04/17/2018

ESPRIT TNG Release 3.6

DP Technology Corp.

Introduction

Welcome to **ESPRIT TNG Release 3.6**.

In this document, find a detailed description of the new features added this Milestone.

Next Milestone will be Release 3.7 scheduled for June 2018.

Performance – What's new?

New Rebuild Mode

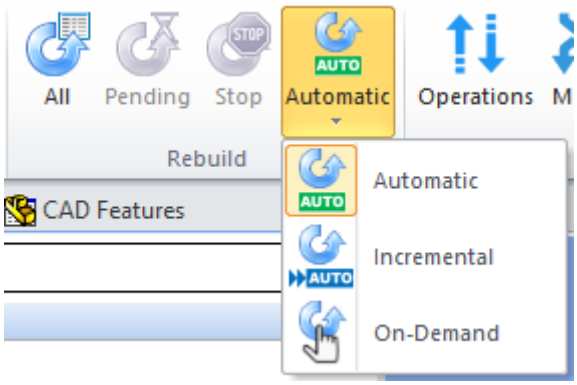
When proving out a program, it is typical to review and update program restarting from beginning. The user updates cutting conditions, strategies… while updating the program, downstream recalculation can cause slow down and often the update is not necessary as user may update the program downstream later.

ESPRIT TNG R3.6 introduces a new INCREMENTAL rebuild mode.

Its purpose is to update only what is necessary after a change, all downstream calculations are paused to prevent unnecessary calculation that could cause lack of responsiveness.

How it works

In Home tab of the ribbon, in the Rebuild Group, Rebuild Mode offers a choice between Automatic Rebuild, Incremental Rebuild and On-Demand Rebuild. The Rebuild Mode controls when operations, stocks and links build after creation of an operation or after modification of the program.



Rebuild Mode

Automatic
Select Automatic and the entire program updates automatically after a change.

Incremental
Select Incremental and program updates automatically upstream to the change. Any downstream items that require an update become pending.
This mode can prevent unnecessary calculation when editing an existing program.

On-Demand
Select On-Demand and nothing updates automatically. Items update on user-request.

If the program has items pending calculation, user can trigger calculation of pending items by either:

- Manually triggering calculation of operation or stock or link. In that case all the dependencies (operations, previous stocks, previous links) build automatically.
- Clicking on Rebuild Pending.

Rebuild Pending
Rebuild Pending rebuilds all pending items of the program.
It rebuilds pending operations but also all pending stock, pending links.

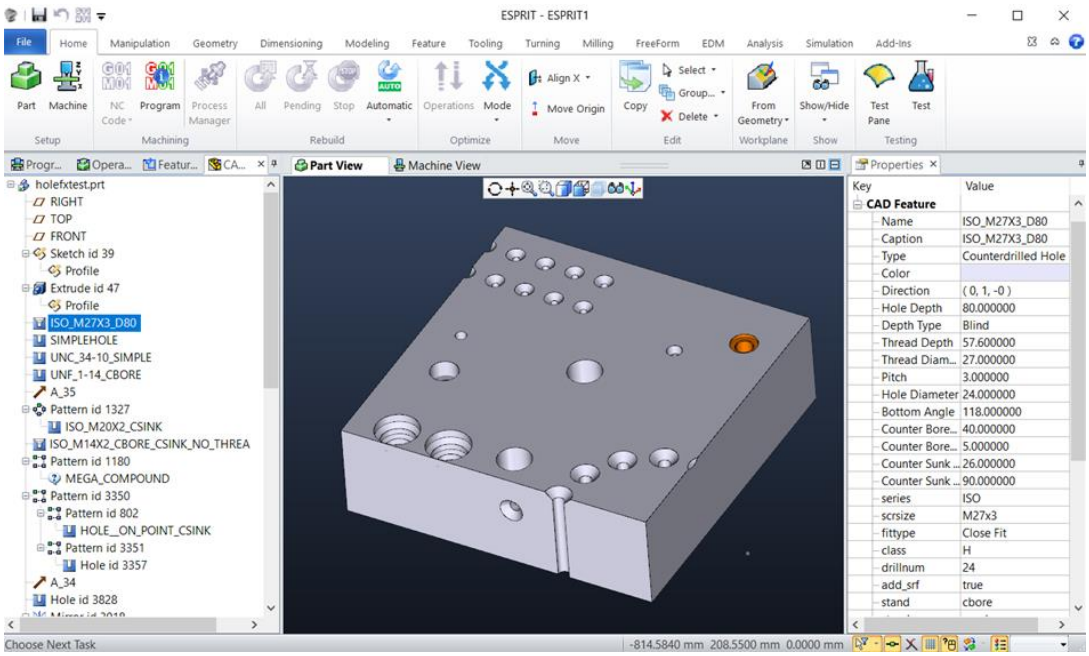
Notes:

- With incremental mode:
  - When creating operation at the end of the program, incremental mode behaves the same as the automatic mode.
  - With incremental mode, when reordering operations in the program, all items between the 2 reordered operations stay pending.
  - With incremental mode, on suppress and delete, impacted items stay pending.
  - When reloading a file save with incremental mode, stocks are not calculated unless user clicks on rebuild pending.
  - You can simulate, verify and post NC code up to the latest calculated operation. Posting of selected operations just requires that operations and links are calculated for the selected items.
- Rebuild all is different from rebuild pending: Rebuild all rebuild all operations, rebuild pending updates only pending items with an hourglass symbol.
- The rebuild mode is saved with the document.
- To change the default mode of a new file, create a new template with the desired rebuild mode.

Usability – What's new?

ETNG FX For Catia, UG, ProE

FX in TNG now supports Catia, UG and ProE.



Stock dimensions in Post

ETNG is now able to output stock dimensions into the CL file and ultimately into the post. This was requested to satisfy the need of certain machine tools which can fully simulate NC code, with stock removal. These machines require stock information to be output in the NC file, so it can be drawn for simulation.

How it works

We have added CL files for CLFileWorkPiece in Begin All Workpieces to allow the post writer to output the stock dimensions.

A list of all CL files used can be found below, along with a sample post processor code to output the correct information specifically for BLK FORM.

BEGIN All Workpieces	
BEGIN Workpiece	
Set-Up Stock type(3475)	Returns values from the enumeration below: (Block = 1 Bar =2 Tube = 3)
Set-Up Stock Minimum X(3476)	Returns minimum X value of the bounding box of block, STL and File stocks with respect to the first work offset on the workpiece
Set-Up Stock Minimum Y(3477)	Returns minimum Y value of the bounding box of block, STL and File stocks with respect to the first work offset on the workpiece
Set-Up Stock Minimum Z(3478)	Returns minimum Z value of the bounding box of block, STL and File stocks with respect to the first work offset on the workpiece

Set-Up Stock Maximum X(3479)	Returns maximum X value of the bounding box of block, STL and File stocks with respect to the first work offset on the workpiece
Set-Up Stock Maximum Y(3480)	Returns maximum Y value of the bounding box of block, STL and File stocks with respect to the first work offset on the workpiece
Set-Up Stock Maximum Z(3481)	Returns maximum Z value of the bounding box of block, STL and File stocks with respect to the first work offset on the workpiece
Set-Up Stock Vector X(3482)	Returns the X vector of the extruded bar stock with respect to the first work offset on the workpiece
Set-Up Stock Vector Y(3483)	Returns the Y vector of the extruded bar stock with respect to the first work offset on the workpiece
Set-Up Stock Vector Z(3484)	Returns the Z vector of the extruded bar stock with respect to the first work offset on the workpiece
Set-Up Stock Bar Radius(3485)	Returns the radius of the bar stock.
Set-Up Stock Bar Length(3486)	Returns the length of the stock as the maximum point on the bounding box with respect to the work offset direction and position
Set-Up Stock Bar Distance(3487)	Returns the length of the stock as the minimum point on the bounding box with respect to the work offset direction and position
Set-Up Stock Bar Inner Radius(3488)	Returns the inner radius of the bar stock.

Example Code:

```
MiscFormat1     : X_     N4YN4   N5YN3   1   1N00
MiscFormat2     : Y_     N4YN4   N5YN3   1   1N00
MiscFormat3     : Z_     N4YN4   N5YN3   1   1N00
MiscFormat4     : R__    N4YN4   N5YN3   1   1N00
MiscFormat5     : L_     N4YN4   N5YN3   1   1N00
MiscFormat7     : DIST_  N4YN4   N5YN3   1   1N00
MiscFormat8     : RI_    N4YN4   N5YN3   1   1N00
```

```
****************************** Defines*********************************************************************

Define VectorOuptutBlockForm

If (ClFileWorkPiece(3482,1)=1) "X" ElseIf (ClFileWorkPiece(3483,1)=2) "Y" Else "Z" EndIf \\

EndDefine


Define OutputTubeBarBlockForm

R__*(CLFIleWorkPiece(3485,1)) L_*(ClFileWorkPiece(3486,1)) DIST_*(ClFileWorkPiece(3487,1))
RI_*(ClFileWorkPiece(3488,1)) \\

EndDefine

****************************** Examples *******************************************************************

Ex_MainStart        : If (ClFileWorkPiece(3475,1)=3)  ## Tube

                    : "BLK FORM TUBE" $VectorOuptutBlockForm $OutputTubeBarBlockForm

                    : ElseIf (ClFileWorkPiece(3475,1)=2)  ##Bar

                    : "BLK FORM CYCLINDER" $VectorOuptutBlockForm $OutputTubeBarBlockForm

                    : ElseIf (ClFileWorkPiece(3475,1)=1)  ## Block

                    : "BLK FORM 0.1 Z" X_*(ClFileWorkPiece(3476,1)) Y_*(ClFileWorkPiece(3477,1))
Z_*(ClFileWorkPiece(3478,1))

                     : "BLK FORM 0.2 " X_*(ClFileWorkPiece(3479,1)) Y_*(ClFileWorkPiece(3480,1))
Z_*(ClFileWorkPiece(3481,1))

                    : EndIf
```

Report Generator Enhancements

In ETNG, we are now able to output the Min/Max XYZ of the Part, Stock and Operations in a report. As a programmer, I rely on simulation and collision detection to ensure my tools are long enough and that my part won't be damaged during cutting. It is easy to observe all this data while programming, but when you get on the shop floor, it is important to be able to relay this information via a report. It is important to note that the XYZ min and max will be output individually for each work offset that exists under the workpiece. The templates used to create the sample reports seen below are available in the Report Generator training script.

MACHINING REPORT

Part Name	**Test_File1 (4).esprit**	Machine Name	**5x Trunnion Mill - L**
Program Number	**1**	Material Standard	**---**
Programmer	**davids**	Material Class	**---**
Date	**1/25/2018 2:51:41 PM**	Material Condition	**---**
Unit	**Inch**	Cycle Time (min)	**51.3191501886688**
Program Comment	**---**		

Min X of Part	Max X of Part	Min Y of Part	Max Y of Part	Min Z of Part	Max Z of Part
0	10	0	20	-30	0
Min X of Operation	**Max X of Operation**	**Min Y of Operation**	**Max Y of Operation**	**Min Z of Operation**	**Max Z of Operation**
-0.313	10.725	-0.313	20.313	-1	0.1
Min X of Stock	**Max X of Stock**	**Min Y of Stock**	**Max Y of Stock**	**Min Z of Stock**	**Max Z of Stock**
0	10	0	20	-30	0

MACHINING REPORT

Part Name	**TestReportMold_MultipleParts.esprit**	Machine Name	**5x Gantry Mill**
Program Number	**1**	Material Standard	**---**
Programmer	**Marco**	Material Class	**---**
Date	**1/16/2018 3:44:57 AM**	Material Condition	**---**
Unit	**Metric**	Cycle Time (min)	**12424.3631795129**
Program Comment	**---**		

Min X of the Part	Max X of the Part	Min Y of the Part	Max Y of the Part	Min Z of the Part	Max Z of the Part
Workpiece1	WorkOffset1				
-70	70	-37.5	37.5	-37.5	-0.5
Workpiece1	WorkOffset3				
0	140	-75	0	0	37
Workpiece1(2)	WorkOffset2				
0	140	0	75	0	37
Min X of the Stock	**Max X of the Stock**	**Min Y of the Stock**	**Max Y of the Stock**	**Min Z of the Stock**	**Max Z of the Stock**
Workpiece1	WorkOffset1				
-100	80	-62.5	72.5	-42.5	0.5
Workpiece1	WorkOffset3				
-30	150	-100	35	-5	38
Workpiece1(2)	WorkOffset2				
-30	150	-25	110	-5	38

Split NC Enhancements

The Split NC command has been updated to include a new option to split by, “Operation Name.” This new command will split NC files based solely on consecutive operation name as opposed to looking at which tool is selected.

A new option <OperationName> has been added to the NC File Name options. This allows the Split NC command to name the resulting files based on the operation name. If the user selects this option yet doesn’t use Split NC, they will have an NC file with only the first operation name.

Lastly, we have changed the way we concatenate NC File names for the Split NC Command. Previously when we used Split NC, the file names were concatenated with an underscore, which some machine controls won’t allow. For this reason, we have replaced the underscore with a dash, which most controllers accept.

WEDM Legacy Work Offset Handling

Previously, when a legacy WEDM file was brought into ETNG, all operations were associated with a single work offset, regardless of how many work coordinates had been used in E20xx. This made for a lot of work by the user to correctly recreate the work offsets, identify which operations should be associated to which work offset, and associate the operations. ETNG now automatically does the above upon import of a legacy WEDM file.

Edge Propagation

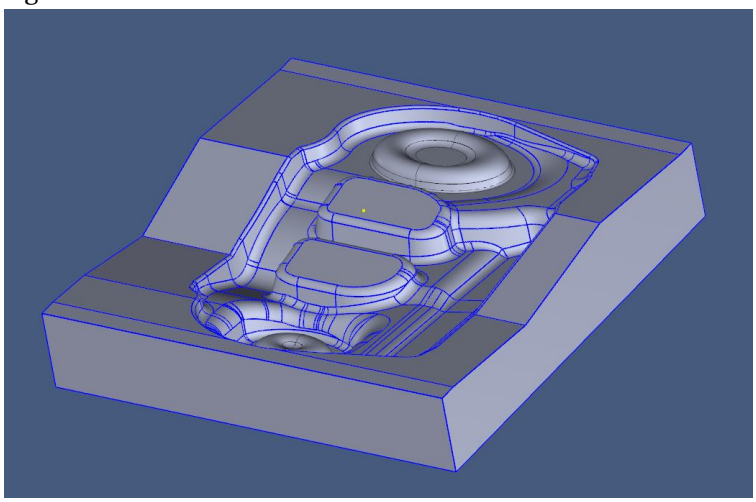
ETNG now can use propagation techniques to select edges. This is a great new enhancement which will make feature creation much simpler.

How it works

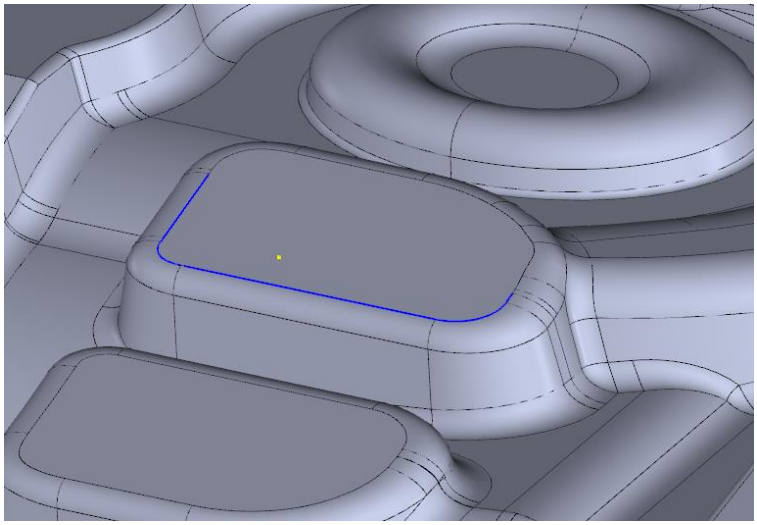
Propagating edges has 4 different options:

- All Edges
- Tangent Edges
- Planar Edges
- Edges in Loop

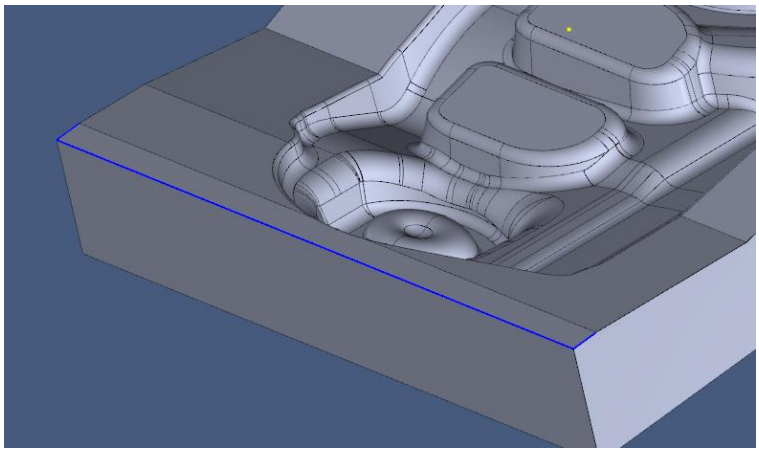
All Edges – all connected edges will be selected.



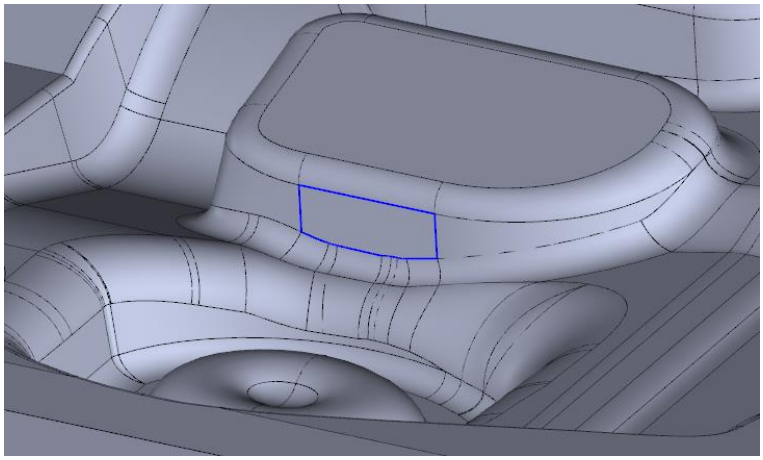
Tangent Edges – All tangent edges will be selected. One can also use the CTRL, CTRL, CTRL+Shift to limit the selection between two edges.



Planar Edges – All edges that are planar to the active work plane will be selected.



Edges in Loop – This option is similar to tangency, except all edges in the loop are grouped. One can also use the CTRL, CTRL, CTRL+Shift to limit the selection between two edges.



Machine Awareness for RTCP OFF

When working in 5 axes, machine-tools can use RTCP ON or OFF basing on the numerical control possibilities. RTCP ON is recommended because it gives many advantages:

- Only need to output tool tip position and tool vector or rotary axis positions
- Machine figures out how to move axes to get to specified position and orientation in the correct amount of time
- Single feed rate is programmed

When is not possible to use RTCP ON, it's necessary to pay attention on the 5-axis kinematic movement of the RTCP OFF, that may damage the part:

- NC directly acts on the machine-tool axes, rather than tool control point
- The axes interpolation may differ from tool control point interpolation

The damage detection is very insidious:

- It is not visible in Toolpath Analysis
- It is not visible in Stock Automation
- It is visible only when running Simulation

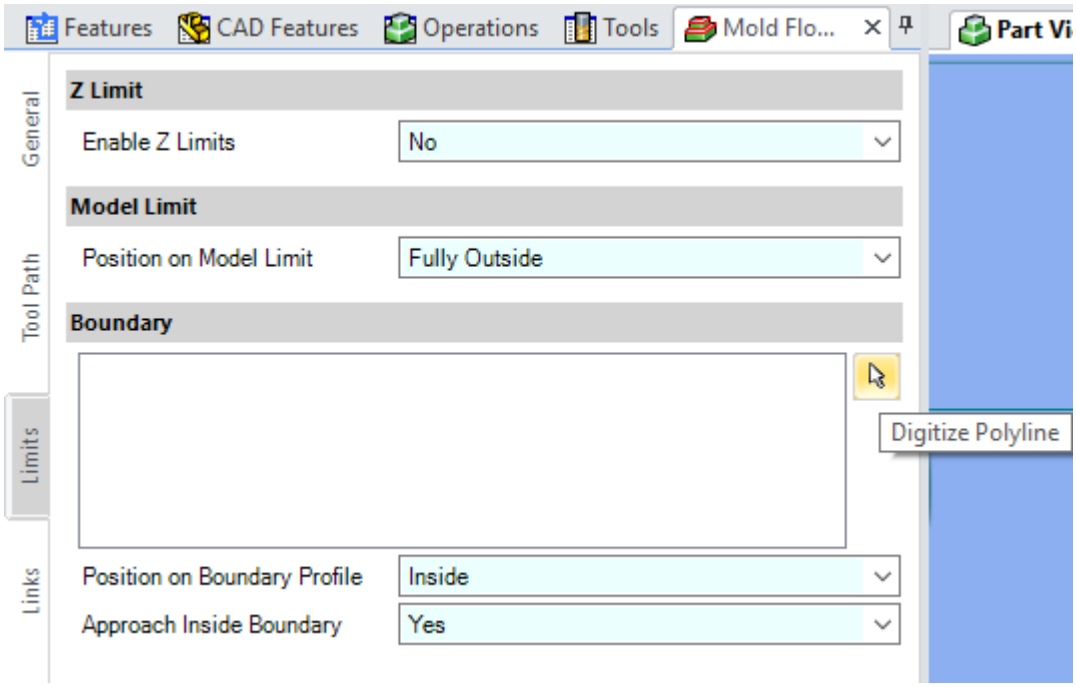
In ETNG R3.6 Machine Awareness is improved for 5 axis operations with RTCP OFF:

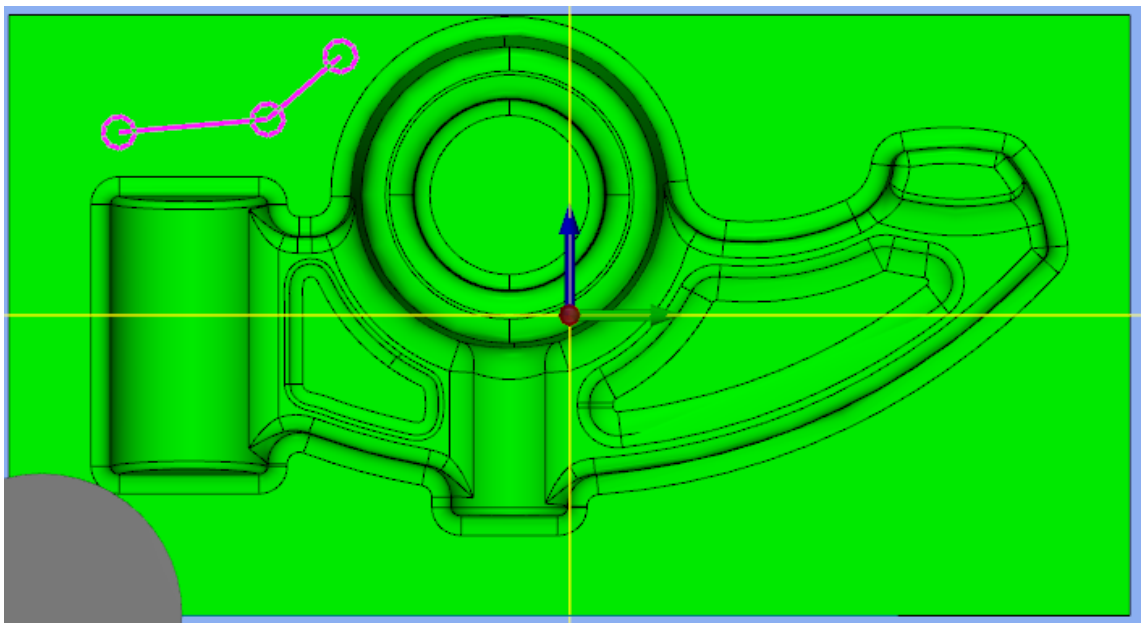
- Machine Awareness uses inverse kinematics to introduce additional interpolated points
- The additional interpolated points ensure that the NC axes interpolation stays inside the operation tolerance

This improvement guarantees that the relative displacement error between the tool tip and the workpiece of the 5-axis movement with RTCP OFF is inside the operation tolerance.

Toolpath Boundary Profile Enhancement

It is now possible to create a limiting polyline directly from any mold cycle's limits tab. Previously a user would need to create a predefined, limiting chain for the boundary section of a mold cycle. Often, a user would only understand the need for such limiting while programming the toolpath. Now the user can use this new functionality to directly create the limiting chain from inside the operation and the limiting polyline is saved as a feature chain to be reused for other operations. While the limiting polyline is being created, the user can see an outline of the tools diameter to properly create the limit.

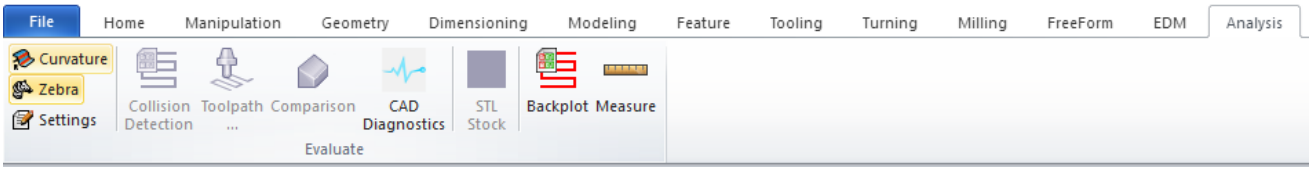




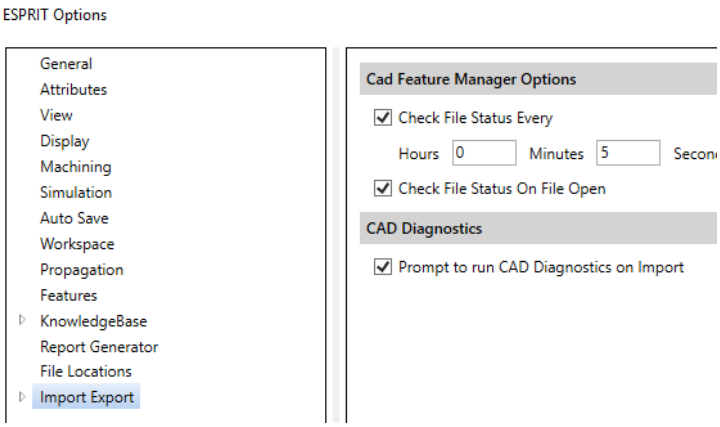
CAD Diagnostics

ETNG can now diagnosis problems on imported CAD files. This helps to ensure the solids being programmed do not have issues such as missing topology or zero length curves. CAD Diagnostics can be invoked upon import or directly on a solid that exists in the document. The command can be found on the Analysis tab.

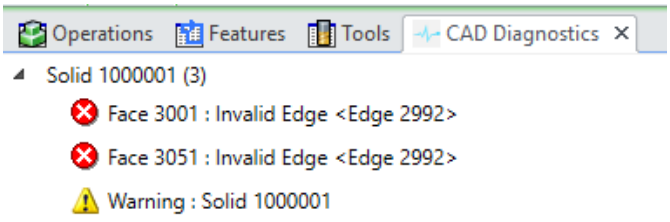
Detects and reports problems with the imported solid



Option to run on import



Displays warnings and errors



2/14/2018

ESPRIT TNG Release 3.5

DP Technology Corp.

Introduction

Welcome to **ESPRIT TNG Release 3.5**.

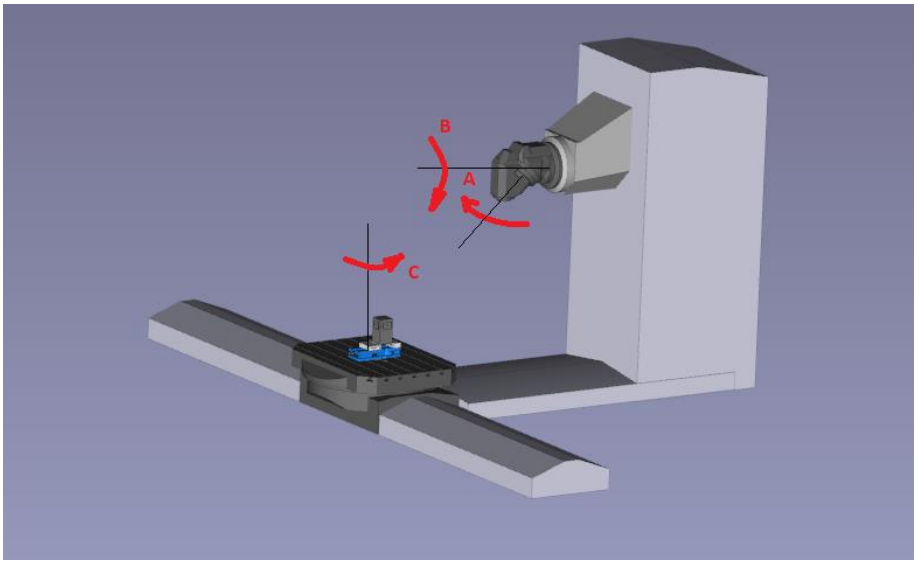
In this document, find a detailed description of the new features added this Milestone.

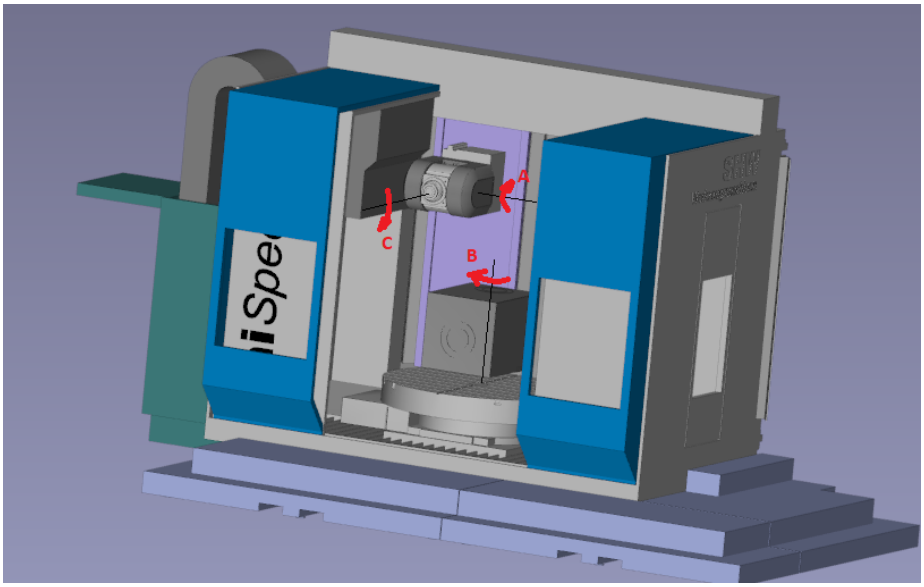
Next Milestone will be Release 3.6 scheduled for April 2018.

Machining and Simulation – What's new?

Management of Machine-Tools with 3 Rotary Axes

ETNG is now able to program machine-tools with 3 Rotary Axes as seen in the example machines below:





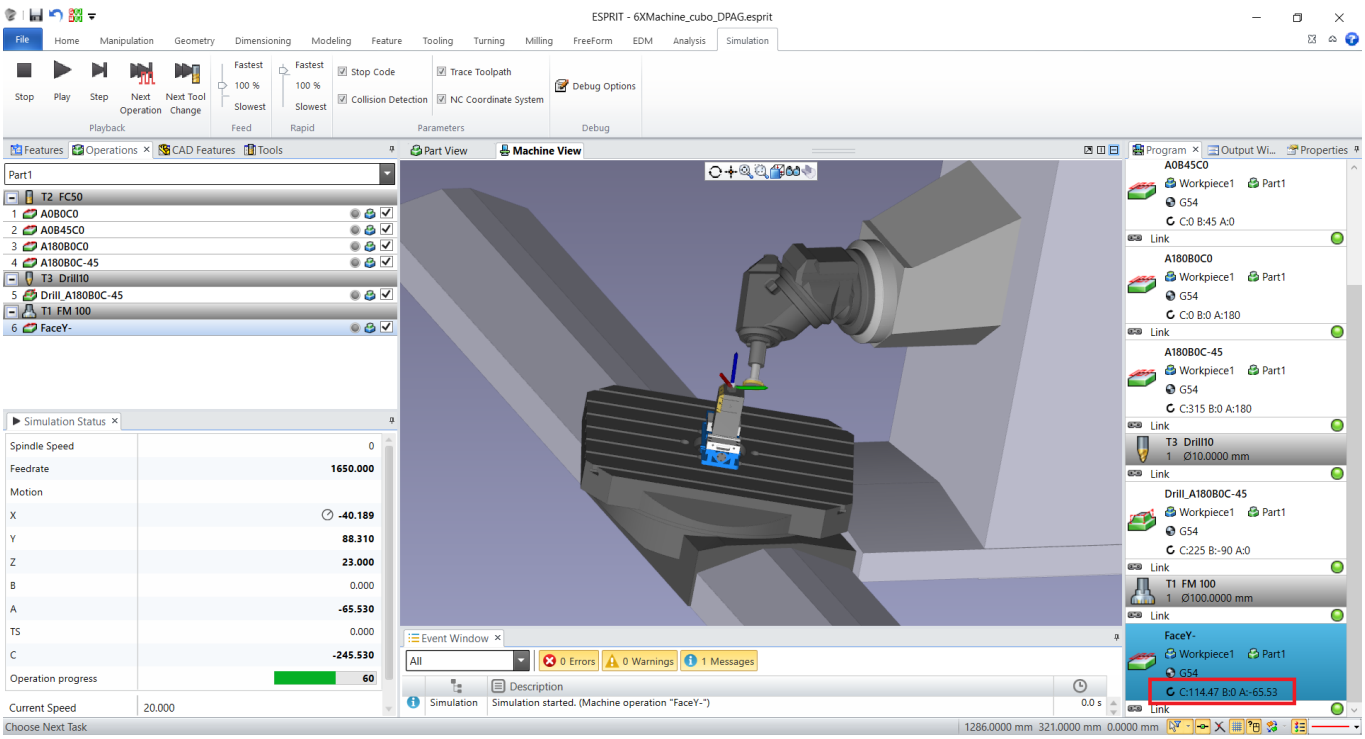
How it works

When a machine having 3 Rotary Axes is opened in the Machine Setup, on its Links tab, a new section for the Tertiary Rotary Axis shows up, having the same parameters as the Primary and the Secondary axes:

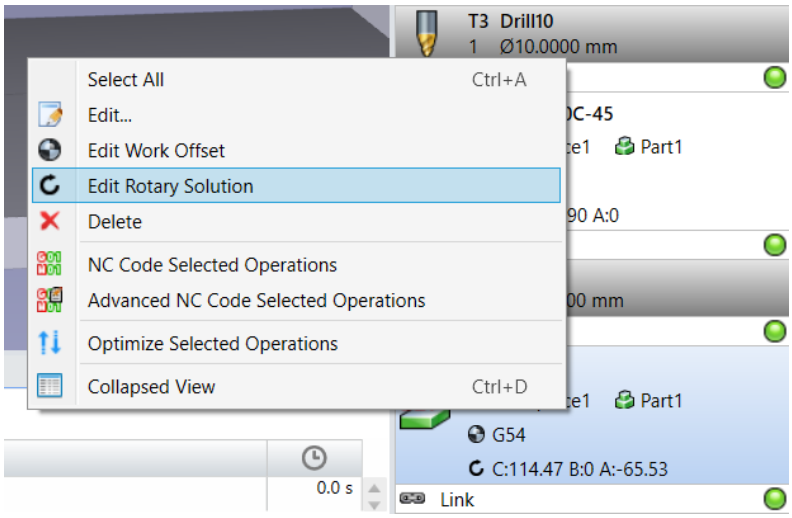
Primary Rotary Axis	
Primary Rotary Axis	C
Minimum Angle Change For Safe Retract	30.0000
Retract For Small Angle Changes	Yes
Rotary Retract Clearance	50.0000
Secondary Rotary Axis	
Secondary Rotary Axis	A
Minimum Angle Change For Safe Retract	360.0000
Retract For Small Angle Changes	No
Rotary Retract Clearance	25.0000
Tertiary Rotary Axis	
Tertiary Rotary Axis	B
Minimum Angle Change For Safe Retract	360.0000
Retract For Small Angle Changes	No
Rotary Retract Clearance	25.0000

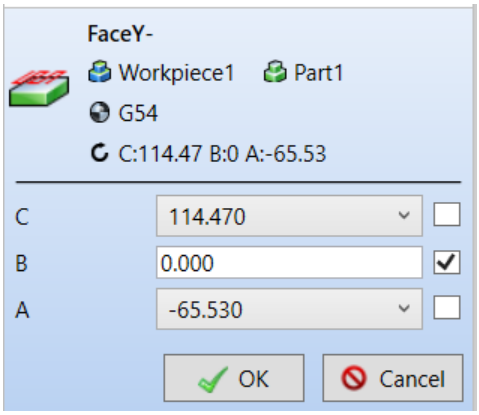
You can set your preferences for the Tertiary, similarly as you do the Primary and the Secondary, and they will affect the links accordingly.

When you create a new cycle, the system autonomously sets 3 admissible values for the 3 rotary axes, and they are shown, for that cycle, in the Program pane:

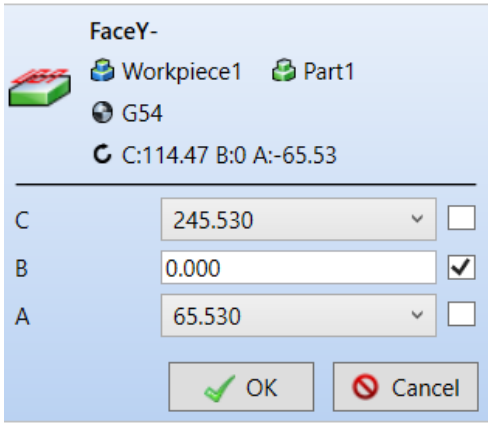


Right-Click on the cycle in the Program pane, and select Edit Rotary Solutions:

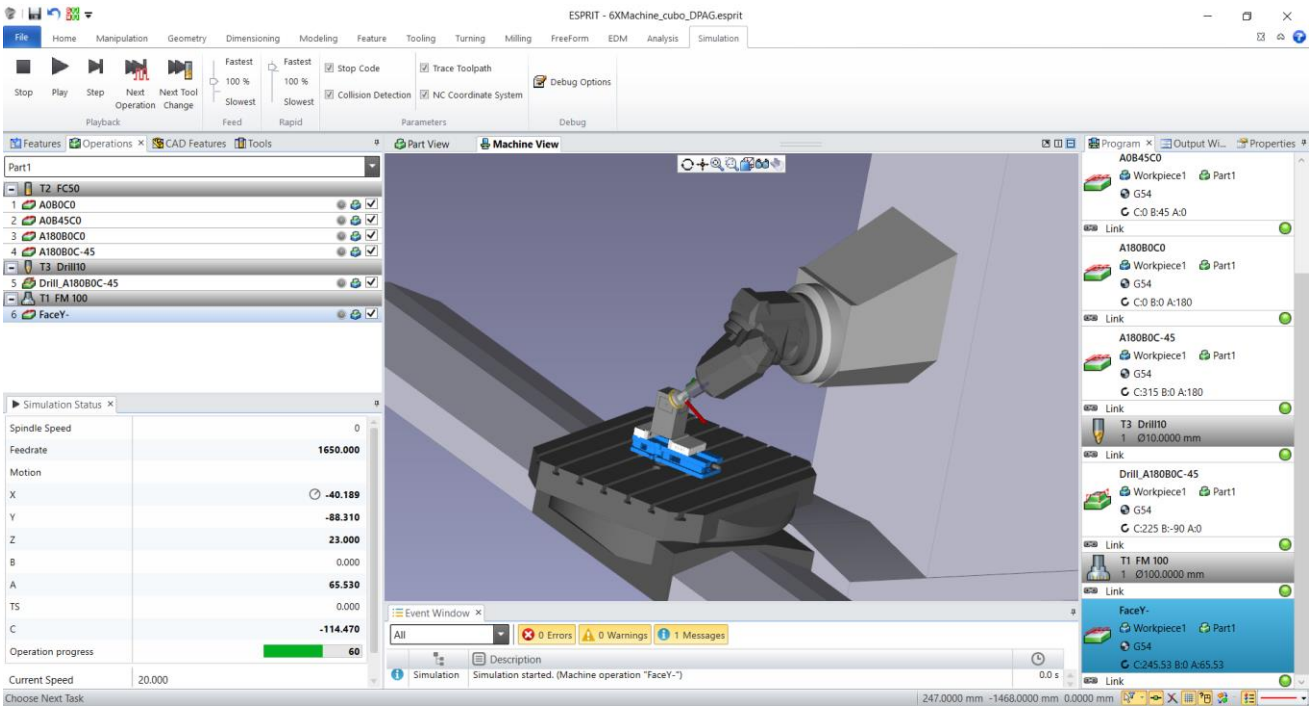




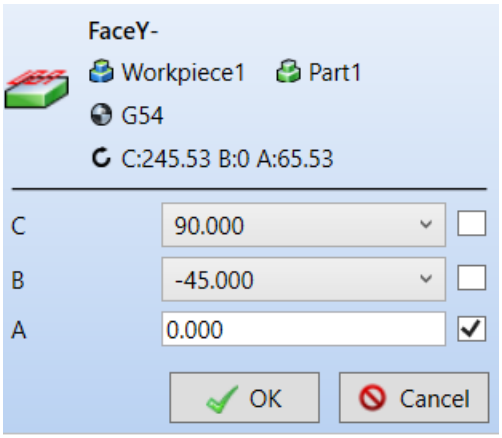
For each rotary axis, its value is shown, and the check-box in the right indicates which is locked. In the example above, the system has locked B to 0, and by selecting the drop down for both C and A, you can change to the alternative solution:



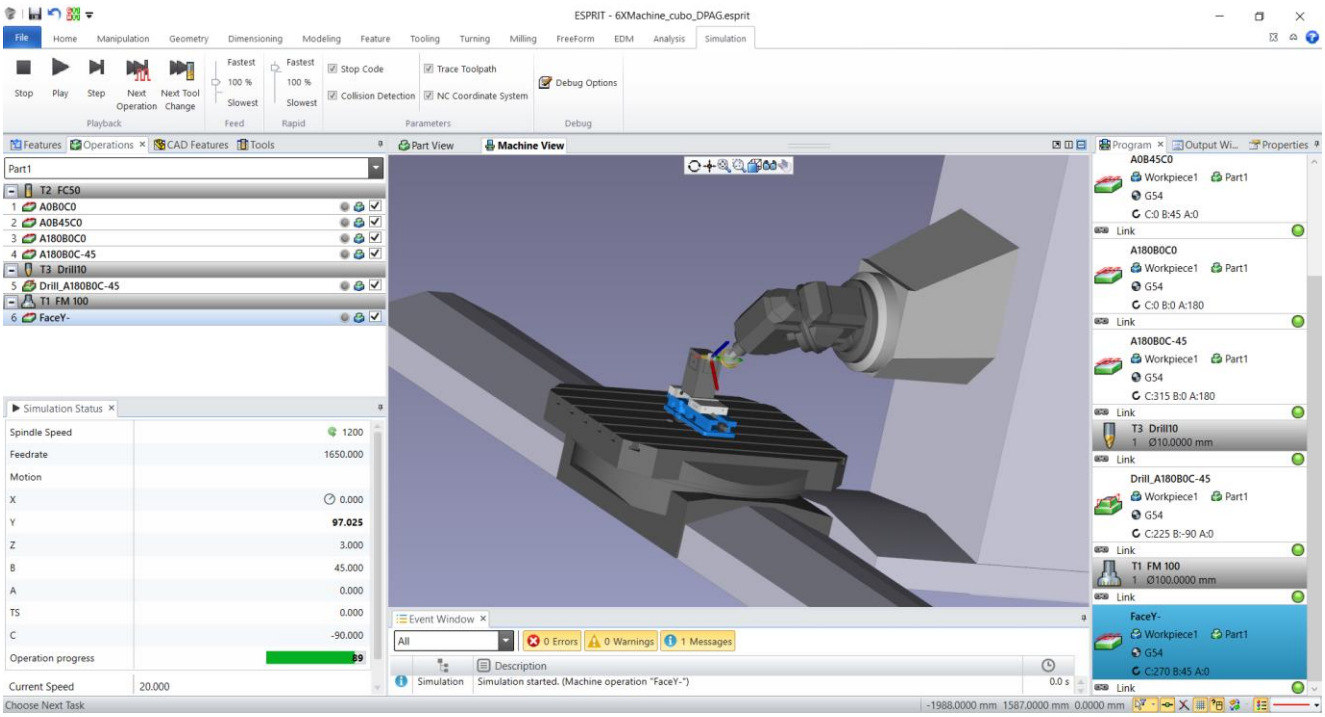
Validating with OK, you save the viewed values. Then the simulation reflects the change:



By editing the Rotary Solutions again, you can change the locked axis and its locked value. For example, if you lock A and set it to 0, then you get C90B-45 or C270B45 as two alternative possible solutions for A and B:



After validation, you get the following during simulation:



Limitations

- Currently, Edit Rotary Solutions can be done only one cycle at the time. In the future we might work on supporting it for multiple selected cycles having the same rotary axes values.
- Currently, continuous 5-axis cycles are not supported by machines with 3 rotary axes. This will be overcome in a future release.

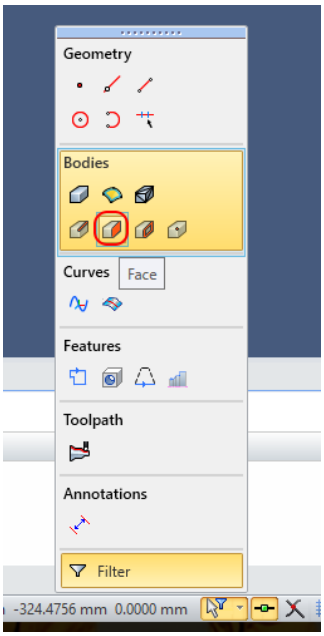
Usability – What's new?

Enhanced Selection

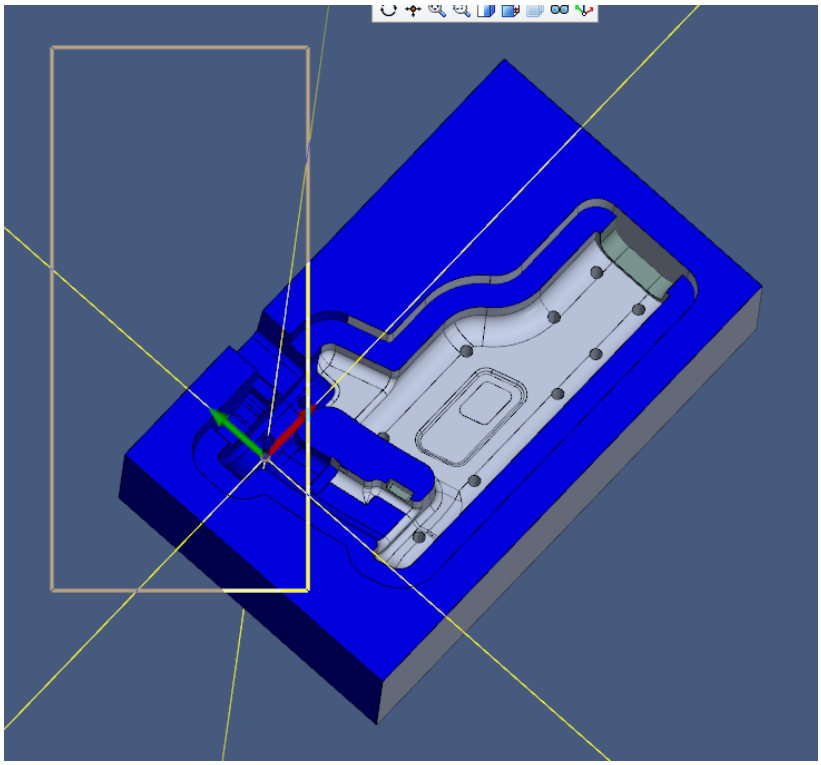
ESPRIT TNG R3.5 introduces enhancements to window selection. This new functionality was initially introduced to allow users to window select while the FreeForm Feature dialogue was open, however, the advancements are always available. The example below shows how to select faces using the new window option, however this method can be applied to any entity inside of ESPRIT.

How it works

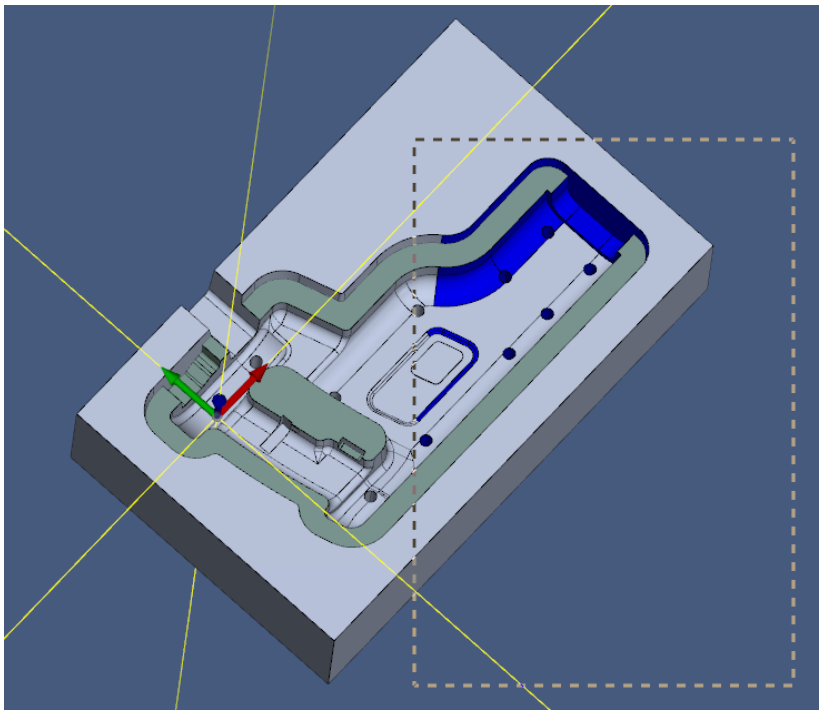
To limit the selection to faces only, use the selection filter. By right-clicking on the Face selection, all other selections will be de-activated.



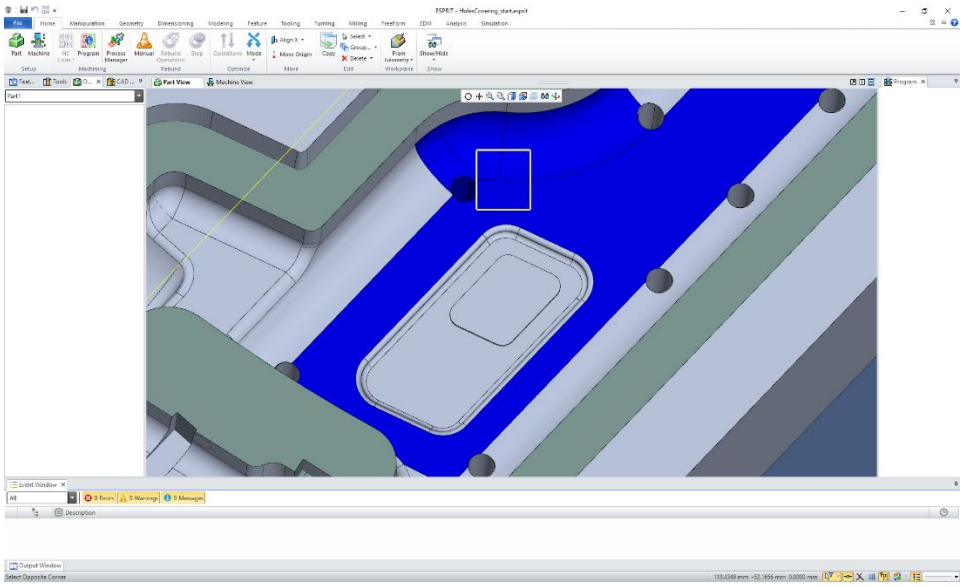
If you select from left to right, the selection box is solid and any faces even partially within the box will be grouped. When a window is drug across the screen, the faces and/or entities that will be selected highlight dynamically. A preview of what will be selected will be shown before the box is validated.



If you select from right to left, the selection box is dashed and only the faces wholly within the box will be grouped.



In order to start a window selection on top of a face, hold down the ALT key and start to drag the mouse.



Updated Solid Modeler UI

The following Solid Modeler commands have been updated to conform to the new user interface style:

- Extrude
- Revolve
- Fillet
- Chamfer

11/27/2017

ESPRIT TNG Release 3.4

DP Technology Corp.

Introduction

Welcome to **ESPRIT TNG Release 3.4**.

In this document, find a detailed description of the new features added this Milestone.

Next Milestone will be Release 3.5 scheduled for January 2018.

Machining and Simulation – What's new?

Enhanced Stock Sensibility in Roughing

For the Roughing cycle, we often want to avoid toolpath passes that don't cut a consistent volume of stock, particularly when we apply it as roughing re-machining.

The parameter 'Minimum Material Thickness' on the 'ToolPath' tab, was introduced to avoid passes in the areas where the stock has a thickness smaller than the parameter value. So, it is a sort of sensibility with respect to the stock.

However, as the help explained, till today such sensibility had no effect on vertical walls and passes on them were not avoided even when the stock left over was smaller than the value set for 'Minimum Material Thickness'. Thanks to this new improvement, the stock sensibility in Roughing will be the same along all the part, independently from local slopes.

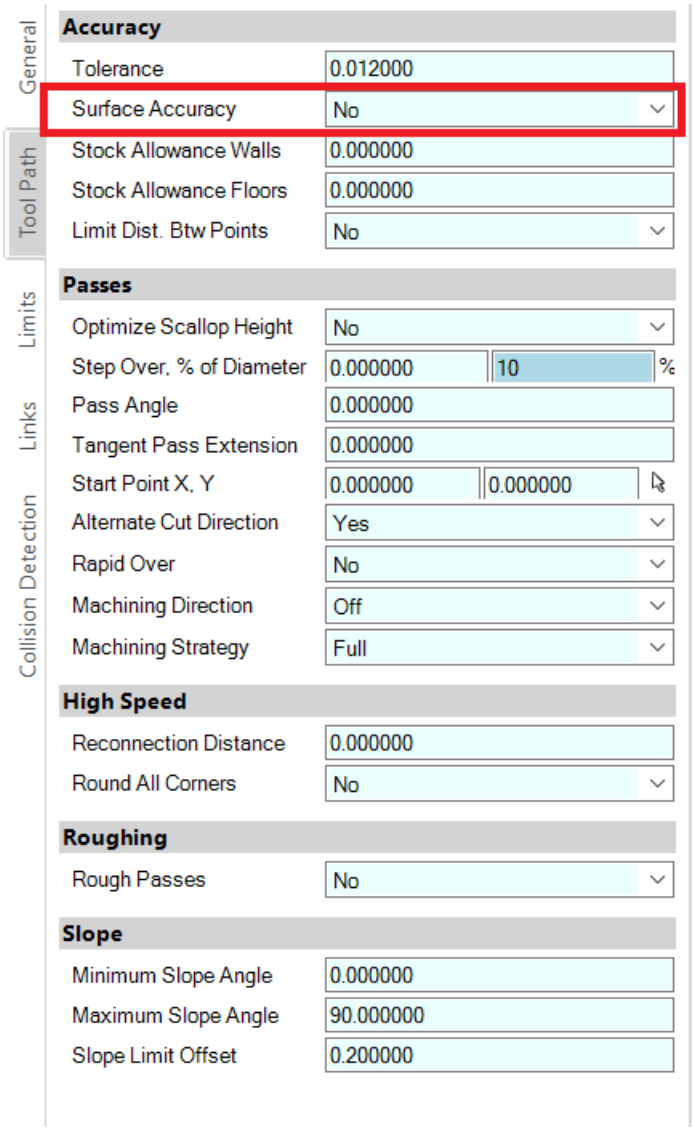
Surface Accuracy in Parallel Planes Finishing

In toolpath calculation, the system considers a triangulated representation of the part. The triangulation is a function of the 'FreeForm Geometry Tolerance' parameter, in File > Options > Machining. This means that any point on any triangle has a distance to the part geometry smaller than the tolerance. The triangulation is an approximated representation of the part to be machined.

In most cases this approximation is acceptable, but there are special cases, particularly in the automotive field, where an extremely accurate finish is needed. We have now realized a new method for which **the toolpath is calculated based on the analytical form of the faces/surfaces of the part**, and not on its triangulated representation.

Before introducing this new strategy, we have successfully tested it, cutting real automotive molds. These tests have shown better surface quality results with respect to the traditional method based on triangulation. In this first phase, we introduced the option to activate the new method only in the Parallel Planes Finishing cycles, but, in the future, we will extend its usage also to other finishing cycles.

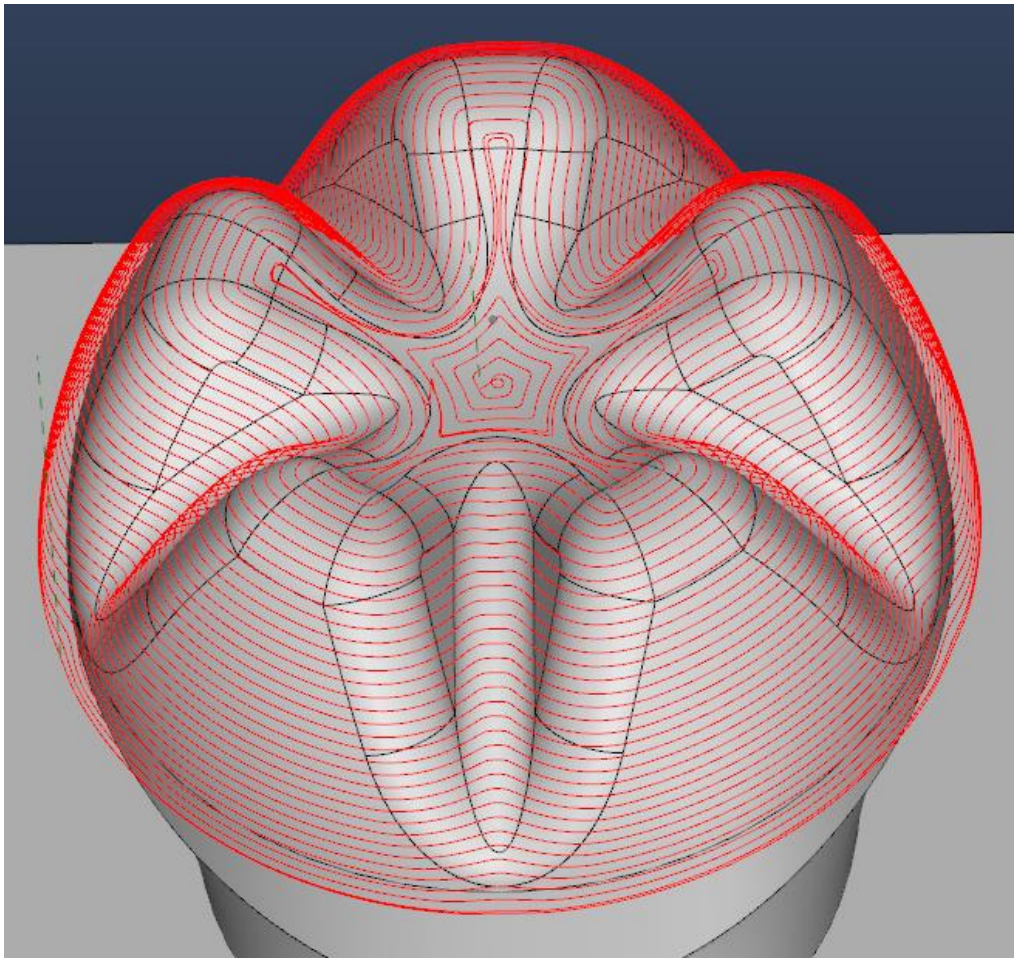
The new method has been introduced as an option because it leads to longer calculation times compared to the traditional method. Hence, we have introduced the new parameter 'Surface Accuracy' in the Accuracy section of the ToolPath tab, of the Parallel Planes Finishing cycle:



Rounding of Sharp Edges - Global Finishing Concentric Toolpath

In 2016 release R2.2, we released the new 3-axis Global Finishing cycle. The part is logically divided in steep and shallow areas, then the system then applies a Z-Level toolpath to the first and a Concentric toolpath to the second. The most prominent feedback from the field has been that, for the Concentric toolpath, internal sharp edges in the trajectory should have been rounded, to allow a real HSC behavior.

The new 'Rounding' parameter is now available to control this behavior.



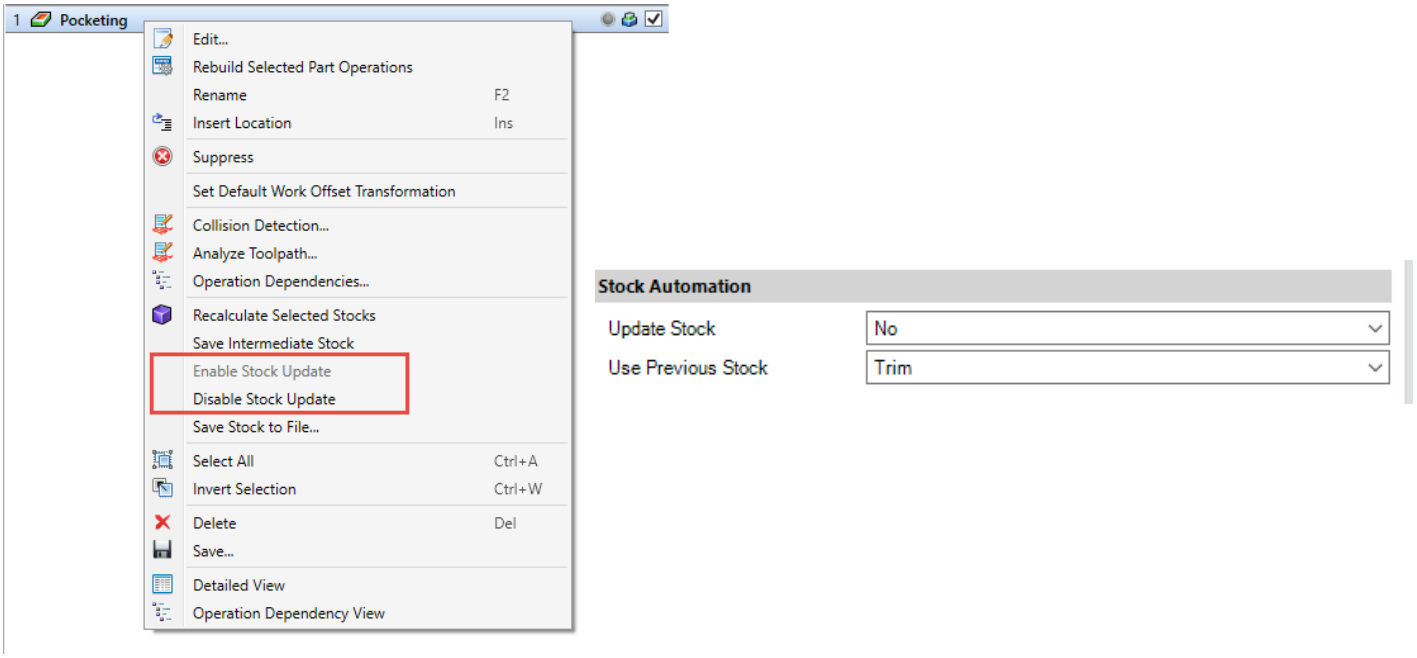
Stock on Demand

The new option, Stock on Demand, was added to operation pages to accommodate different programming workflows. This new option lends itself best to complex Molds and Dies, but also works well for any type or programming. Use of this new feature will yield significant performance benefits.

How it works

If you keep the Update Stock on Yes, the behavior will be the same as before; the stock is calculated on operation creation and the stocks will update after a program update.

If you disable the stock update, stock is not calculated on operation creation. Stock icon stays blank. This option can be toggled by right clicking on an operation on the Operations tab or inside of an operation on the General tab.



Default settings

For the following operations, which usually take long to calculate, the Update Stock will be deactivated by default:

- **3 axis finishing operations**
- **5 axis operations**
- **Lathe threading, Mill Threading**

WEDM Deburr Overlap Technology Change

When programming a Die, there are some situations where a burr would be left where the wire enters and exits the part. Typically, a Skim pass would remove this blip, but sometimes a Skim pass is not desired. To combat the blip that can occur, we introduced the Deburr strategy into 2 and 4 axis contouring operations for all machines excluding Agievision. After feedback from the field, it was determined that this new strategy could use a few enhancements.

The enhancements that were made for this release are only present when the overlap is performed on the rough pass. These settings are meant to control the wire's technology during the overlapped section to control items like over-burn.

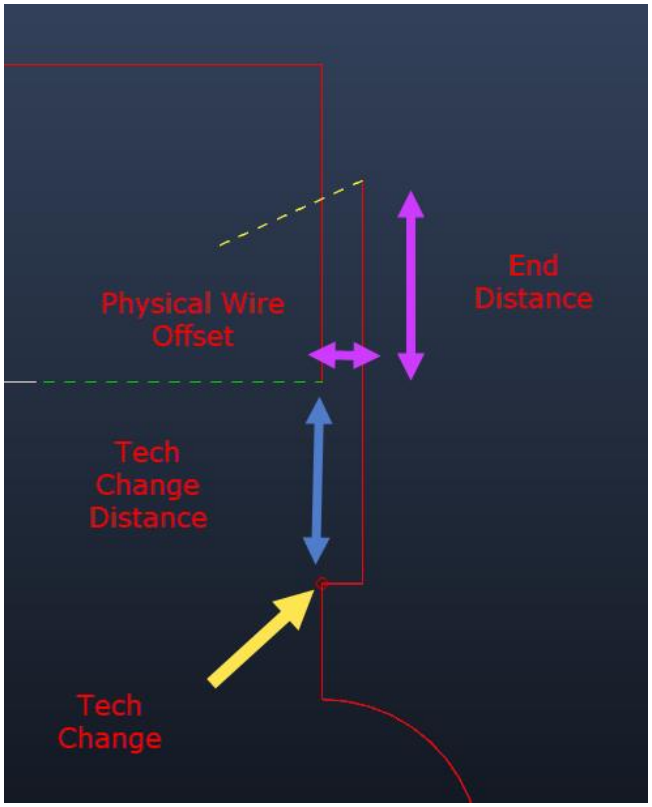
How it works

It is now possible to control 3 new parameters for the overlapped section of the toolpath:

- **Physical Wire Offset** – This is the physical distance in which we want to move the wire into the material. This is used to engage the wire into the material to counteract the forces of flushing. This setting is not required.

- **Technology Change Distance** – This is the distance before the overlap where you will output a technology change.
- **Inline Technology Change** – This text box accepts any string technology change the user desires. For example, you can type in a change in offset or power so that the wire does not over-burn the die cavity. Since each machine may have its own technology change commands, we opted for a text box where anything can be typed.

In the below figure, the new parameters are shown visually on a highly exaggerated toolpath.

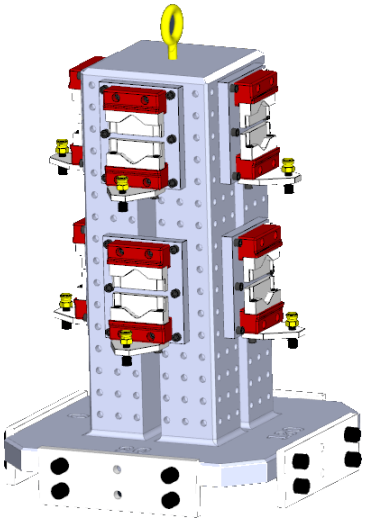


Enhanced WEDM Sorting

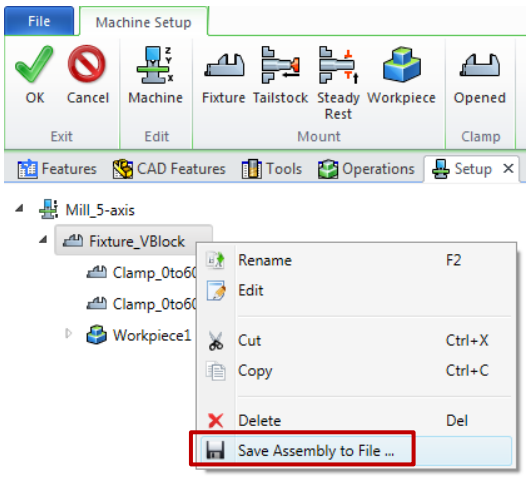
Wire EDM Advanced sorting, Per Cavity, has been enhanced. When one does a Per Cavity sorting, one would expect that all operations within a certain cavity would be grouped together. This has always been the behavior, however when two different operations existed in the same cavity, the order in which they were sorted sometimes resulted in logical, but erratic placement of operations. Now, if operations from two separate features machine the same cavity, they will be grouped together and ordered based on the order seen on the Operations tab. For this to occur, the two operations must share a common start position, within a pre-determined range.

Setup – What's new?

Save Fixture Assembly



Setting up your machine has become easier with the new feature to load fixture assemblies. After you've finalized your fixture setup and want to save it to be reused, just right click on the parent fixture and select Save Fixture Assembly to File. This will save the parent and children fixtures/jaws in their current orientation.



Fixture assemblies are loaded the same way as you would for any standard fixture. In case you want to lock the child fixtures/jaws so that they cannot be adjusted, you still have the option to save the assembly as a flattened file. This means that all the child components will be saved as a single fixture GDML file.

9/29/17

ESPRIT TNG Release 3.3

DP Technology Corp.

Introduction

Welcome to **ESPRIT TNG Release 3.3**.

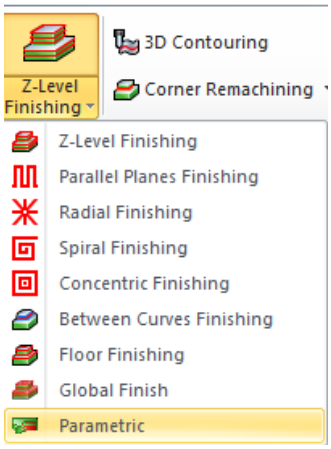
In this document, find a detailed description of the new features added this Milestone.

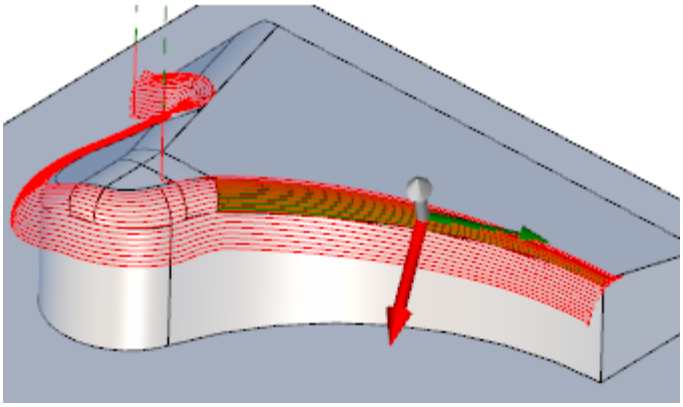
Next Milestone will be Release 3.4 scheduled for November 2017.

Machining and Simulation – What's new?

3x Mold Parametric Cycle

The new Parametric cycle creates a 3-axis finishing operation with cutting passes that follow the parametric flow lines of a selected face on the model.
A single face on the model is selected and a cutting direction chosen that follows either the U or V flow line direction of the face. Toolpath is generated on the model that follows the iso-parametric lines of the selected face.





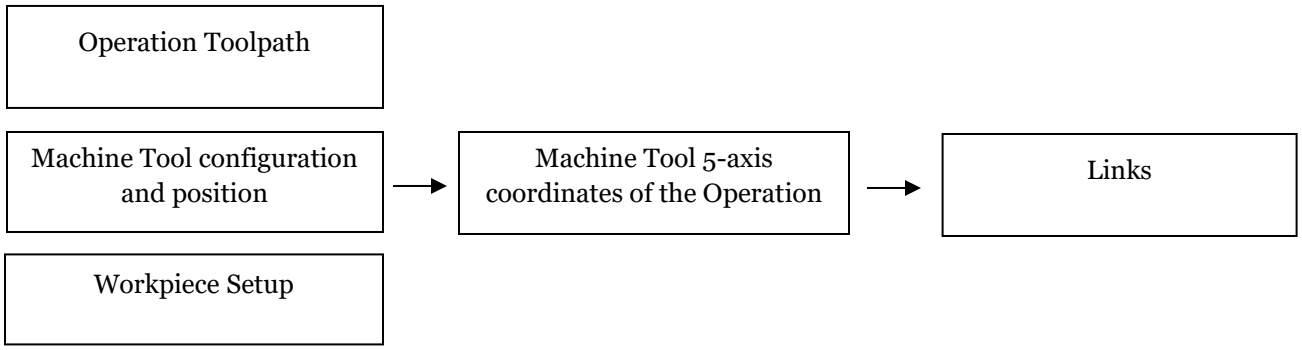
Machine Awareness Improvements

Machine awareness is the capability to calculate 5-axis toolpaths considering the structure of the Machine Tool, the axis limits and the Setup of the workpiece inside the Machine.

Machine Awareness guarantees the consistency between Simulation and Post-Processing, because it defines the optimal angular solutions for the rotary axes considering the Operation Toolpath, configuration and position of the Machine Tool and the Workpiece Setup.

In ETNG R3.3, Machine Awareness is by Default, as it improves the resolution of the kinematic issues which potentially occur in the 5-axis machining process. With Machine Awareness by Default, any change which affects the 5-axis Machine Tool behavior is automatically considered, so the user doesn't need to recalculate the operation.

The links inside the Program list are calculated after the Machine Awareness calculation, which defines of the rotary axes angular values at the start and the end of the operation.



Simulation of NC Compensated Toolpath

Improvements have been made to the algorithm that calculates the compensated toolpath for simulation. These improvements help to ensure the simulation of non-offset, but NC compensated toolpath, matches more closely to the actual machine behavior.

Setup – What's new?

Setup Changes – Split Workpiece

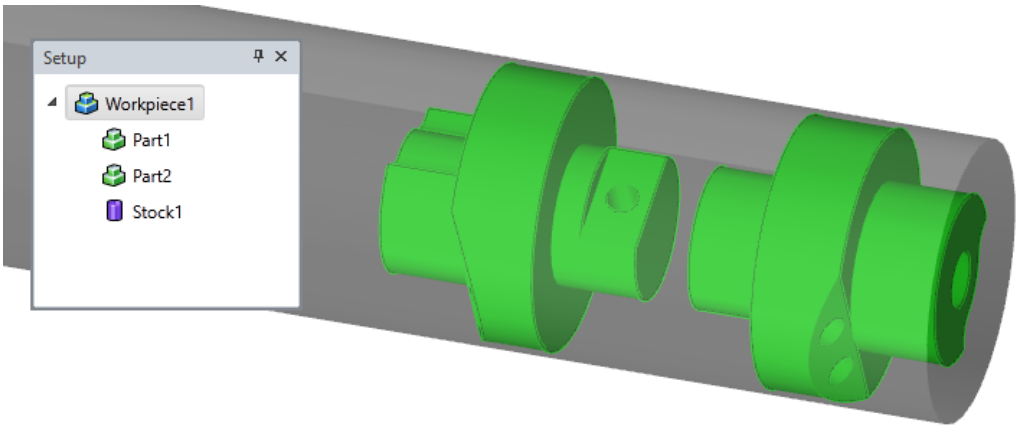
This feature is a continuation of the setup change that was introduced for multi-spindle programming.

When a workpiece contains more than 1 part, if a cutting operation separates the 2 parts, ESPRIT automatically generates a split event with 2 new workpieces.

This happens in turning with multi-spindle programming in 2 scenarios:

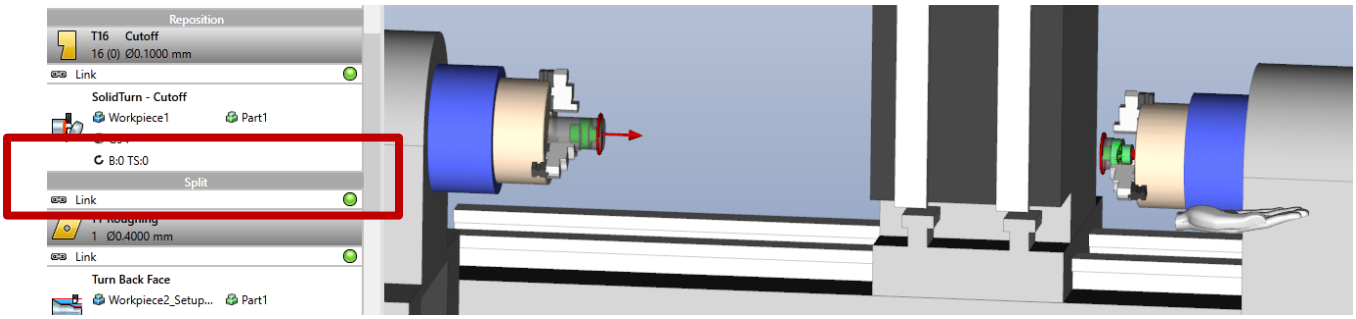
- Cleanup of the bar side after a transfer and to make the bar ready for next part.
- Programming Part 1 and Part 2, one behind the other in the same bar.

In both scenarios, edit the part setup to add a second part in the workpiece.

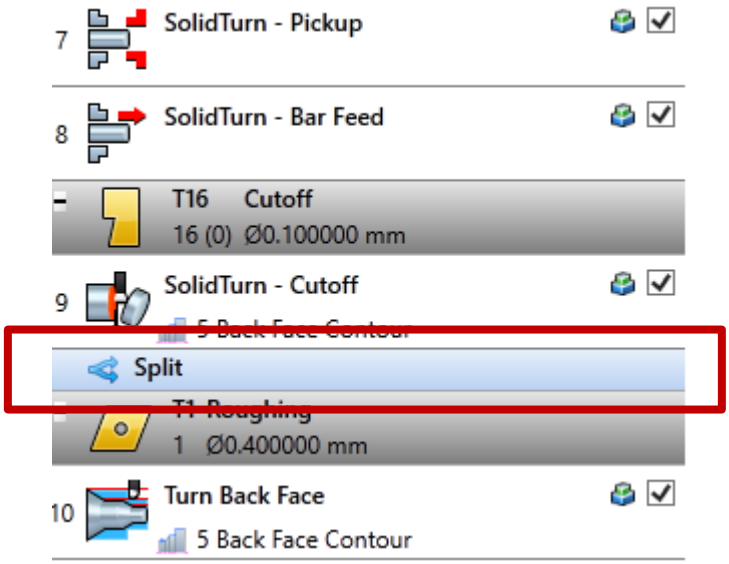


Then program the first part normally as if there was only 1 part in the workpiece.

After programming the cutoff, the system automatically detects the split of the workpiece. A new setup change Split is generated and new work offsets are automatically created following best practices.

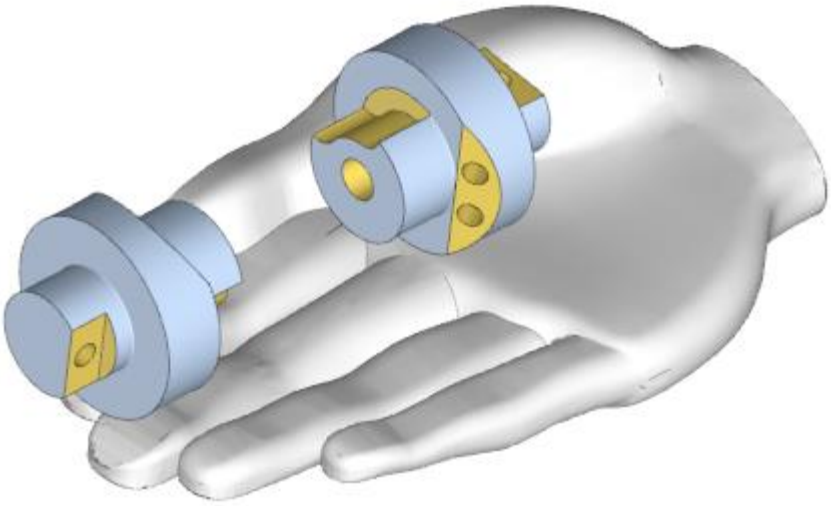


Split event is also added to the part operation list.

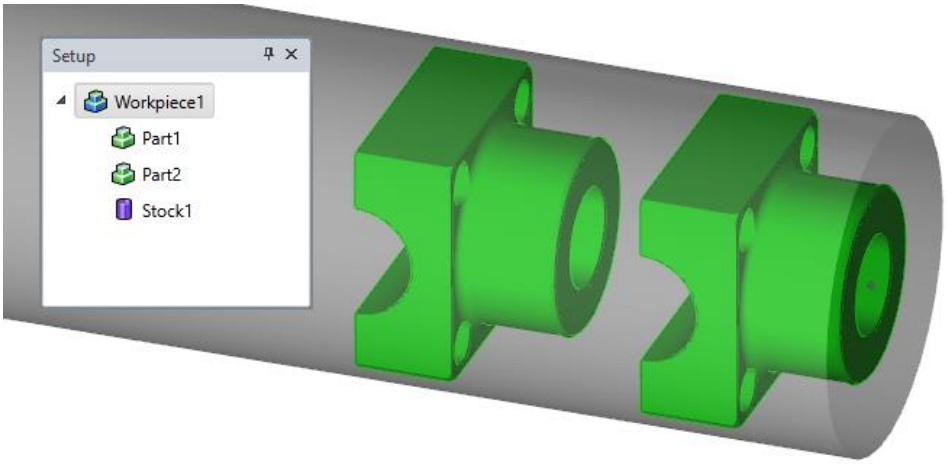


After the spilt, you can switch to Part2 and continue programming with second part.

When simulating a program cutting several parts in a row, ESPRIT aligns the ejected part in the part catcher.

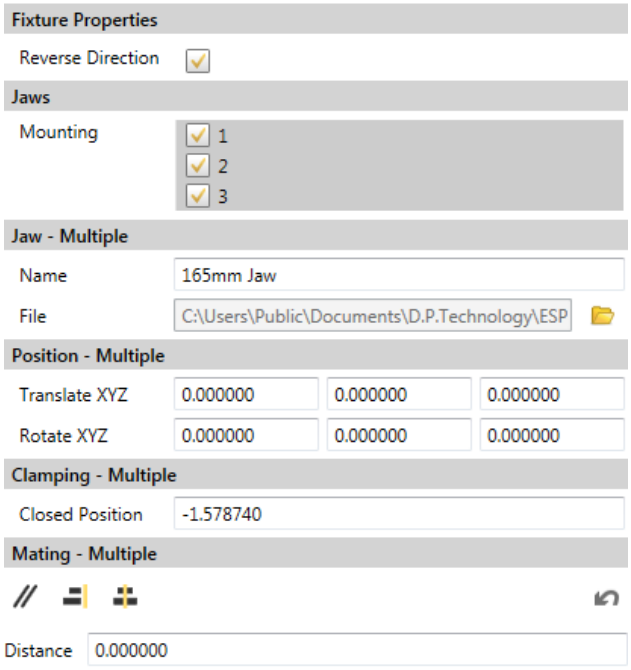


When you want to program just the cleanup of the bar for the next part, proceed with the same method but make a copy of the part behind the original part.



Jaws Enhancements

The Jaws interface has been enhanced to make editing simpler. All the jaw solids can now be edited inside this single interface. You can then use multi-selection to make multiple edits at once by CTRL+clicking the adapters.



The Jaws feature has also been extended to include Steady Rests. Now you're able to swap the arm components for modular enabled Steady Rests.

Better Machine Validation

There are additional enhancements inside Machine Tool Builder to verify if machines are valid for ESPRIT. Any issues will be identified in real time through the output window while creating a new machine or editing an existing one.

Also, ESPRIT has included new checks to make sure axes can be correctly identified so that Links are behaving as expected for simulation. This combined with the validation inside Machine Tool Builder should ensure correct NC code and machine simulation.

Usability – What's new?

Graphics Performance

Significant graphical performance improvements have been made, which directly affects the rendering and propagation of large, complex models.

8/9/2017

ESPRIT TNG Release 3.2

DP Technology Corp.

Introduction

Welcome to **ESPRIT TNG Release 3.2**.

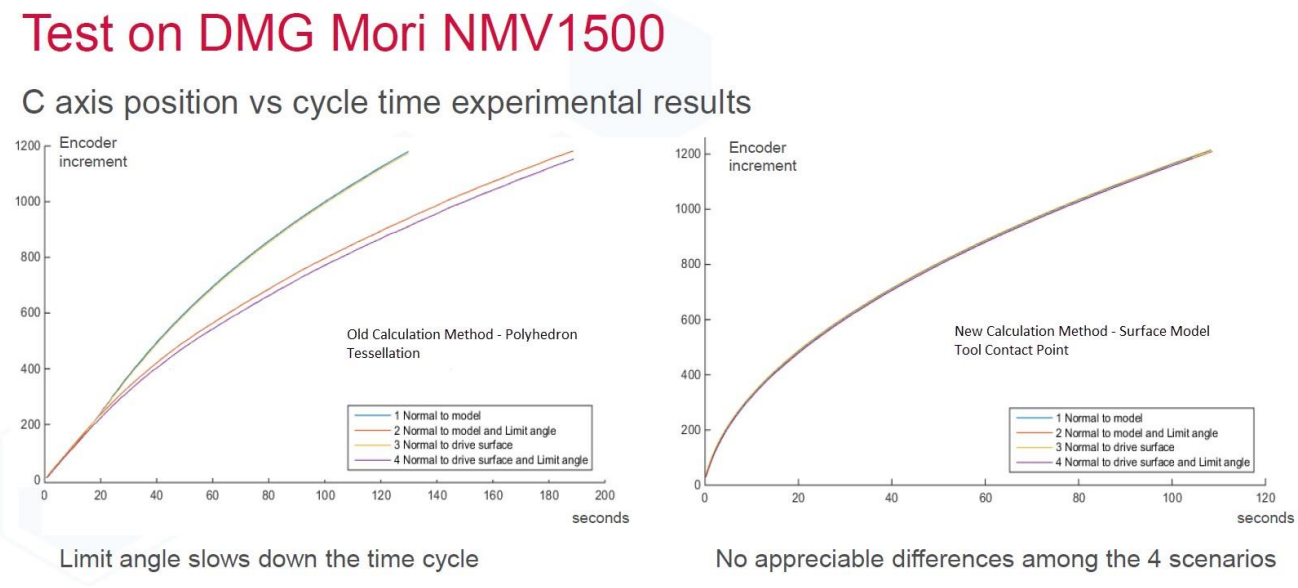
In this document, find a detailed description of the new features added this Milestone.

Next Milestone will be Release 3.3 scheduled for October 2017.

Machining and Simulation – What's new?

Surface Model Tool Contact Point

To give the user the best 5-axis toolpath, a new calculation method has been developed, implemented, and enabled by default. This new calculation aims to improve the surface quality of 5-axis toolpath. The previous calculation method can produce irregular toolpath because the model is tessellated and considered as a polyhedron. The new calculation positions the tool on the real parametric surfaces, which reduces inconsistencies and increases fluidity. The user will notice a slightly longer computational time. However, the tolerance in the operation can typically be set looser. In the past, the tolerance affected the tessellation of the model, which directly affected the surface quality. With the new calculation, the toolpath is calculated directly from the parametric surfaces and the tolerance is the linear interpolation between two points of the toolpath.

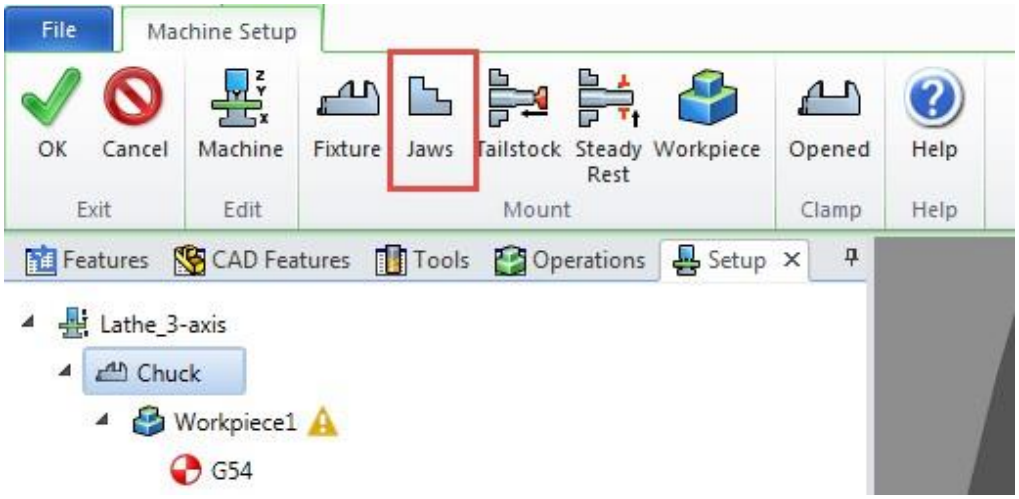


From Machine Tool tests, the new calculation method shows a smoother and more fluid C axis movement.

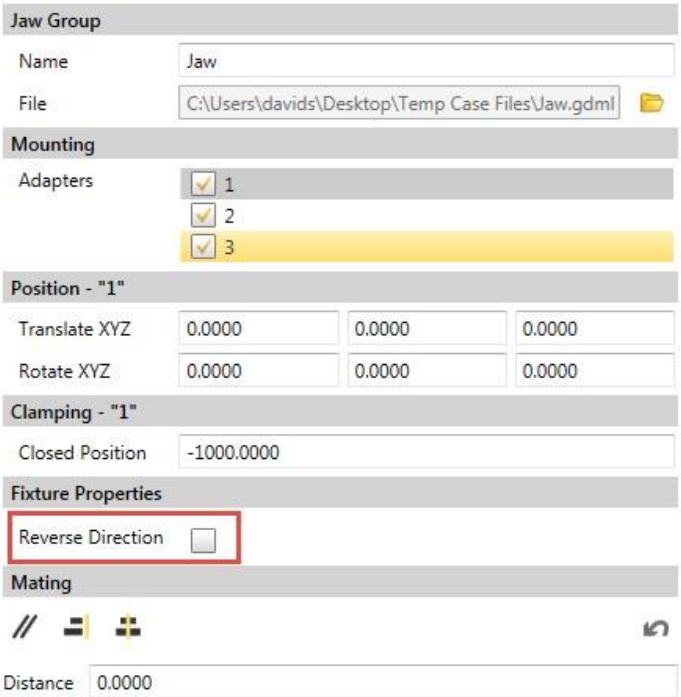
Setup – What's new?

Jaws

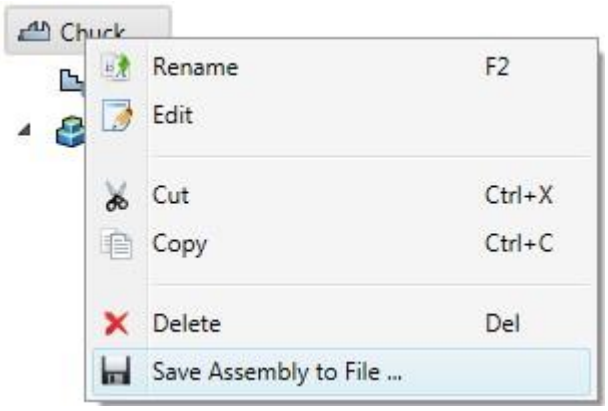
Many improvements have been made to Chuck and Jaw fixtures. Fixtures now accept modular jaws. These modular Fixtures can be easily created within TNG rather than in MTB. For a Fixture to accept modular jaws, it must be saved with certain work planes which denote Jaw adapter locations.



In the past, two different fixtures for ID and OD chucks were required. Now, with a click of a button, the orientation of the modular jaws can be flipped.



It is now possible to translate and rotate modular jaws when adding them to a chuck fixture. Mating can also be used to manipulate the location and orientation of each jaw. It is important to ensure the jaw which is being mated is selected in the Mounting Adapters section.

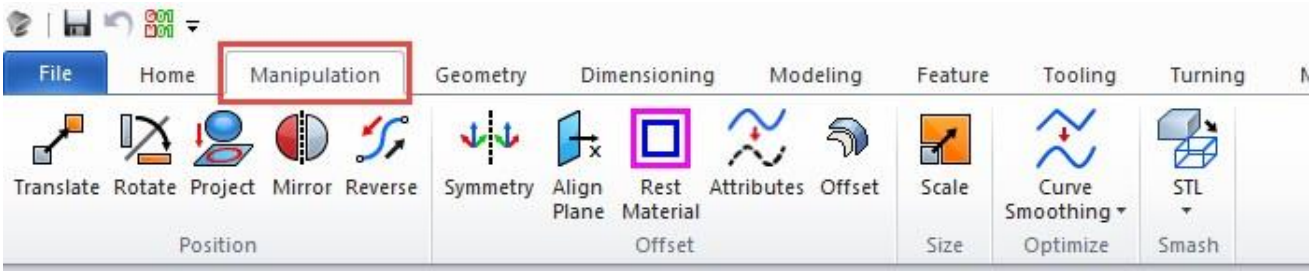


As seen above, it is possible to save a Fixture assembly to file. Once this has been done, the assembly is saved flat and the components, like the jaws, can no longer be swapped. Look for more improvements in the future.

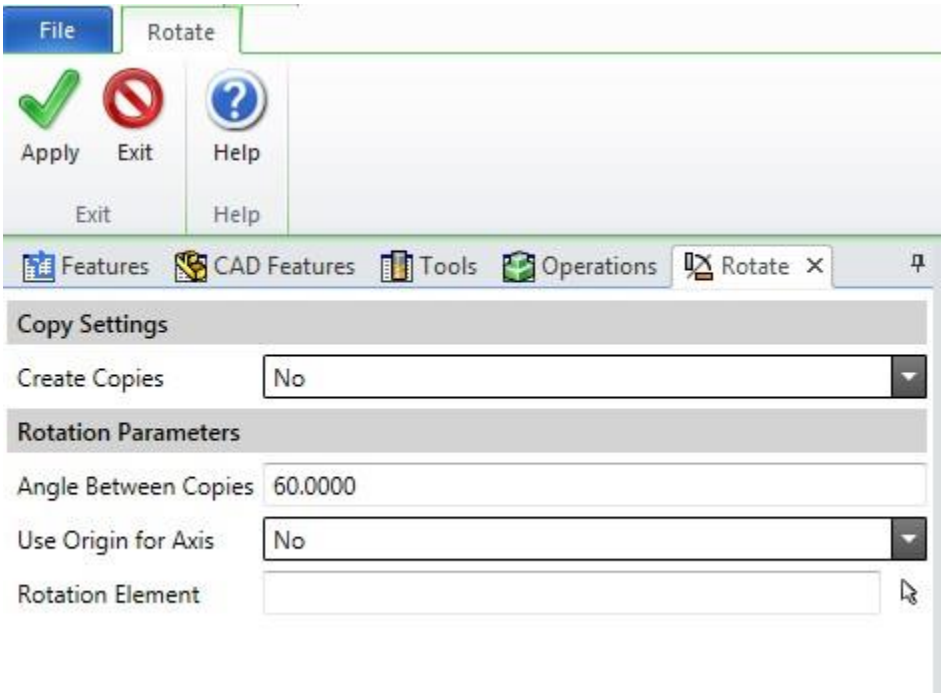
Usability – What's new?

Manipulations UI

The old right-click copy commands have been copied to a new location, the Manipulations ribbon. This new ribbon makes it easy to find the familiar commands like Translate, Rotate, etc. Look in the future for more updates to these commands.



With the old method, the user was limited to applying whichever command a single time per right-click copy. With the new Manipulations ribbon, once a command is open, the user can select items and press apply as many times as necessary. Once the user is finished manipulating, simply press Exit.

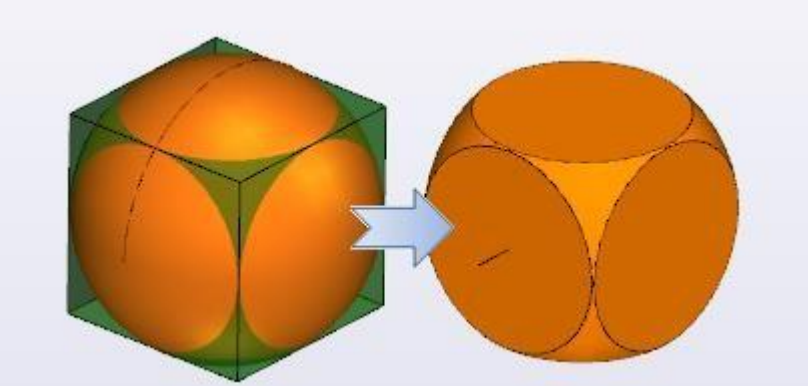


CAD – What's new?

Boolean Solid Modeler

Boolean Solid Modeler combines two or more solid bodies or sheets to create a new result. Boolean functions provide an effective way to create complex shapes out of simple geometric objects.

Three functions are available: Union, Subtract and Intersect. A Main Body is selected that defines the solid body to be modified. One or more Tool Bodies are selected that define the solid bodies that will be combined, subtracted, or intersected with the Main Body.



Machine Tool Builder – What's new?

Features

- Import and Export of Custom Setting Localization
- Added Bar Feeder to Machine Wizard
- Added Steady Rest to Machine Wizard
- Support for Multiple Turrets and Workpiece Carriers in the Machine Project File
- Updated Milling Custom Settings for Machine Wizard and EMS Converter
- Machine Type Set to Generic as Default for WireEDM Machines

Bug Fixes

- Cannot Set Work Offset Transformation: Rotate Auto Yes No Translation
- EMS Converter Doesn't Work for B Axis Machines if B Vector Set to -1
- Tailstock Doesn't Add Axes in the Machine Wizard

6/8/2017

ESPRIT TNG Release 3.1

DP Technology Corp.

Introduction

Welcome to **ESPRIT TNG Release 3.1**.

In this document, find a detailed description of the new features added this Milestone.

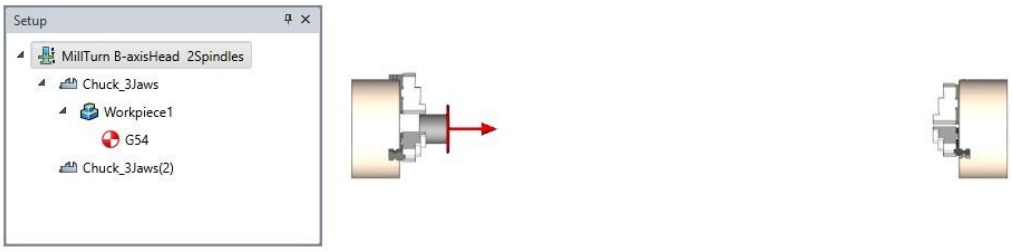
Next Milestone will be Release 3.2 scheduled for August 2017.

Machining and Simulation – What's new?

Program Mode

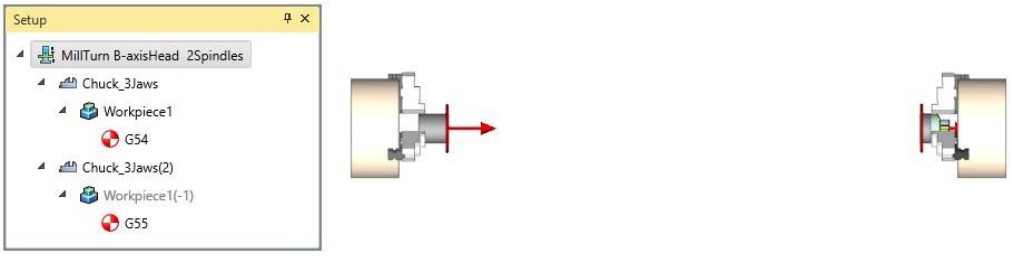
Program Mode allows the user to switch between Sequential and Parallel programming. What does Sequential or Parallel mode refer to?

Sequential Setup:



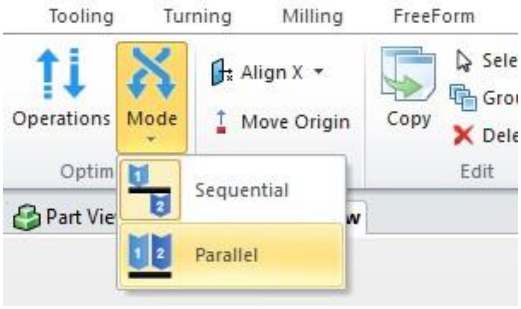
In Sequential mode, the workpiece is loaded in only one spindle.

Parallel Setup:



In Parallel mode, we overlap the start and end of the program. This creates the previous instance of the half cut part loaded into the sub-spindle, thus cutting both sides simultaneously, or in parallel, on the machine.

The Program Mode can be toggled on the Home tab of the ribbon.



To toggle between Sequential and Parallel Mode, there must be an odd number of Transfers and an Eject at the end. When switching between the two modes, work offsets are preserved.

If any modification to the program breaks the requirements for Parallel mode, the program will switch back to Sequential mode.

Program/Operation optimization will minimize tool changes without breaking the part operation order and is very useful for Parallel mode programs. This optimization can be done globally or on selected operations.

Tool Change Position

Tool change positions are now an event of the program. When the same tool is used multiple times in a program, each tool change event can be edited and the tool change position can be altered. Default tool position comes from the machine itself and can be modified in MTB.

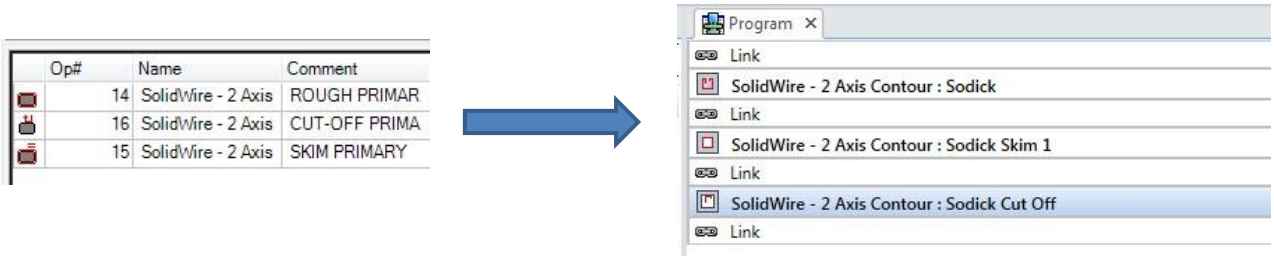
Machine Awareness by Default

Machine Awareness now looks ahead for initial rotary movement to ensure the best rotary solution possible. This helps to limit unnecessary rotary movement.

Machine Awareness also looks ahead to see if a linear axis limit will be met. If the limit will be met, the tool path calculation will take that into account and reorient the C-axis appropriately to make the cut.

Wire EDM Smart Operation Dependencies

Operation dependencies now exist for Wire EDM. No longer can the user reorder children operations into a way which could cause an issue on the machine. In E20xx series, the user can drag children operations into any order desired, which often causes illegal orders and issues at the machine. As an example, a user could program a Contouring operation with 4 skim passes, then take the last skim pass and move it above the rough pass of the same operation. This skim pass would surely break the wire and cause issues when ran on the machine. In TNG R3.1, operation dependencies will not allow these illegal moves to occur.



In the above example, on the left are the operations programmed in E20xx. The user moved the Cut-Off operation above the skim. However, the operation was programmed as [Rough, Skim, Cut-Off] and that order must be retained to remain legal. When a legacy file is opened in TNG, the order of operations is automatically reordered to the legal order, which is seen on the right.

Wall Feature Recognition Enhancements

Wall feature recognition has been significantly improved with increased feature intelligence. Now, both top and bottom blend types, including chamfers and fillets, are correctly recognized. Locations where loops were previously created have been eliminated.

Setup – What's new?

Mating

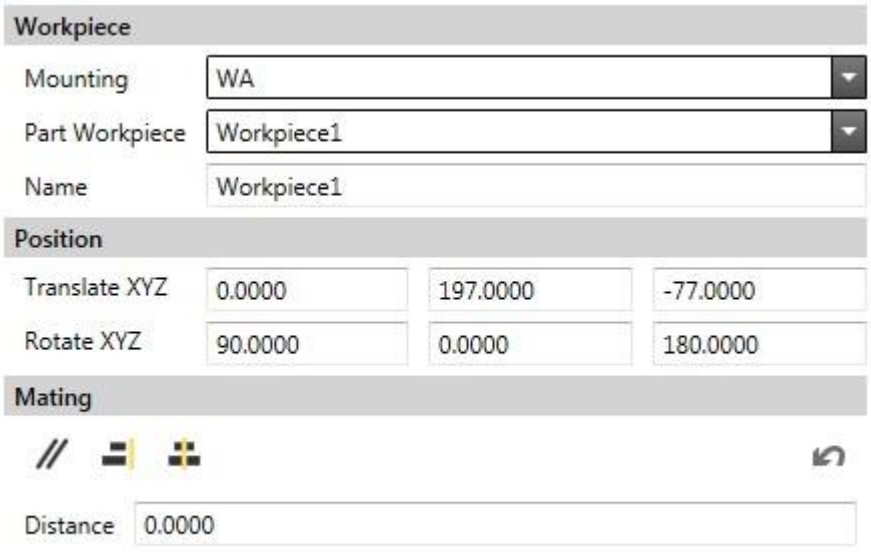
Easily move fixtures and workpieces, without having to guess values, using Mating.

There are three types of mating:

- Parallel //
- Alignment =

- Centering

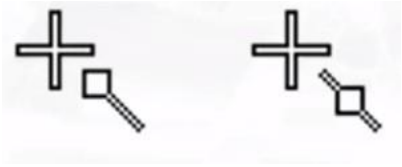
This not only works for the translation but also the rotation fields in the workpiece setup.



Usability – What's new?

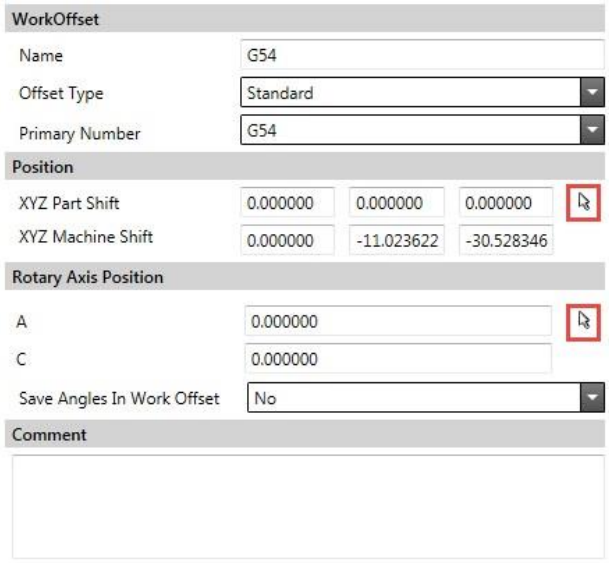
Snap

Snap mode has been enhanced allowing the user to snap on end and midpoints of edges on a solid model. It is no longer necessary to smash the edge of a solid to create snap points. This functionality is available whenever a reference point is required as an input.



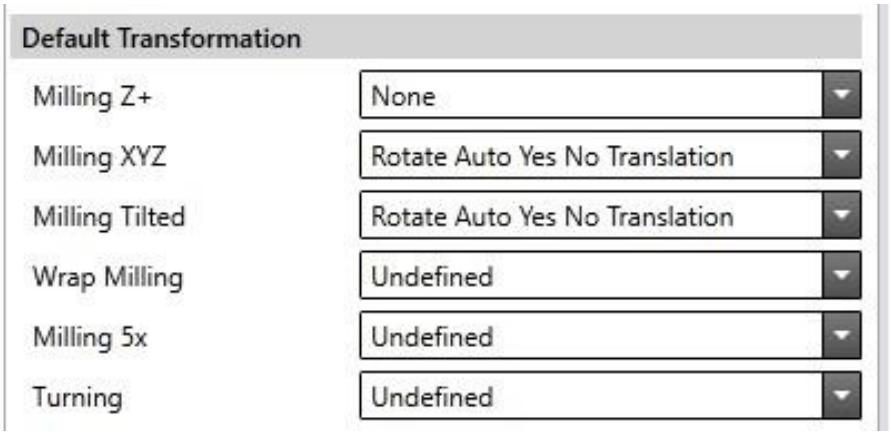
Digitization of Work Offsets

Faces are now valid input when digitizing the Work Offset Position. The Part Shift and Rotary Axis Position can both be defined using faces as inputs.



Work Offset Transformation – Rotate Auto No Translation

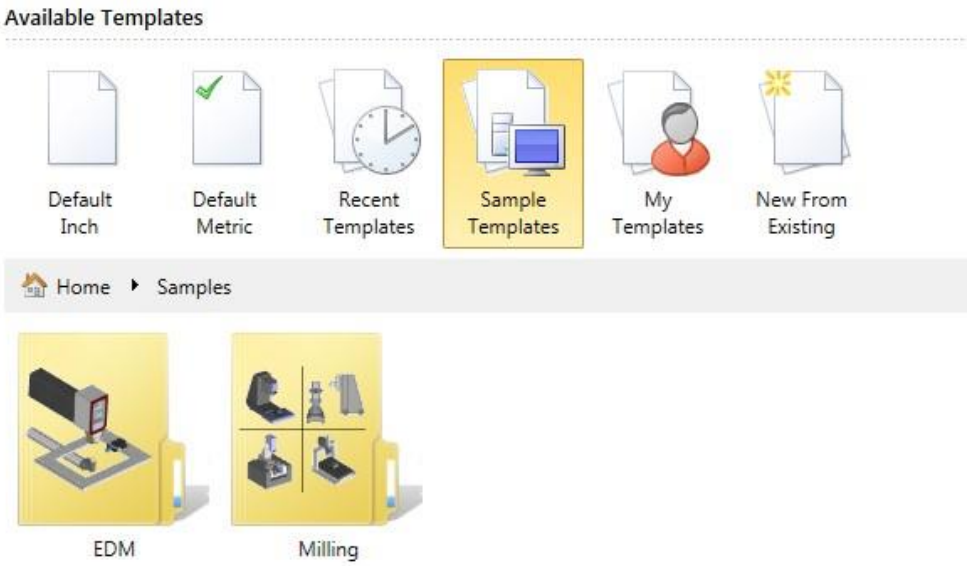
This new Work Offset Transformation mode is available for Mill Z+, Milling XYZ and Milling Tilted.



This mode will yield the same Part Shift values as the original Work Offset but will rotate the I J K based on the operation. If you were to output a G68.2, the NC code would be: G68.2 X0 Y0 Z0 I J K. This produces the same behavior as the Turning Work Coordinate add-in in E20xx.

Starting Files

TNG now comes preinstalled with Sample Templates for the user to start from. Currently, TNG includes both EDM and Milling starting templates. These templates include generic Machines loaded with tools and holders.

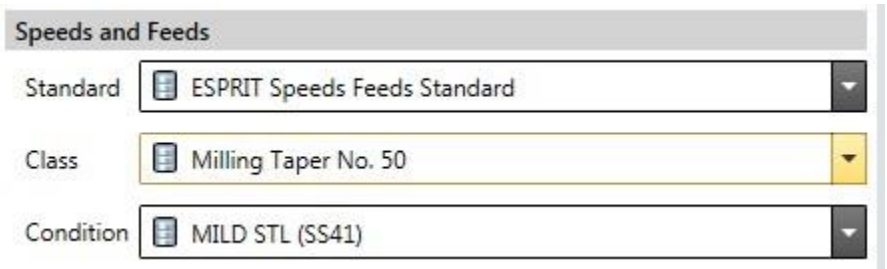


Upon installation of TNG, a variety of sizes of all standard Milling and EDM machines, as well as Holders and Fixtures, are installed. This allows the user to create different templates based on his or her own needs and requirements.

The Standard Cutting Tools in the KBM have also been updated to include realistic cutting lengths. Lastly, the installed Process Technology has been updated to be comparable to E20xx version. The updated Part Type is now named ESPRIT TNG Inch\Metric and functions for both Milling and Turning.

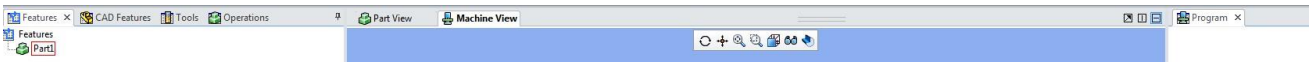
New Feeds and Speeds

CutData has been replaced by an updated Feeds and Speeds database with more relevant data. This is also provided as a courtesy inside of TNG, rather than as an option like CutData in the past. This Standard has been named ESPRIT Speeds and Feeds Standard.



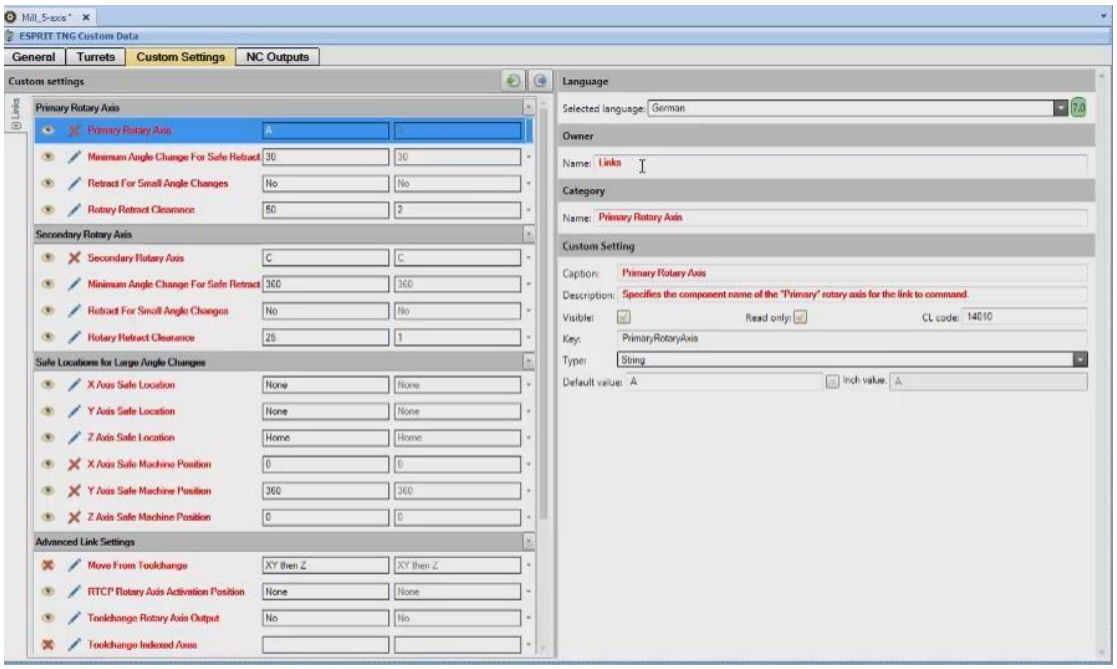
Interface Update

Part operation list is now renamed Operations. Machine operation list is now renamed Program. Part is now renamed Part View. Machine is now renamed Machine View. Prior to R3.1, TNG had several items in the interface with the same name. With the changes mentioned above, these inconsistencies have been addressed and it is easier to understand what each dialogue corresponds to.

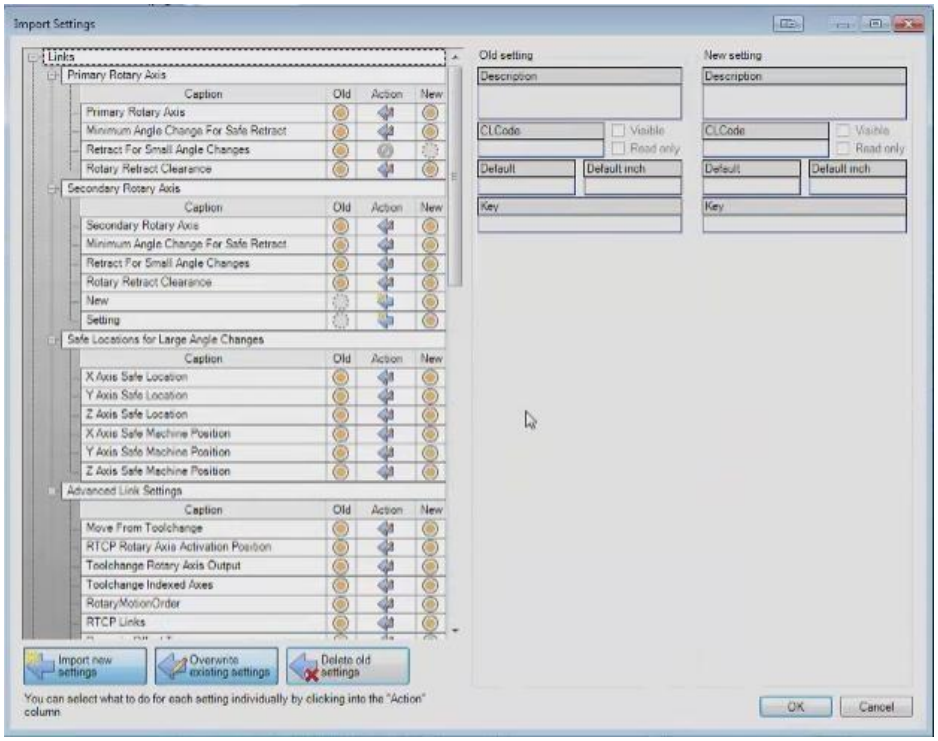


MTB Custom Setting Localization

MTB Custom Settings have a new user interface with new a Custom Data Version. This new feature allows the user to edit multiple languages inside of one MPRJ file, whereas previously multiple MPRJ files would have been necessary. Once the custom data language is changed, un-translated items will show up in red to ensure the user won't forget to translate a field.



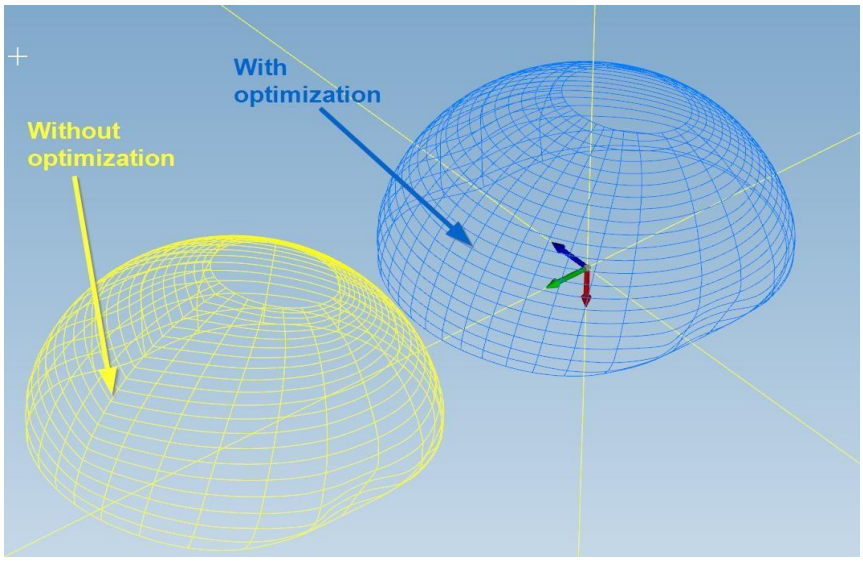
The importer in MTB has been updated and now allows merging of the custom data. The data that is imported will retain its localization.



CAD – What's new?

Knitted Surface Improvement: Seamless Parametric Curves

A new parameter, Optimization, has been added for Closed Knitted Surface creation. This parameter considers the entire surface for parametric lines and removes seams that used to exist. With this new setting active, the calculation time will increase, but when using this surface for projection, the toolpath will be smoother than it previously was.



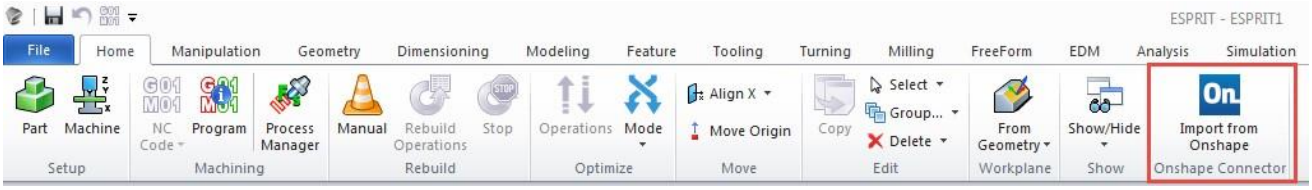
CAD Data Exchange Update

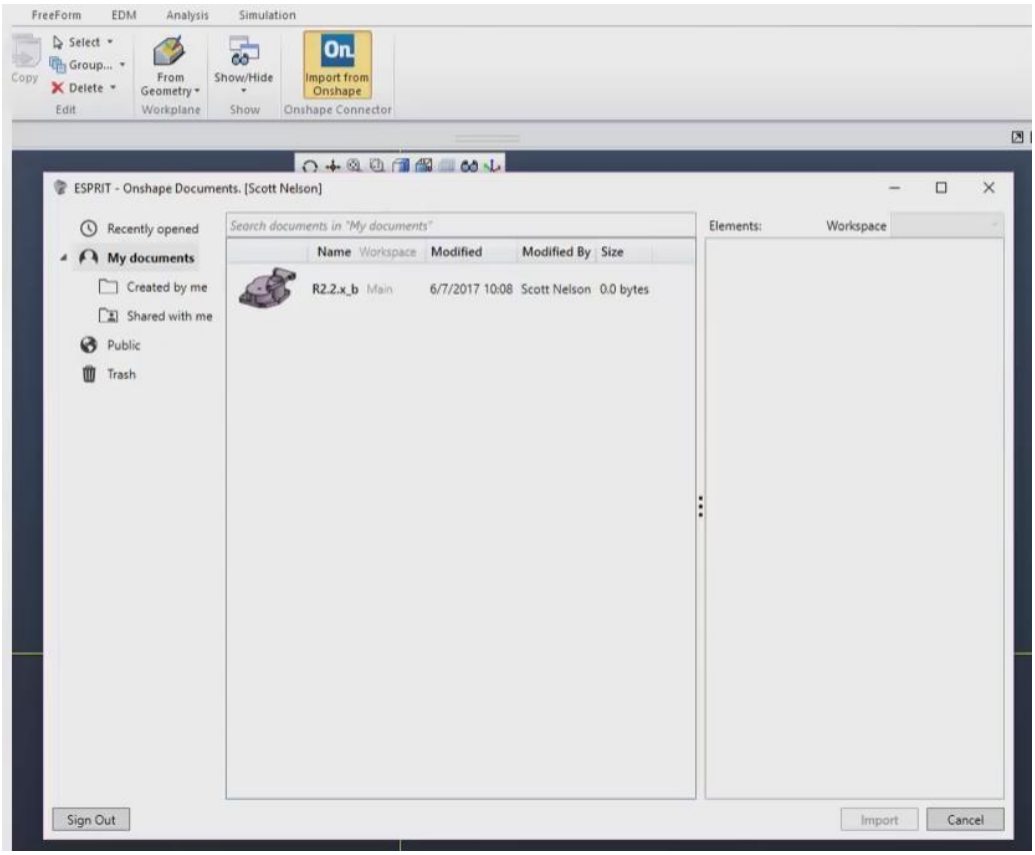
The CAD Data Exchange has been updated and includes many positive changes.

- Improved import quality
- Better provider support, which equals reduced turnaround time for bug fixes
- The ability to import/export STEP files is included in the base license
- **New** support of **3D PDF** files (your license will need to be renewed for this new bit to be active)
- No FX tree for Catia, Pro/E, or UG, however we are in the process of replacing this with superior technology (PMI)

Onshape Connector

Onshape is a new extension we have added to TNG R3.1. The Onshape extension allows you to connect to your Onshape account (cloud service for storing files) and import the files directly into TNG.


ESPRIT - ESPRIT1
File
Home
Manipulation
Geometry
Dimensioning
Modeling
Feature
Tooling
Turning
Milling
FreeForm
EDM
Analysis
Simulation
Part
Machine
NC Code
Program
Process Manager
Manual
Rebuild Operations
Stop
Operations
Mode
Align X
Move Origin
Copy
Select
Group...
Delete
From Geometry
Show/Hide
Import from Onshape
Setup
Machining
Rebuild
Optimize
Move
Edit
Workplane
Show
Onshape Connector


FreeForm
EDM
Analysis
Simulation
Copy
Select
Group...
Delete
From Geometry
Show/Hide
Import from Onshape
Edit
Workplane
Show
Onshape Connector
ESPRIT - Onshape Documents. [Scott Nelson]
Recently opened
My documents
Created by me
Shared with me
Public
Trash
Search documents in "My documents"
Name
Workspace
Modified
Modified By
Size
R2.2.x_b
Main
6/7/2017 10:08
Scott Nelson
0.0 bytes
Elements:
Workspace
Sign Out
Import
Cancel

3/31/2017

ESPRIT TNG Release 3.0

DP Technology Corp.

Introduction

Welcome to ESPRIT TNG Release 3.0.

This is the first Release of ESPRIT TNG with support for Multi-Spindle machines.

In this document, find a detailed description of the new features added this milestone.

Next Milestone will be Release 3.1 scheduled for June, 2017

Multi-Spindle – What's new?

Setup Changes

The machine setup defines the initial setup of the machine at the beginning of the program.

Operations such as Pickup, Release, Bar feed, Cut-off, can modify the setup in the middle of the machining process.

Modifications to the setup are represented by **Setup Changes** inserted in the machine operation list.

Setup Change 1

Setup Change 2

Setup Change 3

To edit a Setup Change, double click on the gray bar in the machine operation list.

When editing a Setup Change, ESPRIT shows the machine in the state of the new setup.

Only work offsets can be edited in a setup change. Fixturing and workpiece position are the result of the initial setup and of the programmed operations.

There are 4 types of setup changes:

- **Reposition:** Result of a bar feed operation. Workpiece is repositioned in the fixture but stays in the same fixture.
- **Transfer:** Result of a pickup/release or pickup/cutoff. Workpiece is relocated into a different fixture.
- **Move:** Workpiece stays in the same position in the fixture but the whole fixture moves relative to the machine coordinate system. Could be the result of a pickup by main spindle of workpiece located in sub spindle.
- **Eject:** Result of a Release cycle. A fixture releases the workpiece and workpiece is not held anymore. Usually happens at the end of the program to eject the completed workpiece.

Automatic definition of Work Offsets after a setup change

When programming a transfer or any operation generating a setup change, the work offset for new setup is automatically generated based on common practice:

- **Transfer Main to Sub:**
  - ESPRIT increments the work offset automatically by default (G54 becomes G55).
  - The new work offset moves to the back face of the part (the face opposite from where it is held).

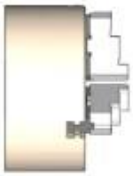
    INITIAL SETUP:

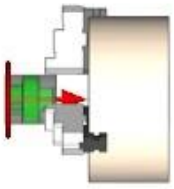
    AFTER TRANFSER:

- **Move (Main spindle picks up workpiece in sub spindle):**

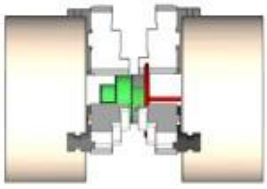
- ESPRIT increments the work offset automatically by default (G54 becomes G55, if G55 in sub, G54 is created).
- The new work offset moves to the back face of the part (opposite face from where it is held).

INITIAL SETUP:



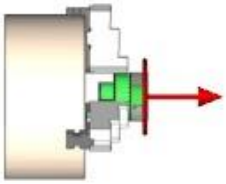


AFTER PICKUP:



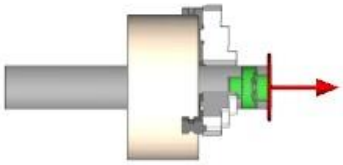
- **Transfer Sub to Main (after Move):**
  - Transfer uses same work offset as Previous Move Setup change.

AFTER TANSFER:

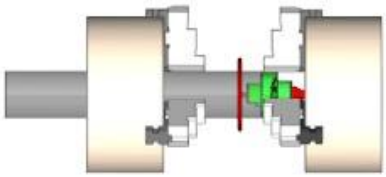


- **Barfeed, Reposition:**
  - Reposition setup change maintains same setup change as in previous setup.
    If G54 on front face, after reposition, program stays with G54. G54 is inside part after bar moved forward.

INITIAL SETUP:



AFTER BARPULL:

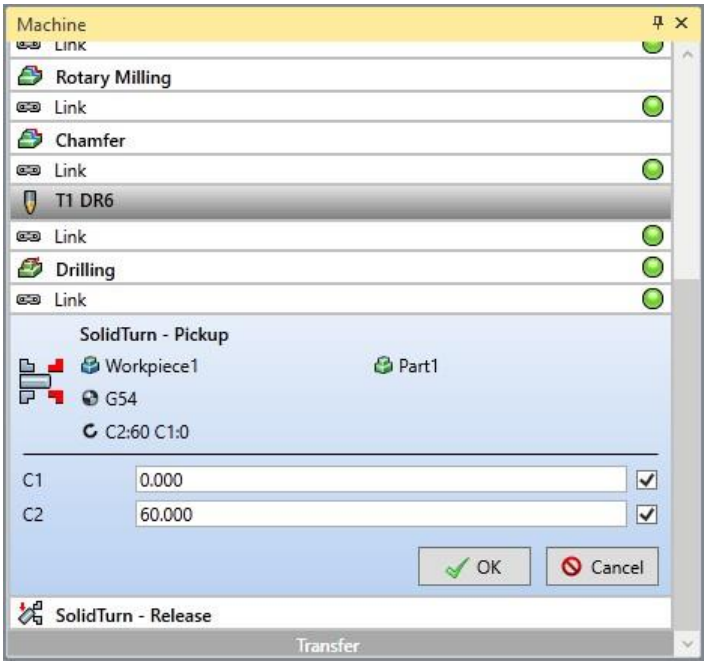


Pickup Cycle and C-axis Indexing

On creation of a Pickup cycle, ESPRIT synchronizes the C-axis of main spindle and sub spindle at 0 degree.

It is possible to change the position of the C-axis during a pickup.

In the machine operation list, select the Pickup Cycle and right click to edit Rotary Axes.



Input the C-axis values for pickup or uncheck the box on the axis line to keep the axis at its previous position.

Links Configuration for Multi-Spindle machines

On multi-spindle machines, the available travel on the Z axis can be very large. When the tool returns to a "Z Axis Safe Location" the user may want multiple machine positions available to avoid unnecessary travel.

How?

- The Z Axis Safe Location (as well as X and Y) can be set per kinematic chain by appending the name of the kinematic chain at the end of the key of the link custom setting.
- The custom setting is optional and may be specified for any number of the machine's kinematics (or none of them). Note that normally, it is expected to only define these parameters for B-axis main/sub kinematics (not turrets).

- The system looks for a matching kinematic chain first, then uses the “normal” Z Axis Safe Machine Position. Legacy data keeps the same behavior.
- All values are measured from the same machine coordinate system 0 position defined by the start node of the kinematic chain.
- Make sure to reference the same MCS for main and sub spindle kinematic chains.

Example:

Assume an example with a single B-Axis and two spindles, thus two kinematics:

1. Main Spindle - B-Axis
2. Sub Spindle - B-Axis

A user can define two custom settings:

- "ZAxisSafeMachinePosition"
- "ZAxisSafeMachinePosition_Sub Spindle - B-Axis"

These parameters would be utilized when the B-axis rotates greater than the "Minimum Angle Change for Safe Retract" and "Z Axis Safe Location" is Machine Position. Note that the user could also explicitly define "ZAxisSafeMachinePosition_Main Spindle - B-Axis".

If 1st and 2nd operations are on the main, the tool will move to the standard "ZAxisSafeMachinePosition".

If 1st and 2nd operations are on the sub, the tool will move to "ZAxisSafeMachinePosition_Sub Spindle - BAxis".

If the 1st operation is on the main, and 2nd operation is on the sub, the tool will move to the "ZAxisSafeMachinePosition".

If 1st operation is on the sub, and 2nd operation is on the main, the tool will move to the "ZAxisSafeMachinePosition_Sub Spindle - B-Axis".

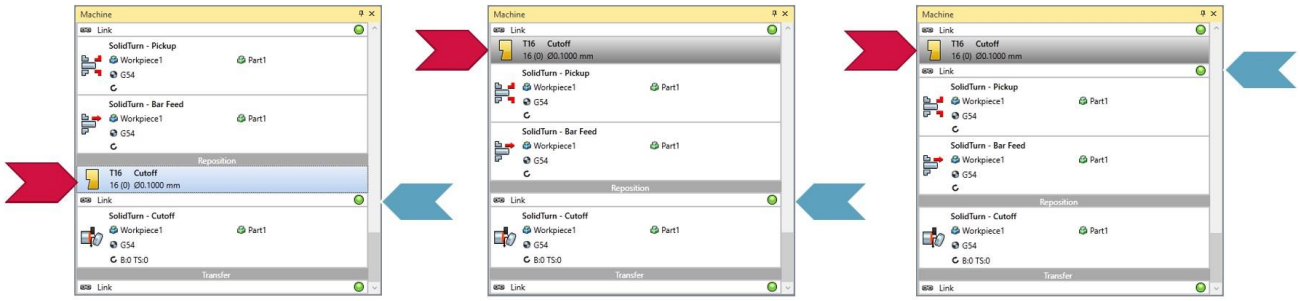
Note that if "ZAxisSafeMachinePosition_Sub Spindle - B-Axis" is not defined in the mprj, the "ZAxisSafeMachinePosition" would be used in lieu.

Links and operations without a tool

With ESPRIT 2000 series, controlling when to drop (cancel) a tool during a transfer sequence was handled by the post processor. It made the post processor logic very complex and did not give the user any control over when the tool gets dropped.

With ESPRIT TNG, you have full control over when the tool change happens and when the tool retracts and approaches.

Drag-and-drop the operation or link to control the approach link, retract link, or tool drop/change in the machine operation list. Set your own sequence of actions and the NC code will follow this sequence.



Multi-Spindle and Post Processing

There are specificities in the post processor for a multi spindle machine:

- **How to know the working spindle and state?**
- SpindleCode : SpindleMain / SpindleAux
- Spindle ID = NextClFile(793)
- SpindleDirection returns specific values for main and sub spindle
- **Setup Changes:**
- E20xx examples are carried over: Ex_Barfeed / Ex_PartRelease / Ex_PartPickup
- **NEW** set of examples for Setup Change:

  Ex_Setup_Transfer

  Ex_Setup_Split

  Ex_Setup_Eject

  Ex_Setup_Reposition

  Ex_Setup_Move

  These examples are output after an operation that caused a setup change.

SolidMill – What's new?

Pocketing and Contouring Stock Automation

The same Stock Automation enhancements applied to Facing have been applied to SolidMill Pocketing and Contouring.

With accurate knowledge of in-process stock, the system can automatically calculate:

- Starting Depth
- Retract for IDepth
- Open Edge Offset
- Entry/Exit Moves

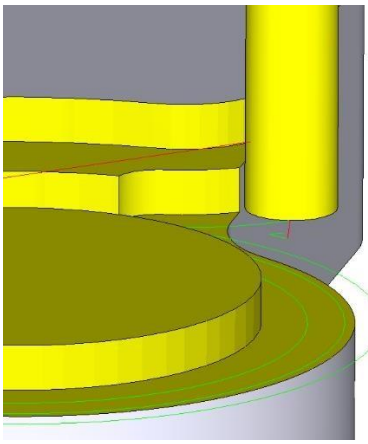


Figure 1. Tool positioning is optimized by the system to reduce cutting time and safely avoid stock at all incremental depths.

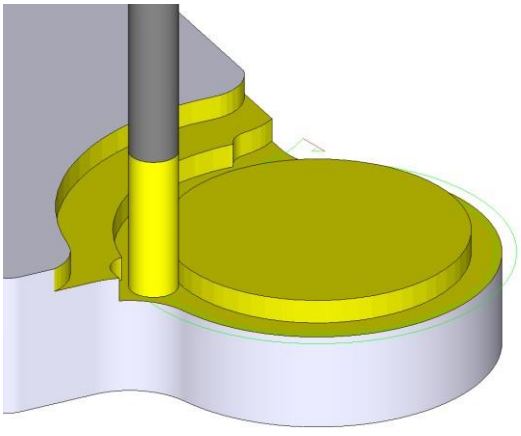


Figure 2. The bottom of the cutter is positioned on open edges to always produce a clean finish.

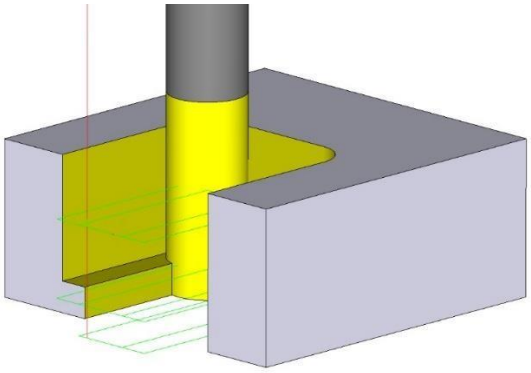


Figure 3. Contouring rough passes are calculated to rough all stock at all levels. There is no need to enter the number of rough passes.

Stock Automation

Choose whether to enable the automatic calculation of stock.

Type	Value	Description
	**Yes**	The current state of the stock model defines the boundary for cutting passes. The selected feature or solid face simply defines the position of the work plane. Cutting passes are created only for areas where stock remains. The system automatically sets certain parameters to optimize the toolpath to the shape of the stock model. Settings such as starting depth, retraction modes, retract plane, and lead-in/leadout distances are calculated automatically by the system and hidden from view on the technology page. ***Note:*** Stock Automation has no awareness of the target part. If the selected face includes islands or is bounded by a wall that must be avoided, make sure to select island features that define the areas to avoid.
	No	The selected feature is used as the stock boundary.

	Trim	The selected feature is used as the stock boundary. Toolpath is trimmed to where stock exists. *This is the same as Trim to Stock = Yes in previous version of ESPRIT TNG.*

ProfitMilling Upgrade in Closed Cavities

The technology behind ProfitMilling was revisited to improve performance and reliability in closed cavities. Key improvements are a larger initial spiral to open the pocket faster and tighter control of the initial channel to reduce machine decelerations.

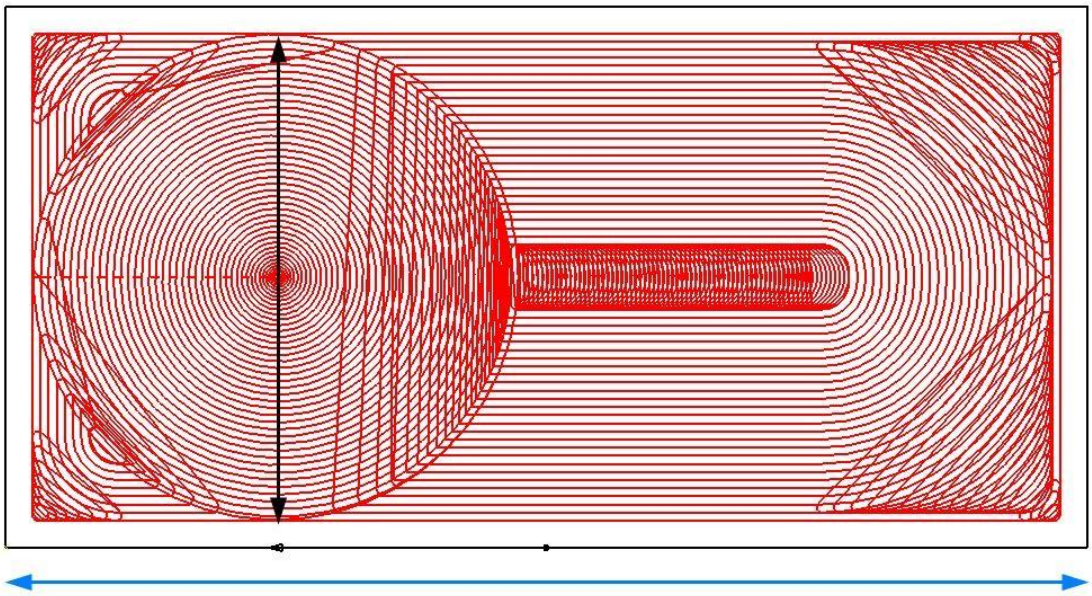


Figure 4. A larger initial spiral opens closed cavities faster for better chip evacuation and coolant access.

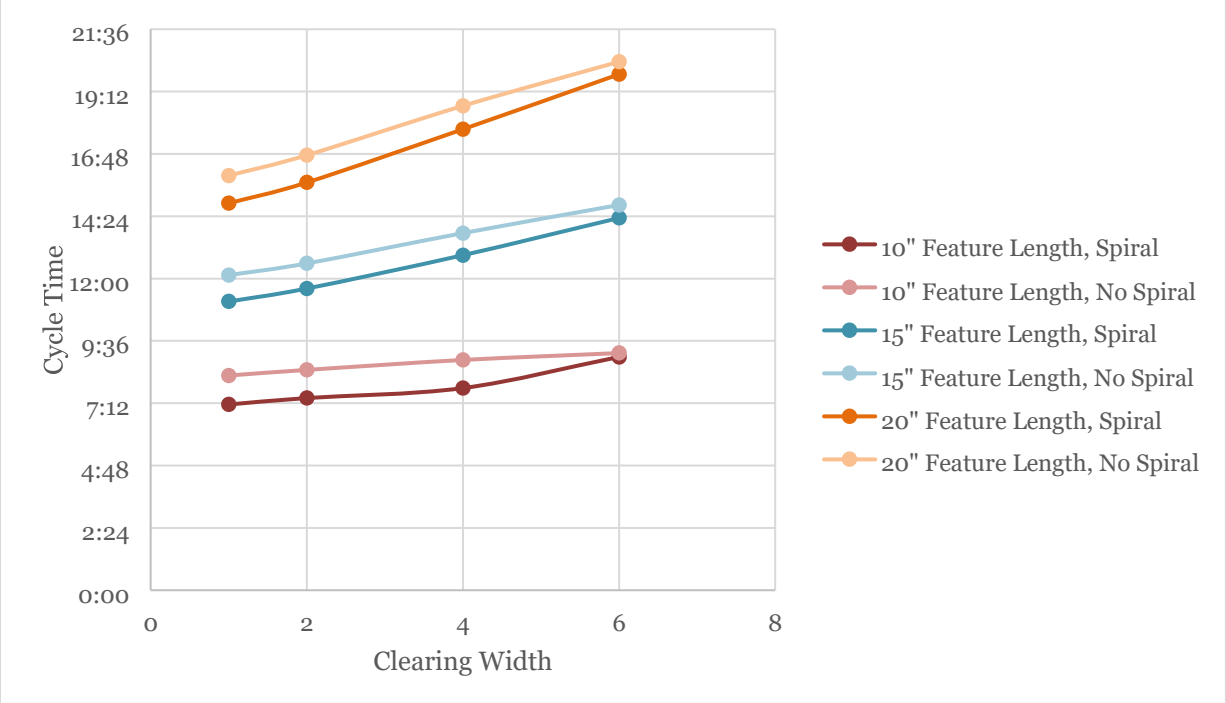


Figure 5. *An initial spiral out resulted in the fastest cycle times.*

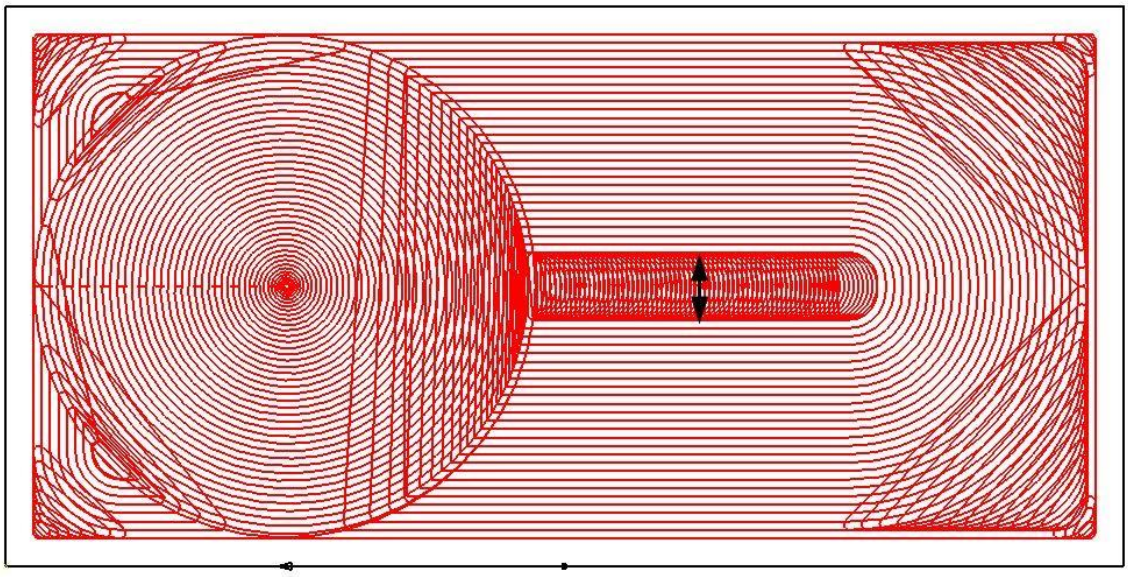


Figure 6. *The width of the initial channel is optimized for faster attainment of programmed feed rates and fewer machine decelerations.*

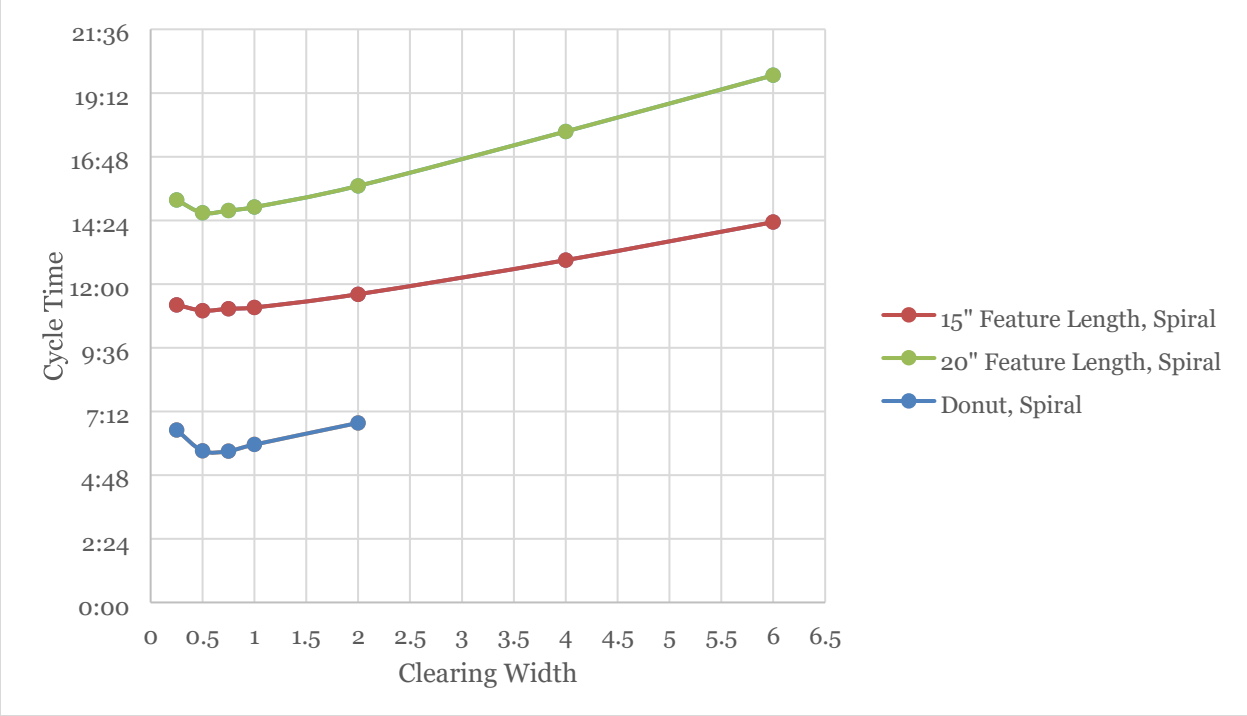


Figure 7. The fastest cycle time is achieved with a clearing width of 0.5 – 0.75 x Tool Radius. ESPRIT is 1.0.

Wire EDM – What's new?

Wire EDM Open Pocketing

ESPRIT has long supported open pocketing and morphed toolpath for milling operations. Now that technology is applied to wire EDM to free EDM programmers to cut more types of profiles more easily.

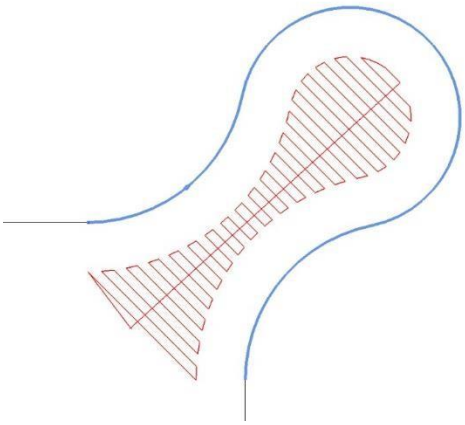


Figure 8. Efficiently burn small openings regardless of whether the profile has a closed or open shape.

Open pocketing makes it easier and faster to cut small openings along a contour. When the wire is too big to fit into a small opening, simply rough the contour first in G40 mode with look-ahead to leave the opening intact. Then follow up with a pocketing operation on the open profile to burn away only the small amount of material that remains.

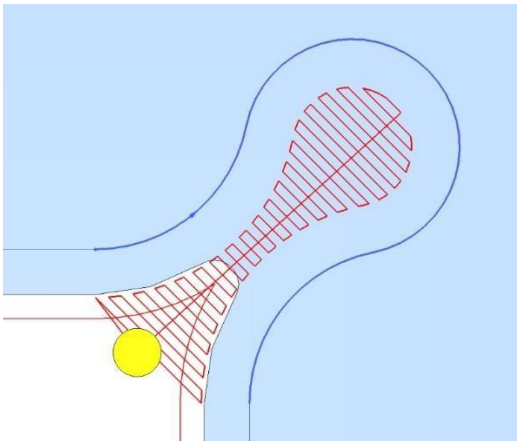


Figure 9. *Combine contouring and open pocketing to efficiently cut awkward profiles.*

No-Core Pocketing from Internal Cavities

Sometimes, a big hole or a groove is pre-machined inside a pocket cavity to remove as much material as possible and, at the same time, release the stress within the material. Other times, a model has a cast pocket cavity and the inside profile could be of any reasonable shape. Pocketing passes must recognize and adapt to the shape of an internal cavity to avoid cutting air.

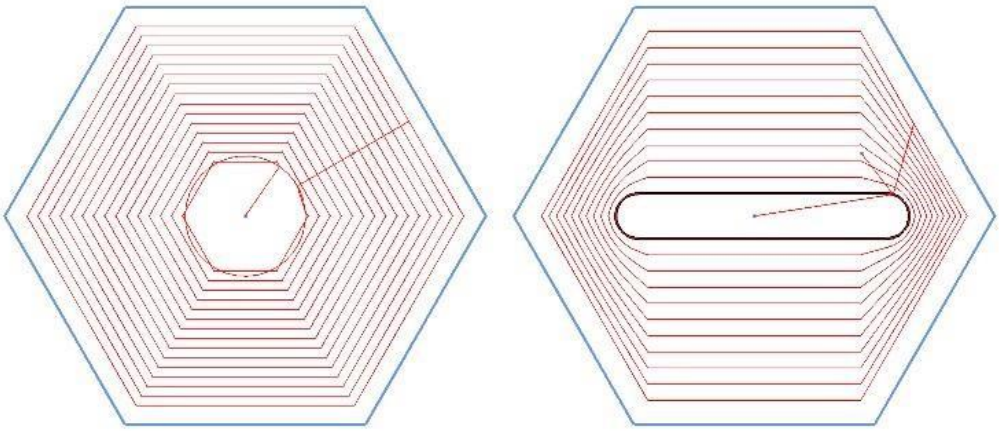
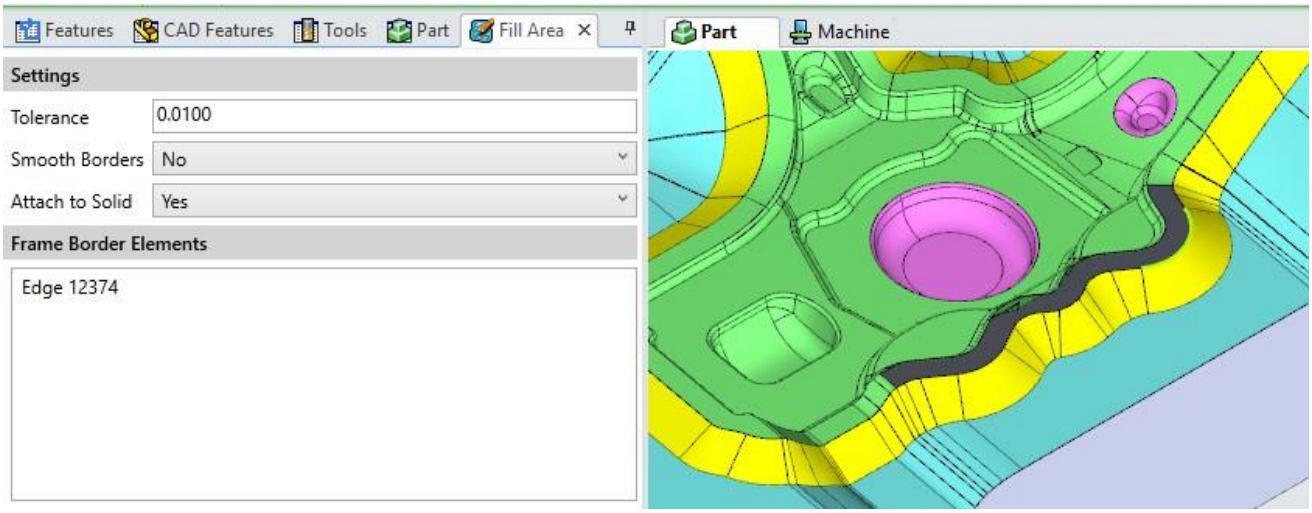


Figure 10. *No-core pocketing offers the option of a constant offset around a pre-drilled hole or a smooth transition between the shape of an internal cavity and an outer profile.*

Modeling – What's New

Fill Frame Enhancements

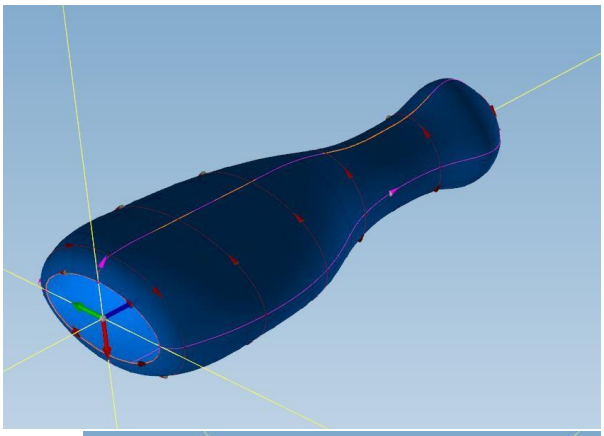
When using Fill Frame on a solid with Attach to Solid set to YES, it is not necessary to select all edges limiting the hole on the solid. Now, 1 edge of the border of the hole is enough; ESPRIT detects automatically the hole border for filling the gap.

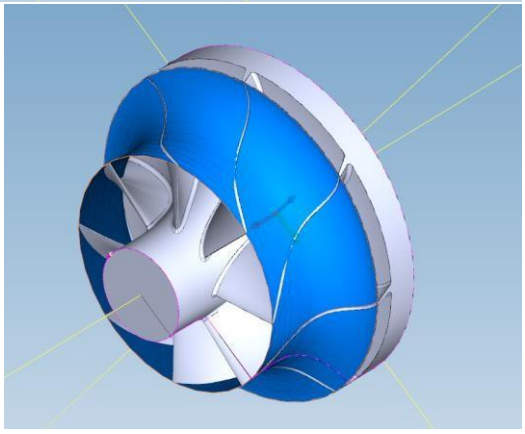


New Loft Command

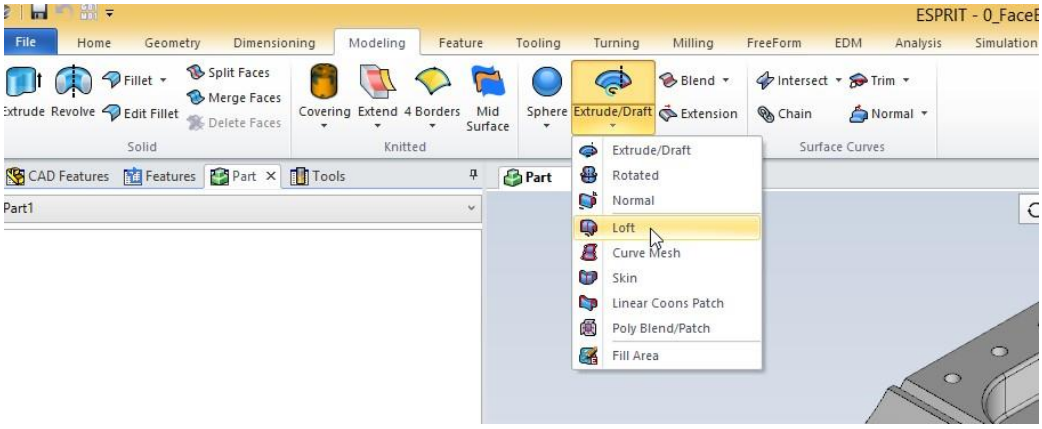
When programming a Part in ESPRIT, it sometimes occurs that the CAD solid model is incorrect or we need to create a surface to help the programming (for example: pilot a 5 axis toolpath or add a part or check element to a FreeForm Feature).

In those situations, the new Loft Surface command will help the user to create some surfaces or solids:





The Loft surface command is located in the Modeling tab, under the “Surfaces” group:



Parameters:

Tolerance

Defines the accuracy for surface creation.

Closed Results

Yes: Choose Yes if the result is closed (it has only 2 edges).
No: Choose No if the result is open (it has 4 edges).

Tangency Condition

Choose the Tangency condition for either the start of the surface, the end of the surface, or both.

Start Tangency Extension
Start Tangency extension is a factor of the tangent vector of the surface at the start.

End Tangency Extension
End Tangency extension is a factor of the tangent vector of the surface at the end.

Tangency Faces/Surfaces
Choose the faces or surfaces that are going to define the Start/End tangency. They can be selected in any order.

Add more Profile Elements

Yes: choose if you have more than one profile element.
No: choose if you only one profile element.

In case the profile that you want to loft is composed of more than one element, select the first one in "Profiles (first Element of Profiles), and select all the other ones in "additional Profile Elements". The order is not important.

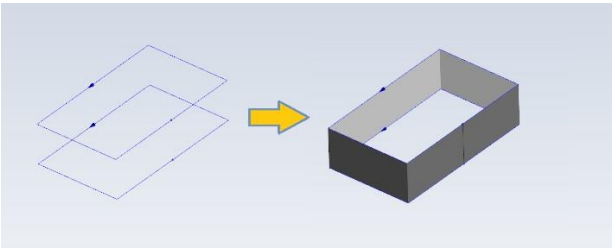
Create As

Solid
Choose only if profiles are closed. Result loft element will be a closed solid.

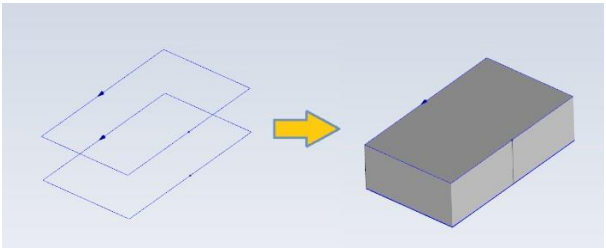
Surface
Resulting loft elements will be surfaces.

Sheet

Resulting loft elements will be sheet solids (2D solid).



create as sheet



create as solid

Topology Type

This setting is only available if the creation type is Solid or Sheet.

Minimal
Merge all resulting faces of the Solid/Sheet as much as possible

Along Profile Direction
Merge faces of the Solid/Sheet only in profile direction. Guide curves act as separation.

All Loops
Does not merge any faces together.

Additional Features

Use this option if your surfaces need either synchronization points or guides curves.

None

No Synchronization points or guide Curve.

Synchronization

When you choose this option, you can then select points or vertices to synchronize the surface. Select pairs of vertices (or points) whenever you need a synchronization line.

Guide Curves

When you choose this option, you can select or several guides curves (or features) to guide the surface.

Both

When choosing this, both the synchronization and the guide curves are available.

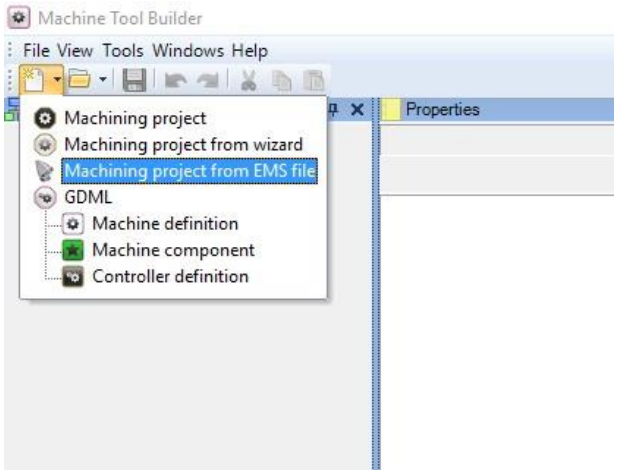
Machine Tool Builder – What's New

EMS Converter Lathe Support

Machine Tool Builder now supports conversion of a lathe setup from E2000-series.

Under File/New, select **Machining project from EMS file** to convert an existing EMS setup.

The conversion supports Multi-Spindle machines. However, it won't convert tailstock components.



In addition, there were several minor enhancements to Machine Tool Builder:

- Updated Lathe Custom Settings in Templates
- Empty Node Visibility
- Improved Error Trapping

12/9/2016

ESPRIT TNG Release 2.2

DP Technology Corp.

Introduction

Welcome to ESPRIT TNG Release 2.2.

This is the third Release of ESPRIT TNG with support for Turning 1 Channel 1 Spindle.

In this document, find a detailed description of the new features added this milestone.

Next Milestone is scheduled for February 17, 2016

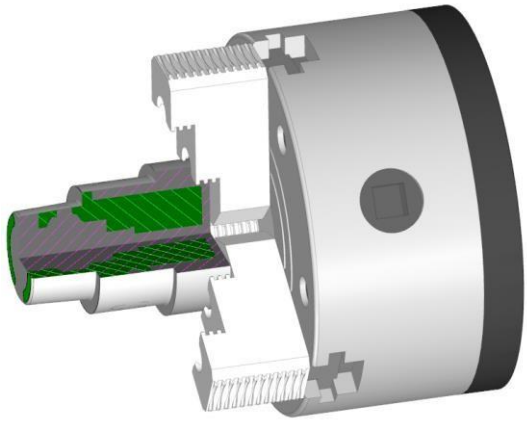
SolidTurn - What's new?

Clip Plane

When performing internal cutting, it can be difficult to inspect the motions in simulation.

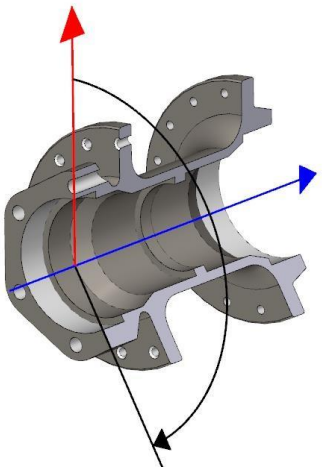
The New Clip Plane functionality give you a simple way to visualize a section of the workpiece in machine view for inspection.

Clip plane cuts the workpiece: Part and stock.

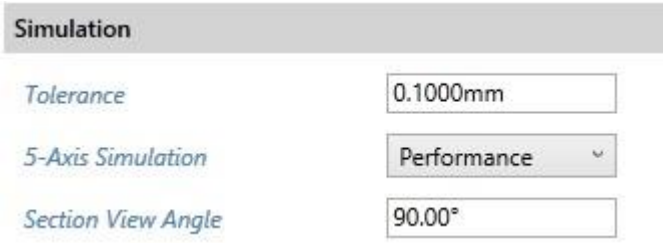


Clip plane feature is available only on turning machines (machine with a turning axis).

The section view starts from a plane defined by the machine turning axis (blue arrow) and the plane contains the X-axis of the machine coordinate system.



In options/Simulation, **Section View Angle** defines the angle span of the section.

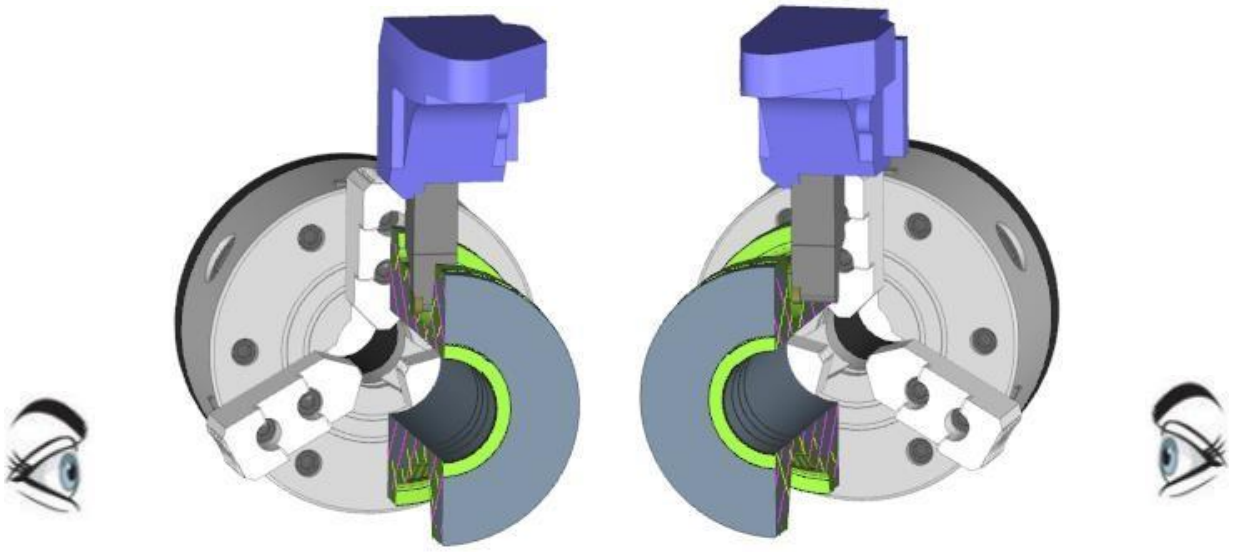


Note: If X-axis of the machine is parallel to the turning axis then second direction is set by the Z-axis of the machine coordinate system.

Section view angle and clip plane state are saved with the document.

Clipped section changes with the view.

The clipped section changes with the orientation of the view. It dynamically removes the front section.



SolidMill – What's new?

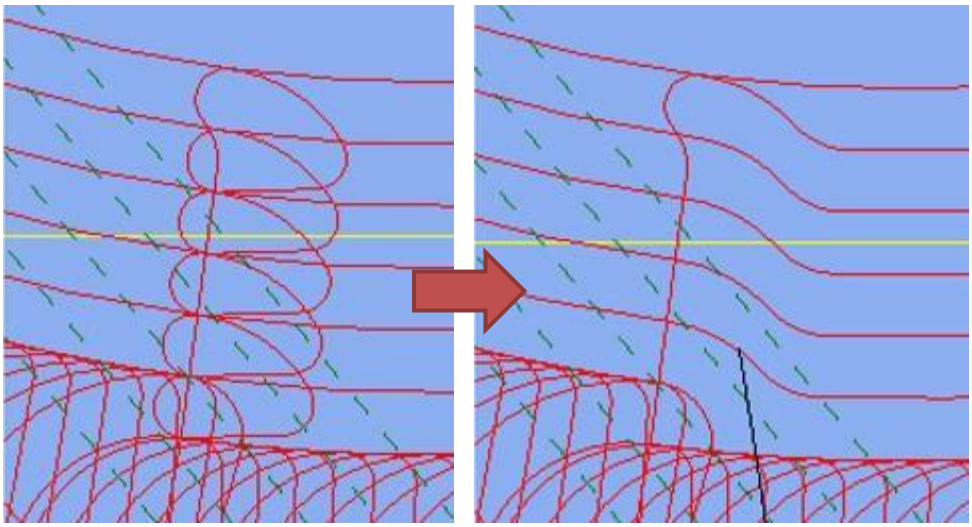
Stock Automation Performance

Calculation times for cycles using stock automation has been improved, especially when applying machining operations to complex stocks. Improvements of up to 25 times faster toolpath calculation can be achieved, compared to previous versions, and can be expected for:

- Facing (Stock Automation = Trim or Yes)
- Pocketing (Trim to Stock = Yes)
- Contouring (Trim to Stock = Yes)
- Z-Level Roughing

ProfitMilling

ProfitMilling toolpaths will now link moves between offset passes without using a "trochoidal" motion. The new motions maintain consistent cutter load and keep the tool engaged in the material, while reducing toolpath length and overall cycle time.



Facing Stock Automation

A new option has been added to the Facing cycle to recognize the in-process stock and create tool path optimized for that shape.

When using this parameter, the user interface has been simplified by determining Tool Overhang and Leadin/Lead-out automatically.

New for ESPRIT R2.2, Starting Depth and Clearances will also be automatically applied.

Stock Automation

Choose whether to enable the automatic calculation of stock.

Type	Value	Description
	**Yes**	The current operation calculates the remaining stock material based on previous machining operations. Cutting passes are created only for those areas, not the entire part. Starting depth, Incremental Depth Calculation and Retraction modes, and Retract Plane will be calculated automatically. Overhang and Overhang Direction will be applied automatically based on the Step Over. Lead-In and Lead-Out will be applied automatically. Lead-In ~ 50% of tool diameter; Lead-Out ~ 0% of tool diameter; Last Pass Lead-Out ~ 50%.
	No	The selected feature is used as the stock boundary.
	Trim	The selected feature is used as the stock boundary. Toolpath is trimmed to where stock exists. *This is the same as Trim to Stock = Yes in previous version of ESPRIT TNG.*

More Information on Custom Holes

ESPRIT TNG exposes more information on custom holes of a hole feature. This information may be used for automation with KnowledgeBase.

Custom Hole Definition

A Custom Hole can be defined by a succession of sections of different types: Straight (vertical face), Taper (chamfer), Blend (radius), Face (flat). A custom hole can be defined by as many sections as necessary. Those sections can be read and added through the API, and they all have several specific properties: A straight section is defined by a diameter and a depth, whereas a taper will also require the taper angle information.

List of the different section type and parameters

1. Straight (type 1)
    a. Height
    b. Diameter
2. Taper (type 2)
    a. Height
    b. Start Diameter
    c. Taper Angle
3. Blend (type 3)
    a. Height
    b. Start Diameter
    c. End Diameter
    d. Blend Radius
    e. Direction
4. Face (type 4)
    a. Start Diameter
    b. End Diameter

In TNG, we now give access to all the Custom Hole section properties through the API, so that the user can easily automate his programming through API or KBM. DP Technology Corp.

API Example in TNG:

```
Public Sub OutputCustomHoleInformation()

Dim hole() As ComFeatureHole

Dim hitems() As ComFeatureHoleItem

Dim holes As ComFeatureHoles

Dim htest As HolesFeature

Dim obj As GraphicObject

Dim i As Integer

Dim j As Integer

For i = 1 To Document.Group.Count

Set obj = Document.Group.Item(i)
```

```
If obj.GraphicObjectType = espHolesFeature Then

Set htest = obj

Set holes = htest.ComGraphicObject
hole = holes.GetHoles  hitems =
hole(0).HoleItems  With
Application.EventWindow

Call .Clear

For j = 0 To UBound(hitems)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "----- Section " & j & " -------")

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Type : " &
getsectiontype(hitems(j).Type))

Select Case hitems(j).Type

Case 0

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Type of face is unknown")  Case
1

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Height : " &
hitems(j).StraightHoleItemData.Height)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Radius : " &
hitems(j).StraightHoleItemData.Radius)  Case 2

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Height : " &
hitems(j).TaperHoleItemData.Height)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Starting Radius : " &
hitems(j).TaperHoleItemData.StartingRadius)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Taper angle : " &
hitems(j).TaperHoleItemData.TaperAngle)

Case 3

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Blend Radius : " &
hitems(j).BlendHoleItemData.BlendRadius)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Direction : " &
hitems(j).BlendHoleItemData.Direction)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Starting radius : " &
hitems(j).BlendHoleItemData.StartingRadius)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Ending Radius : " &
hitems(j).BlendHoleItemData.EndingRadius)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Height : " &
hitems(j).BlendHoleItemData.Height)
```

Case 4

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Starting Radius : " & hitems(j).FaceHoleItemData.StartingRadius)

Call .AddMessage(espMessageTypeInformation, "CH_Section " & j, "Ending Radius : " & hitems(j).FaceHoleItemData.EndingRadius)

End Select

Next j

End With

End If

Next i

End Sub

Public Function getsectiontype(i As Integer) As String

Select Case i Case 0

getsectiontype = "Unknown (0)" *Case 1*

getsectiontype = "Straight (1)" *Case 2*

getsectiontype = "Taper (2)" Case 3

getsectiontype = "Blend (3)" *Case 4*

getsectiontype = "Face (4)"

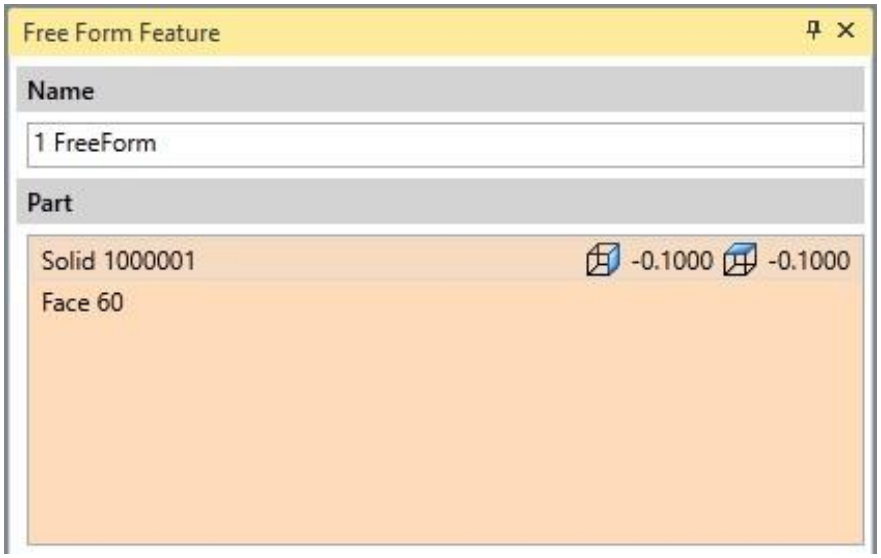
End Select

End Function

FreeForm – What's new?

FreeForm Feature Different Stock Allowances

Different Stock Allowance is a new feature available in the Free Form Feature editor, and consequently in some 3x Mold cycles, which allows to get specific stock allowances only in particular faces of the part.



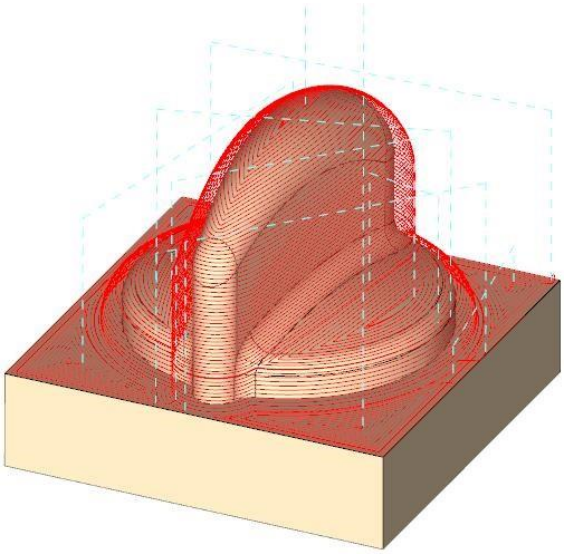
- FreeForm feature Stock Allowance and operation stock allowance are cumulative:
    - Use FreeForm Feature stock allowance for model modification. For instance, electrode needs to be smaller than the final part.
    - Use operation stock allowance for process related allowance. For instance, roughing process needs to leave 0.5mm of material for finishing pass.
- If you select a solid and apply an allowance on it, if then you select a Face of that same solid and apply another allowance value, then the allowance set to the Face is applied (while, for all the other faces, the value set on the solid will be applied).
- FreeForm feature stock allowance is not supported by all the FreeForm cycles. When a cycle that doesn't support DSA is launched on a FFF with DSA set on its elements, then an error is trigged.

Mold Cycles supporting DSA	Mold Cycles NOT supporting DSA

• Roughing • Z-Level Finishing • Parallel Plane Finishing • Radial Finishing • Spiral Finishing • Concentric Finishing • Floor Finishing • Global Finishing • 3D Contouring	• Corner Remachining • Pencil Tracing • Between Curves Finishing • Autotilt • All 5x cycles

FreeForm 3-Axis Global Finishing Cycle

Global Finishing cycle generates a toolpath in both the steep and shallow areas of the part.



The user sets a Slope Threshold Angle: part areas having a slope above such set value are considered steep; the other areas, having a slope below such set value, are considered shallow.

On the steep areas, the system applies a Z-Level type toolpath; on shallow areas, it applies Concentric type toolpath. Both types of toolpaths can be covered in spiral way.

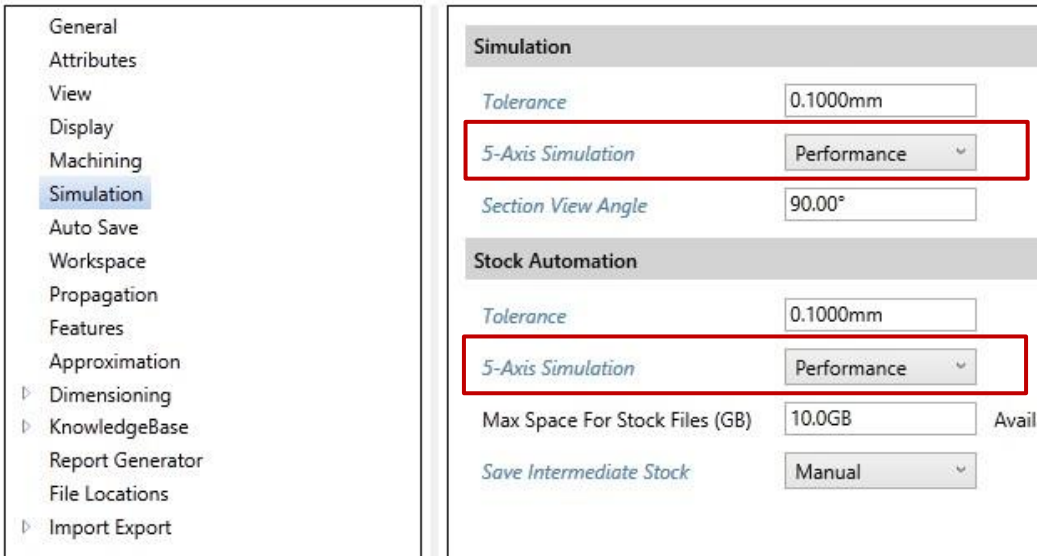
The system tries, as much as it cans, to minimize the number of detaches and re-approaches, linking together when possible the Z-Level toolpath areas with the Concentric one.

New 5-Axis Simulation options: Performance vs. Quality

Simulation of 5-axis motions can be optimized for either performance or quality.

The optimization can be set independently for stock automation and for full machine simulation.

When optimizing stock automation for performance, stock automation can be up to 3 times faster than in previous Milestone.



5-Axis Swarf Cycle Improvements

The 5-axis Swarf cycle has been improved to generate a smoother toolpath.

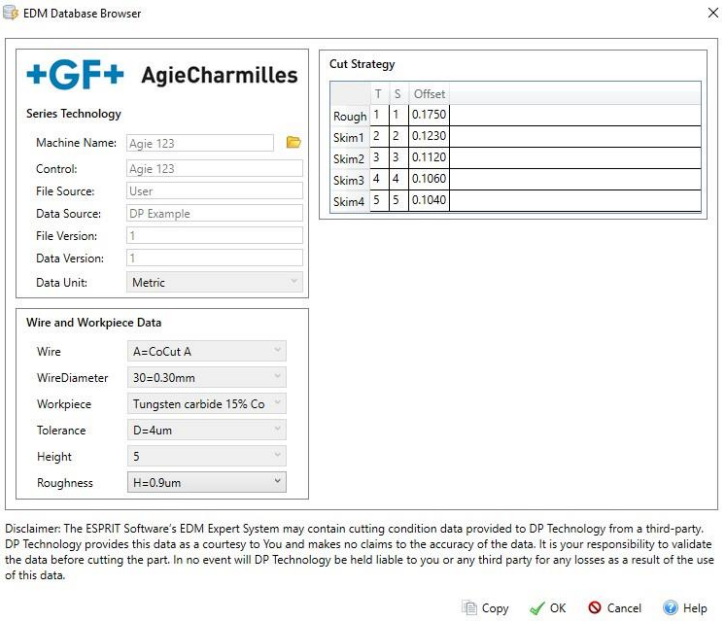
The new toolpath avoids discontinuities in accelerations/decelerations in the axial motion.

This new functionality does not imply any change in the User Interface nor in the cycle programming. The effect will be visible as a general improvement of the resulting part surface quality.

Wire EDM – What's new?

Expert System for Agie 123

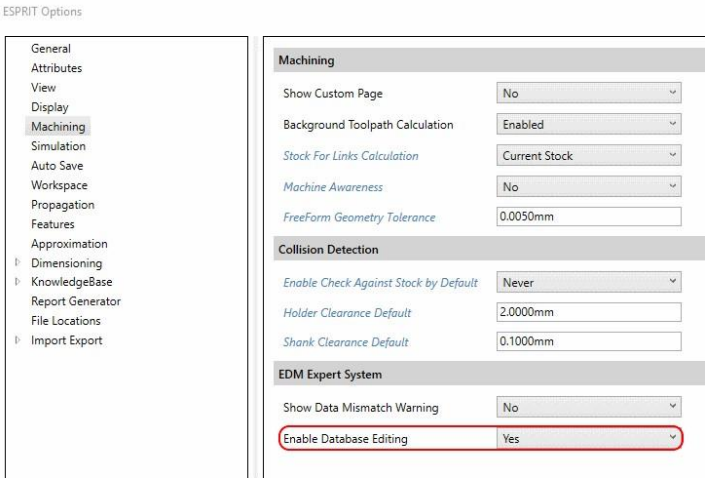
Wire EDM Expert System is now available for old Agie (123). Although these machines were last manufactured in the mid-1990's, they are extremely durable and the field requested that we continue to provide support for them. T (Power register), S (Feedrate register), and offset are typically included in the database.



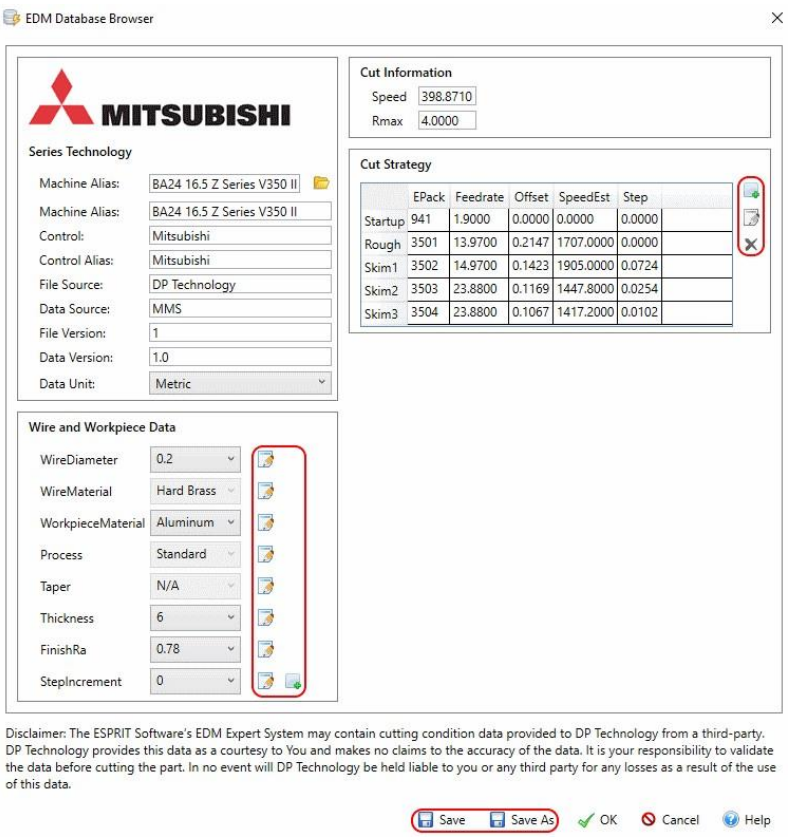
Edition of Wire EDM Database from Expert System

It is now possible to edit wire EDM .xml databases directly from the Expert System. This development is a result of a request from the field. It was possible with the legacy databases to edit them directly from the Expert System.

User must enable the edition in the options.



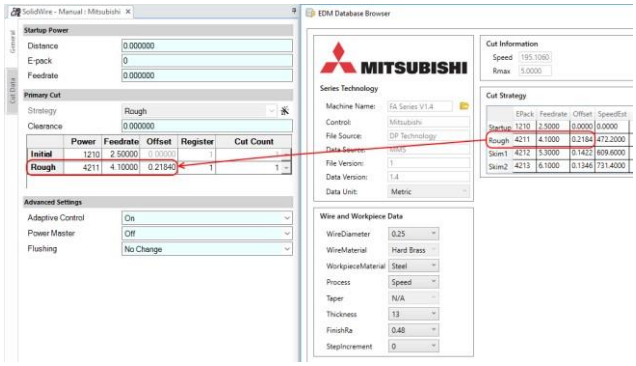
Once enabled, the user will see additional options in the interface.



Wire EDM Expert System for Manual Cycle

Wire EDM Expert System is now available for Manual EDMing. The workflow is similar to using the WEDM Expert System for other operation types (2x contouring, 4x pocketing, etc.), simply –

- Enter the wire and workpiece data
- Cut strategy is selected by the Expert System
- Startup (Initial) and Rough data are used by the Manual Cycle



What else is new?

New option to show CL Code

In ESPRIT 20xx, you can run ESPRITSettings.exe to activate display of CL code value in technology pages.

This option has been integrated as a standard option in ESPRIT TNG.