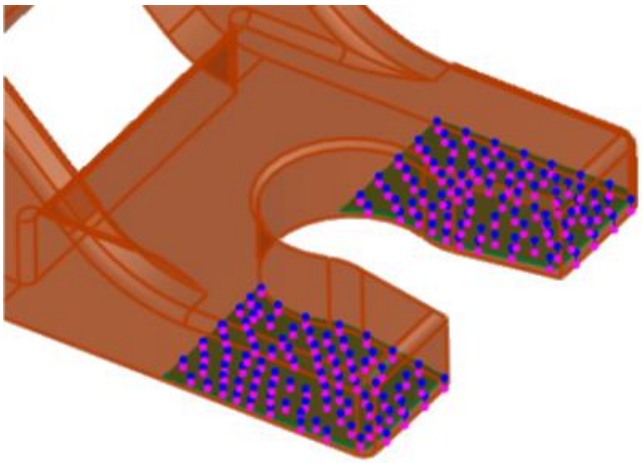
The deviation map tool shows the “areas of interest”.



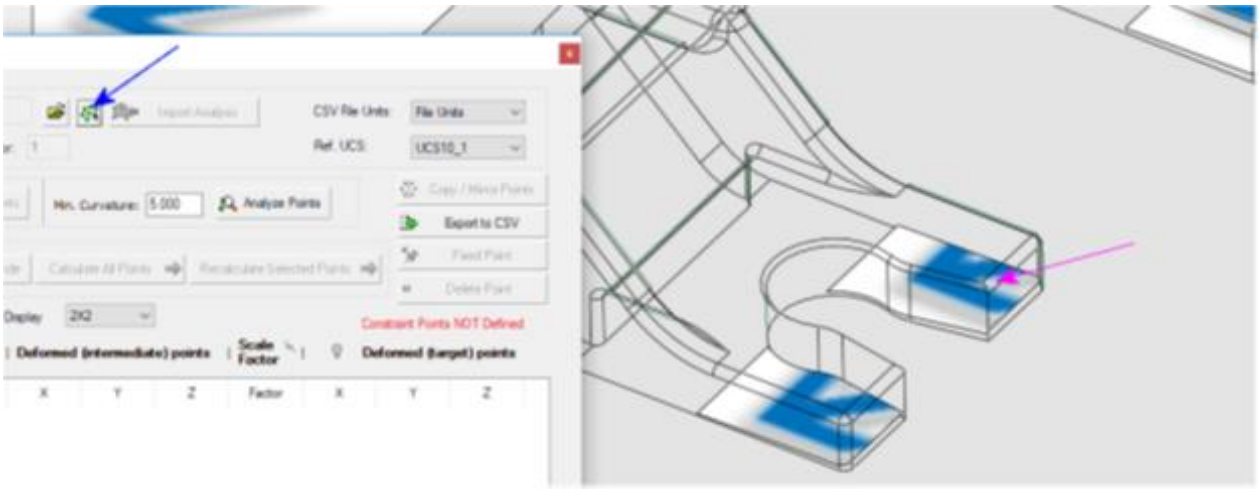
The user then creates “local meshes” (skin mesh) on the areas of interest using the Split/Copy Mesh tool:



These local meshes can be used to find pairs of points, using the Match Points on Skin tool:

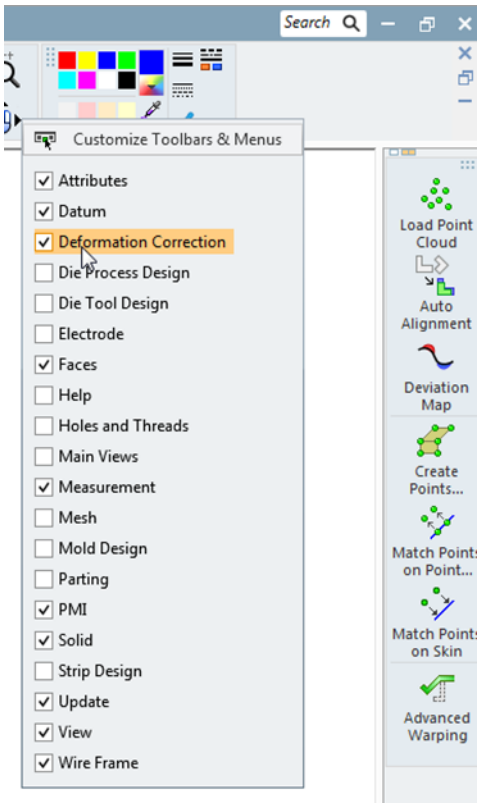


The Advance Warping (Springback Deform) tool can then be used to compensate (reverse direction) the original B-rep by picking the local mesh object with paired data:



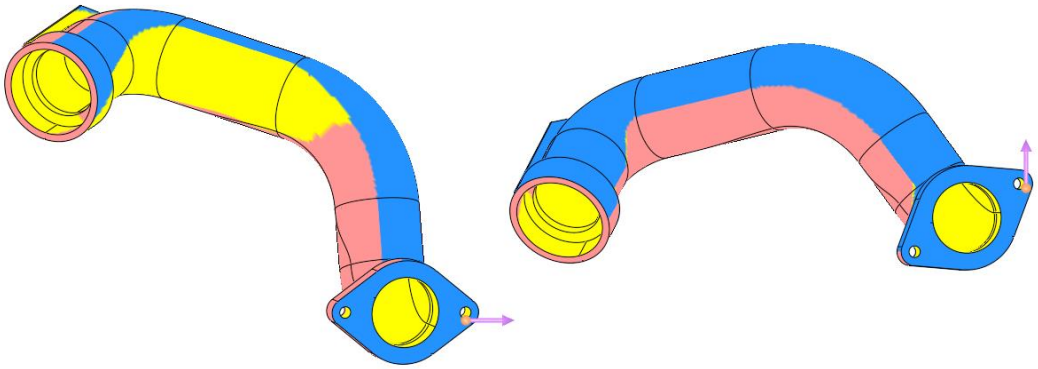
2.3 New Deformation Correction toolbar

The tools required for the above mentioned functionality of Mold Correction were gathered in a new toolbar called Deformation Correction. The toolbar is not turned on by default and in order to turn it on – right click the toolbar area and select it from the context menu.



2.4 Find best orientation for opening direction

The system can now provide a fast analysis to automatically find the best orientation for the mold's opening direction (a crucial initial step of mold design) with the least amount of undercuts. This is especially useful for parts with an ambiguous or hard to find parting line (for example – automotive pipes).



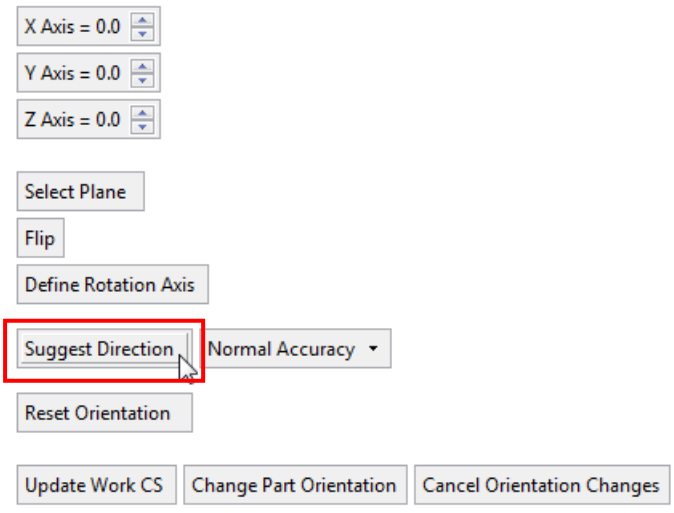
Blue and red areas can be extracted with no undercut as core and cavity. Yellow areas are undercuts. On the left the part is in its neutral orientation and opening direction. On the right it is after automatic optimization of orientation and opening direction.

The new orientation analysis is available in the Direction Analysis tool. Clicking it will run an analysis looking for possible orientations that will yield the least amount of undercut area.

When used in the 2nd stage it will result in a direction arrow pointing at the optimal main opening direction.

When used in the Orientate Part sub-stage, it will actually rotate the part so that the optimal opening direction is on Z direction.

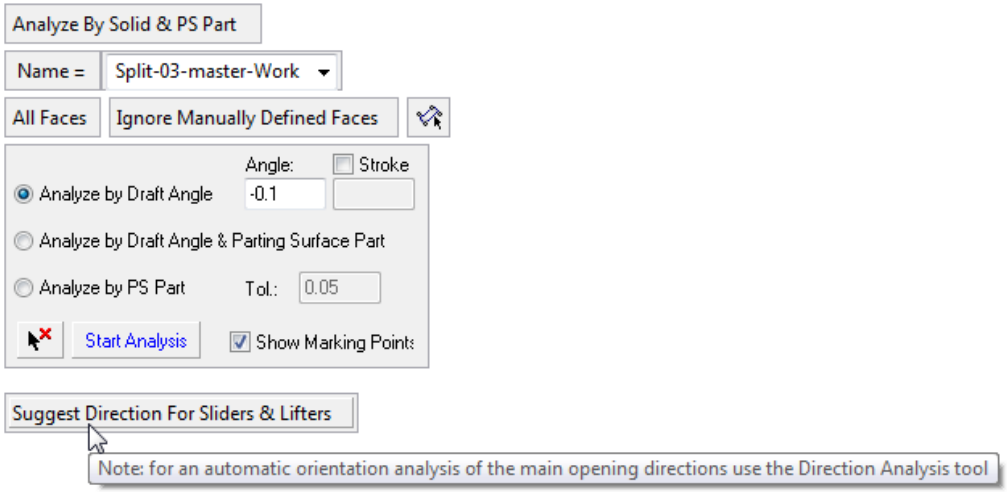
Note that you may set the accuracy level of the analysis, which of course, affects the duration of the analysis.



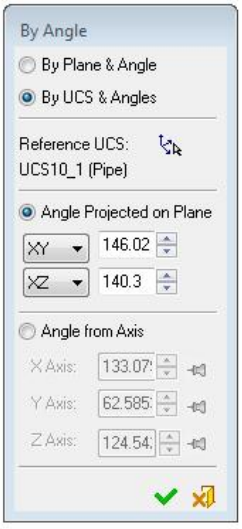
Also note that you may select an entire object or specific faces. This is relevant when you do not want to take internal faces into account (of pipes for example).

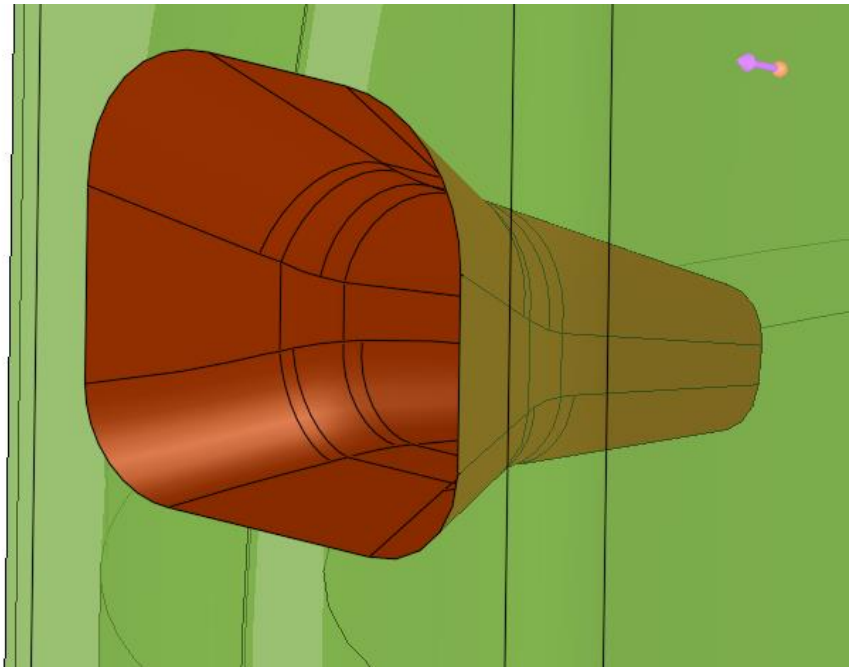
2.5 Quick Split - Find best extraction orientation for Sliders and Lifters

A new option in the Quick Split tool allows to find the best opening direction of selected Slider and Lifter faces. This takes into account all selected faces and finds the direction with least amount of undercuts. This is especially useful when all the faces of a slider or lifter have draft angles and the opening direction is very hard to define for them.

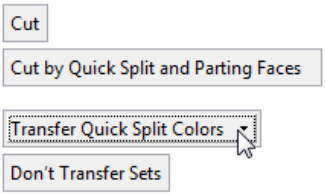


Clicking the new Suggest Direction For Sliders & Lifters button will find the optimal direction for the selected faces and open the By Angle direction dialog to allow rounding of angles and small modifications.





2.6 Transfer colors and sets in cut active



The system now allows for more control over the transfer of colors between the work part and active parts, allowing to retain color coding defined in the work part.

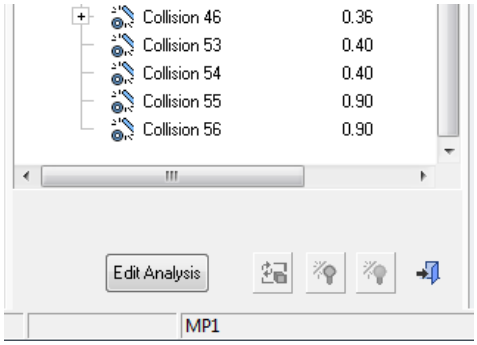
In the last stage of the Cut Active tool, new options now allow to control the transfer of colors and sets to the cut part.

The user may transfer the original face color, the Quick Split Color, or no color at all, and transfer or not transfer sets in which the faces take part.

2.7 Hole Analysis Tools Improvements

2.7.1 Working without the Pane commands

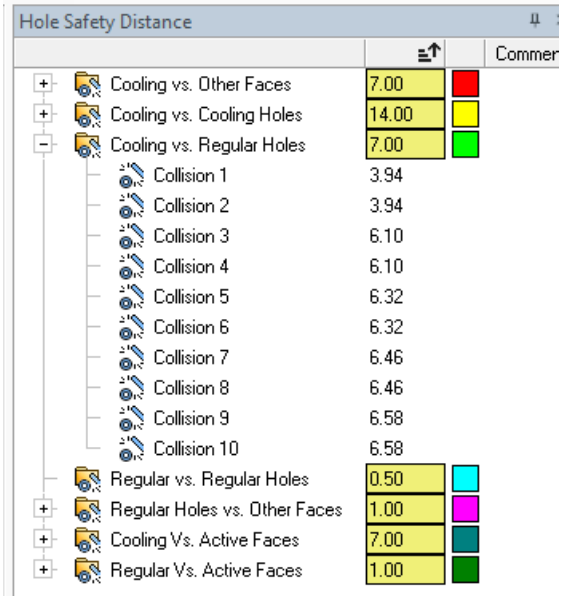
In both the Hole Safety Distance and in the Hole Verification tools, the commands that opened the analysis pane after analysis where removed. Instead, the tool commands themselves open the analysis pane once an analysis exists. In order to rerun the analysis, an Edit Analysis button was added at the bottom of the analysis pane, along with an Exit button to leave it.



2.7.2 Hole Safety Distance – Sort by Distance

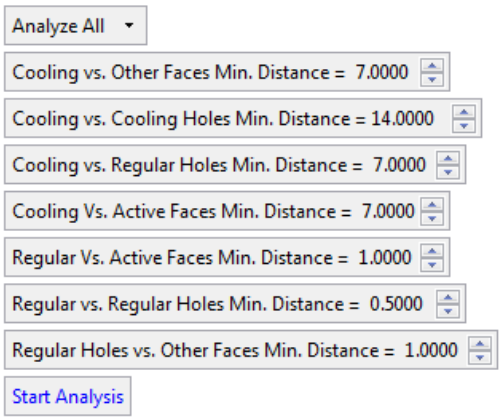
To allow for better understanding of the results of this analysis, and for a quicker recognition of problematic areas, when the result tree is created, the items within each folder (and sub items within them) are automatically sorted by distance.

The user may change the order from ascending to descending by clicking the top of the distance column.



2.7.3 Hole Safety Distance – Separate analysis for active faces

The system now allows to separate the analysis of distances from non-hole faces to Active faces (touching the plastic) and "regular" (other) faces, and to define different thresholds for them. This allows for more control over the analysis results and a finer analysis, and useful when the user wants to ensure a higher distance of holes from active faces than from other faces.

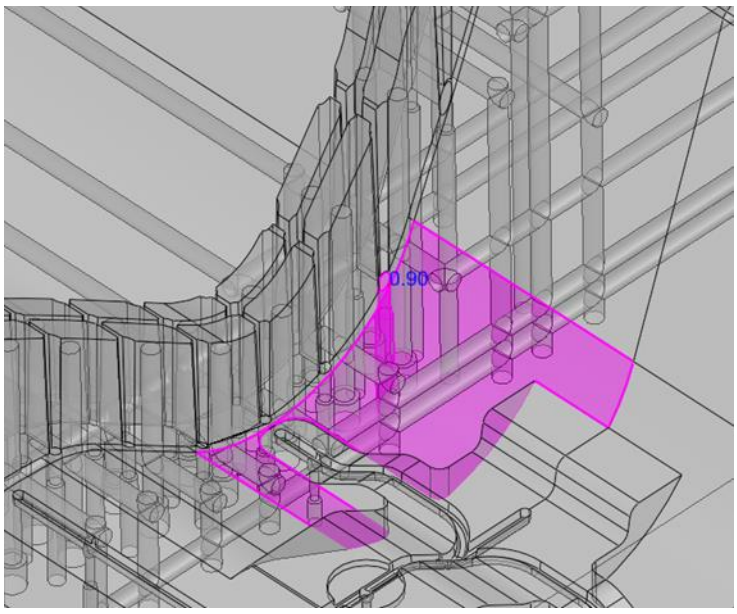


Wherever previously there was an analysis of distance from "Other" faces (anything that is not a hole), version 15 now has split to analysis of distance from Active faces, and distance from Other faces (anything that is not a hole or active face).

This means two new analysis result folders and options were added; **Regular Holes vs. Active Faces** and **Cooling Holes vs. Active Faces**.

2.7.4 Hole Safety Distance – Improved rendering

The rendering of selected items is now clearer than before.



2.8 Collect text features in a folder

In different tools such as Ejector table, Baffle table, and the new Cooling Labels tool, there may be cases of multiple text features within the same part (cavity or ejector plates for example). This can create a very long feature tree that is hard to manage. To simplify the tree, those features are now collected into a feature folder.

2.9 Cooling Items with Channel Improvements

2.9.1 Round length downwards

When creating the hole for the cooling items, the hole length values are now rounded downwards to ensure there is no penetration or getting too close to walls as a result of rounding to the nearest value which may be upwards. This is relevant for the options “No Interference” and “Max Length".

2.9.2 Go to higher level assembly in Add Cooling Item with Channel

When creating the hole for the cooling items in a deeply nested sub-assembly, finding reference faces against which to define the length of the hole, may prove difficult if those faces reside in a higher nested sub-assembly. To solve this issue, the system now looks for faces in higher level sub-assemblies when it can't find any in the current sub-assembly. This is relevant for the options “No Interference” and “Max Length".

2.10 Implement Add Diameter as Part Size for cooling parts (especially Baffles)

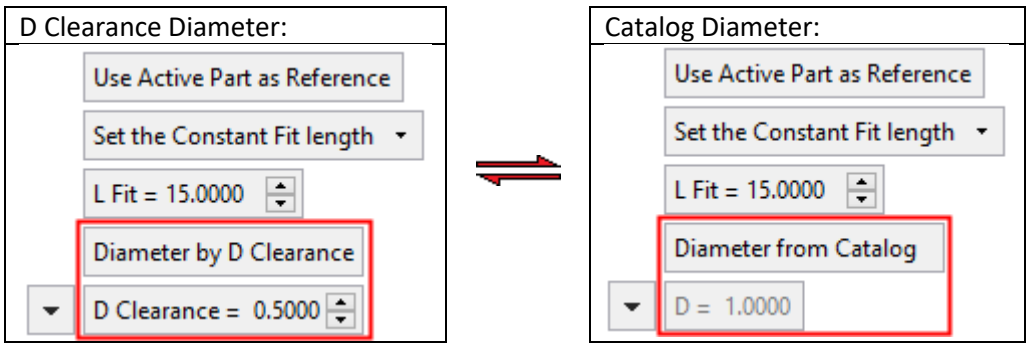
In the Add tool, when adding on a face, a new option was added in version 14 to change the diameter of added parts according to the diameter of the circle created on the Add Sketch. This option was initially implemented for ejectors, but was now extended to also support other components including cooling components. This is often required for baffles which may require different diameters in the same add operation.

2.11 Ejector Pocket improvements

2.11.1 Use catalog diameter for ejector pocket

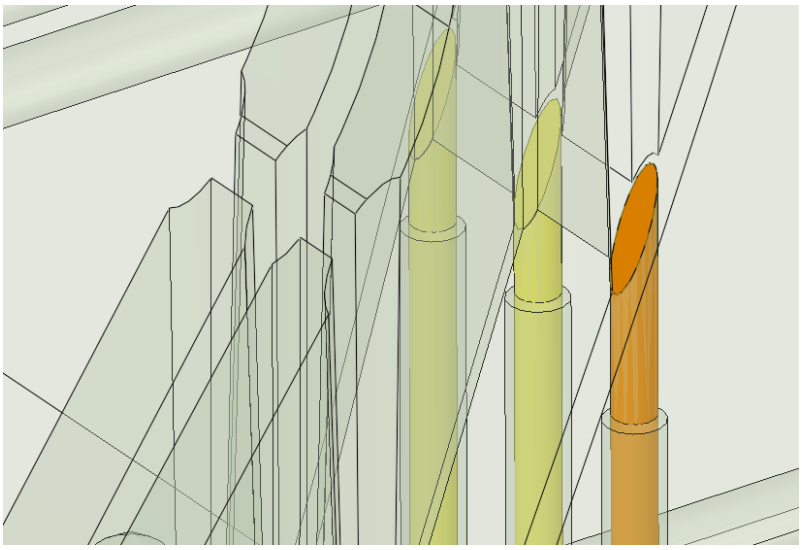
When using the Ejector Pocket tool, the offset from the ejector in the part of the hole that is loose, was set as a constant offset value (*D clearance*). A new option now allows to take this value from the catalog (this would normally be the *d1 cut* value). In some catalogs, it may also be called *d cut* or *p cut*. This allows for standardization of clearances in different molds by using a value that comes from the catalog. It also allows using different clearances for different ejectors based on their diameters.

Note: The standard catalogs themselves do not contain the *d1* value. All of our catalog ejectors have this parameter, but its size is a certain delta above the *d* size. This feature allows users to define these *d1* values so that they can always get a specific *d1* size for a specific *d* size. In order to do that, users will need to edit the catalog ejector according to their preferred sizes.



2.12 Measure minimum distance in ejector pocket

Measure minimum distance in ejector pocket from the bottom end of the ejector and not from the middle.



When measuring fit length, for ejectors that have a steep face at the top, the fit length may turn out short when the minimum fit length is measured from the middle of that face. It is now measured from the lowest point on the face.

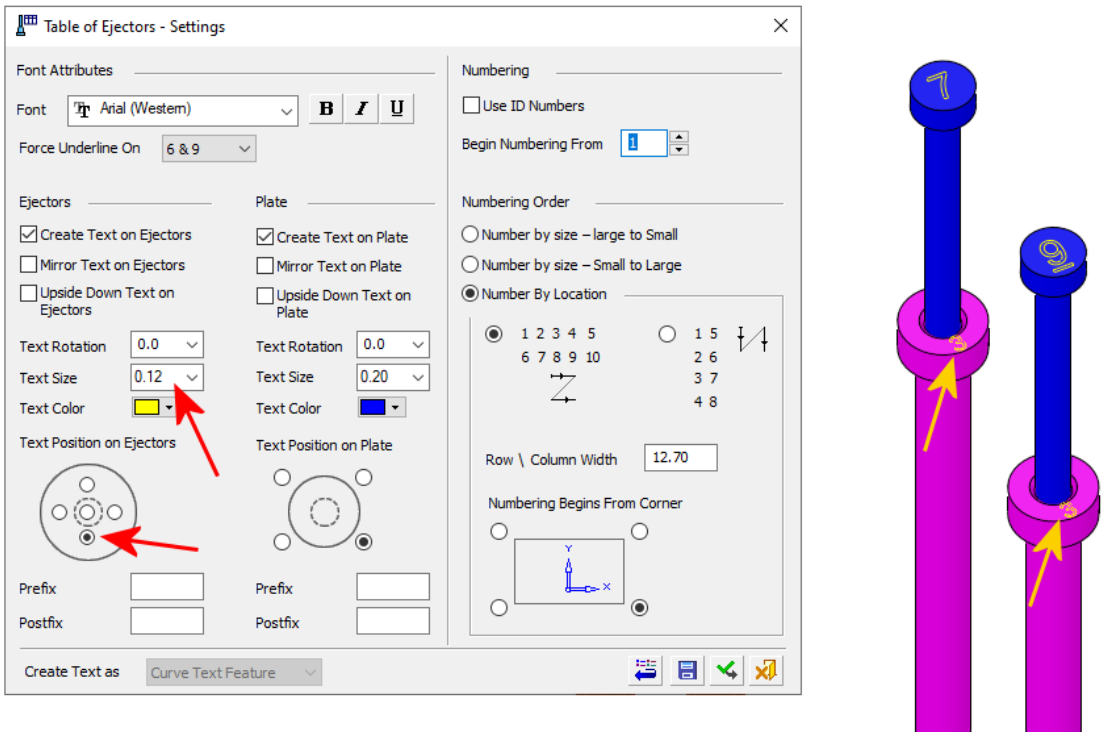
This change results in a safer ejector pocket with the defined minimum fit length applied correctly.

Note that this will only take effect on new features and will not change existing ones.

2.13 Correct Numbering of Sleeves in Table of Ejectors

The Table of Ejectors tool knows to find ejectors and sleeves that go through the same hole and make sure they get the same number. This works perfectly when sleeves are independent parts rather than instances of the same parts (as is usually the case with ejectors). However, when sleeves are all instances of the same part – the results were less than perfect. We have now adjusted the numbering mechanism to give a unique number to all sleeves that are different than the one given to the ejectors in this situation.

Another small improvement is the placing of numbers on sleeves. You may now position text outside of the center of an ejector (where a sleeve will have a hole) and it will be placed close to the edge of the base of the ejector.

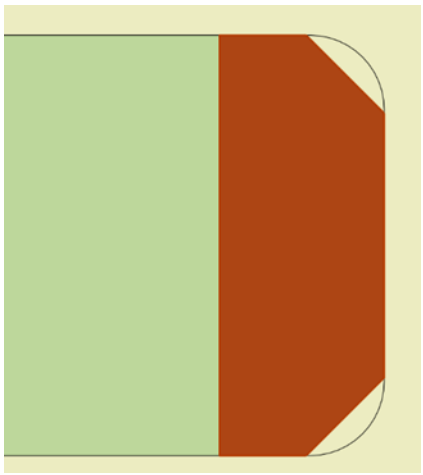


The Table of Ejectors Settings define different positions and sizes of the Ejector text

2.14 Insert Design Improvements

2.14.1 Chamfer for Wedges

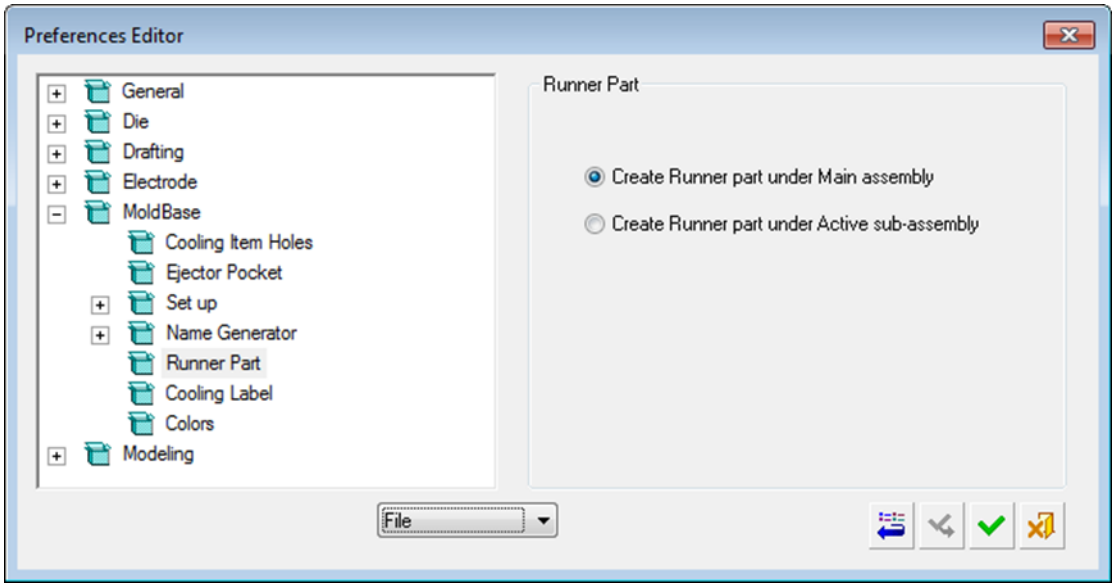
Up to version 15, the corners of wedges could have been rounded. The user now has more flexibility and can either round or chamfer them. This is useful when the pocket for the insert is rounded and the user wants to insert corners to be chamfered.



2.15 Create Runner part in the active sub-assembly

Until now, Runner parts have always been created in the main mold assembly. This makes sense since they sometimes have to cut both sides of the mold. However, when working in Concurrency mode, this situation may be hard to manage as the main assembly is often not owned by any of the users.

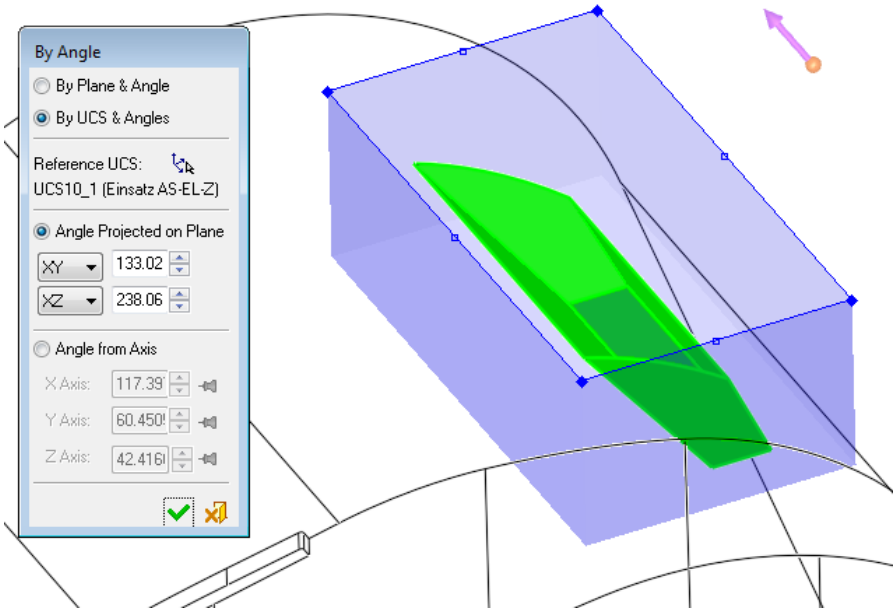
A new option in the Preference allows creating the Runner part under the main assembly or creating it under the active sub-assembly.



3 Electrode Design

3.1 Find electrode extraction direction

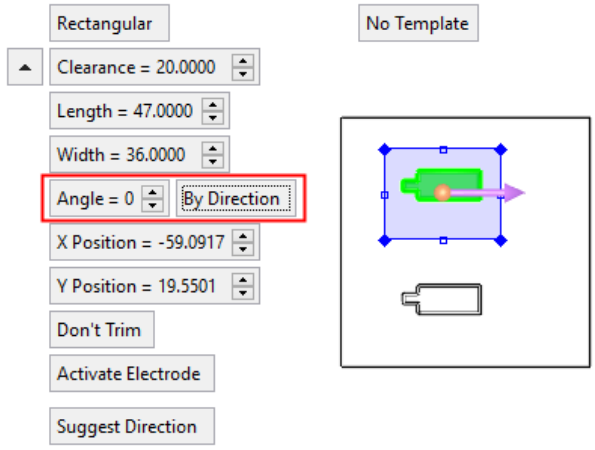
Some electrodes require an extraction direction that is hard to find. Clicking the new Suggest Direction button will find the optimal direction for the selected faces and open the By Angle direction dialog to allow rounding of angles and small modifications.



3.2 Allow to define Rotation Angle using direction arrow

When extracting an electrode, the Reference Line option to define the rotation angle of the blank was now replaced with a button called **By Direction**, which opens the direction arrow allowing to define any direction as the +X direction of the blank. This gives the user much more flexibility in defining direction.

Note that this option currently only appears in the Extract Electrode tool. It will later be made available in the Blank tool as well.



3.3 Electrode – Add rotation angle to cylindrical blanks

Cylindrical blanks can now be rotated by angle or by line, the same way rectangular blanks can. The purpose is to have this angle flow downstream to the Electrode and NC UCSs to affect the way the electrode is placed on the EDM machine and on the milling machine.

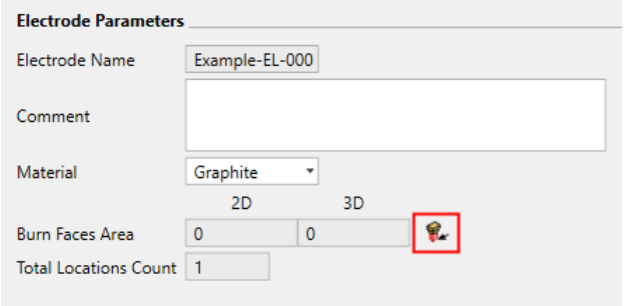
If an angle is defined for the marking, it is “added” on top of the blank angle.

3.4 Allow Select All / Manual Selection for burning faces in EDM Setup

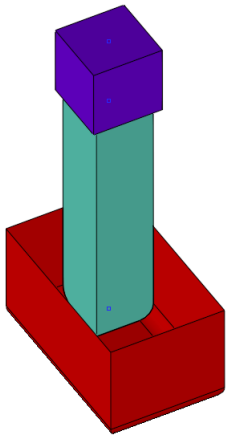
One of the key features of Cimatron’s EDM Setup is the automatic calculation of the electrode’s area.

This value is required by various EDM machines to optimize the burning process.

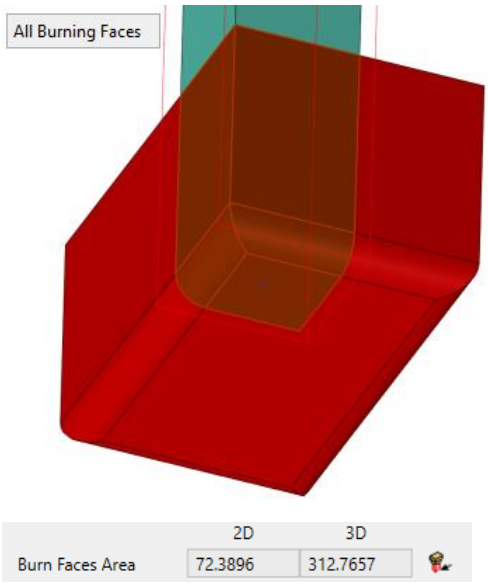
In the EDM Setup, select the electrode and click the Calculate 2D and 3D Area button.



Once clicked, the system selects all the electrode’s burning faces but if required, you can manually change the selection (add or remove faces). This may be needed, for example, if the electrode covers a portion of the originally defined burning area.



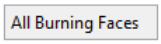
Unselect the non-relevant faces and click OK on the Feature Guide to get the 2D and 3D surface area into the EDM Setup dialog:



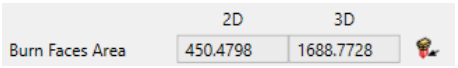
Starting from Cimatron 15, the last selection of faces for each electrode is kept.

As a default, the system still selects all burning faces, however the system always keeps the last selection status for each electrode – so if the user selects/unselects faces, in Edit mode the last selection is available.

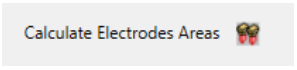
To reset the selection (set to default) click the new option 'All Burning Faces' (this picks all burning faces of the electrode).



In the above example, clicking All Burning Faces will return higher surface area values.

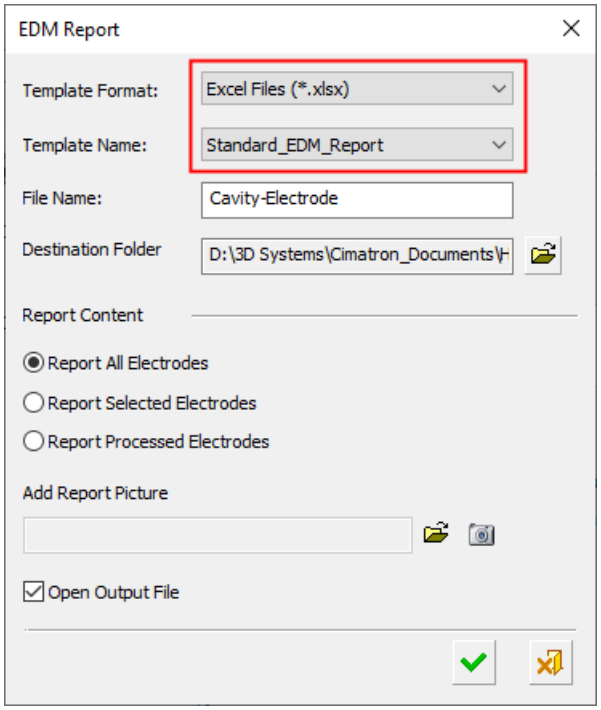


The Calculate all Burning Areas has been updated to consider the last selected faces, so it works based on the last selection of each electrode (up to Cimatron 14.0, it picks all burning faces).



3.5 Add Pictures in EDM Excel Report

An EDM Excel report can now automatically include pictures showing each part in shaded isometric view allowing for easier recognition of parts on the report.



Note that an appropriate template including the picture should be used for this option to be available.

Note that the creation of pictures may take a while on large assemblies.

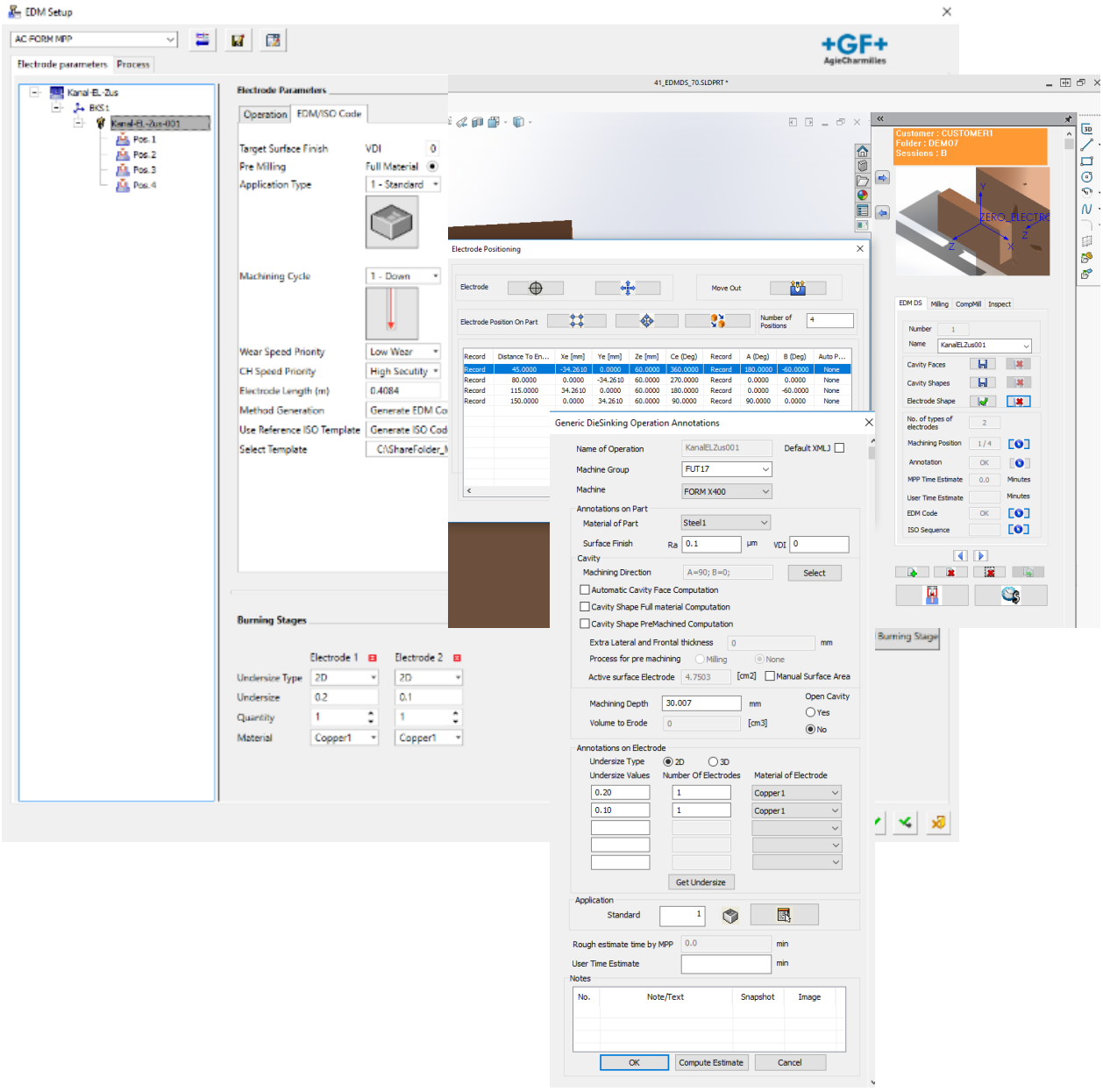
3.6 EDM Setup - Output from Cimatron's EDM Setup to GF Machining Solutions' Multi-Process Preparation (MPP) software

GF Machining Solutions' Multi-Process Preparation (MPP) software enables to prepare and optimize EDM machining processes from 3D CAD to AgieCharmilles die-sinking EDM machines.

Cimatron's EDM Setup can now output project data to GF MPP. For this purpose, a new EDM profile has been added. This profile generates the project data and CAD files and then launches MPP. Once MPP's calculation is over, MPP generates the NC code for the machines.

In order for the Cimatron-MPP integration to work, GF Machining Solutions' MPP and AC Form HMI software must be present on the same PC, where Cimatron is installed.

The image below left shows the new Cimatron EDM Setup profile for MPP. The other images show some MPP interactions, with project data received from Cimatron's EDM Setup.



3.7 EDM Setup – Interface with GF Machining Solutions' MPP

Based on the EDM's project parameters, such as machine type, materials, and surface roughness, the GF MPP software can calculate and suggest best practice parameters for an optimized EDM process.

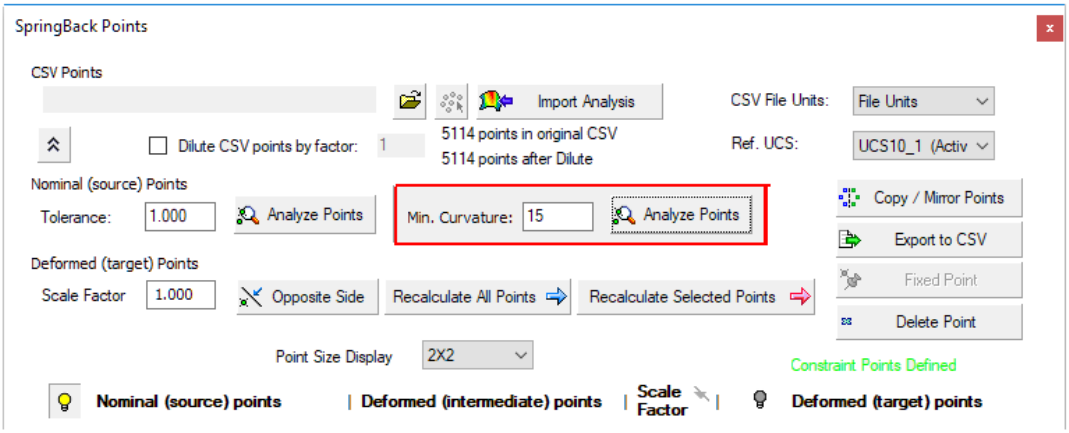
Cimatron's EDM Setup can send (via dedicated functions) selected EDM data of each electrode to GF MPP. GF MPP automatically computes the optimized EDM values, such as the optimum electrode undersize values for each tool. These optimized values are then retrieved by Cimatron's EDM Setup, and updated for each electrode tool.

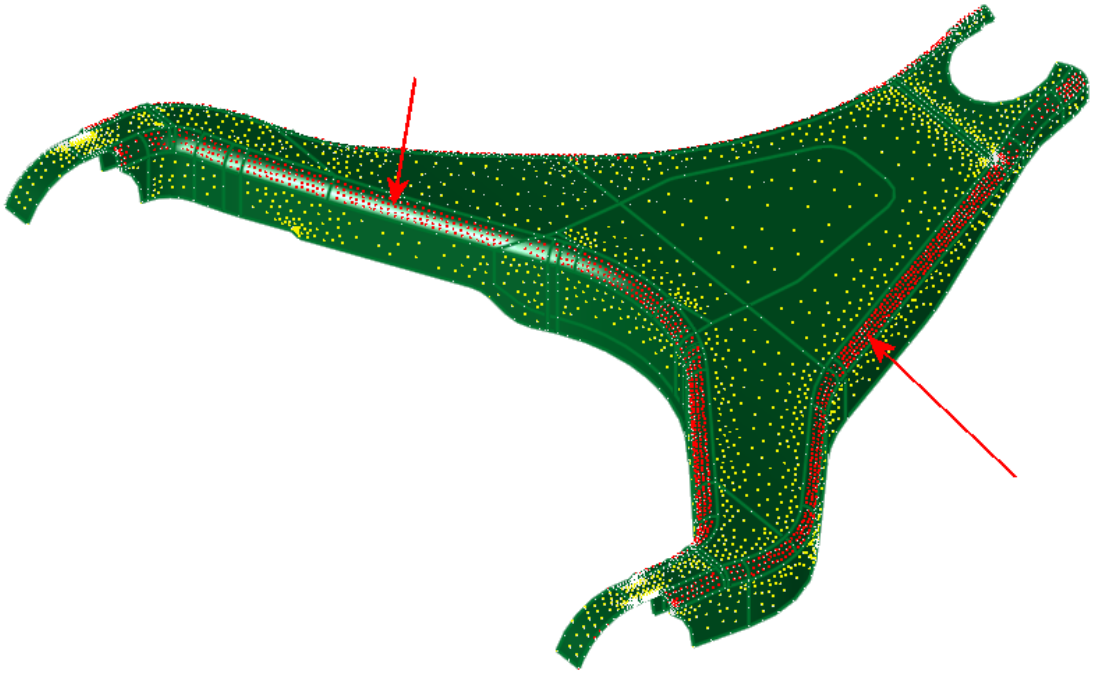
For some EDM parameters, based on the data sent from Cimatron, MPP can send back to Cimatron's EDM Setup a list of possible/suggested values for optimized EDM.

4 Die Design

4.1 Springback

An option has been added to detect points that lie on high curvature areas. Generally, areas of low curvature work better with the springback tool than those with high curvature. This option allows finding points that may be problematic. These points may then be deleted so that they are excluded from the operation.



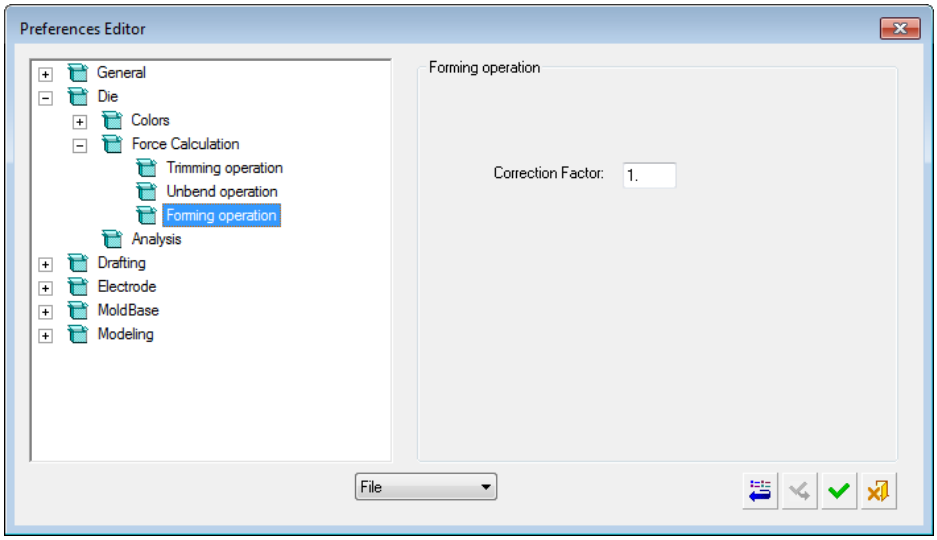


4.2 Match Points on Skin

Warpage and Springback Compensation were improved. A general ability (not only for Die) to compensate B-rep against Mesh rather than point cloud. The results are saved within the Cimatron file rather than in an external CSV file and can later be used for Springback Deform.

4.3 Add Correction Factor for FTI (forming) Forces

A new correction factor was added to the calculation of forces for forming (similar to the existing factors for Trimming and Unbend). This factor will affect the result of Die Forces and allows the user to have more control over it.

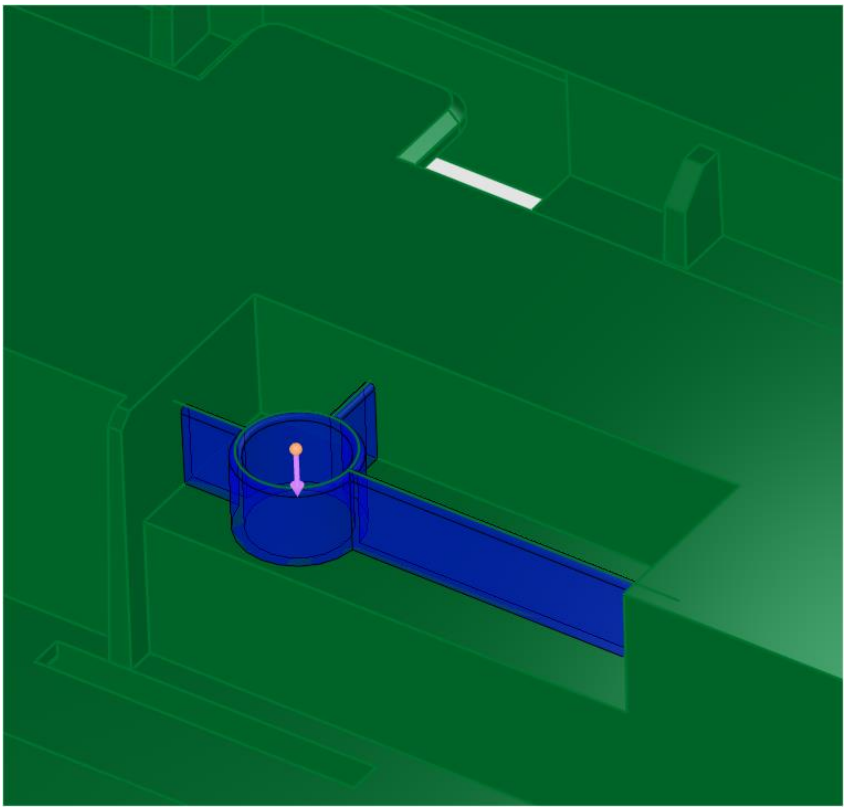


5 Part Design

5.1 Rib

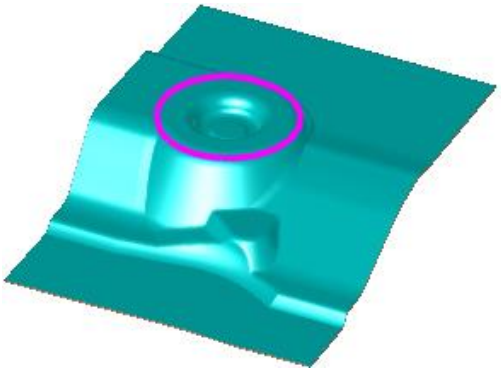
The long awaited Rib tool is now available. The user can now create a rib to strengthen regions of the part.

The rib is based on a 2D wireframe with a defined thickness, draft angle, direction and a flat or rounded top.

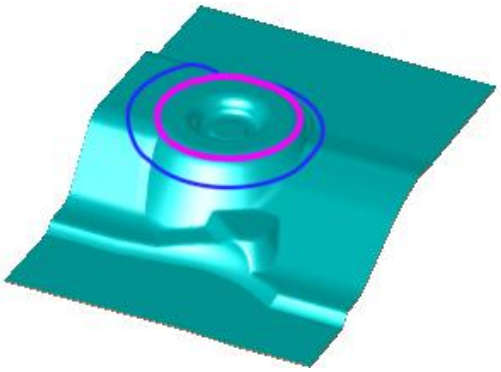


5.2 Offset Wire Along Faces

A new tool that allows offsetting a curve along 3D faces (or mesh) so that the offset value is kept along the selected faces. This is useful for parting surface splitting at a certain distance from the parting line, as well as for similar operations in die design.



Cyan geometry to be offset

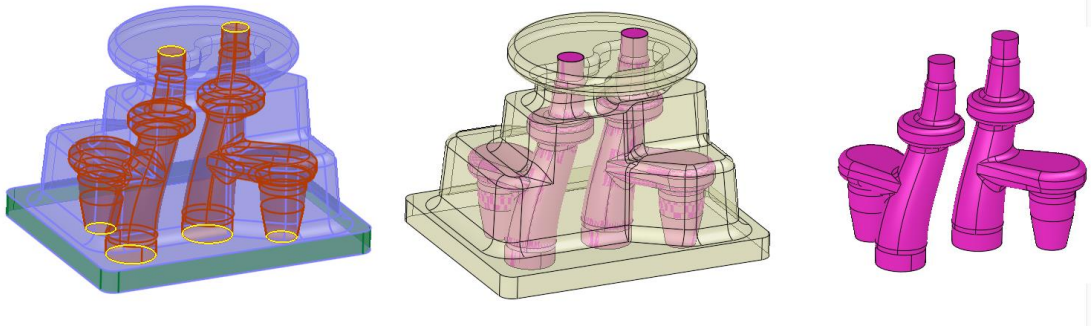


Result – blue geometry offset along faces

5.3 Volume Inside Shell

This new tool allows for the creation of an object inside a shelled part, filling up the shell.

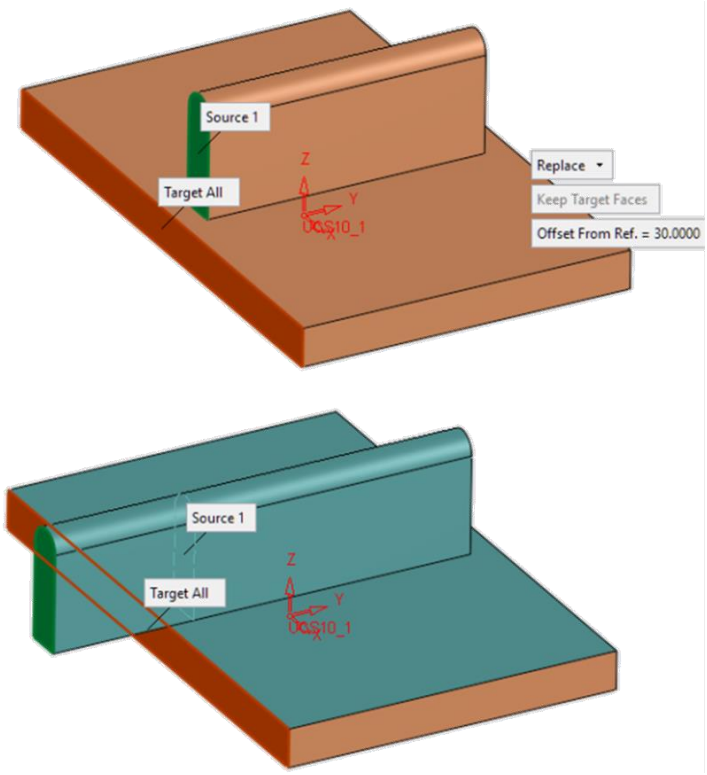
This is useful when you get a shelled part and need to create an object holding its inner volume. It is particularly useful for the creation of cores for sand casting and may also be useful for other casting technologies (investment casting for example).



5.4 Direct Modeling and Extend Object

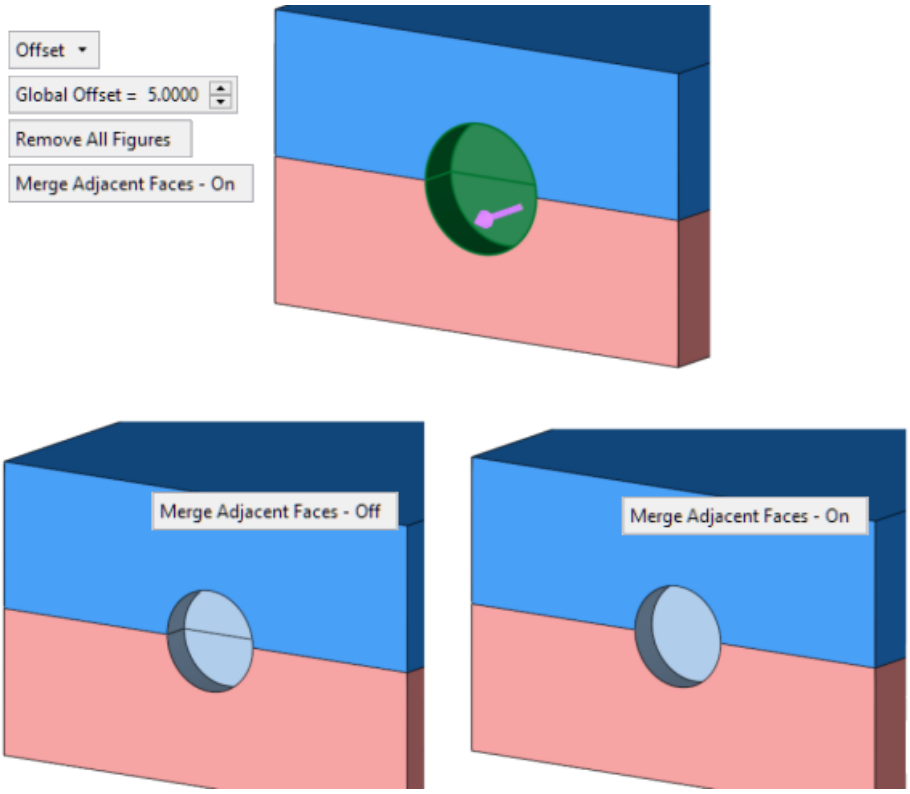
5.4.1 Offset From Reference

In the Offset From Reference option in the Direct Modeling and Extend Object tools, the user can now add an Offset value from the Reference (as in Extrude to Reference), allowing more flexibility in the definition of the target face.



5.4.2 Merge Adjacent Faces

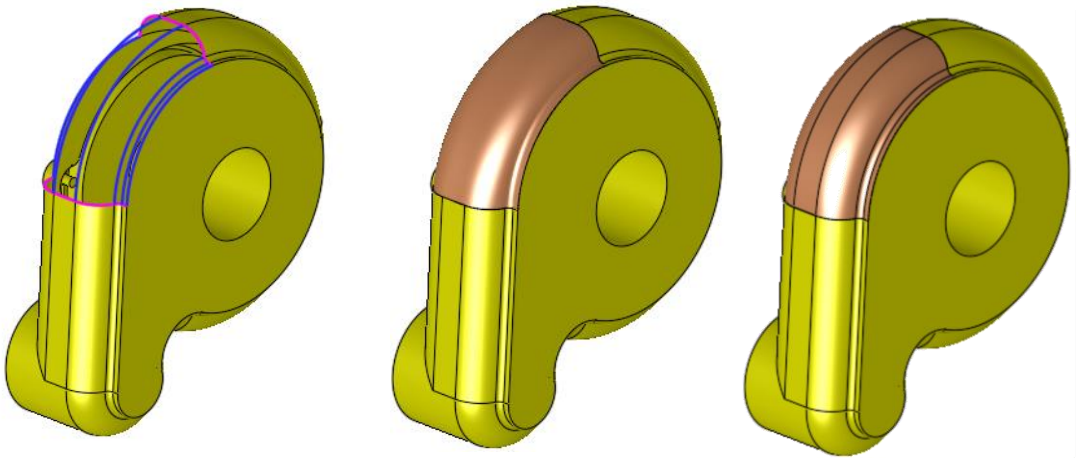
In the Merge Adjacent Faces option in the Direct Modeling and Extend Object tools, the user can merge adjacent faces after selected faces have been removed. This extend the adjacent faces to fill the gap of the removed faces.



5.5 Surface Design Improvements

5.5.1 Blend - Single/Multi Faces

The result of the blend tool can now be multiple faces allowing for higher quality results.

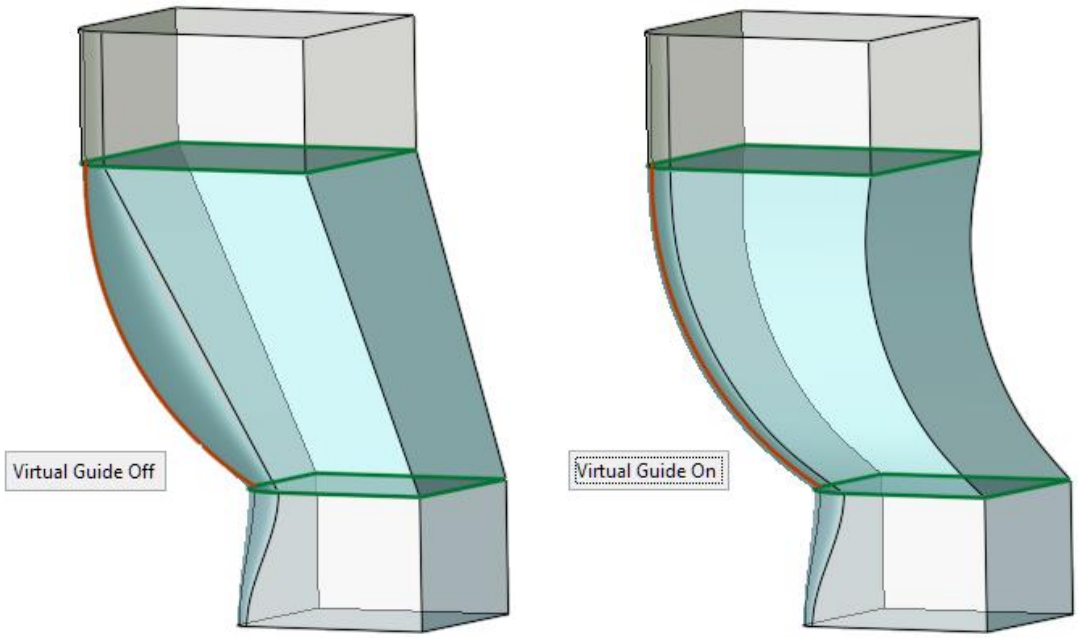


Before Blend *Single Face* *Multi Faces*

Note that the option mentioned below for Virtual Guides in the Skin tool was also added to the Blend tool.

5.5.2 Skin – Virtual Guides:

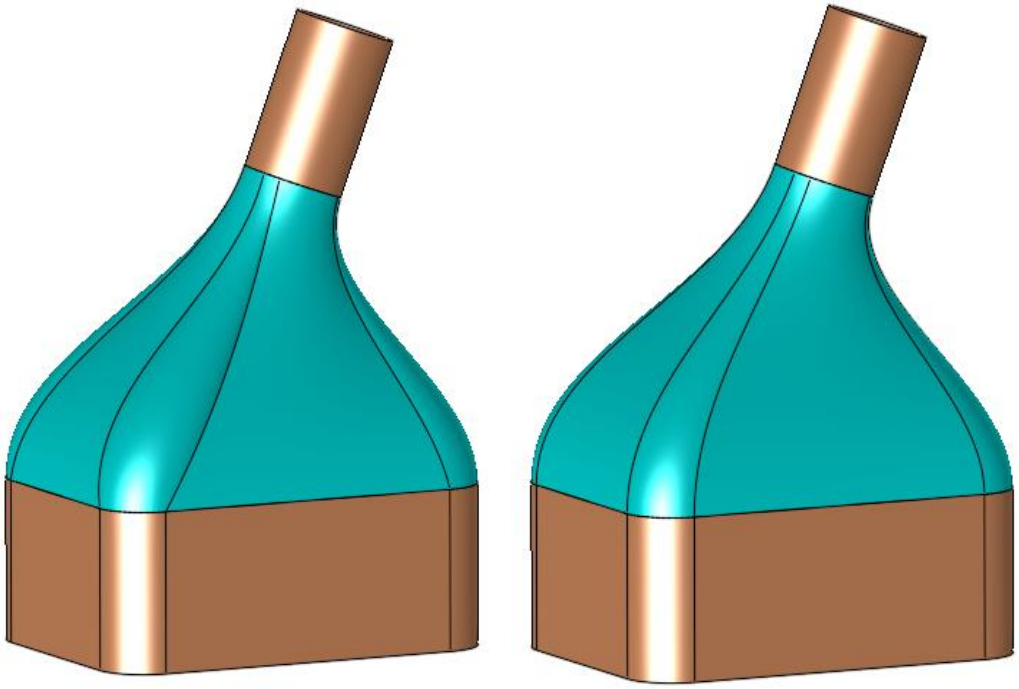
An option to define virtual guide curves was added. The selected guide curve defines the shape of the skin face and sets the progression for the entire section. This allows the user to create a single guide curve instead of several guides.



Virtual Guide OFF *Virtual Guide ON*

5.5.3 Loft – Smooth/Free Faces Connection:

A smooth connection to the faces is now available resulting in a more streamlined model with less sharp corners.

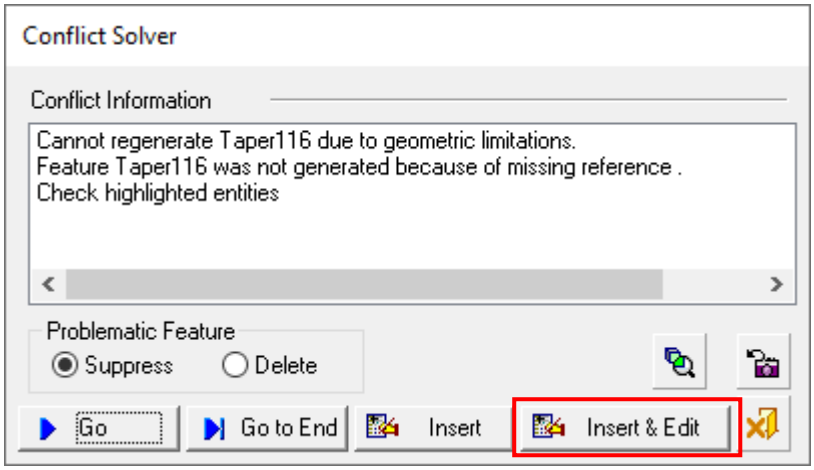


Free Faces Connection *Smooth Faces Connection*

5.6 Conflict Solver

The **Insert & Edit** option has been added to directly edit a feature that caused the failed execution.

When this button is clicked, the Part file becomes active and the feature directly enters edit mode.



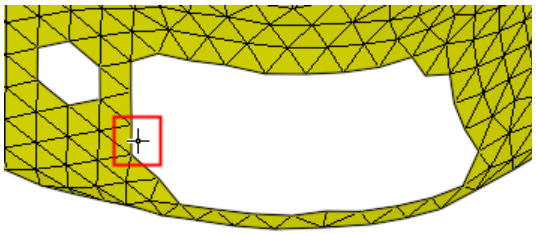
5.7 Mesh Design

5.7.1 Composite on Mesh

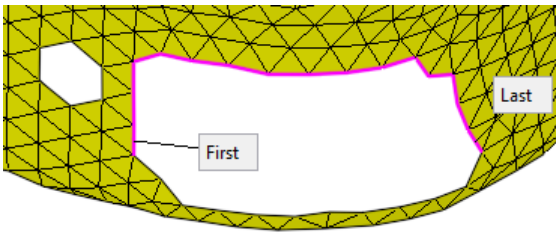
A new tool was added allowing the creation of composite curves by selecting mesh facet edges the same way as defining a composite curve on edges of faces. The created contour can be used as input for any solid design, surfacing or mesh design operation.

As in the Composite Curve tool of faces, the composite curve can be defined Along Open Edges, closed or open, with or without simplification.

Options specific to this tool are the shortest Path Along Points (find the shortest path along facets between selected points) and Planar Approximation (available when selected points are not on a plane).



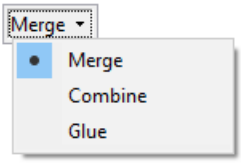
Contour Start Point



Composite curve result

5.7.2 Unite Mesh – New Options

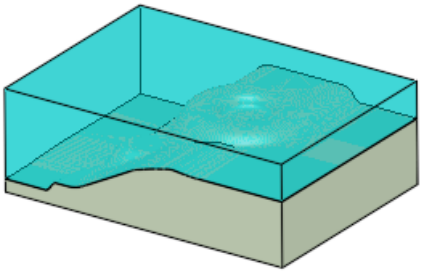
New options have been added to the Unite Mesh function. This allows for a higher success rate of the unite operation.



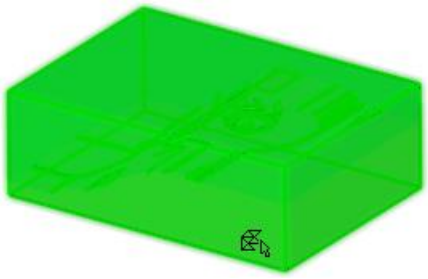
Merge: Merge the selected objects using boolean operations.
This is the existing option.

Combine: Combine the selected objects using non-boolean operations. This option may improve the results for mesh closing operations. If the resulting mesh object is self-intersecting, a message is displayed.

Glue: Unite the selected objects when they are almost coincident with each other (where there are small gaps between the objects).



Two mesh objects



The two objects are united

5.7.3 Smooth Mesh

A new function allowing smoothing a mesh object or selected facets to remove sharp edges and simplify the object. 5 levels of smoothing are available. You may define the smoothing to occur with or without shrinkage. You may also keep the current mesh faceting or re-facet the mesh according to a defined facet edge length.

Smoothing level

Smoothing Level = None

Smoothing Level = Low

Smoothing Level = Medium

Smoothing Level = Medium-High

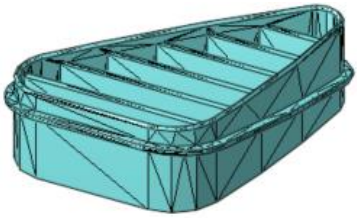
Smoothing Level = High

Allow or Avoid Shrinkage of the selected entities during the smoothing operation:

Avoid Shrinkage
Smoothing Level = Medium-High

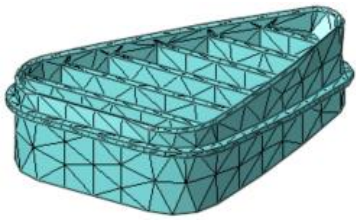
Allow Shrinkage
Smoothing Level = Medium-High

Keep the current mesh faceting or re-facet the mesh according to a defined facet edge length.



Keep Mesh

Re-facet Mesh
Target Edge Length = 0.1

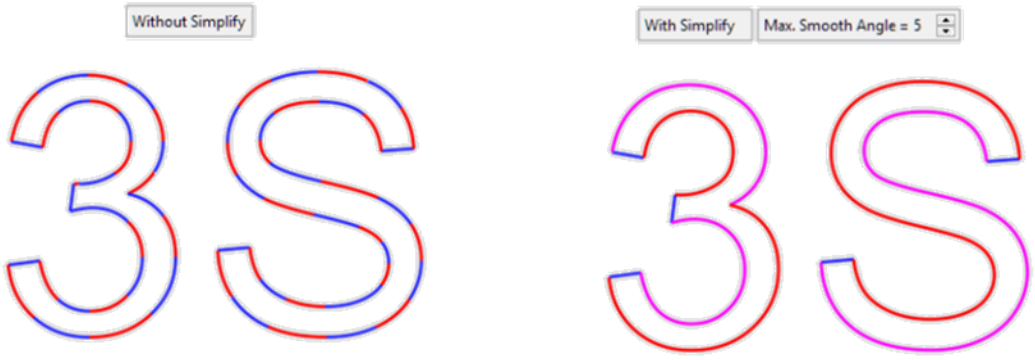


Smoothing Level = High
Target Edge Length = 0.4

5.8 Minor Wireframe Design Improvements

5.8.1 Simplify wireframe

In the Project and Text tools, a new option was added to simplify the resulting contours to allow for a higher quality result with less edges.



5.8.2 Multiple Wires

In the Intersection Curve, Silhouette Curve and Project Curve tools, a new option allows to create a single or multiple wire bodies. This allows the result to be comprised of multiple wire bodies, which makes it usable in cases where indeed it is comprised of multiple lumps (sections), without the need to unmerge them first.

5.8.3 Skeleton Curve

The Skeleton Curve tool now allows the selection of planar faces and creates a skeleton curve based on the boundary edges of the selected face, eliminating the need to first create a composite curve on those edges.

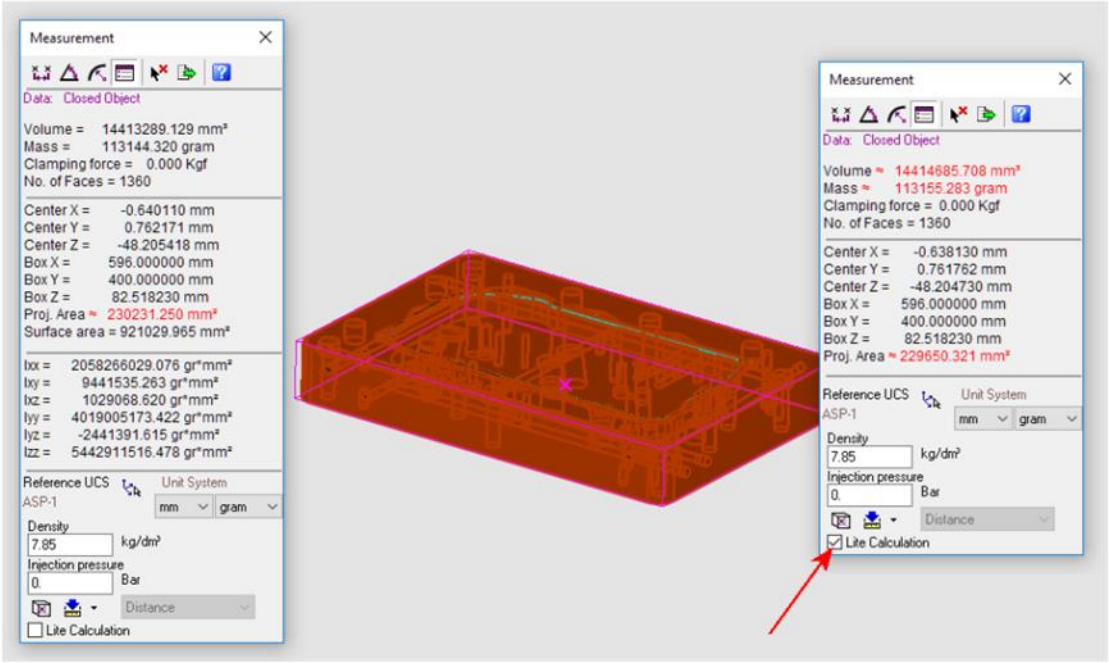
5.8.4 Plane

When creating a Parallel Plane with the option By Offset, the new “Both Sides” option allows creating planes in both offset directions in a single operation.

5.9 Analysis

5.9.1 Measuring - Faster Calculation of Mass properties

A new option called Lite Calculation was added to the Data tab of the Measurement tool. This option shortens calculation time of large objects to be up to 50 times. Note that when using this option, the Moment of Inertia and Face Area are not calculated, the accuracy of volume and mass may be off by up to 2%, and bounding box may be off by up to 0.01 mm.



5.9.2 Interference Check on Mesh

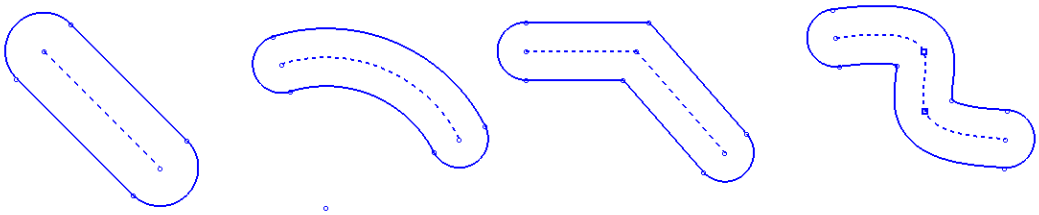
The Interference Check tool can now work on mesh objects and check their interference with other objects.

5.10 Sketcher

5.10.1 Slot Design

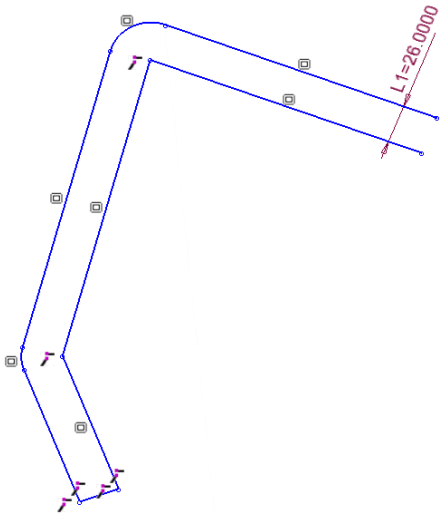
A new tool in the Sketcher allows the design of slots with a single operation rather than creating the slot elements separately.

A slot may be defined dynamically by interactively defining its spine line or arc (in any of the methods available in the Sketcher to define them), or by selecting any sketcher geometry as its spine. The slot's thickness (offset from spine) may be defined as a dimension or by free hand.



5.10.2 Improve associativity of offset operations

A new Offset constraint is added to entities created as a result of an Offset operation in Sketcher. This improves the associativity of offset operations after trimming or modifying source or target entities. This means the result of offset operations is much more robust – it behaves better when the offset value changes and after entities are trimmed or modified.



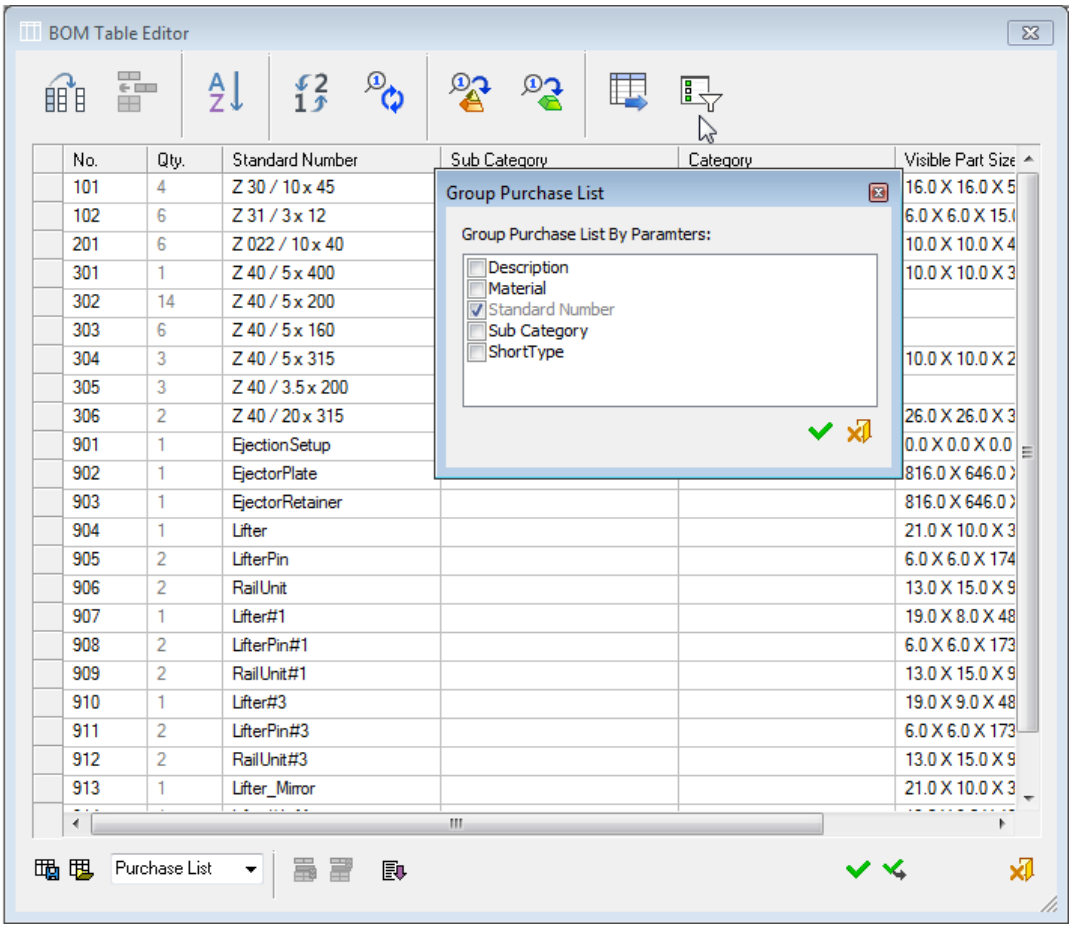
6 Assembly Design

6.1 BOM Enhancements

6.1.1 Add grouping parameters in purchase list

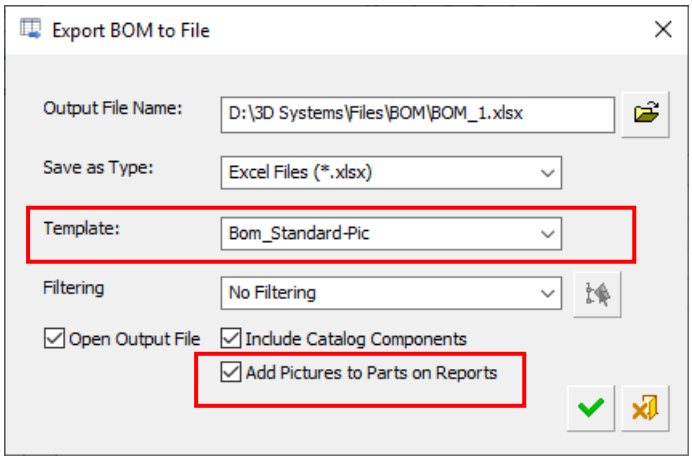
When creating a Purchase List BOM, a new option called Group Purchase List allows grouping parameters based on additional criteria. For example – as always, all parts with a specific standard number will appear by default as a single row, but the user may additionally want to separate them to different rows based on their material or the supplier they come from. This provides an additional layer of control over the purchase list, allowing the user to see and show more information based on catalog data.

The grouping parameter may be any BOM column. The list of checkboxes in the dialog can be controlled using a new column in the BOM_Template.csv file, called Grouping List.



6.1.2 Add a picture of each part in the Excel report on the table

The Excel report of a BOM (in modeling only) can now include pictures showing each part in shaded isometric view allowing for easier recognition of parts on the report.



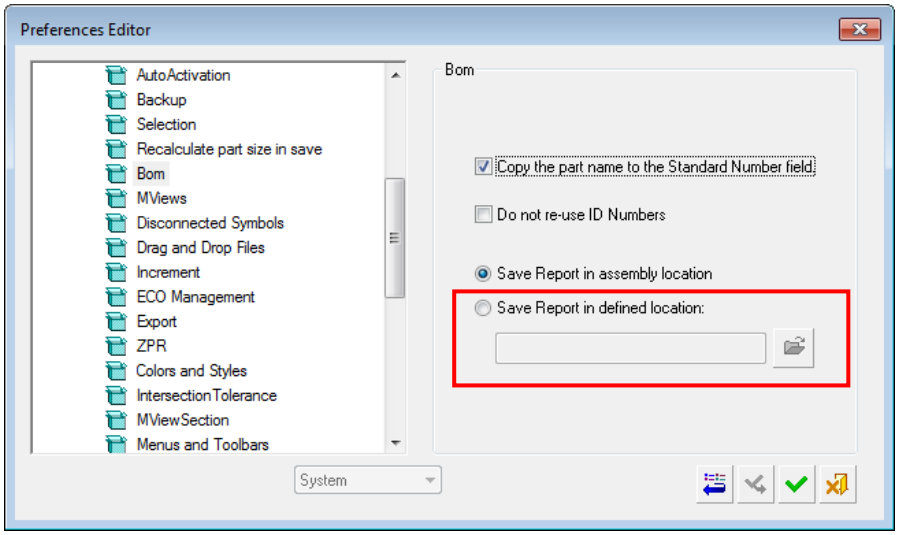
Note that an appropriate template including the picture should be used for this option to be available.

Image	ID Number	Quantity	Standard Number	Sub Category	Category	Visible Part Size	Material
	1	1	K 20 /246 346 /36	Cavity Plate	Assembly Plates	246.0 X 346.0 X 36.0	1.1730
	2	1	K 10 /246 346 /27	Clamping Plate	Assembly Plates	296.0 X 346.0 X 27.0	
	3	1	K 70 246 346 22	Ejection Retain	Assembly Plates	158.0 X 346.0 X 22.0	1.1730
	4	1	K 20 /246 346 /36	Cavity Plate	Assembly Plates	246.0 X 346.0 X 36.0	1.1730
	101	1	Z 31 / M8 X 18	Cap Screw	Screws & Bolts	13.0 X 13.0 X 26.0	DIN 912 ,10.9

Note that the creation of pictures may take a while on large assemblies.

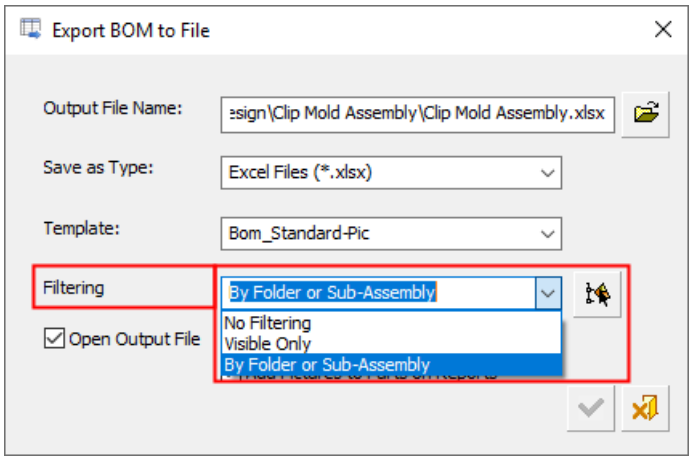
6.1.3 Allow to save report with assembly or in designated folder

A new preference option allows to save BOM reports in the assembly folder or in a constant predefined folder for user who prefer all reports to be concentrated in a single location. This sets the default of the report's save location. The actual location of each report may still be changed in the report creation dialog.



6.1.4 Allow to select more than one folder or SUB-ASSEMBLY

In the BOM report tool, under the filtering option By Folder or Sub-assembly, it was possible to select an assembly folder or sub-assembly that will define the content going into the BOM. It is now possible to select multiple folders or sub-assemblies, allowing for a more flexible definition of the scope of the BOM report.



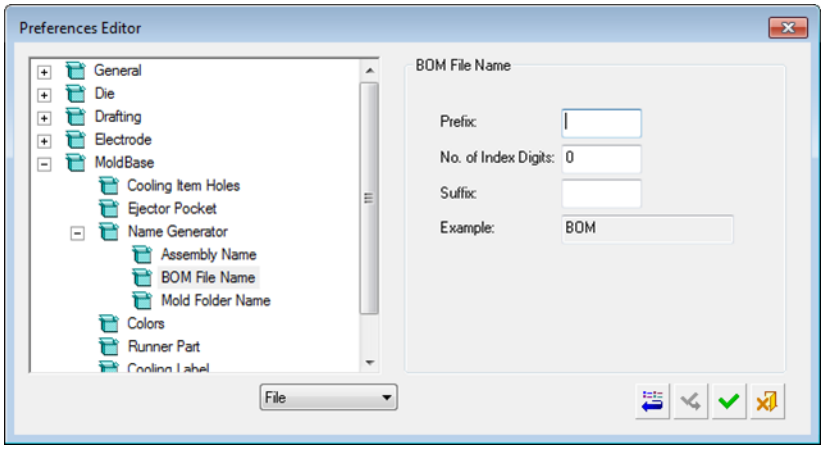
6.1.5 Allow ignore in BOM for sub-assemblies

The Ignore in BOM checkbox is now also available for assembly, allowing the removal of whole sub-assemblies from the BOM.

6.1.6 BOM File Name Generator

It is sometimes useful to keep multiple versions of the BOM so that you can track changes on it. You may set the BOM Name Generator to add an index number to the report files created (by changing the No. of Index Digits from zero to a higher number). When doing so, any report you will be creating from your BOM will get an index number in its name, and when you save another report to the same location, it will not overwrite the previous report, but rather raise the index number.

Note that this is part of the Mold Design name generator and requires a Mold Design license to be accessed.



6.2 Project Number

A new tool allows adding a project number to the assembly (it doesn't have to be comprised of numbers only, of course). This number will be attached to any component on the assembly and to any component that will be added to it.

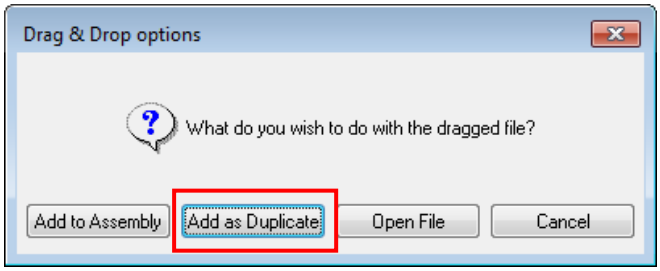
The number is attached as an attribute to each of the files. It can be viewed in Cimatron Browser, and as any attribute – can be used in different tools – for example – it can be used as a symbolic text and be added to any part as such (it can also be added beforehand as symbolic text to catalog parts and get the right number after the tool is used).

Note that the tool is only available when the main assembly is active.

6.3 New Add Duplicate Options

6.3.1 Drag & Drop Add

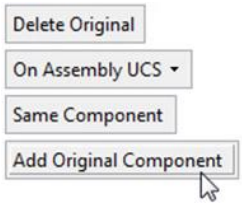
When dragging a component into an assembly, when the preference option of Drag and Drop is set to Ask Me, a new option was added to bring in the component as a duplicate of the dragged component. This is useful when the user brings in a part that is used in another assembly and would like to have an independent copy of it without having to first copy it manually.



6.4 New Add This Component Options

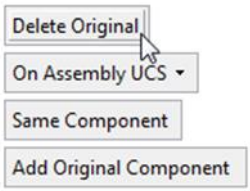
6.4.1 Add as a Duplicate Copy

When using the Add This Component tool, when adding as Same Component, a new option allows to add the original component or add a duplicate component of it. This is useful when we want to add several instances of an existing component. We want all of them to be instances of a single part, but we don't want it to be the original part, but rather a copy of it.

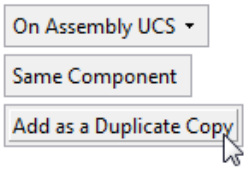


6.4.2 Delete Original Component

When bringing an existing component into the assembly and positioning it, you now have the option to remove the original component. By doing so you have actually moved the component from its old location to the new location(s). Note that this option has been available in past version and was removed, and was now brought back.

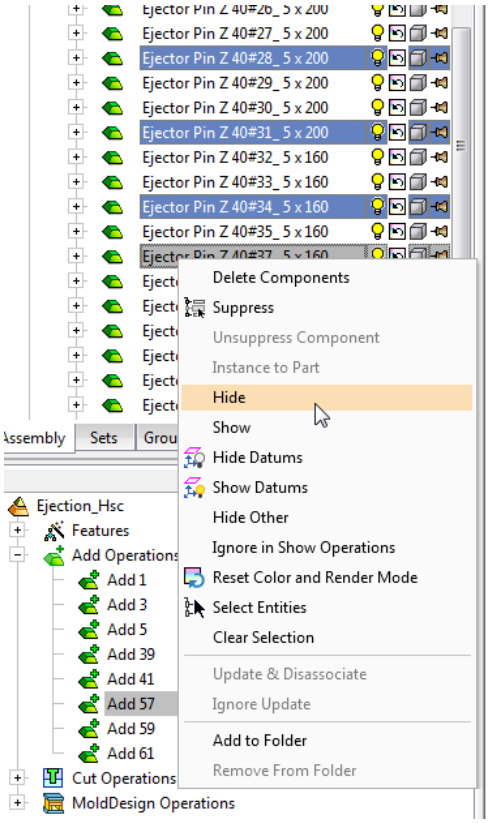


When using the Add This Component tool, when adding as Same Component, a new option allows to add the original component or add a duplicate component of it. This is useful when we want to add several instances of an existing component. We want all of them to be instances of a single part, but we don't want it to be the original part, but rather a copy of it.



6.5 Hide / Show Multiple objects

When selecting multiple components on the assembly tree and using the context menu on them, new Hide and Show options are now available to hide and show all selected components.

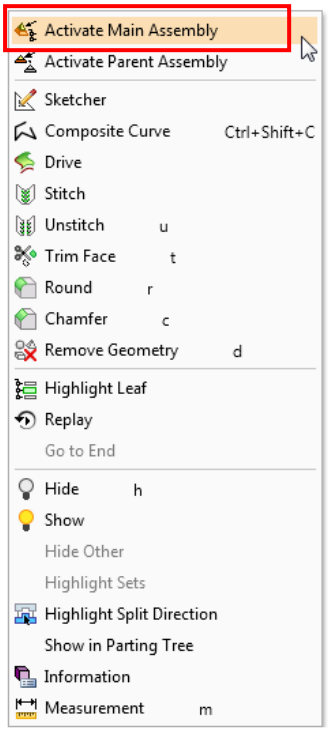


6.6 Nested Add – Add on Cone Face

The Nested Add option is now also available when adding assemblies with the option Place on Cone Face (rather than only on Add on Face). Other than this new placement option allowing to position the nested assembly on cone faces, it works exactly the same as before.

6.7 Restore Activate Main Assembly as part of the RMB

The option Activate Main Assembly was missed by many users and has now returned to the context menu when the main assembly is not active.

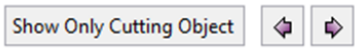


6.8 Cut Improvements

6.8.1 Hide and Show Cutting Objects

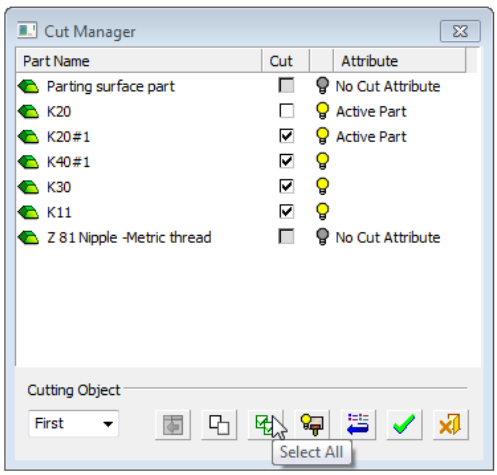
In order to allow more efficient work with cutting objects, the Assembly Cut by Selection tool was added the ability to hide everything but cutting objects so that they are easier to pick. Clicking the button again returns the hide/show situation to the way it was. Previous and Next buttons for the hide/show status have also been added.

Note that the same capabilities have been added to the Cut tool while working in an assembly environment.

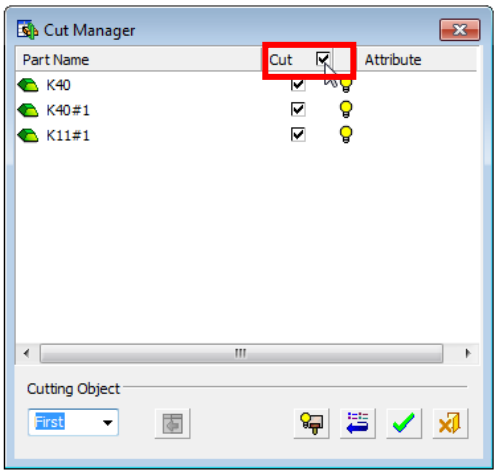


6.8.2 Cut Manager: Select All / Clear Selection - changed to a checkbox

The old Select All / Clear Selection buttons on the Cut Manager were replaced with a checkbox at the top of the column.



Cimatron 14



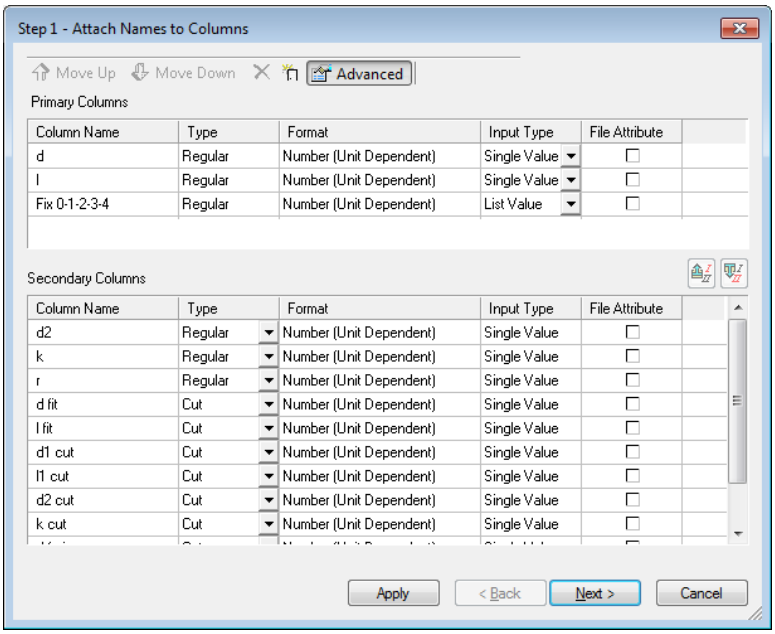
Cimatron 15

6.9 Catalog Parts

6.9.1 Show catalog columns as file attributes

When creating a catalog part, any column can be marked as a File Attribute. Once a column is marked as such, parts created out of the catalog part will have that column data as part of their attributes. By changing system definitions using the Attribute Manager and relevant files (BOM_Template.csv, sym_text.csv), this data can be presented in the attribute section of the browser, in the BOM, or as symbolic text.

This can be useful when you want to add information regarding catalog items that should appear in the BOM for example (such as the specific part vendor from where you are getting this specific catalog record).

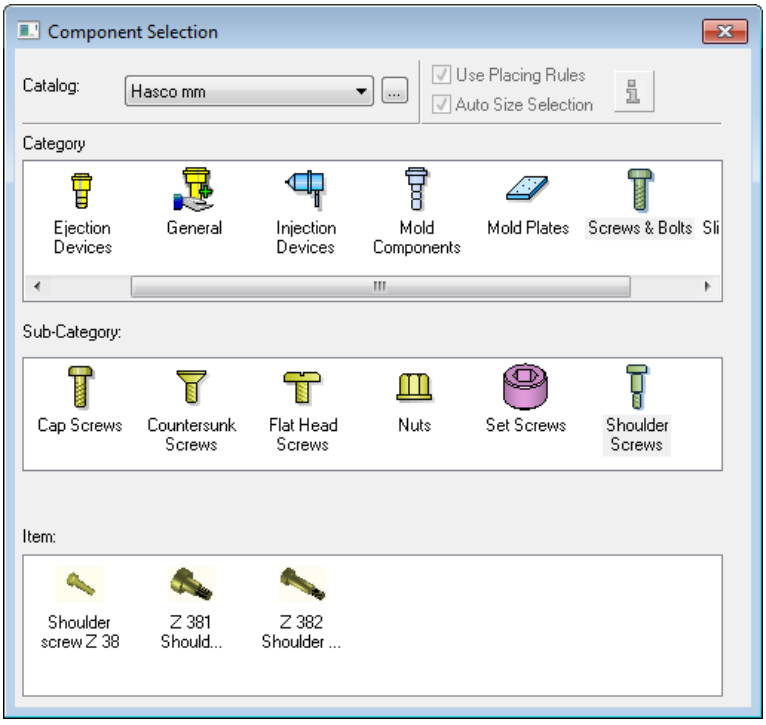


Click the Advanced button on the first stage of the catalog save wizard to expose the File Attribute option.

In addition, the window of the catalog definition wizard can now be resized.

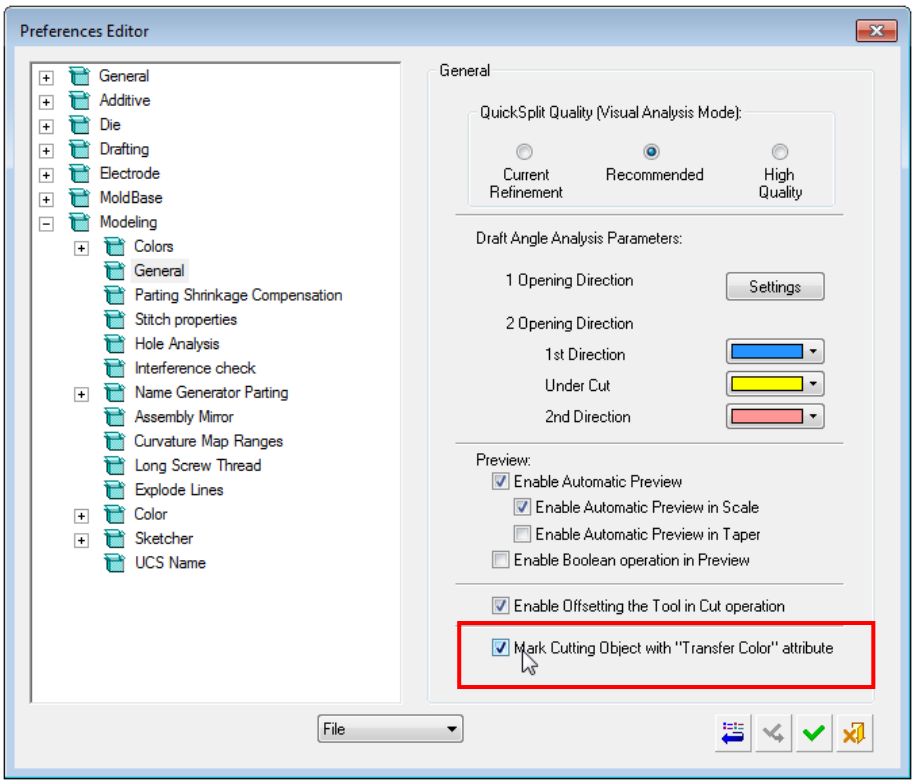
6.9.2 Catalog picture in Add Mold/Die Component

In the Add Mold Component and Add Die Component tools, the last row showing the actual items, now shows a small picture of the catalog part rather than a generic icon of the sub-category, allowing for easier recognition of items.



6.9.3 Cutting object with or without “transfer color"

When marking an object as a Cutting Object (usually used for catalog parts) there is an option to define whether the color of the cutting object will be transferred to the cut part when the cut is performed. This may be required for color coding of holes, for example.



When creating the cutting object using the Cutting Object tool (under the Solid menu), this option can now be controlled from the preference. A new option under Modeling >> General allows to turn Mark Cutting Object with “Transfer Colors” attribute, on or off.

6.9.4 Catalog Updates

New Mold and Die part catalogs can now be downloaded or existing catalogs can be updated.

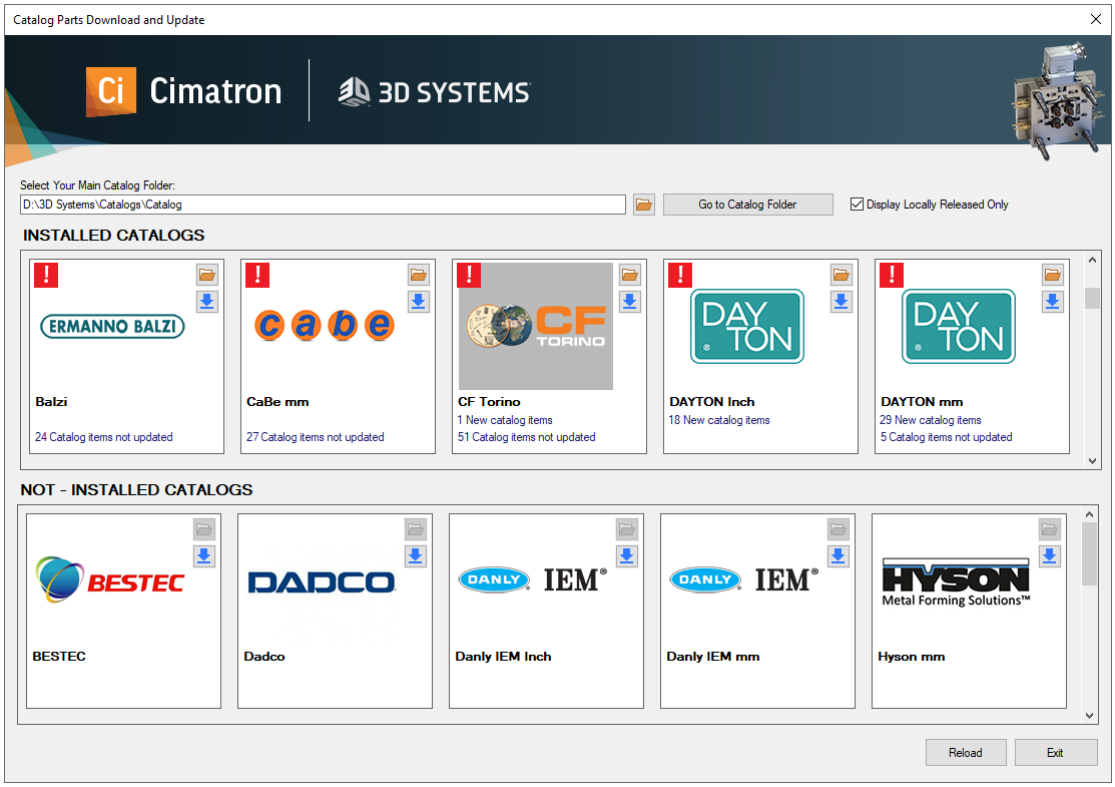
The Catalog Updates tool was unofficially released after the Cimatron 14 Official release. This tool is now officially released with version 15.

New or updated parts are constantly released by 3D Systems and uploaded to the server. Once released, the parts become immediately available for all Cimatron users, and can be downloaded directly onto your PC or local network.

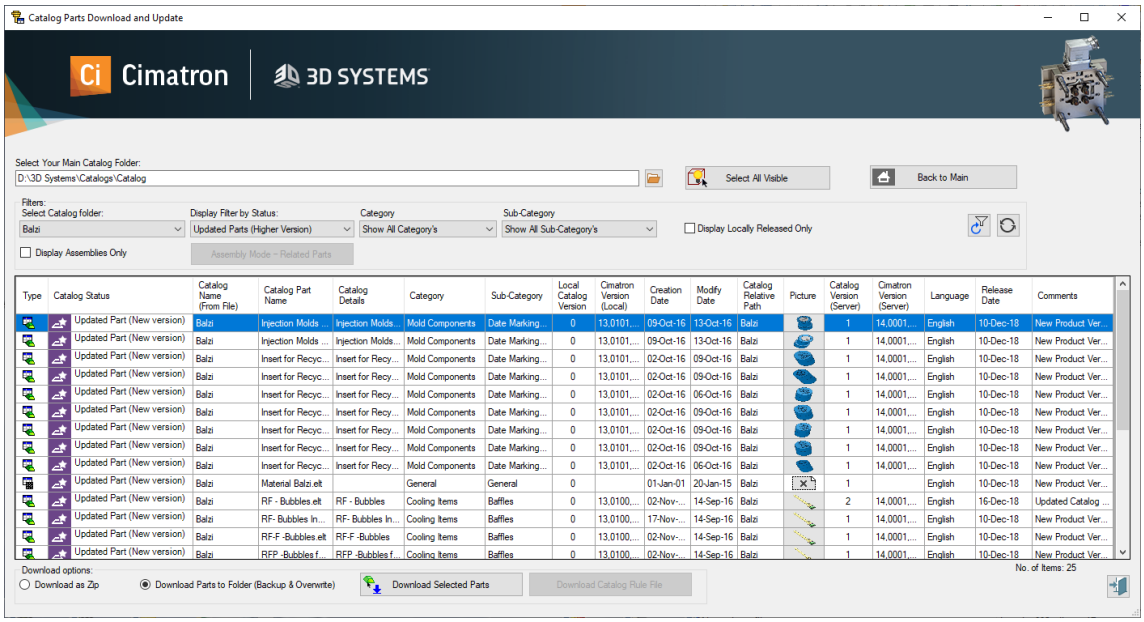
A Cimatron Provider or Reseller can upload additional catalog parts to the server, or upload their translated parts (these are the same parts as released by 3D Systems, but with their names translated to the local language).

The **Cimatron Catalog Updates** utility has two levels (or modes) of views; a **Higher Level** and a **Lower Level** view.

The **Higher Level** catalog update mode shows the status of all available catalogs, those already installed on your computer (or your network folder) and those that have not been installed and are available for downloading from the server. This enables you to quickly see how many catalog parts are new, updated (with respect to those previously downloaded), or to download a complete catalog.



The **Lower Level** catalog update mode shows data for a specific selected catalog. This displays a comparison between the catalog's locally installed parts (on your computer or your network folder) and the latest parts available from the server. You can see which parts are new or updated for a specific catalog and selectively download them.



6.9.4.1 New Functionality

The following new functionality was added to the Catalog Update tool for Cimatron 15:

Higher Level dialog:

Go to Catalog Folder button: Click this button to open the windows browser on the defined catalog folder. This is a shortcut link to the defined catalog folder.

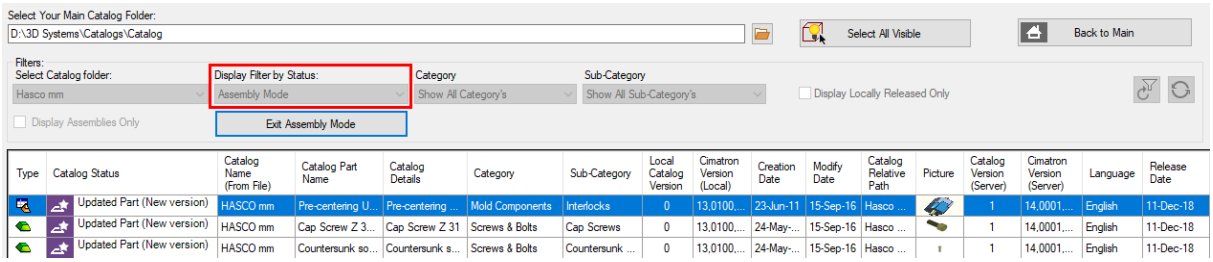
Display Locally Released Only checkbox: This checkbox, already existing in the Lower Level dialog, has also been added to the Higher Level dialog.
When this checkbox is ON, this displays only those catalog items uploaded by the local Provider or Reseller. For example, if a Provider or Reseller partially translated a catalog, local users can see only these translated items.
When this checkbox is OFF, and only some catalog items are translated, the view is a mixed mode view, non-translated catalog items are shown in English and the translated items are shown in the local language.

Download entire catalog folder : When downloading an entire catalog folder to your PC, this includes all the catalog parts, assemblies and configurations.

Lower Level dialog:

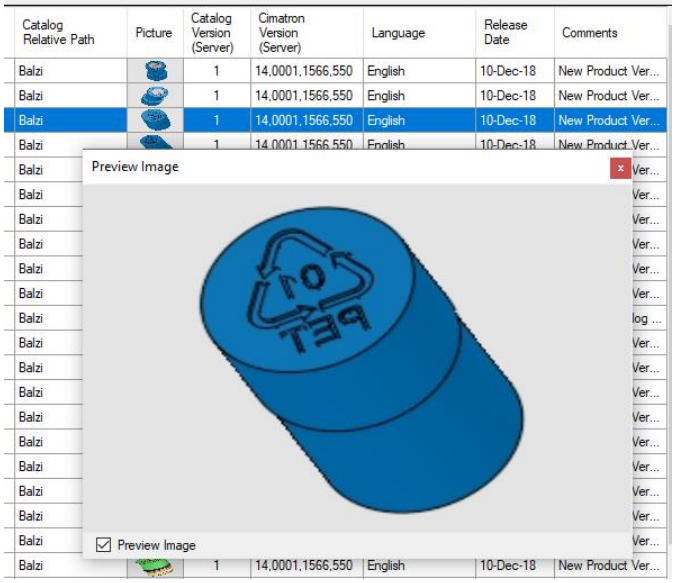
Display Assemblies Only checkbox: When ON, only catalog assemblies are displayed in the dialog table.

Assembly Mode - Related Parts / Exit Assembly Mode toggle button:
This new feature enables to find all the assembly related components.
This button is enabled when a single catalog assembly is selected. Clicking the **Assembly Mode - Related Parts** button displays only the selected assembly and ALL its related components. In this case, the **Display Filter by Status** field shows the text 'Assembly Mode'.
Note: Related components can be shown only if the assembly file exists locally (has been downloaded).
Clicking the **Exit Assembly Mode** button returns the dialog to its previous display status (before Assembly Mode was invoked).



Download Catalog Rules File: This button downloads the catalog rules file if it is available. When adding a catalog component, the catalog rules enable you to automate the process by applying certain rules regarding the location and/or size of the added component.
This button is enabled when either an English rules file exists for the selected catalog, or if a local (translated) file exists (when the **Display Locally Released Only** checkbox is ON).

Preview Image: When enabled in the Customize Columns dialog, the Preview Image dialog is displayed when hovering over an image of a catalog item. This dialog will continue to display images of catalog items until disabled. A checkbox has now been added to the dialog to easily disable it. Click the checkbox to OFF and close the dialog to disable it.



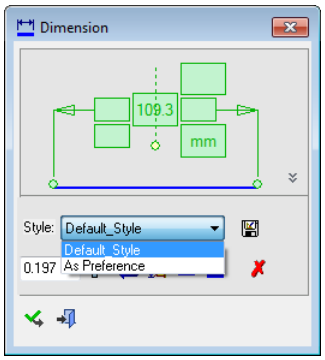
7 Drafting

7.1 Predefined Styles for Symbols

Each drafting symbol (such as Dimension, Geometrical Tolerance, Surface Roughness, Marking, etc.) holds a lot of parameters that control its data and visibility. Often times we want to save the parameters as sets and reuse them.

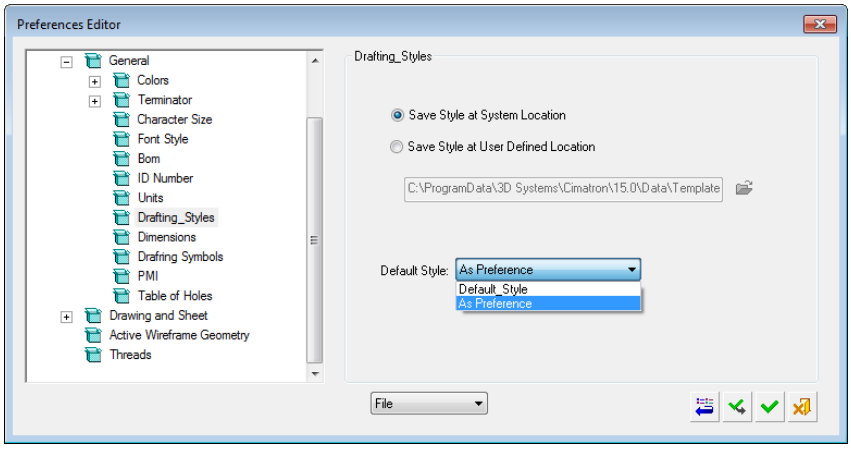
The new Symbol Styles allows you to save any set of parameters for any symbol and reuse them at will, saving tedious definition work and time.

In each symbol dialog you will now find a Style section where you can select the desired style from a dropdown menu, or save the parameters currently defined in the symbol to an existing or new style. Note that the option As Preference is always there and allows you to work with the parameters and definitions set in the preference, essentially eliminating the symbol style functionality.



When you save a style file, a normal save dialog opens up allowing you to save a new style file or overwrite an existing one – in case you want to update the parameters of a specific symbol type. Note that each style file can hold the parameters of many symbol types. Whenever you perform a save, you are only saving the parameters of the symbol style you have used when clicking the save button. If you have chosen an existing symbol file, the system will check if the specific symbol type you are saving already exists in it, and will ask you if you wish to overwrite it if it does. Note that each dimension type is considered a different symbol type (Linear Dimensions, Radial Dimensions, etc.).

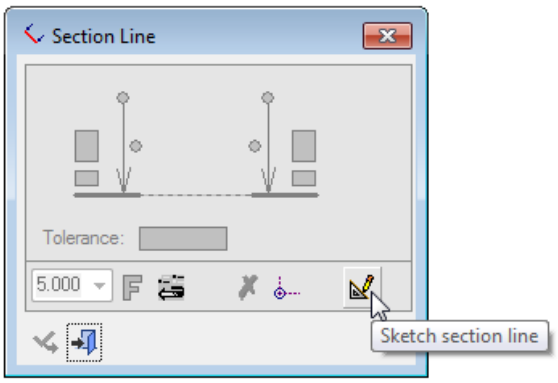
In the Preference, you can set the default style you want to use (including the option As Preference which means you don't want to use styles) and the location in which the style files will be saved. This is important as the dropdown menu will look for style files in that location.



7.2 Section line as sketcher

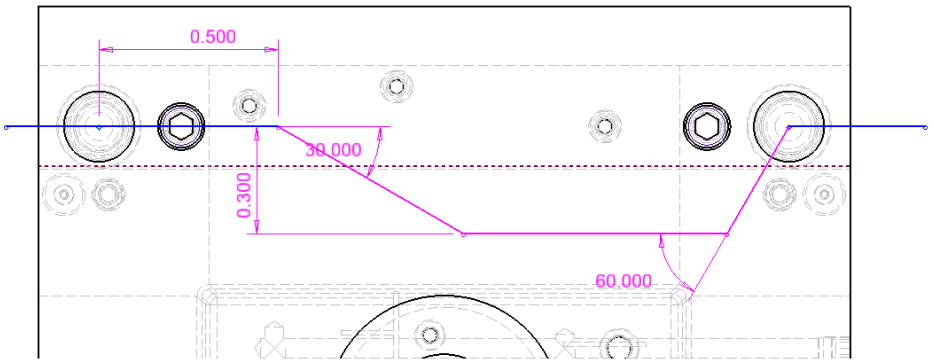
A new option on the section line dialog allows to generate the section line using the Sketcher tool. This allows for much easier creation and editing of complex section lines. It also allows adding dimensions to control distances and angles.

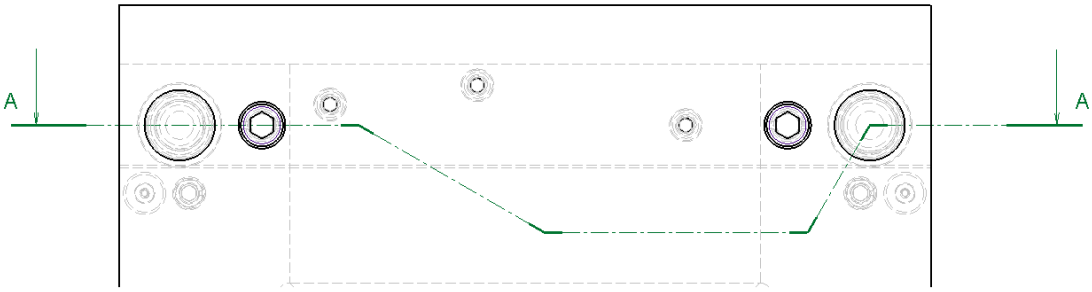
Click the Sketcher icon on the Section Line dialog to enter this mode. Select the view on which the section should be created and use the standard Sketcher tools to sketch the section line.



Note that acquiring a reference point requires hovering above the point for a short time or using the Add Reference tool. The Add Geometry tool may also be used to create lines.

Exit this mode using the context menu and selecting the OK or CANCEL options.

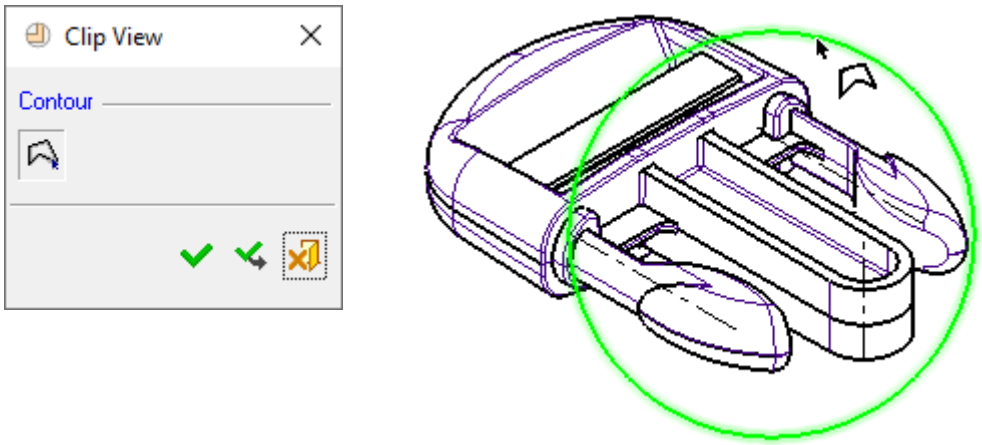




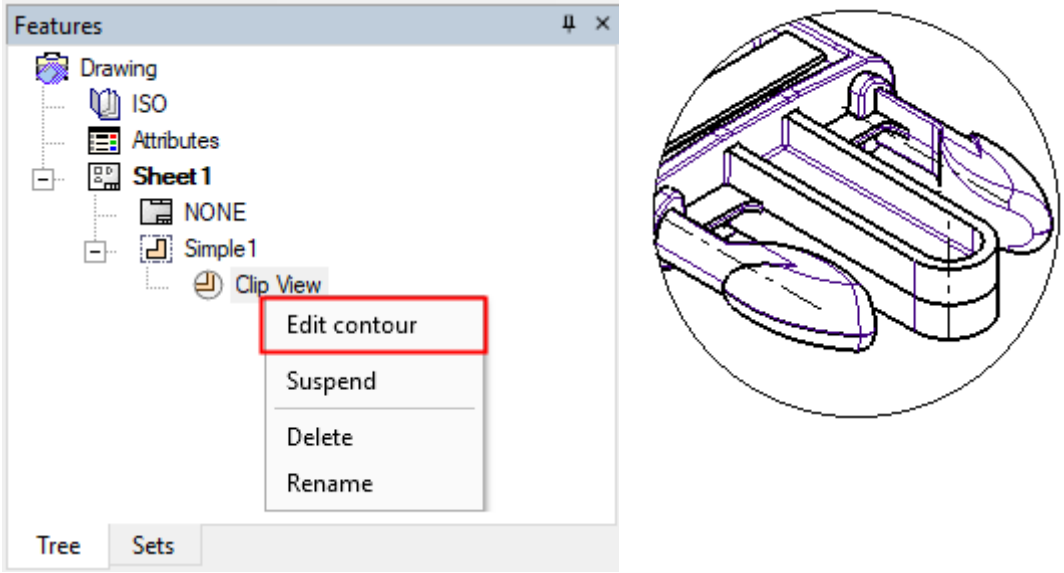
7.3 Clipped View

Views can now be clipped using the new Clip View tool.

It works similarly to the Local Section tool. A Contour must first be defined, and then selected as input for the Clip View tool, defining the boundaries of the clipping.



The clip view is created as a child of the original view and deleting it will reverse the clipping operation. The clipped result can be edited by right-clicking the Clip View item in the Drawing Tree.

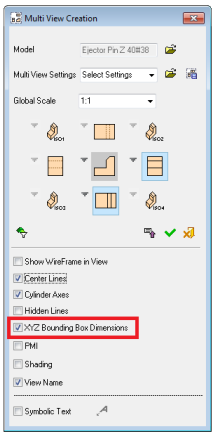


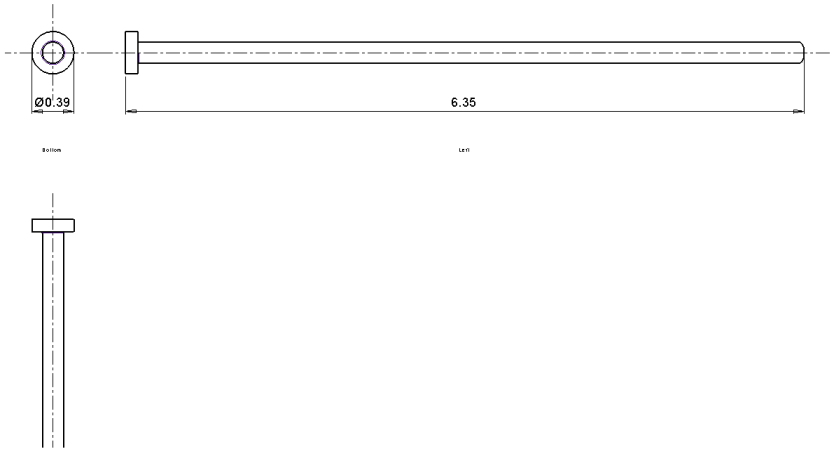
7.4 Multi View – Bounding Box Dimensions

When creating a multi view, a new option allows creating the 3 bounding box dimensions of the part (X,Y,Z).

The dimensions are normally created on the top view (if it is created) and one perpendicular view. In the case of revolved parts, the system creates a single diameter dimension instead of the X and Y dimensions.

This is especially useful when creating a multi-frame drawing holding multiple parts.

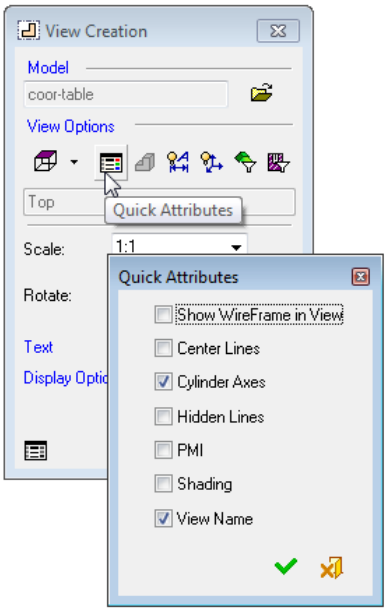




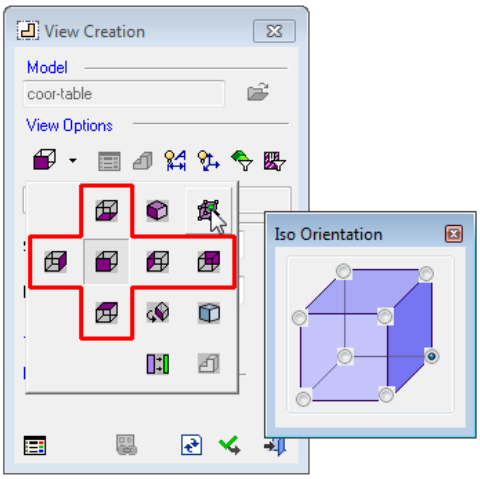
7.5 Implement multi view options in a single view

The Multi View Creation dialog has some enhanced options that were missing in the View Creation dialog, and were now added to enhance uniformity between the tools.

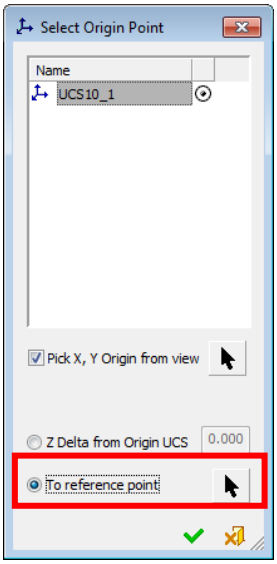
A new option called Quick Attributes opens a list of the attributes available in the Multi View Creation dialog.
This is only available before the view was created. After the view is created these parameters can be controlled through the View Attribute manager.



In the Simple View Projection Type dialog, the –Z Iso option was replaced with a new option allowing the user to choose the desired iso orientation (as is available in the Multi View dialog).



7.6 Define depth of origin point using another view

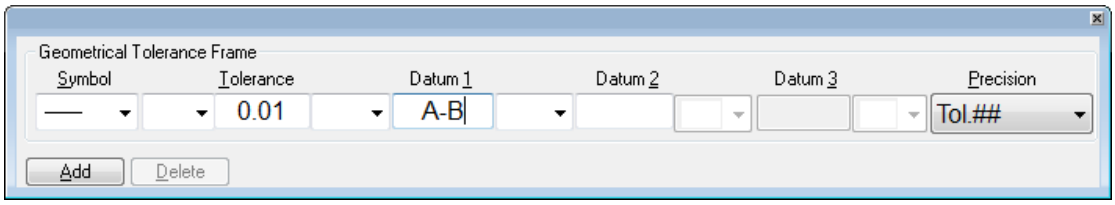


In the Table of Holes and Label of Holes tools, the origin point can either be defined by selecting a UCS, or the XY origin point can be picked on the view (with the Z origin defined by a delta value from the Origin UCS).

The Z origin can now also be defined by selecting a reference point on a view perpendicular to the view on which the TOH or LOH are created, which is often more convenient than defining a delta.

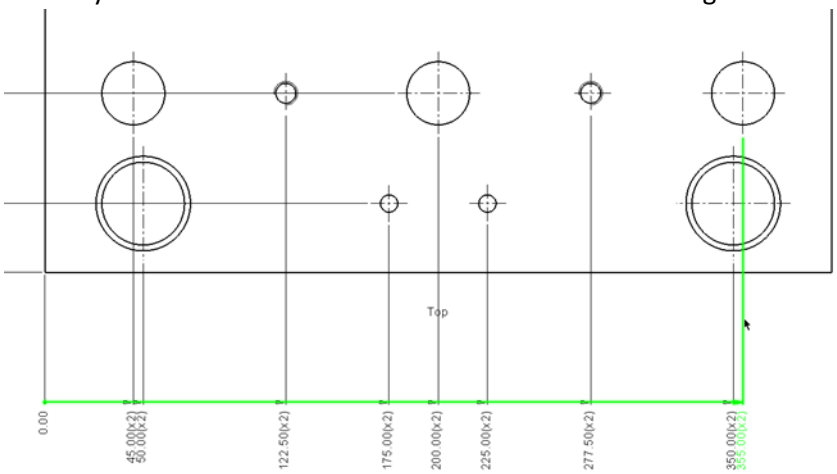
7.7 More Characters in Geo Tol and Geo Datum

The Datum boxes in the Geometrical Tolerance and Geometrical Datum tools can now show up to 5 characters allowing for more elaborate datum definition.



7.8 Start Center lines automatic ordinate at line end

When creating ordinate dimensions on Center Lines, the dimension lines will start from the end of the center line, rather than the center of the circle, to create a clearer view. Note that this only works when Center Lines are selected in the dialog and Circles are not.



7.9 Insert Picture – support PNG

When using the Insert Picture tool to insert pictures to a drawing, you can now also use pictures of the PNG format on top of the JPG format that has always been available.

8 General Enhancements

8.1 Data Management

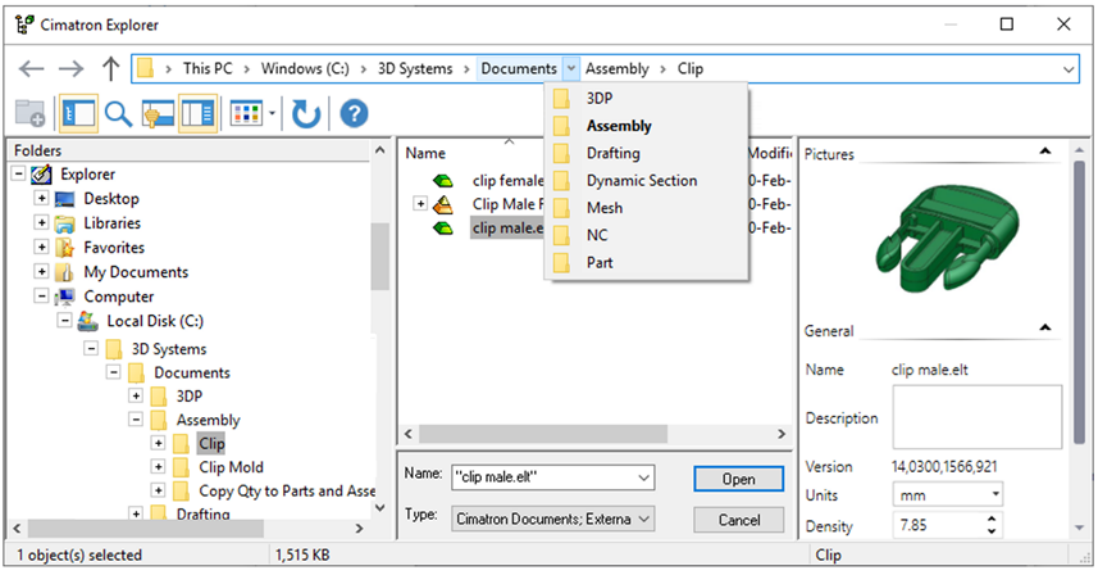
8.1.1 Browser Improvements

The “Bread crumbs” functionality offered in the Windows browser was added to the address bar. Bread crumbs is the separation of the path shown in the address bar, to a series of folder names that can be clicked to go up the path folders or opened to see the sub-folders under each folder. This behavior is more standard with current browsers and more familiar.

The look of the different icons has also been improved to enhance clarity. In addition the Cut, Copy, Paste and Delete icons have been removed to resemble the Windows Browser.

The top filter has been removed when opening or loading a file. All filtering operations can now be performed using the bottom filter.

The name of the “root” of the tree has been changed.



Note that the browser can now open and save Data Interface formats. See more on the Data Interface section on page 76.

8.1.2 Save part images when saving an assembly

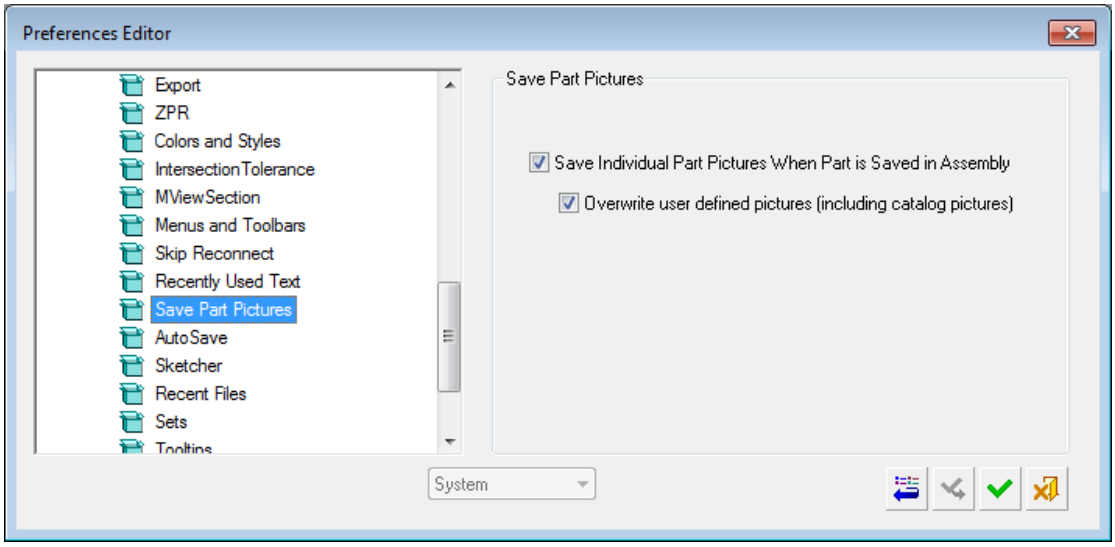
You may now save images of parts when saving the assembly. This will serve to show a preview picture of the part when clicking it in the file browser.

Only parts that have had their geometry or attributes (including hide/show of entities or color change) modified will have their pictures saved when the assembly is saved. Obviously, parts that have not been changed will not be saved at all and their picture will not be saved.

A new preference option controls whether part images will be saved. Note that this takes a small toll on save performance.

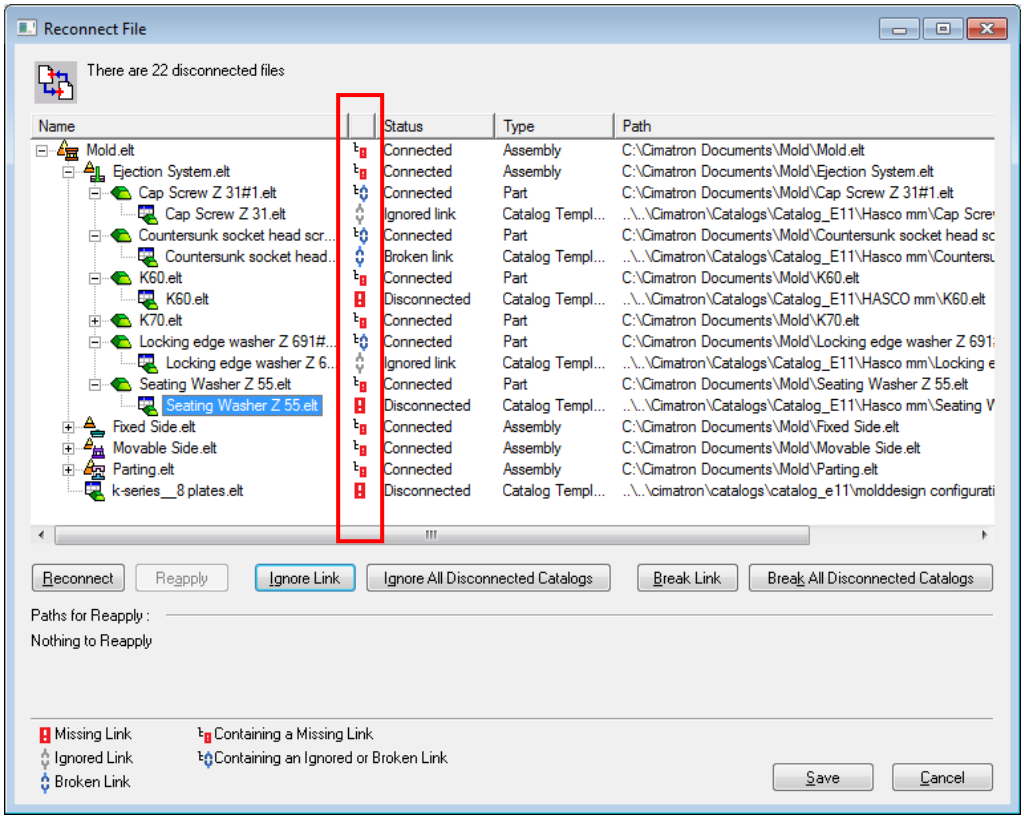
Another option determines if parts that have a manually created image (meaning that it was created with the Add Picture tool) will be overwritten. This is particularly relevant to parts coming from a catalog. When this option is OFF, they will retain their original catalog picture. When it is ON – if they were changed, the change will be reflected in their picture.

By default both options are turned ON.



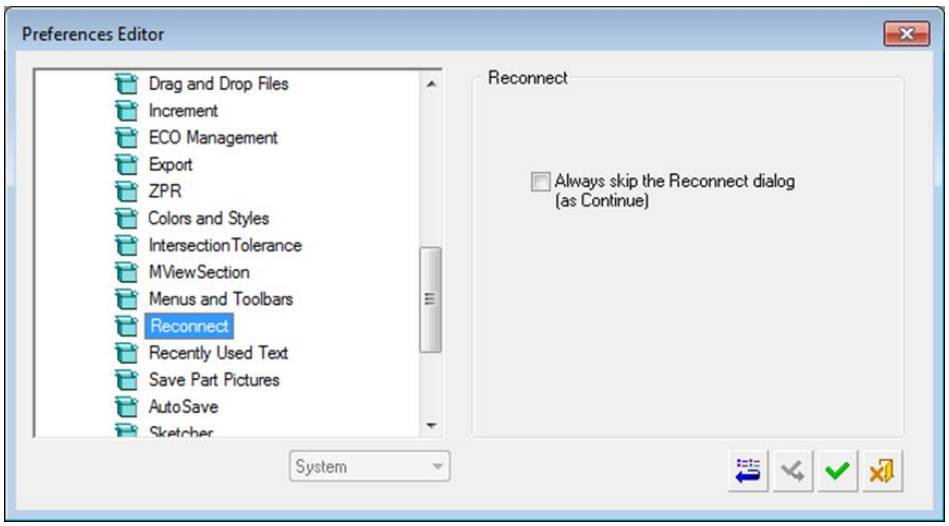
8.1.3 Reconnect icons in a separate column to make them bigger and clearer

The status icons in the Reconnect dialog have been put in their own column to make them bigger and clearer.



8.1.4 Do not open the Reconnect dialog

For some users the Reconnect dialog that opens up when you open a dialog with missing connections, is a redundant nuisance. A new option in the Preference dialog allows not having it open by the system. You can still open it manually by right clicking a file and selecting the Reconnect option from the context menu.



8.1.5 Concurrency – deal with duplicate names

When running a concurrency project, it is quite likely that two users will create files with similar names (for example – Part1). When the first user checks in such a file, the part on which the other user is working will have the part marked as an earlier or later version of the same file, although the files are not related in any way.

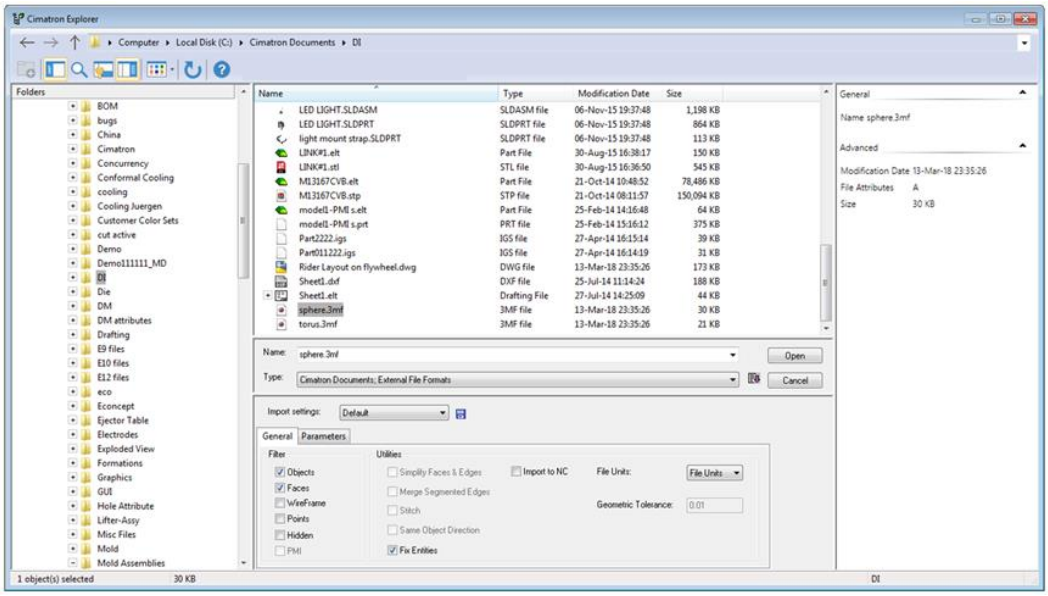
In order to solve this conundrum, the system now pays attention as to how and when the file was created. When encountering such inconsistency, it will let the user know that the two files have the same name but that they are not two versions of the same file.

8.2 Data Interface

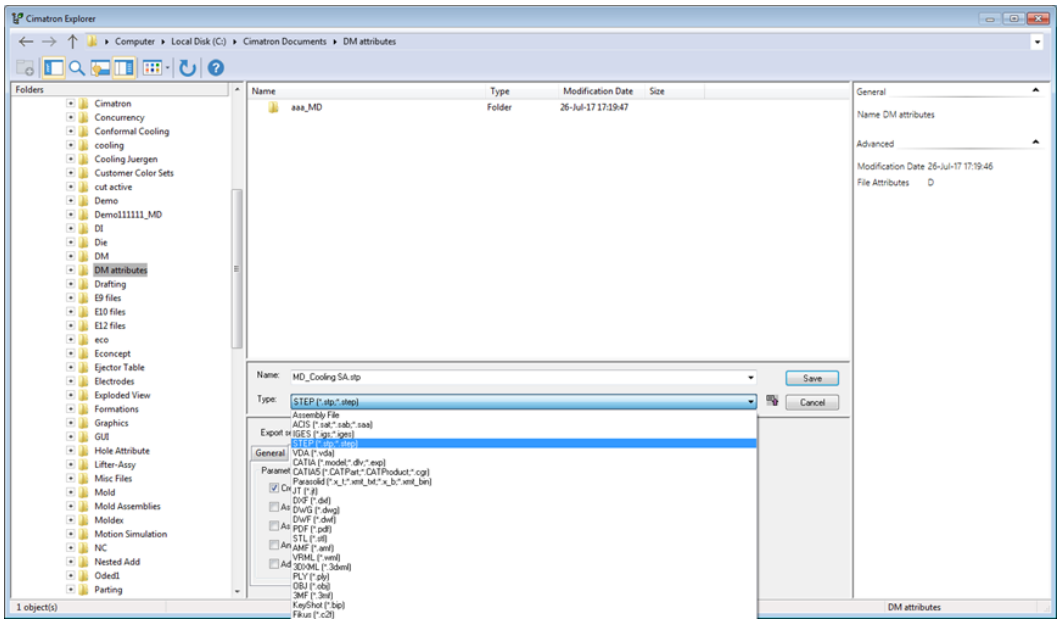
8.2.1 Import and Export using the Cimatron Explorer

The Cimatron Explorer can now be used to perform import and export operations with greater ease and with standard behavior.

When opening a file you may select an external format file and then an expand button will appear next to the Type dropdown menu. Clicking it will open the familiar parameters options of the Import Dialog allowing you to control the import options.



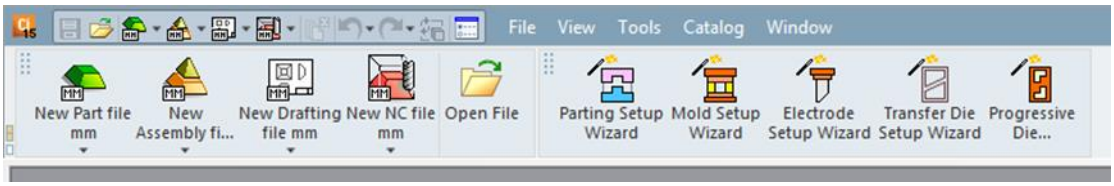
Similarly, when using the Save As tool, the type dropdown menu allows you to select an external format to which to export the file, and again, the extend button will allow you to control export parameters.



Note that not all operations are available with the new interface; namely, the ability to import and export multiple files by batch. For these operations, use the regular Import and Export tools.

The old tools will also appear in different import operations such as the import options in Setup Wizards. They will be replaced with the browser soon.

Note that the old Import and Export commands have been removed from the top Toolbar of the system when no file is open. The commands are still available through the menu.



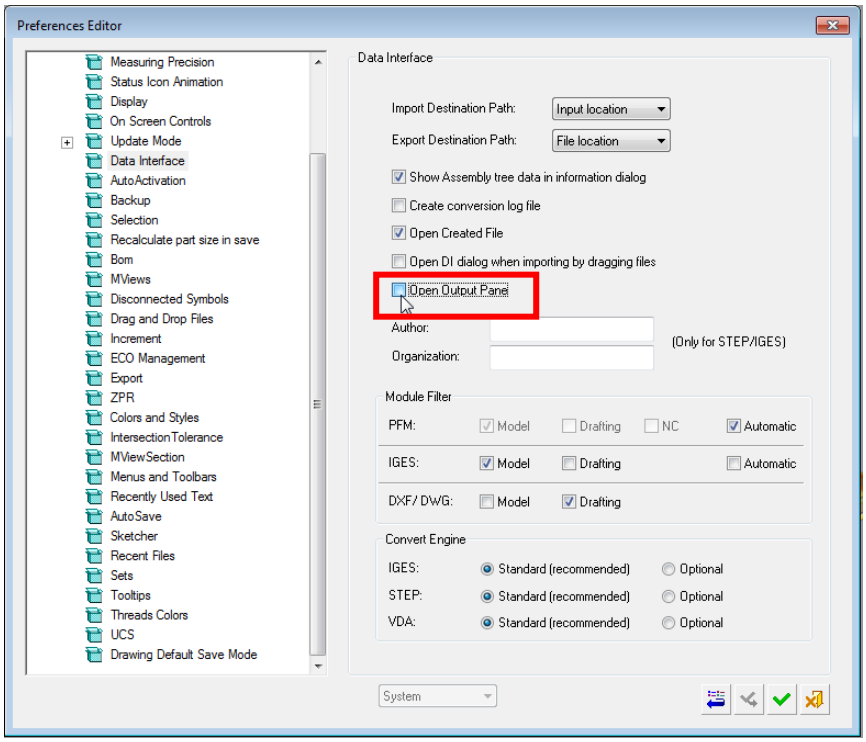
8.2.2 Imported file created as temporary file

In order to avoid creating unnecessary files, imported files are now created as temporary files – which means that when you import a file, the created part/assembly/drawing are not saved to the disk. They behave the same way as when creating a New File, and will not be saved to the disk unless you invoke the Save command. This is useful for when you only want to view a file – you may open it (import) and then close it without saving and no file is created on the disk. This is also the standard way of operation for other files in the system and for most software. Note that this only applies to files opened using the browser. The regular Import tool works the same way it always did.

8.2.3 Notify the user when DI operations fail

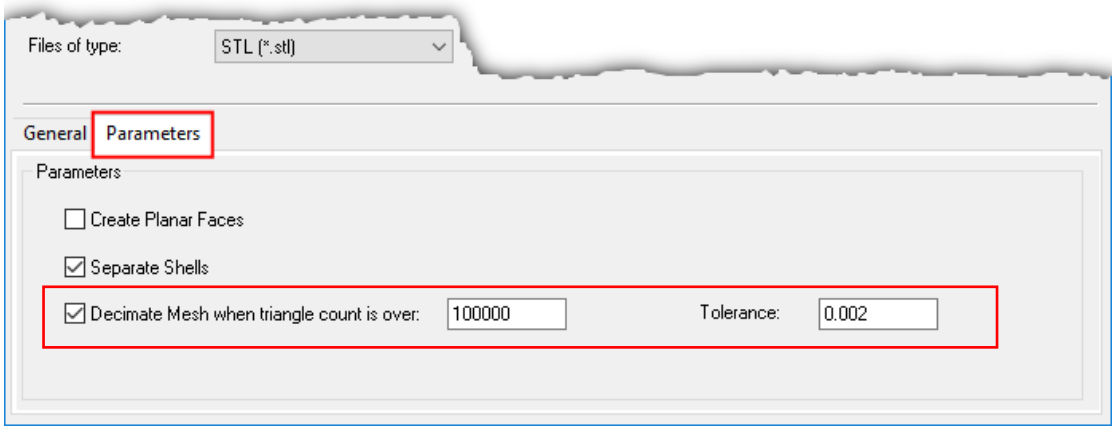
In order to let you know more clearly that a DI operation has failed, the output pane is no longer opened when importing or exporting files from an external file. Instead, a warning message pops up if the conversion operation fails. The message details information about the failure.

Note that control was added to the preference allowing to open the output pane as before.



8.2.4 Decimate Mesh when triangle count is over

The **Decimate Mesh when triangle count is over** checkbox option has been added to the Import Data Interface dialogs for the following Mesh formats: 3MF, OBJ, PLY, STL allowing to perform the decimate for these formats as well.



When this checkbox is ON, this reduces the mesh size to improve mesh performance on support and other operations. The meshes that are decimated are those that exceed the defined Triangle Count and Tolerance thresholds. The checkbox is turned ON by default.
The Number of Triangles default is 100,000
The Tolerance default is 0.002mm (inch 0.0001) Min. Tol – 0.0001 mm Max. Tol: 10 mm
When this checkbox is OFF, the Triangle Count and Tolerance fields are grayed out.

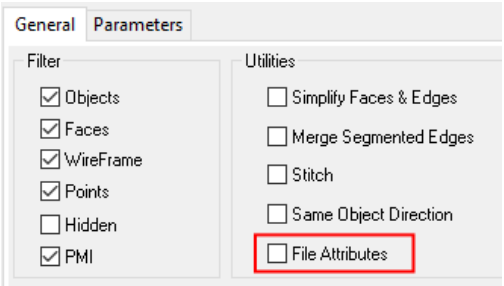
8.2.5 Read in pictures from DWG and DXF

DWG and DXF files with drawings in them will now be read properly and the pictures will appear in the right location on the drawing.

8.2.6 File Attributes

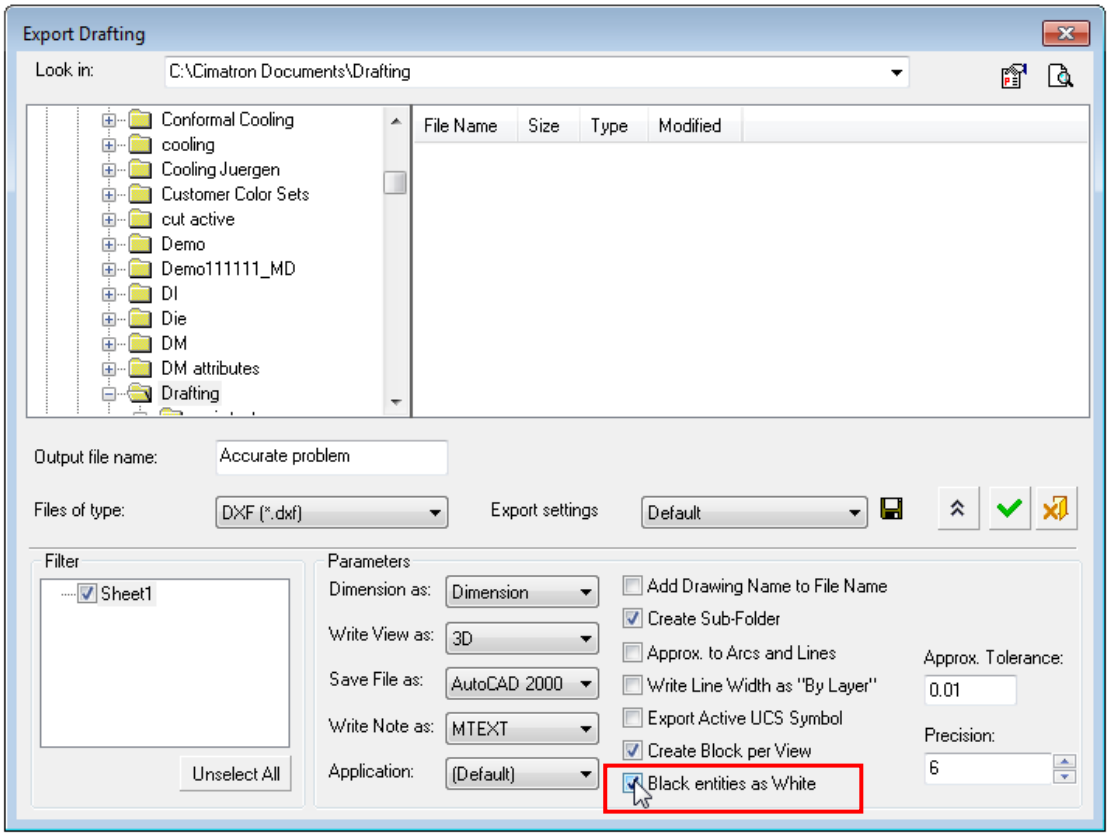
The File Attributes checkbox has been added to import dialogs.

When this checkbox is ON ☑, during the import process of converting the external file format into the Cimatron format, all external file format attributes (such as BOM attributes) will added to the Cimatron Attribute Translation Map resource file (AttrTrnMap.xml).
Default = OFF ☐.



8.2.7 Add control over turning black edges to white when writing to DWG/DXF

When converting to DWG or DXF, black edges where turned to white automatically. There is now control over this operation in the export dialog allowing to allow viewing the result on a white or black background.

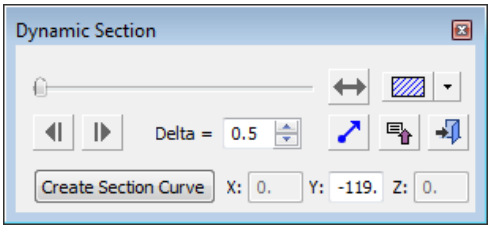


8.3 Graphics

8.3.1 Dynamic Section

The new Dynamic Section tool replaces the existing one allowing you to leave the section mode on and continue working and perform any operation, so that you can better view hidden areas.

The tool offers the same capabilities offered by the previous tool.



The new function is displayed as a floating dialog. By default, the dialog is displayed docked to the tree pane, above the top tree. It can be dragged to another location.

The Step Forward and Step Backwards button allow you to progress the section plane by the delta set next to them.

The default section direction is the Z direction. In order to change it, click the Arrow icon and select a planner face or use the direction arrow to set the direction normal to the section plane. The selected direction can be changed by clicking the Flip button.

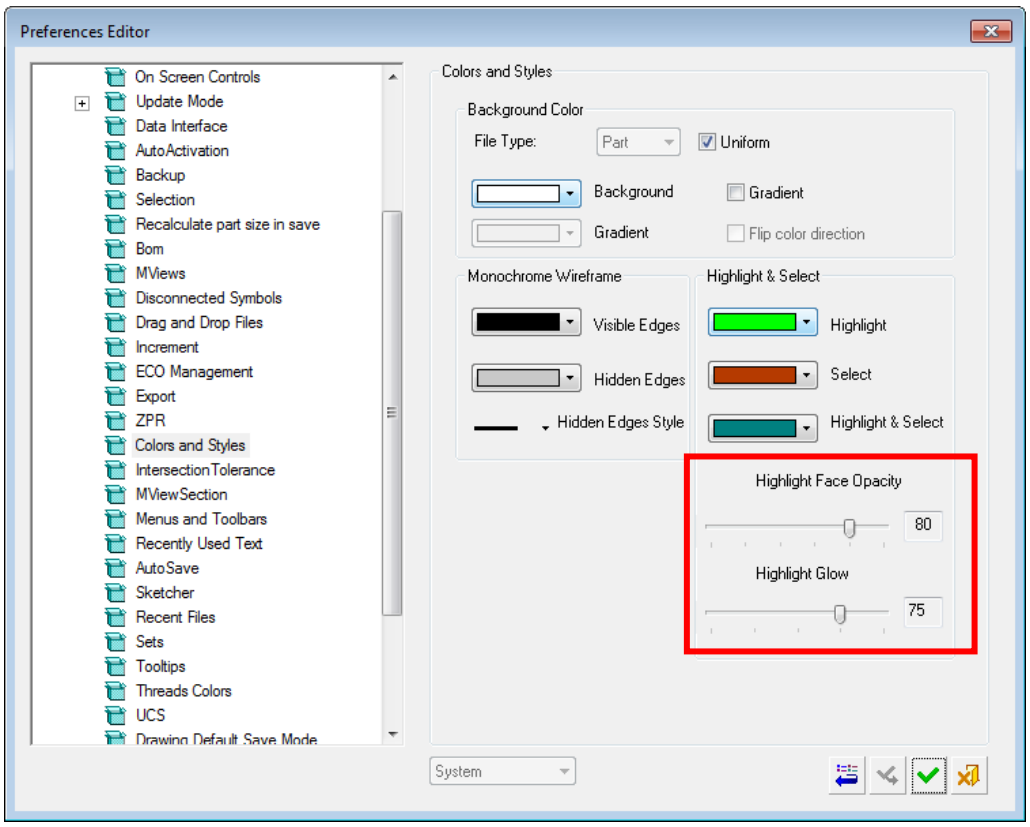
The rendering of the section plane can be set with the render dropdown to Hatch, Transparent Hatch, Shaded Fill, Transparent Fill, or no fill.

In order to create a section curve, click the expand button and then click the Create Section Curve button. The X Y Z fields next to it allow you to set the precise location of the section plane before creating the curve.

8.3.2 Highlight style

Until now, the user had control over the opacity of highlighted faces.

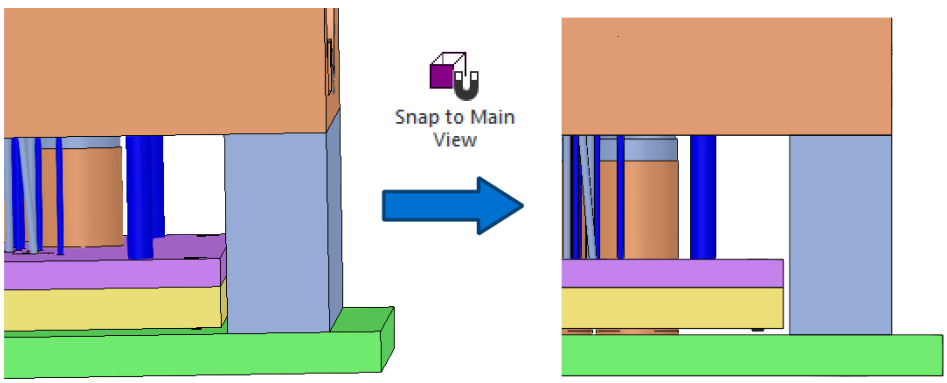
A control has been added also on the glow level given to highlighted entities.



8.3.3 Snap to Main View

When your viewing orientation is close to one of the main views (for example almost on the Front view) and you want to get it to the closest main view without having to click the interactive UCS, you may use the new Snap to Main View command. It will rotate the view to the closest main view without changing the zoom level.

For those who want to use this option, it is recommended to assign a shortcut key to it.

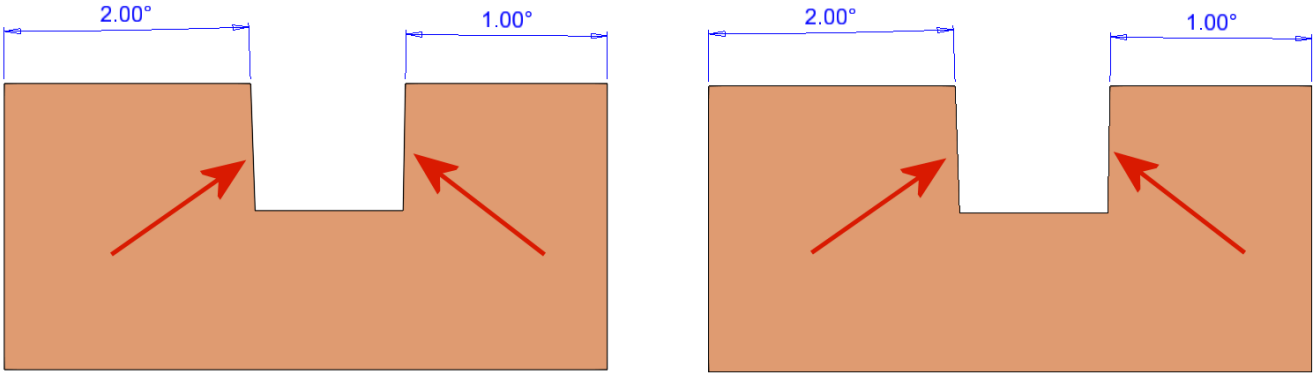


8.3.4 Display Jagged Edges

In the past, diagonal lines used to get a jagged look made out of clear pixels. In all modern software, including Cimatron, when creating the graphic display of edges and curves, they are always smoothed so that they look nice and continuous (also known as anti-aliasing).

However, this makes it quite hard to see if a line is horizontal or vertical, or has a slight angle. For that purpose only, the old fashion display was more useful, and when it comes to mold design and detecting whether a face got a draft angle or not – this capability is quite significant.

In Cimatron 15, we've added a display option allowing you to view edges as smoothed or jagged.

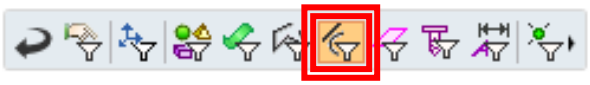


Smoothed (anti-aliased) edges on the left, and as Jagged edges on the right

8.3.5 Filter Improvements

8.3.5.1 Select a single filter

On the main filter toolbar you may now double click a filter to select it and unselect all other filters. For example – if you only want to have Edges & Curves filter turned on – double click it and all other filters will be turned off.



To return to the default state of the filters, click the Reset Selection Filters button.

8.3.5.2 Point by Intersection

The Intersection Point filter now allows the selection of 2 axes.

9 NC

9.1 Turning Application

9.1.1 Overview

There is a new application for creating Turning procedures (lathe), inside the Cimatron CAM environment. It is aimed to support milling and turning together, on Mill-Turn machines and milling machines with turning capabilities.

The Turning application uses GibbsCAM as the engine for the Turning operations. The system supports 2 axis machining with full functionality for roughing, high-speed roughing, contouring, threading, center drilling, tapping and boring.

In Turning, the workpiece is placed on a spindle spinning around the Z axis of the reference UCS, which is defined in the NC Setup. The tool, which is not spinning during the procedure, can be positioned in any fixed orientation within the work envelope of the 5X machine, including a different orientation around the tool's axis.

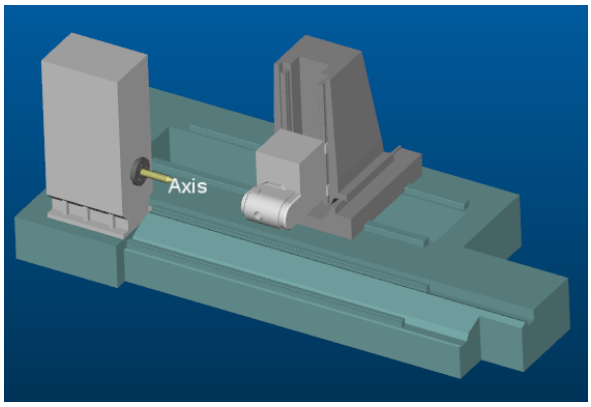
The Turning procedures are integrated into the entire NC process, and can be combined with milling and drilling procedures. The stock is updated throughout the process both for milling and turning.

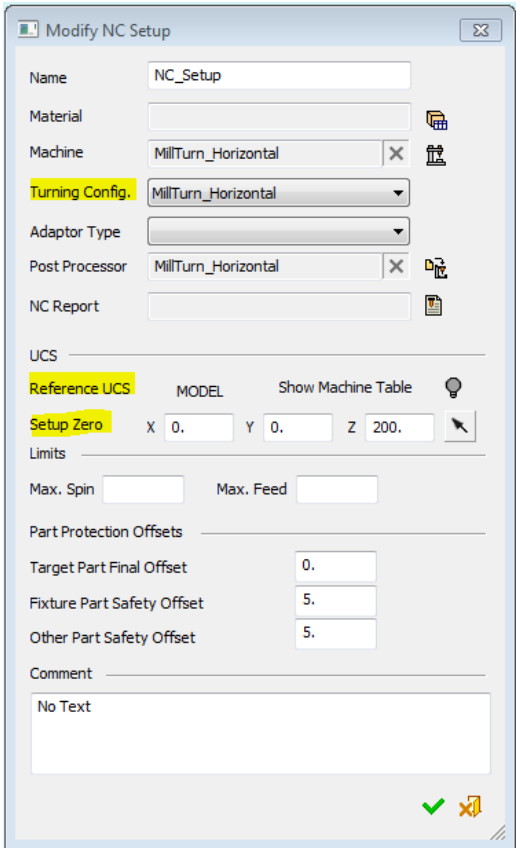
9.1.2 Workflow

The workflow consists of 6 steps: Defining the NC Setup, creating Turning cutters, Defining the Stock, Part and Procedures, and finally review and output of the toolpath.

9.1.2.1 NC Setup and Configuration

The user must define the machine and the Turning configuration prior to creating a Turning cutter or procedure. The Turning configuration is selected in the NC Setup:





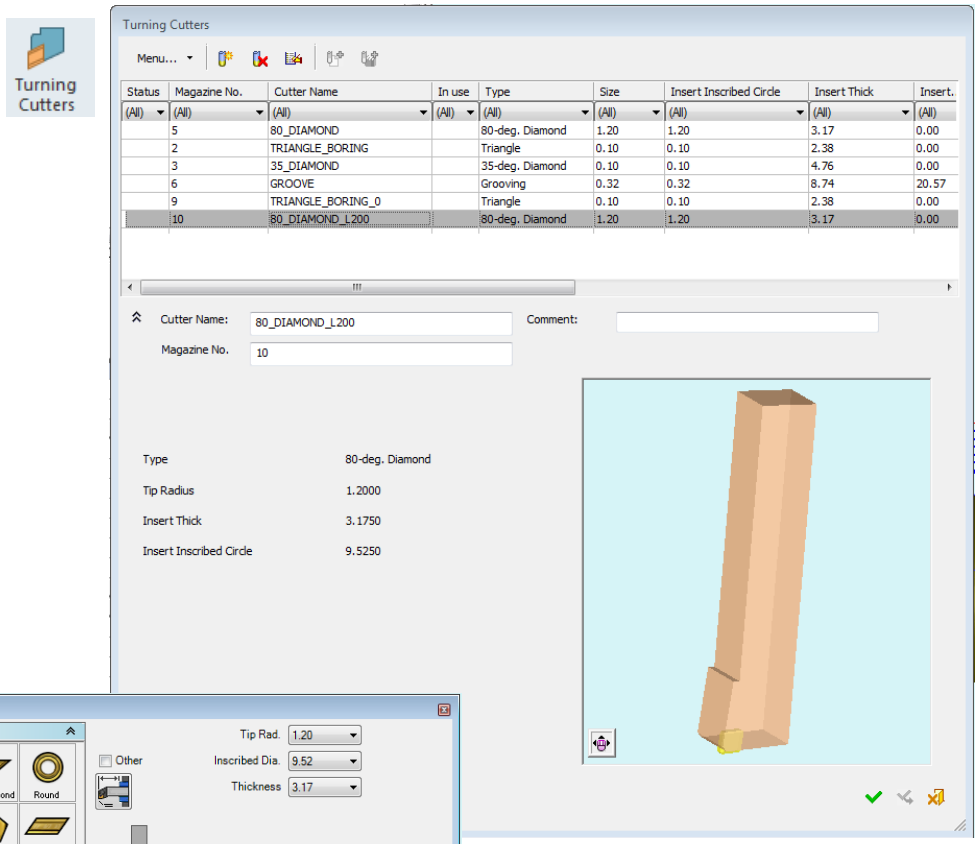
Notes:

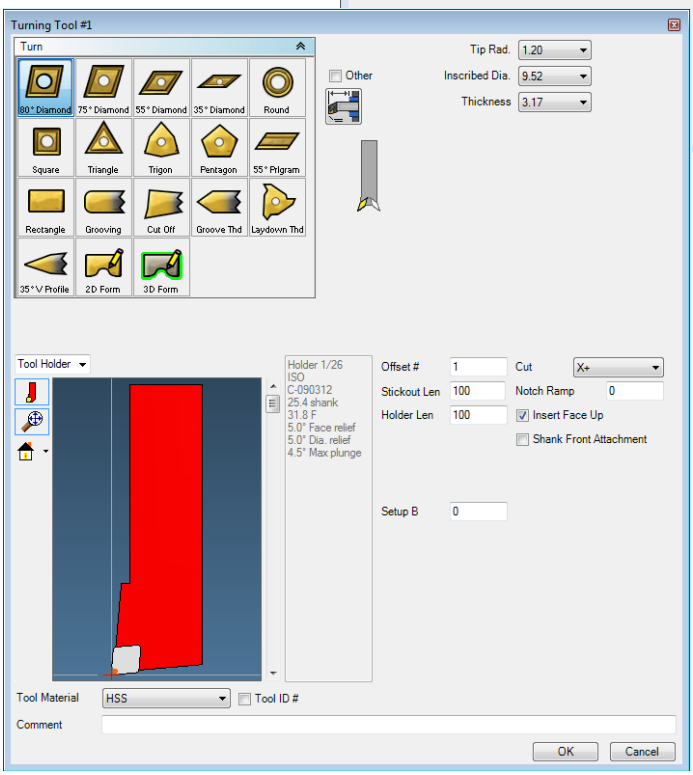
- *In this version, we support machines with one turning spindle and one tool changer.*
- *Machine simulator is supported for Automatic Tool Changer (ATC) only. There is no simulation of Turrets.*
- *The Setup Zero position must be on the rotary axis of the turning spindle.*

9.1.2.2 *Turning Cutters*

Turning cutters should be defined separately using the Turning cutter table.

For creating a new Turning cutter, or editing an existing one, a GibbsCAM dialog is used. This is a graphical user interface to define tools, which shows the user key dimensions to allow easy and error free tooling creation. The system supports a wide variety of predefined common tool shapes, as well as user-defined form tools.

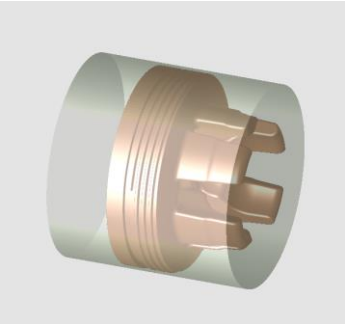




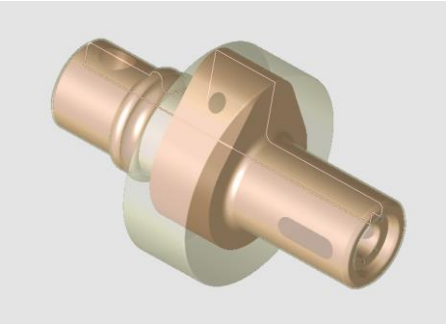
9.1.2.3 Stock Definition and Update

a. New stock type can be defined – Stock by Revolve.
   Four options are available:
   - Simple Cylinder
   - Bounding Cylinder
   - By Contour
   - By Spun Silhouette (of the surfaces)

   The stock is a revolved body, defined by selected surfaces with an offset. The revolve axis is always identical to the Z axis of its UCS.



Bounding Cylinder

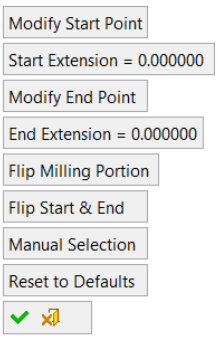


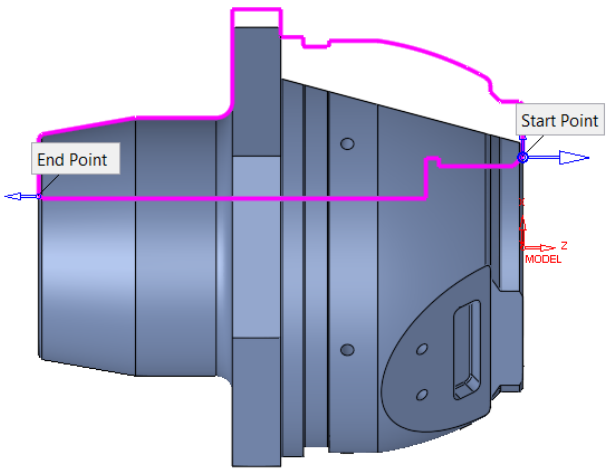
Revolve by Spun Silhouette

b. The Turning rough procedures take into consideration the remaining stock before the procedure for creating an easy and efficient Turning toolpath.
c. The stock model is fully updated throughout the process, both for Turning and milling procedures.

9.1.2.4 Geometry and Contour Creation

a. The spun silhouette of the part is selected automatically to be the default contour for the turning procedure, which makes the programming process of the Turning fast and easy.
b. The start and end points of the contour, as well as the contour's extension and direction, can be defined easily in the selection tool.
c. The contour can be changed manually using all of Cimatron's CAD capabilities and contour creation tools.

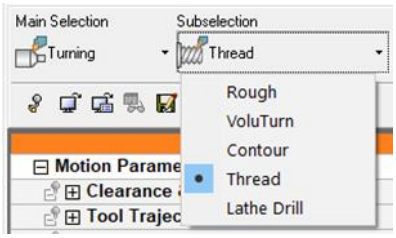




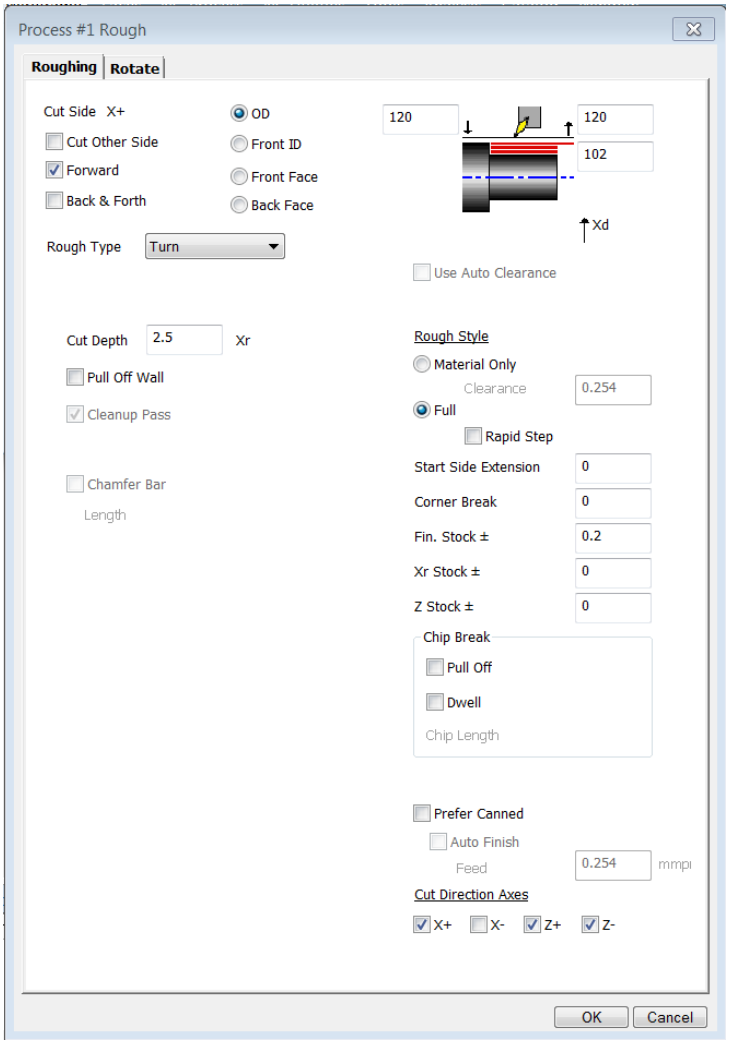
9.1.2.5 *Turning Procedures*

Cimatron 15 supports a large variety of Turning procedures. It is possible to define outside and inside diameter cutting as well as front and back facing. The user can define toolpaths in a way that the tool cuts in both, forward and reverse directions or in one direction only. The following Turning procedures are supported:

- **Rough** – include turning, plunging and grooving
- **VoluTurn** –high-speed roughing using round insert
- **Contour** –for finishing operations
- **Thread** – for lathe threading
- **Lathe Drill** – for center holes (X0, Y0 coordinates)



a. The cutting parameters and the procedure orientation are defined using GibbsCAM dialogs, such as the following:

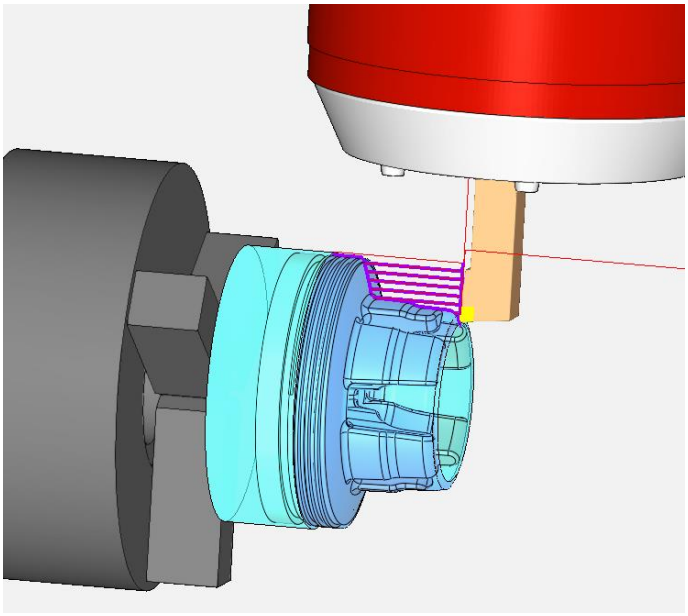


9.1.2.6 Review and Output Toolpath

Once the toolpath is created (the procedure was executed), it is a part of the Cimatron CAM environment, and all operations on the toolpath are supported, including Navigator, Simulator & Machine Simulator, Post and Report.

Notes:

- *Motion Editor and Transformation are not supported at this stage.*
- *Some changes are required in the post and the report template in order to support Turning.*



9.1.2.7 License: Turning Module and Standalone Turning

The Turning application is available for sale in two options:

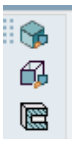
- Turning Module – this is an expansion of the Cimatron NC package or full package. The turning module includes the ability to create turning cutters and turning procedures, counting all options listed above.
  This module is aimed for customers with Mill-Turn machines that would like to benefit from all the machine's capabilities.

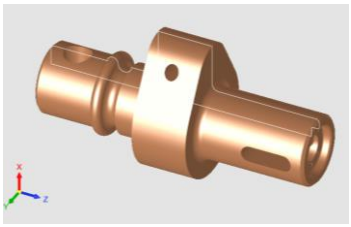
- Standalone Turning – this is a separated base package that does not include the entire Cimatron NC functionalities.
  This package contains the ability to create all types of cutters: turning, milling and drilling, and generate turning procedures. A set of very simple milling and drilling procedures are also available in this package, such as pocket, profile and legacy drill.
  This seat is suitable for customers with simple turning machines (one tool changer and one spindle), that would like to use Cimatron separately for these machines.
  The package includes GPP2 advanced 5X and machine simulation modules.

9.1.3 Custom Toolbar: Lathe Views

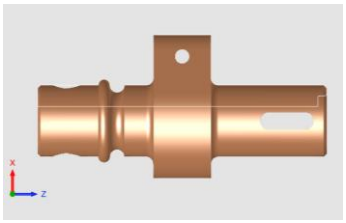
For a simple visibility of the Turning part on the Turning plane (ZX), we added a new toolbar called Lathe Views. The toolbar contains three new commands:

- Lathe Isometric view – for isometric view of the ZX plane
- Lathe Front View – for a front view of the ZX plane
- Lathe Section – for a section view of the part and stock in the ZX plane

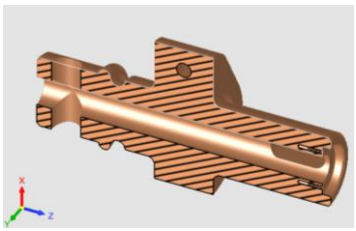




Lathe Isometric View



Lathe Front View



Lathe Section

9.2 5 Axis Machining

9.2.1 3+2 Axis Roughing Automation

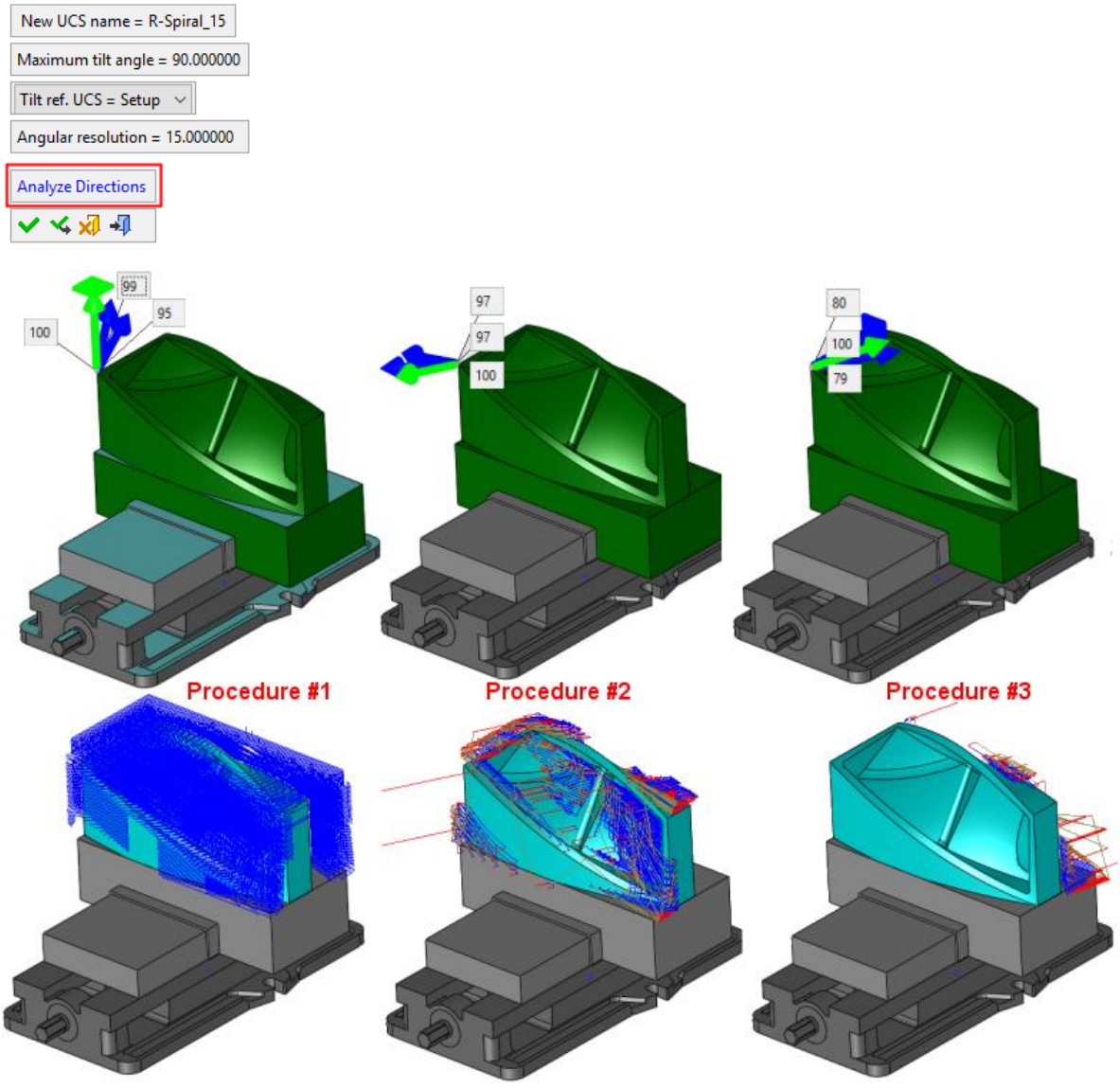
The Rough procedures can analyze the part, stock and cutter and suggest a best milling direction.

The relevant procedures are Rough Spiral and Rough Parallel.

The project has 2 optional levels:

- **Interactive** analysis within the Rough procedure that suggests the UCS for that procedure at interaction time (on the opened procedure).
- **Automatic** analysis within the Rough procedure at run time, during execution.

The examples below show three Rough procedures, using the **Interactive** mode. For each procedure, the user has calculated the preferred direction (top line) and received the best results (bottom line).



The **Automatic** mode in Rough 3+2 axis procedures can be used for creating multiple Rough procedures that will rough the entire part from different directions.

This can be achieved by creating multiple Rough procedures (typically by a template) with the same cutter and setting the Milling Orientation to Automatic. Each procedure in its turn, calculates the best direction for removing the stock, and then calculates the toolpath according to that direction. The remaining stock is then calculated to be used by the next procedure.
Once no significant stock remains, the calculation stops.

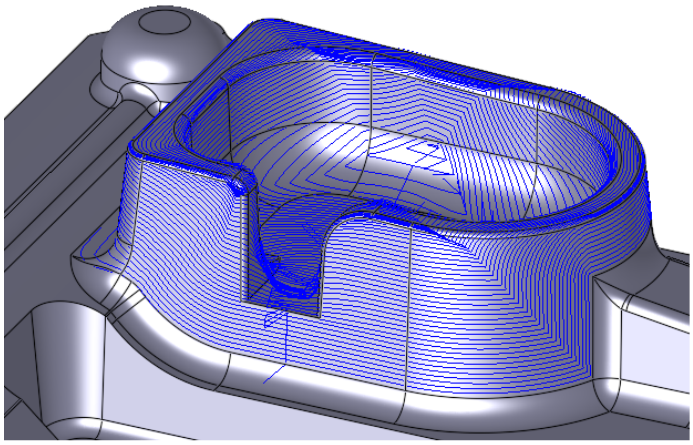
9.2.2 New Geodesic Procedure

A new high quality toolpath calculates an even 3D step over complex parts. It includes support for guide curves, undercut and considers the holder to prevent collisions.

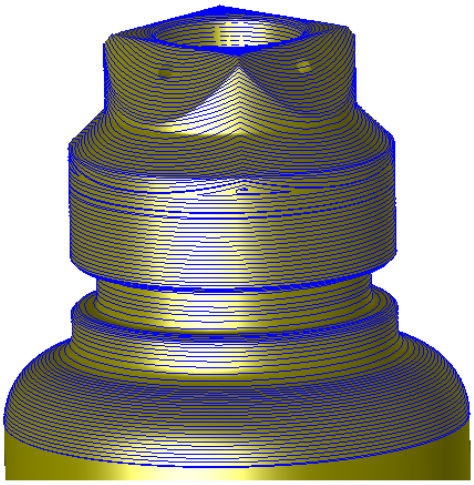
Users can use it for 3 axis, 4 axis and 5 axis toolpaths, typically used for parts that require high quality surface finish such as mold and die cores and/or cases where a continuous toolpath start-to-end without re-entries, is required.

For a 3 axis toolpath, it uses 3D step on multiple surfaces for achieving good surface quality.

For a 5 axis toolpath, it can also mill undercuts as shown in the picture below.



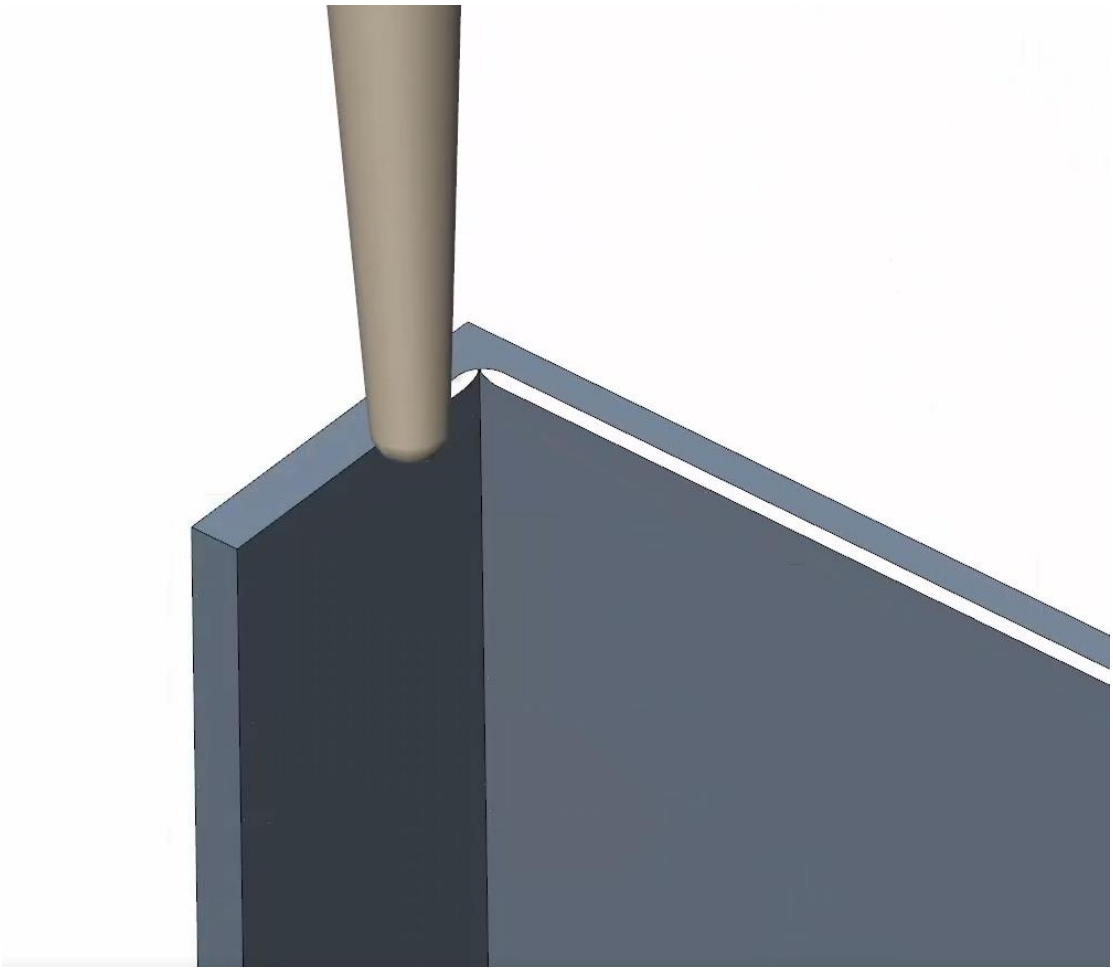
Geodesic 3X



Geodesic 5X

9.2.3 New Deburring Procedure

Automatically program deburring operations by selecting geometry, surfaces or an entire model. The toolpath considers the machine axis limits and holder to create a collision-free toolpath. Users can use it for 3 axis, 4 axis and 5 axis toolpaths.



9.2.4 Enhanced Multi Blade Basic procedure

The Multi Blade Basic procedure does not require a special license and is now available for all users that have the - 5X_AEROSPACE license.

9.2.5 5 Axis Tilting, more control

5 axis Tilting can be calculated for “All Motions” or “Only when Necessary”.

When calculating "For All Motions", a new check box called "Avoid Tilting on Horizontal Planar if not required" is shown, allowing refraining tilting on planar horizontal regions.

9.3 Plate Machining and 2.5 Axis Milling

9.3.1 New Slot Procedure

A new procedure that slots the center line (skeleton) of a slot-shaped pocket.

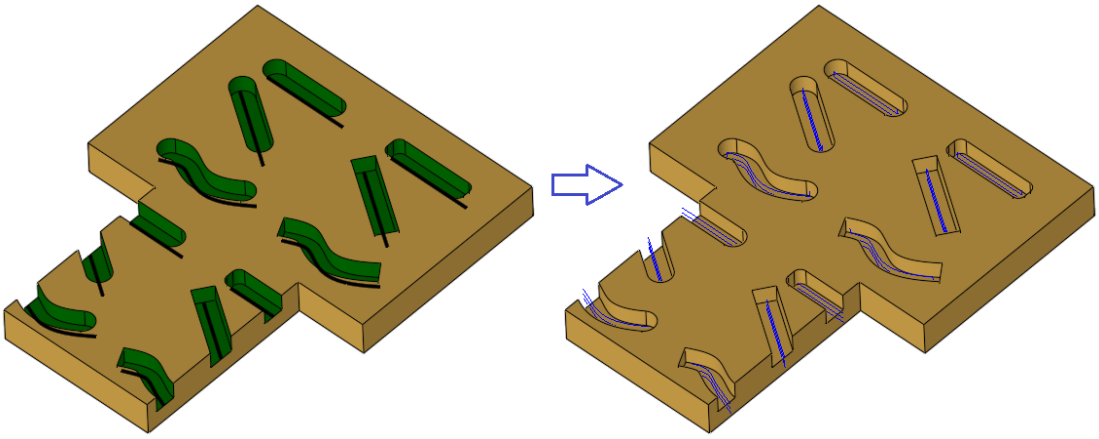
The procedure input may be a pocket as well as any open or closed contour.

Milling a slot may take several stages: Slotting, Roughing, Finishing the walls, Finishing the floor. The new procedure deals with the Slotting stage. The new procedure may be used as a standalone procedure or combined within a template that applies to all stages (maybe with different cutters).

The slotting procedure will:

- Recognize open and closed slot ends.
- Support a few alternative methods:
  - Simple Slotting
  - Trochoidal
- Support plunging (Helical or Ramping).
- The slotting path is based on the Skeleton ability of Cimatron CAD.

Each pocket in the Pocket Manager gets a “Slot likelihood”, ranging between zero (not a slot) and 100 (definitely a slot) defining how much it is a “slot”. The rank is displayed as a new column in the Pocket Manager, called “Slot Likelihood”.



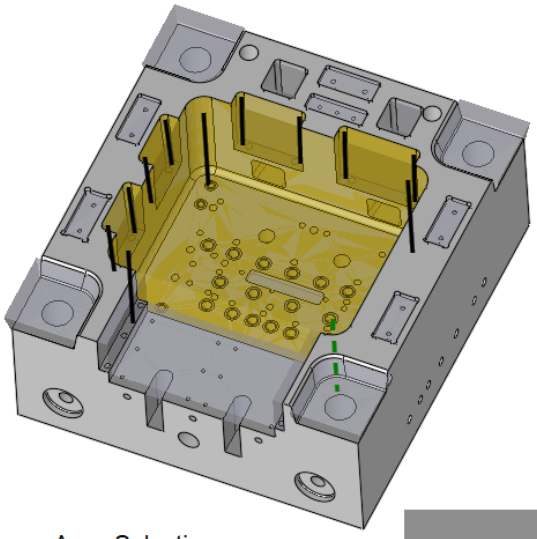
9.3.2 New Corner Plunging Procedure

This new procedure creates plunging motions with a plunge cutter at rounded corners of pockets. The input comes from pockets created by the Pocket Manager.

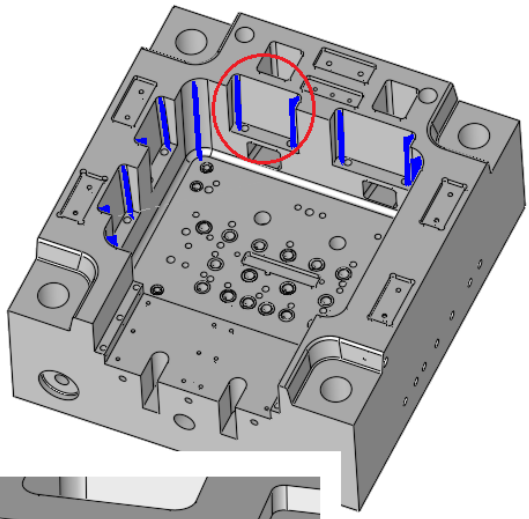
Plunging Motions are parallel to the cylinder of the corner that may be vertical or slanted.

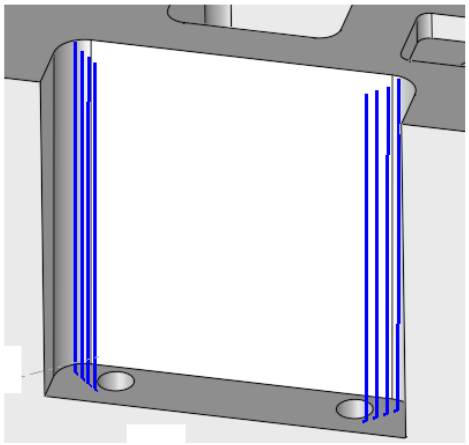
The approach is parallel to the cylinder, retract is horizontal.

The procedure performs a gouge check.



Axes Selection



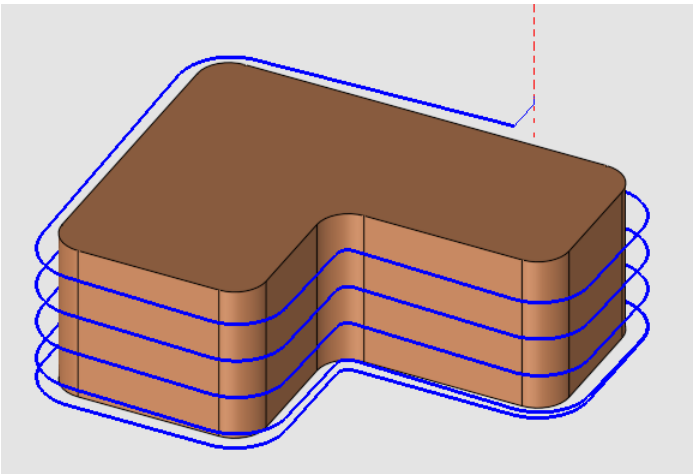


Plunging Toolpath

9.3.3 Helical Milling of Closed Profile

Helical Closed Profile, under the 2.5 Axis main selection. This new procedure works on closed contours.

The procedure allows milling the entire part in a single pass, without approaching and retracting each layer.



9.3.4 New Safe Profile Procedure

These new 2.5 axis profile procedures also implement Check Surfaces, Safe Milling and Holder Usage control to avoid gouging the check surface and avoid holder collisions.

The new procedure allows safer machining. It also allows setting the Z top and the Z bottom of the machining with respect to the check surfaces.

9.3.5 Pocket Manager Enhancements

Creating Pockets on main planes

A new option that allows creating pockets in the main planes of the active UCS.

In previous versions, the user could create pockets in many directions, by using “by Criteria” and only if all directions had a pre-defined UCS.

Now, a new option allows creating pockets in main planes, perpendicular and parallel to the active UCS (Top, Bottom, Front, Back, Left and Right), without creating pre-defined UCSs for each direction in advanced.

9.3.6 Plate Machining Pro seat

The Plate Machining Pro seat is a new saleable item, based on the Plate machining seat.

It includes all the abilities of the regular Plate machining seat and some CAD functions, such as Extrude, Cut, Merge and more. It also adds some NC abilities that are not included in the basic Plate Machining seat, such as 5X Simulator and 4X Automated Drill.

9.4 Programing Automation

9.4.1 3+2 Axis Remachine Automation

The system now suggests directions for selected Remachine Segments for Guided Cleanup.

The user can control the following:

- Snap to existing directions and Snap to existing UCSs.
- Whether to split segments.
- Overlap size.
- Whether to create UCSs for suggested directions.
- Preferred and Maximum angle.

9.4.2 Add Spark Gap to Remachine Segments

The Spark Gap is considered when machining electrodes and acts like a negative offset.

A Spark Gap has been added to the Remachine Segments, allowing the Guided Cleanup procedure to machine electrodes.

9.4.3 Select Z Limits by Criteria

Z-Top and Z-Bottom can now be defined by Criteria (new option) as well as Manually (old option).

This adds flexibility and automation by selecting Z Limits of the machining by entities selected manually or by criteria.

9.4.4 Max-Part-Z – new parameters

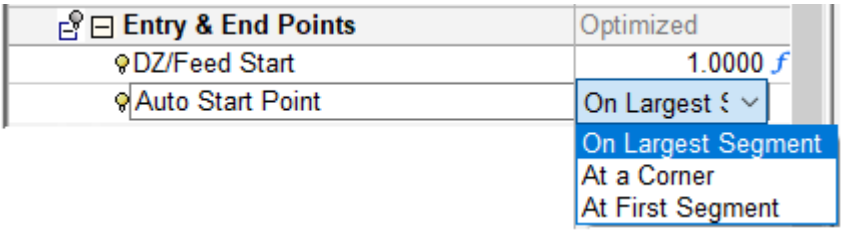
Six new global parameters have been added:

- maxtrgtz, mintrgtz, maxfixtz, minfixtz, maxothrz, minothrz
- They are defined to be the highest Z and the lowest Z, respectively, of all points in the Target, Fixture and Other respectively, as defined in the NC Setup.

9.4.5 Closed Profile Start Point – better user control

Two options are available for selecting the start point of closed 2.5X profile procedures: **Manual** and **Automatic**. These options are available in the following procedures: Profile Closed Contour, Safe Closed Profile, Helical Closed Profile and Chamfer Closed Contour.

1. The **Automatic** mode enables 3 automatic options:
   - On Largest Segment – Machining starts at the middle point of the largest segment.
   - At a Corner – Machining starts at the sharpest outer corner.
   - At First Segment – Machining starts at the middle point of the first segment.



2. The **Manual** selection of the start point is enabled through the Contour Manager. Selecting a point manually overrides all automatic options.

9.4.6 Finish by Layers New Options

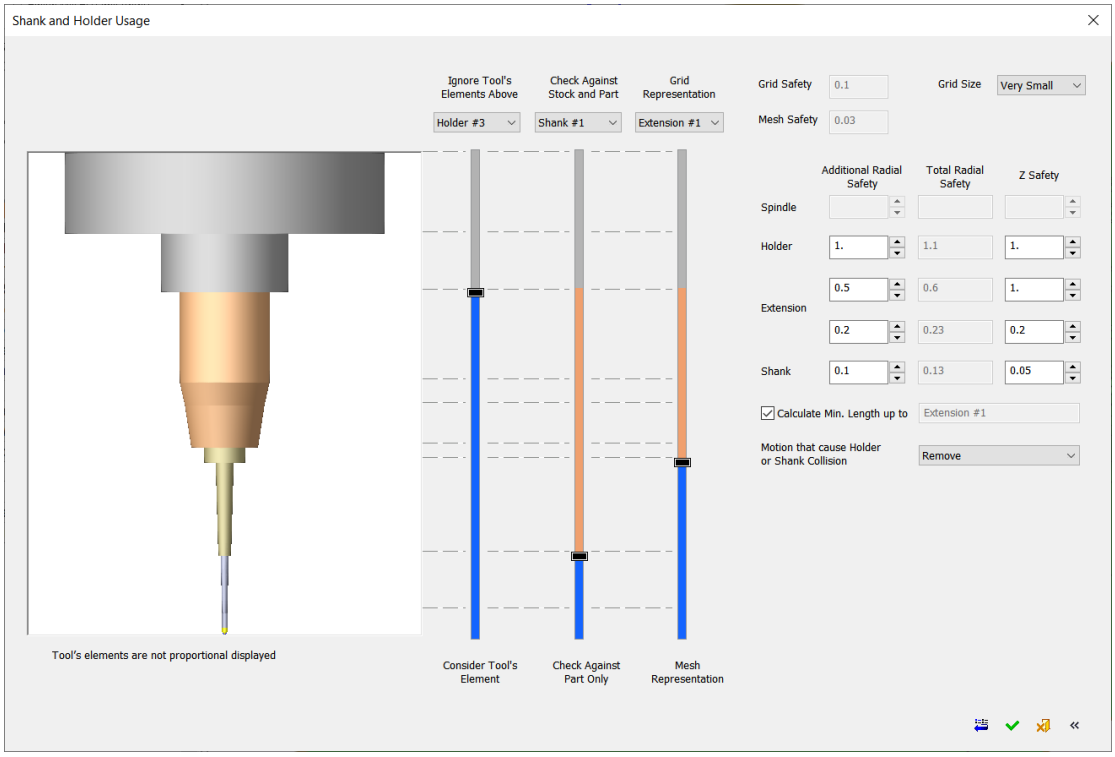
Mill only N bottom layers (and all beside N bottom layers):

In Finish by Layers, a specified number of Bottom layers may now be defined for machining.

9.5 Easier Programing

9.5.1 Shank and Holder Dialog

An advanced mode under the Shank & Holder branch opens a dialog that helps users better understand the meaning of all parameters.



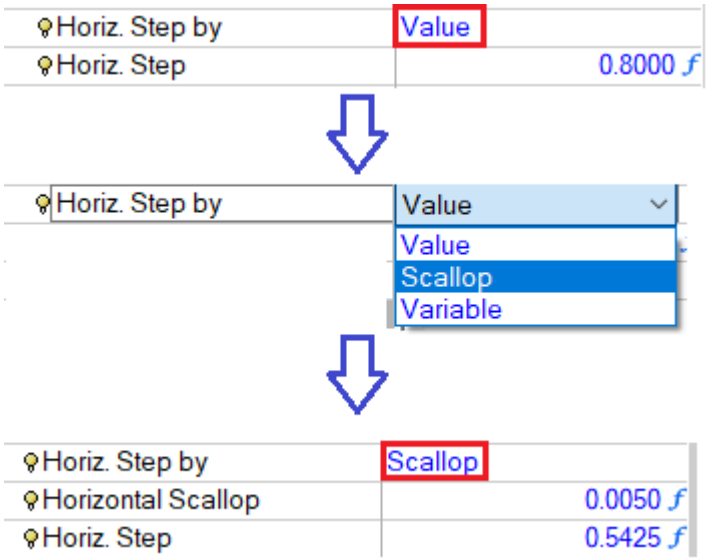
9.5.2 Use scallop in Finish procedures

The scallop defines the surface quality of Finish milling.

Currently, the scallop is defined indirectly by “Horiz. Step”, “Vertical Step” and “3D Step”. To achieve a desired scallop, the user is required to define the steps “off line”.

The new feature allows defining the scallop directly, and the Step size is calculated by the system automatically, depending on the cutter and the average slope.

In the example below, the user asks for a scallop of 0.005 mm and the system calculates the horizontal step, considering the cutter:

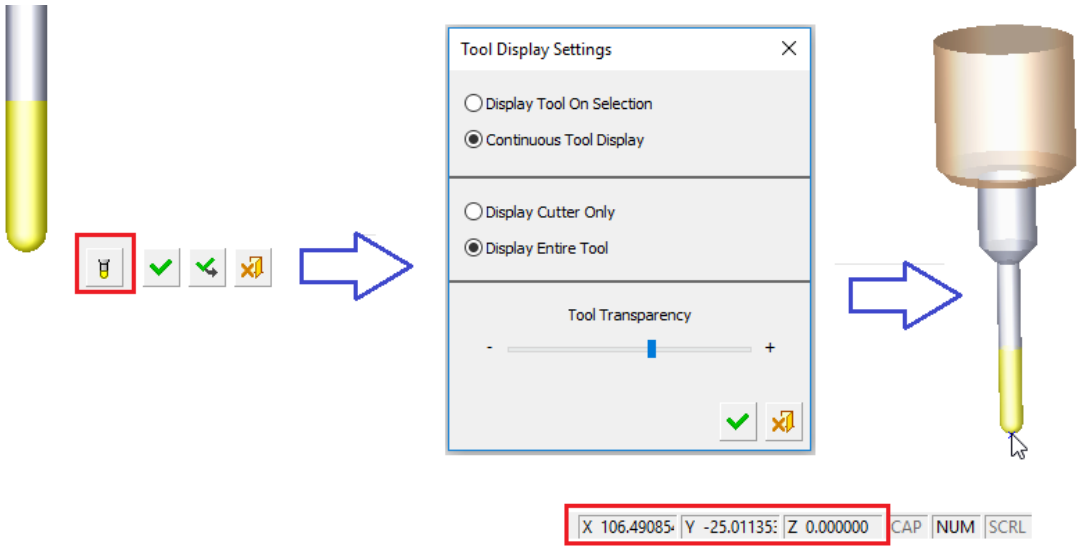


9.5.3 Improved Cutter display within the Cutter Table

When a cutter is selected in the cutter table, it can be displayed in a continuous way, adjusted to the cursor. The cutter can be displayed with or without its holder.

The user controls the transparency of the cutter as well.

In addition, the X, Y and Z coordinates of the tool tip are displayed in the information bar.



9.5.4 Conic tool - better definition

In previous versions, the Cut Length of a Conic tool determined the taper height and so set the cutter geometry.

Now, the Cut Length can be set separately from the taper height, giving more flexibility.

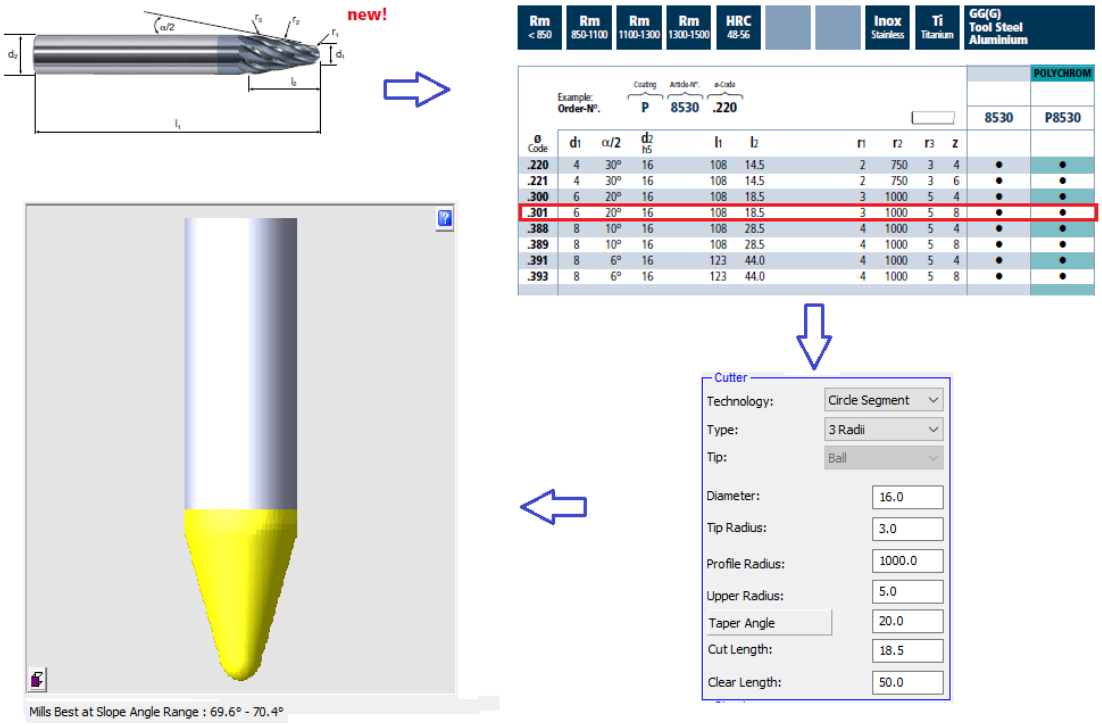
9.6 Faster Machining

9.6.1 Rough Efficiency with AFC

The stock removal rate and the chip load may vary according to the part geometry. AFC can be compensated automatically by controlling the feed rate, resulting in smoother and safer motions, faster Roughing machining time, less drastic changes on the machine spindle and axes and longer tool life. The Tool Load calculations was improved in V15 and it is more accurate and reliable.

9.6.2 New Circle Segment Cutters

Barrel, lens, oval and now also 3 radii segments cutting tools can be utilized by Cimatron toolpath strategies. Using any of these cutters will shorten the machining time for a given scallop while achieving even better surface quality.

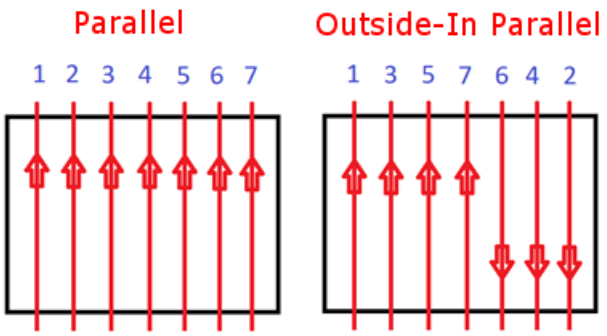


A new Circle Segment cutter from the cutter manufacture is built in Cimatron's cutter table

9.6.3 Facing Efficiency

The Facing procedure is now able to parallel machine bidirectionally, in Climb or Conventional mode, thus shortening machining time.

For example, the milling order of the passes in Climb mode would be as follows (each region is milled in By Pass mode):



9.6.4 Rough Efficiency

The machine time has been reduced by removing “air milling” passes.

9.6.5 Rough Smoothness

Connections in Rough procedures are now smoother, allowing smooth motions of the machine and reducing machining time.

9.6.6 Fast Motions more Control

The following methods may now be selected for calculating the Z height clearance of fast motions:

- **Bounding Box:** Calculation of the Z height for fast motions on a bounding box that contains the 2 connected points. This is used because the exact path between the points in Fast Feed is not certain in most old machines.
- **Straight Line:** Most modern machines do those connections in a straight line and allowing, in many cases, to connect at a lower height.

9.7 Faster Calculation

9.7.1 Rough Performance

The Rough performance has been improved in some special cases; for example, a big cutter, with no holder.

9.7.2 Stock Update Performance

Stock calculation performance and stability has been improved by up to 30%, especially in cases where many small procedures update a complex stock. The new mechanism passes stock data from one procedure to another faster, without losing precision.

9.8 Enhanced User Control

9.8.1 On Machine Inspection – Read back Report

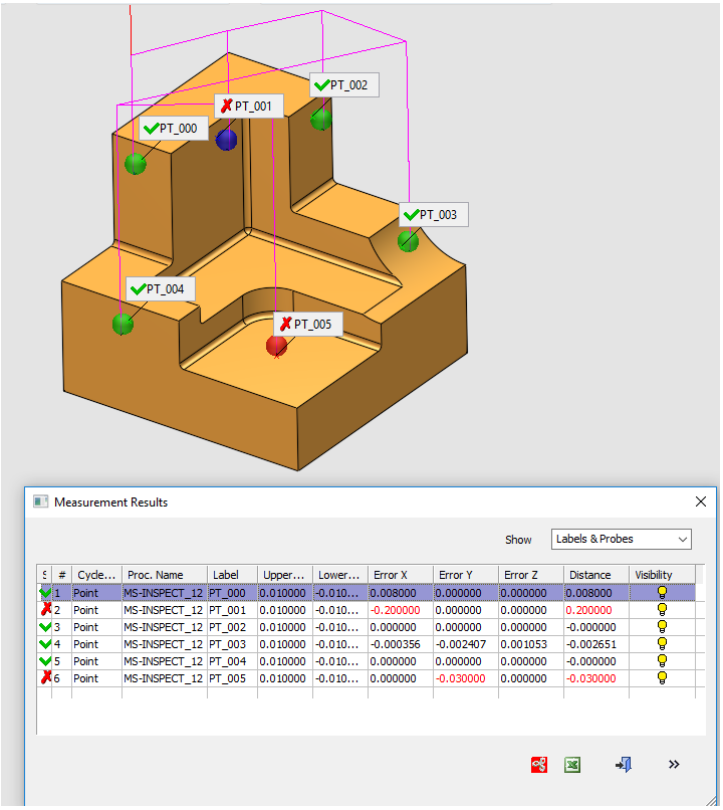
A new option in Cimatron creates a report for On Machine Inspection (OMI) procedures.

Process description:

- The OMI procedure creates measurement motions that are posted to the CNC machine.
- The measurements are executed on the machine.
- The measurement results are imported from the CNC controller back into Cimatron and associated with the OMI procedure.
- A new command: “Import Measurements Results” imports the results file.
- A new command: “Inspection Report” can be invoked by the user.

The results are shown on the graphic screen, with labels and coded colors. The details are displayed in a table.

The user can export the results to Excel and to PDF files.



9.8.2 On Machine inspection – Highlight Probes

The probes associated with selected rows in the Measurement Cycles table are highlighted and vice versa.

9.8.3 Stock and Part information in the NC Report

The user can now select Stock and Part procedures while creating a report and the information about these procedures will be available for output to the report and also for the GPP2 post processor.

Stock information includes the stock type (box, bounding box, by surfaces, revolve), stock size, offset, etc.

Part information includes the part type (target, fixture, other), number of faces, bounding box, etc.

If the part or the stock were created by criteria from a set, the set name is also available.

In order to show this information, new templates should be created with a different hierarchy. New and improved Excel (XLSX) demo templates are available in the version.

NC Setup Sheet

Cimatron 3D SYSTEMS

Program Name	
Program Comment	
Date	3/12/2019
Time	20:59
User Name	Haran
Cimatron File Name	1General PostTest_v15_demo
Document Path	C:\Cimatron_Documents\Gpp2\Post Stock and Part
Reference UCS	MODEL
Job Number/Name	1 /
Setup Name	SETUP_MODEL
Setup Comment	No Text
Material	

TP Name	SETUP_MODEL	TP Comment	No Text

TP Procedures

1	Procedure Name	Target Part_1	Procedure Comment	No Text	UCS Name			
Tech	Part Type	No. Faces	Part Set Name					
	Target Part	41						
Misc	X min	X max	Y min	Y max	Z min	Z max		
	0	100	0	100	0	74.92		
2	Procedure Name	Stock - Auto_2	Procedure Comment	No Text	UCS Name			
Tech	Stock Type	Offset	Stock Set Name					
Misc	X min	X max	Y min	Y max	Z min	Z max		
	0	100	0	100	0	74.916		

TP Name	TP_MODEL	TP Comment	Rough+Fin

TP Procedures

3	Procedure Name	R-Spiral_83	Procedure Comment	No Text		UCS Name	MODEL		Axes Number	3	
Tool	No.	Name	Tool Type		Diameter	Corner Rad.	Cut Len.	Free Len.	Whole Len.	Holder Name	
	T1	FLAT 10	Flat		10	0	15	50	50	HOLDER1	
Tech	Side step	Down Step	Cont. Tol.	Cont. Off.	Part Tol.	Part Off.	Check Surf. Tol.		Check Surf Off.	Clearance	Spin	Feed
	6.5	7		0	0.5	0.25				125	1000	500
Misc	X min	X max	Y min	Y max	Z min	Z max	Z max (Feed)	Coolant	Feed Time	Air Time	Total Time	Tool Total Feed Time
	-5.633	106.904	-6.843	106.059	11.916	125	77.079	M08	00:23:46	00:00:31	00:24:16	T1 / 00:23:46
4	Procedure Name	F-Slope_84	Procedure Comment	No Text		UCS Name	MODEL		Axes Number	3	
Tool	No.	Name	Tool Type		Diameter	Corner Rad.	Cut Len.	Free Len.	Whole Len.	Holder Name	
	T5	BN10R3	Bull nose		10	3	10	60	60	HOLDER3	
Tech	Side step	Down Step	Cont. Tol.	Cont. Off.	Part Tol.	Part Off.	Check Surf. Tol.		Check Surf Off.	Clearance	Spin	Feed
	2.5	1		0	0.2	0.1			-0.3	125	1000	500
Misc	X min	X max	Y min	Y max	Z min	Z max	Z max (Feed)	Coolant	Feed Time	Air Time	Total Time	Tool Total Feed Time
	-1.946	102.711	-1.971	101.989	9.999	125	76.824	M08	00:28:30	00:00:21	00:28:51	T5 / 00:28:30

TP Milling Tools

Number	Name	Comment	Diameter	Corner Radius	Taper Angle	Drill Angle	Cut Length	Clear Length	Tool Type	Holder Name	Holder Comment
T1	FLAT 10	NO COMM	10	0	0	0	15	50	Flat	HOLDER1	
T5	BN10R3	NO COMM	10	3	0	0	10	60	Bull nose	HOLDER3	

TP Summary

X min	X max	Y min	Y max	Z min	Z max
-5.633	106.904	-6.843	106.059	9.999	125

Program Milling Tools

Number	Name	Comment	Diameter	Corner Radius	Taper Angle	Drill Angle	Cut Length	Clear Length	Tool Type	Holder Name	Holder Comment
T1	FLAT 10	NO COMM	10	0	0	0	15	50	Flat	HOLDER1	
T5	BN10R3	NO COMM	10	3	0	0	10	60	Bull nose	HOLDER3	

Motion Limits

X min	-5.633	X max	106.904
Y min	-6.843	Y max	106.059
Z min	9.999	Z max	125
Z max (Feed)			77.079

Statistics

No. of Procedures	4
No. of Tools	2
Total Air Time	00:00:52
Total Feed Time	00:52:15
Total Time	00:53:07

9.8.4 Utility Procedure

The Utility procedure is a new type of procedure that appears in the Process Manager among other procedures.

The procedure can serve any purpose defined by the its creator; normally for sending information to the post processor and the NC report.

The procedure does not contain toolpath motions nor cutter, only a set of parameters that are passed to the post processor. The new procedure is intended for turning machines, but can be used for any machine as well.

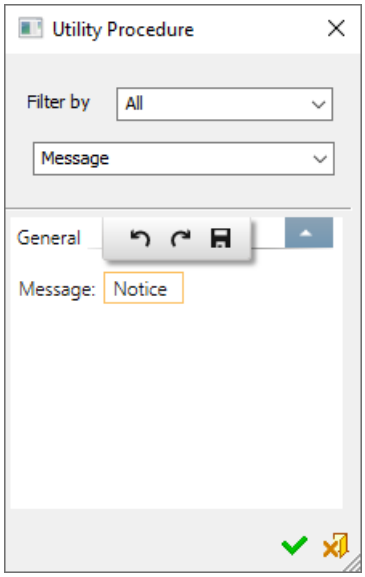
The procedure contains a flexible dialog. The first parameter in the dialog allows the user to select which operation the utility is to perform. The remaining parameters are operation dependent.

Three utilities are available for all machines:

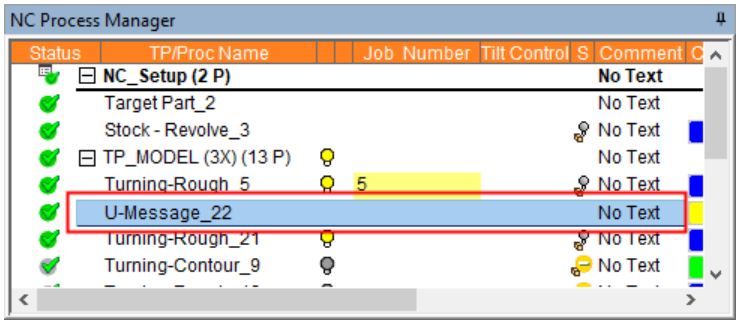
- Message
- Insert G-code With Sequencing
- Insert G-code Without Sequencing

Other utilities are machine dependent.

Once a Utility Procedure is created and saved, it is displayed in the Process Manager.



Message defined in the Utility Procedure

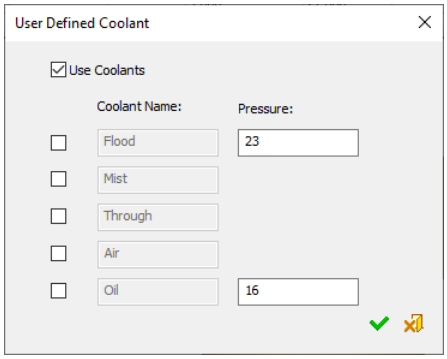


Utility Procedure appears in the Process Manager

The Utility procedure can be prepared by a Cimatron Provider/Reseller using a dedicated Utility Procedure Editor.

9.8.5 More Coolant and Multi Coolant

Additional user-defined coolants are available. Multiple coolants can be selected to work simultaneously.



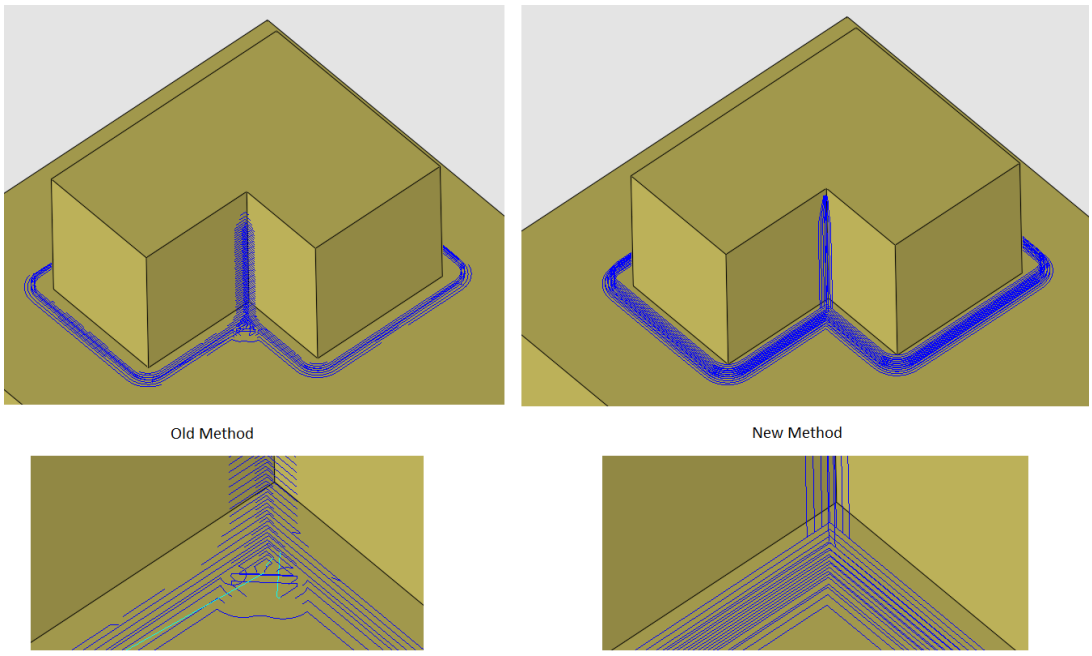
The defaults are defined in the Preferences.

9.9 Surface Quality

9.9.1 New Rerough Strategy in Cleanup Procedure

When a Cleanup procedure is required to remove much stock, a Rerough operation is needed prior to the finishing passes.

In Cimatron, a new strategy for Reroughing was developed, based on offsets of the finishing passes.
The new strategy creates a smoother and more regular toolpath than the old one. The old method is also available, and user can choose which method to use.



9.9.2 Finish - 3D Cutter Compensations

The usage of 3D cutter compensation allows controlling the actual tool radius on the machine, without re-programing the toolpath.

The normal of the touch point of the tool to the part surfaces is now kept in the toolpath and used on the machine for compensation.

9.9.3 Finish by 4 Limit Angles

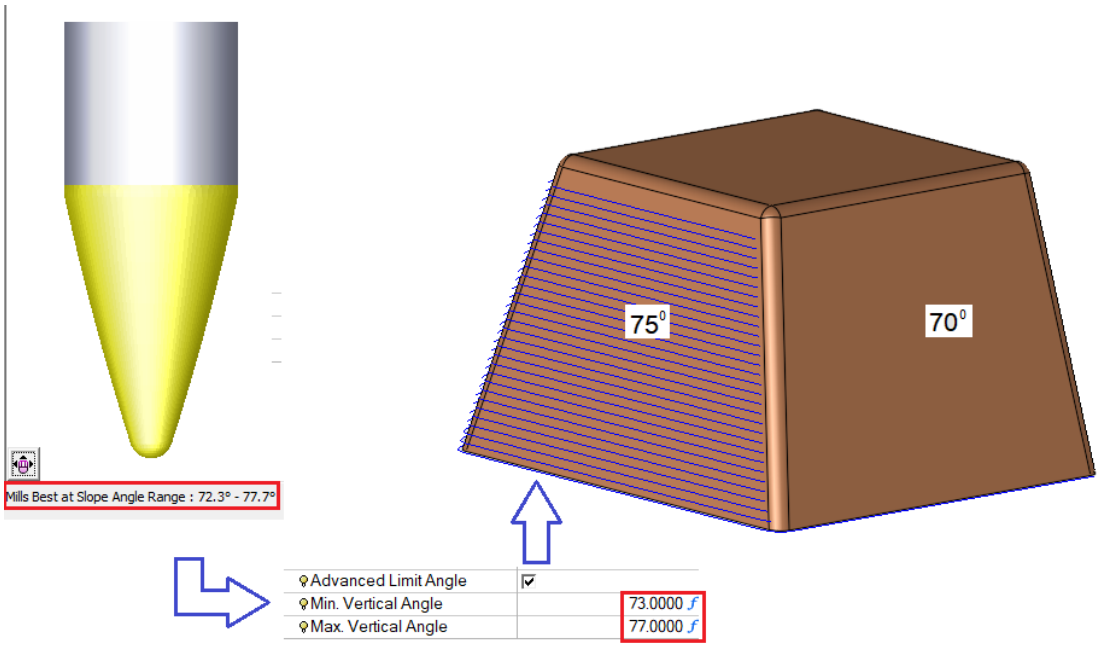
The Finish by Limit Angle and Rest Milling procedures now have an option to control the Minimum and Maximum angles for horizontal and for Vertical regions (a total of 4 angles).

This allows circle segment tools to mill at their best slopes.

This also allows better control of angles for regular tools (ball, bullnose).

In the example below, due to its geometry, the three radii cutter mills best on walls at an angle of 75^0.

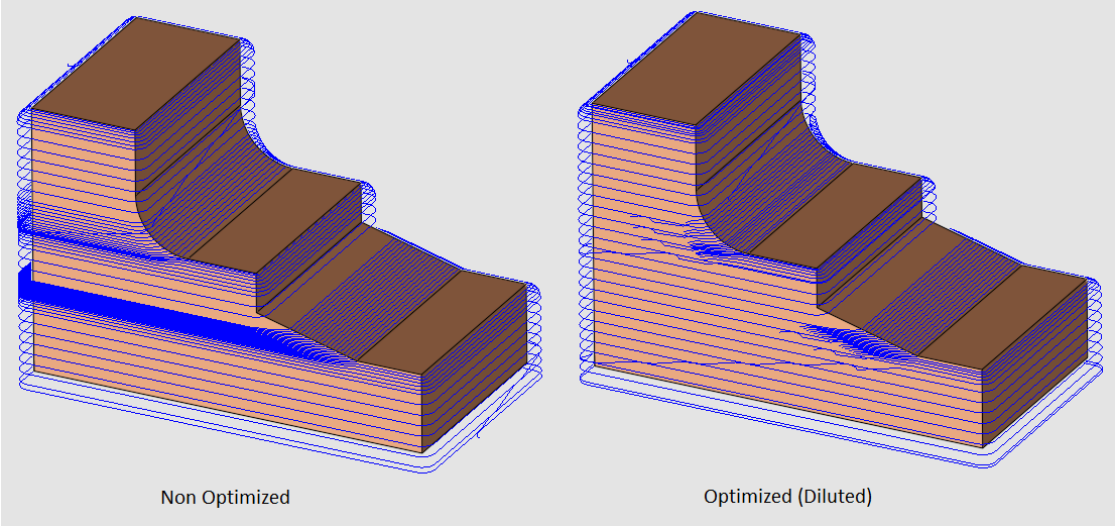
The Finish by Limit Angle set the slope between 73^0 and 77^0. This mills the wall with the desired angle and avoids milling walls at a different angle.



9.9.4 Optimized Variable Down Step for Finish by Layers

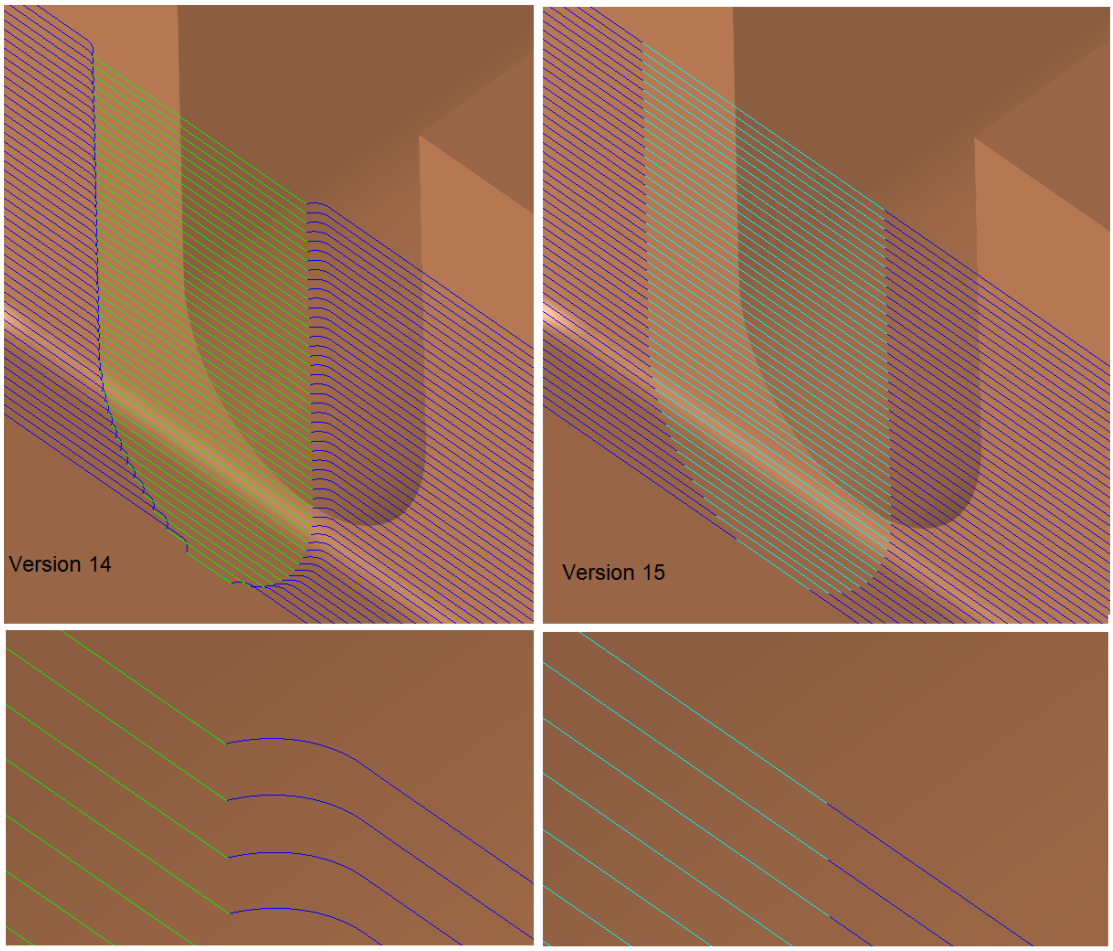
A new option to optimize the variable down step was added to the Finish procedure. Using this new option will reduce machining time.

When a specific surface quality requires shallower slopes, this results in smaller down steps. In the past, each layer was milled entirely. So, when shallow slopes existed in some places of the geometry, the Down Step was defined for the entire layer, creating redundant milling and thus increasing the machining time. The new “Optimized” option dilutes the unnecessary motions, thus reducing machining time. However, notice that using this option will cause the cutter to leave the part more often.



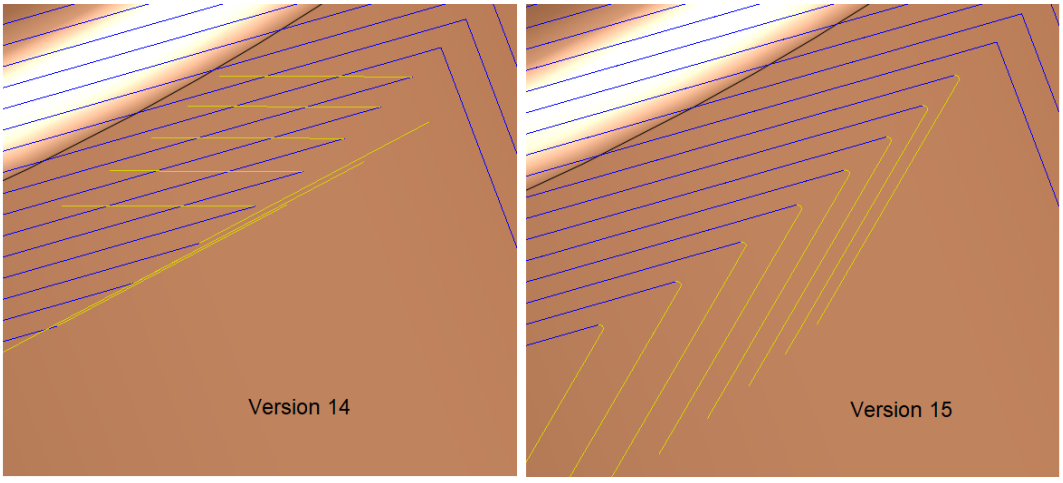
9.9.5 Finish - Short Connections for coincident layers

Connections have been improved in Finish by Layers, where layers are more or less coincident.



9.9.6 Finish - Avoid sharp approach and retract

When the approach or retract fails, round corners are created by shortening the milling passes.



9.9.7 Cleanup - Round Corners

Round motions create smoother motions and create better surface quality.

In addition to other procedures, rounding is now possible for all motions, in Cleanup procedures: Cleanup, Guided Cleanup, and Multi Axis Guided Cleanup as well.

9.9.8 Chamfer - Opposite tangential approach

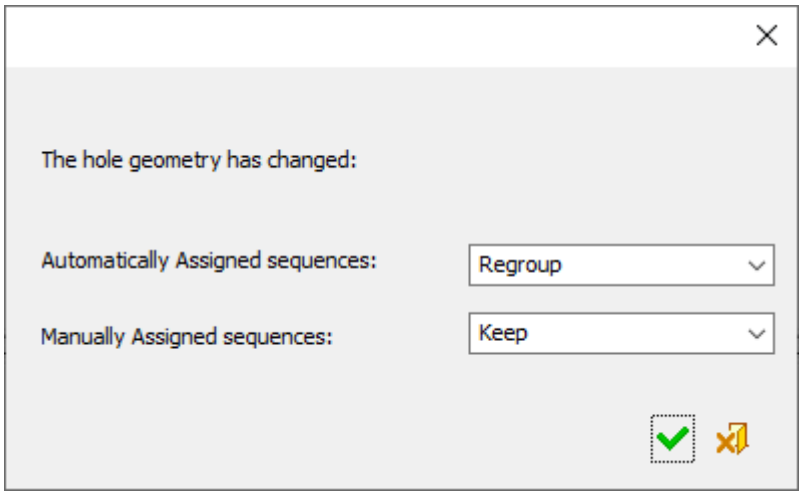
In the Chamfer procedure, when the toolpath is truncated and cut, for example by check surfaces, it normally cannot perform a tangential approach or retract. In that case, the next priority is now opposite tangential.

9.10 Automated Drill

9.10.1 Automated Drill - Manually assigned sequences are not lost

Sequences that were manually attached are no longer lost when the part geometry or the stock are changed.

A dialog enables the user to either keep or remove the manually assigned sequences.

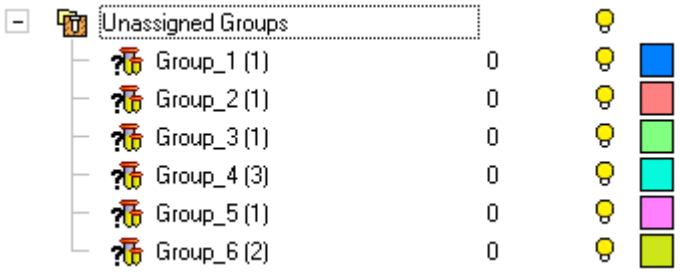


9.10.2 Automated Drill - Automated Names of Groups

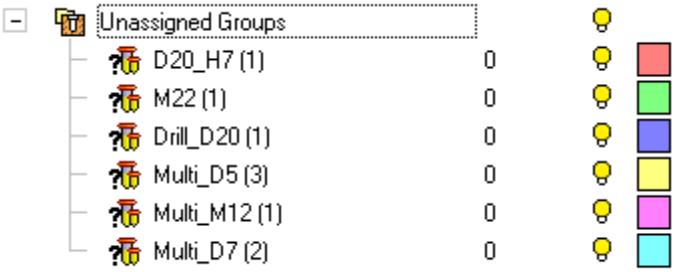
Groups in AutoDrill get significant names, based on hole geometry and attributes.

Having significant names helps users to understand the meaning of each group.

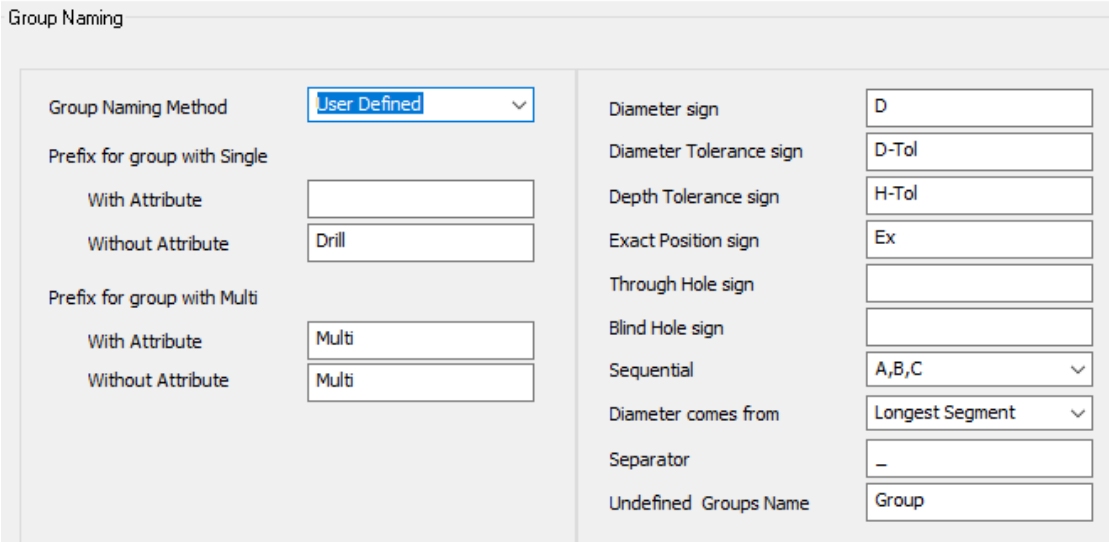
Group_1, Group_2, …:



Get significant names:



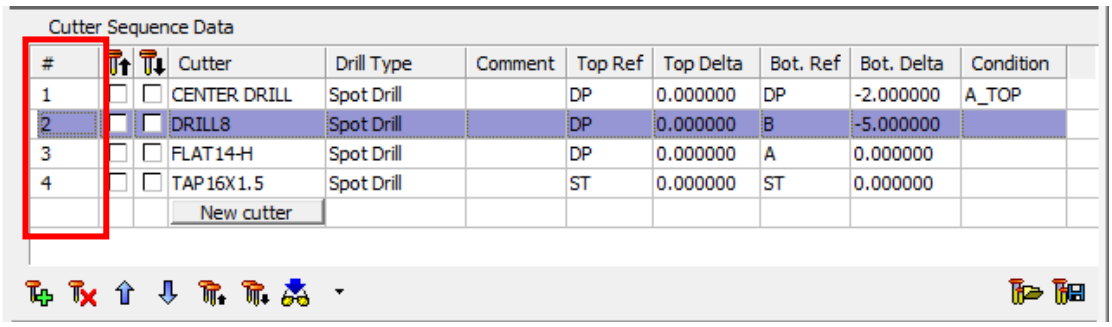
The naming rules are defined in the Preferences.



9.10.3 Automated Drill - Add Sequential numbers for operations

A new column has been added to the Cutter Sequence Data table that shows the operation sequential number.

This helps the user to identify a problematic operation.



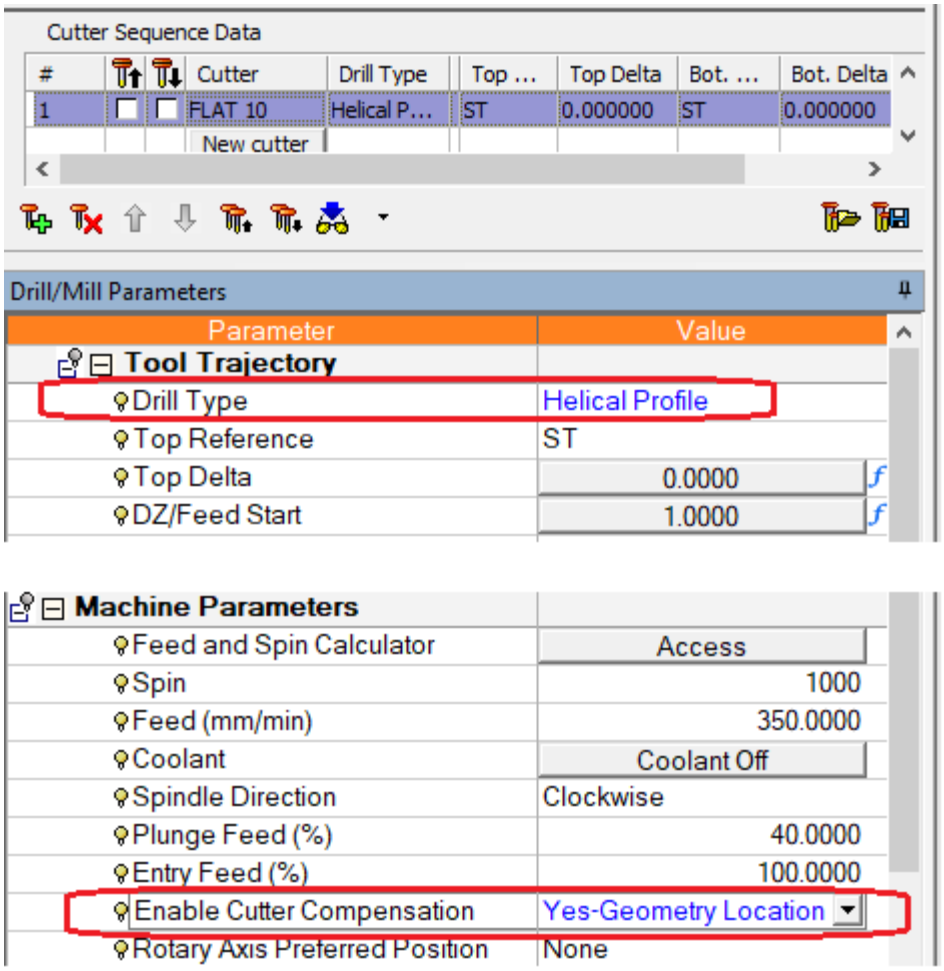
9.10.4 Automated Drill – Split by Cutter

When splitting an AutoDrill procedure by cutter, the comments of the newly created procedures now contain the sequence name, hence helping the user to understand each procedure's role.

9.10.5 Cutter Compensation for Automated Drill Helical Profile

The Helical Profile operation in AutoDrill now supports 2D Cutter compensation.

Using cutter compensation enables accurate hole diameters, using any milling cutter.



9.11 Simulator

9.11.1 Simulator - Saving deviation table, colors and safety

Deviation table values and colors, as well as safety-offset values of the simulator, can now be saved and loaded. These are saved as “template” files in a specific folder.

These values are used for newly created files (depends on the Preference) and can be loaded for existing files.

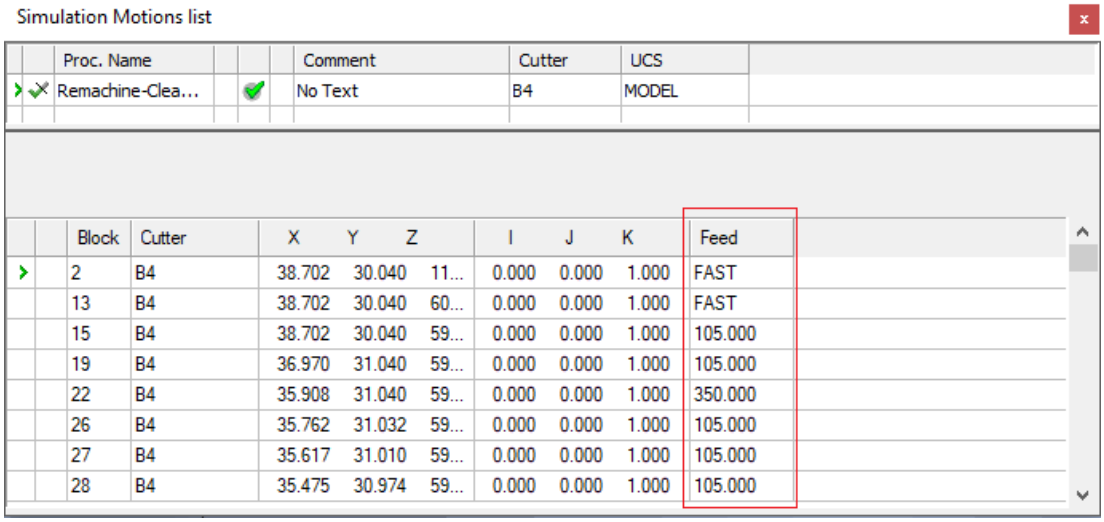
9.11.2 Simulator - Deviation on initial stock

The Deviation was previously shown only for simulated procedures. It will be shown from now also for procedures before them.

9.11.3 Simulator – Display the current feed rate in the Motion List table

In the simulator, the feed rate of each motion is displayed in the Motion List table. This option helps the user to see where and how the feed rate changes.

The new option is mostly useful while simulating Rough with Automatic Feed Control. The user can then track the feed of each cutter movement and compare it with the cutter load.



9.11.4 Simulator - Performance and movie smoothness improvements

Performance and movie smoothness improvements.

9.11.5 Prevent false errors on imaginary chamfer

Running the simulator on the Chamfer procedure that uses imaginary contours will no longer show false gouges.

9.11.6 Support Shaped Cutters

Shaped cutters are now supported in the Cimatron interfaces for Eureka and Vericut.