CAD/CAM SOFTWARE

What's New in ESPRIT 2017

All brand or product names or proprietary file types mentioned in this document are trademarks or registered trademarks of their respective holders. Contact the appropriate companies for more information regarding trademarks and registration.

DP Technology Corp.

1150 Avenida Acaso
Camarillo, California 93012
USA
Tel: +1 805 388 6000
Fax: +1 805 388 3085
www.espritcam.com

Printed in the United States of America

Contents

Faster editing of features

The way features are edited in ESPRIT has gotten more straightforward to help you remove geometry you do not want to machine. Features are now edited quickly and easily by simply deleting elements anywhere along the path: start, end, or middle. Feature elements that surround the newly made gap are automatically extended and connected while retaining their original attributes.

- Quickly edit chains, pocket features, and turning profiles by simply deleting sub-elements
- Remove multiple sub-elements at once
- Delete a starting element without reversing the feature

Until now, the process of editing a feature required multiple steps. The feature first needed to be opened for editing using the Manual Chain command and then edited with the Move Back command to remove elements from the end of the chain. Removing a starting element required a reversal of the feature direction before edits could be performed.

To quickly edit a pocket feature, chain feature, or turning profile:

1. Make sure Sub-element selection is enabled.
2. Use Highlight mode to select the sub-element you want to delete.
3. If needed, hold down the Ctrl key to add other sub-elements to the group.
4. Press the Delete key.

Figure 1. Pocket Feature Recognition detected all the walls of this pocket, including the holes along the walls.

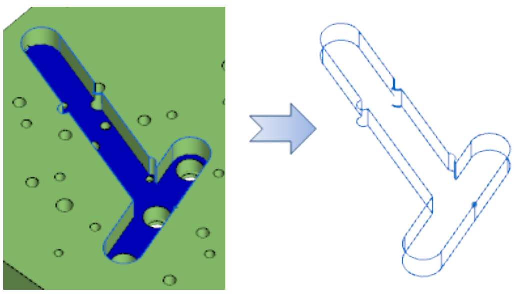


Figure 2. To create smooth walls for a pocketing operation, sub-elements are grouped and deleted to modify the shape of the pocket.

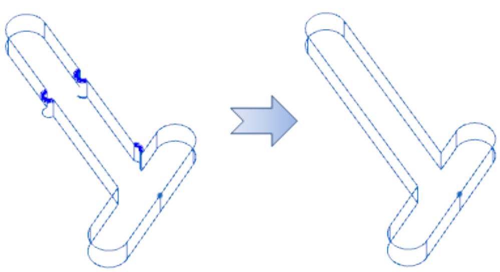


Figure 3. When a sub-element is deleted, previous and next sub-elements are extended and connected to close the gap.

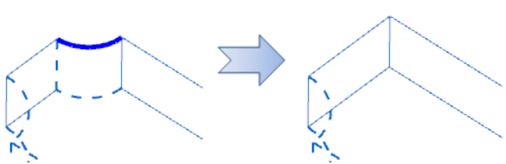


Figure 4. Extended sub-elements retain their original attributes, such as open edges.

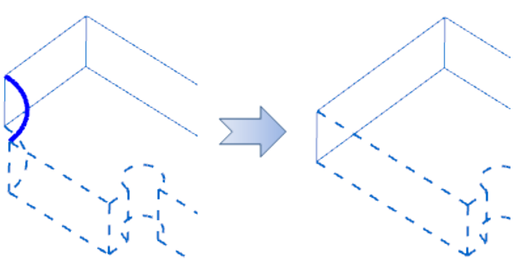


Figure 5. When previous or next elements do not intersect, a new sub-element is created in between. The new sub-element copies the properties of the next element in the feature.

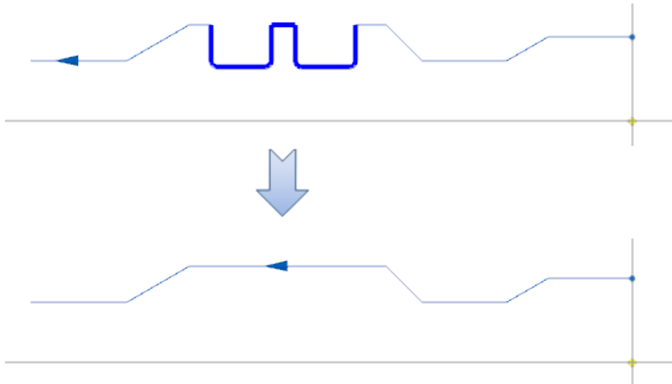


Figure 6. The Move Back command is no longer needed to remove an ending element.

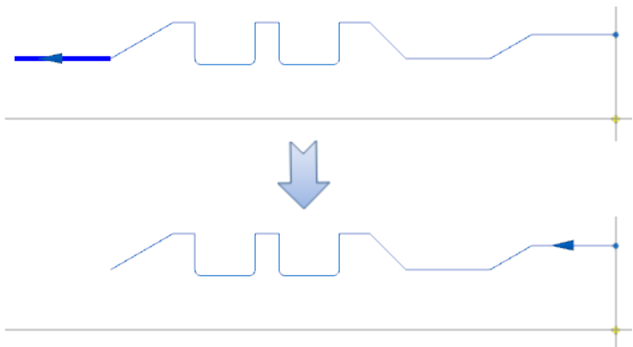


Figure 7. When the starting element is deleted, the feature start point is automatically moved forward.

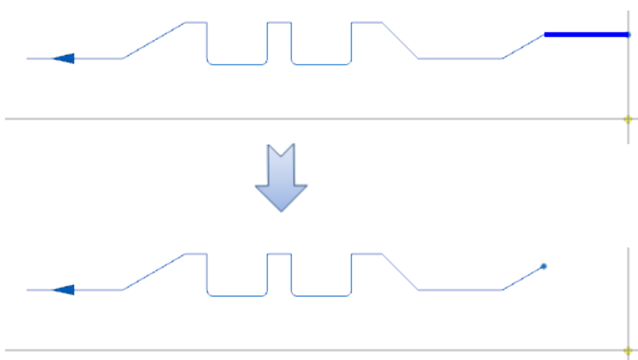
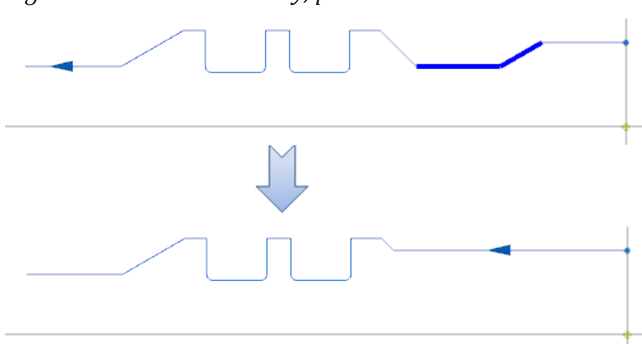


Figure 8. When necessary, previous or next sub-elements are trimmed to the intersection point.

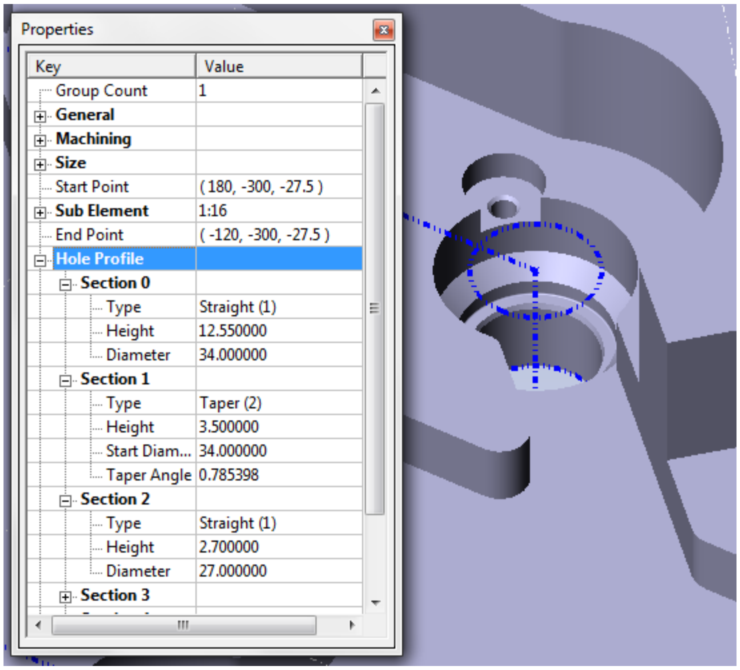


Better recognition of custom holes and their properties

Complex holes are recognized as Custom Holes and are now defined by a succession of sections that are identified as either cylindrical, chamfered, rounded, or flat. Each section and its properties are exposed in the Property Manager and the ESPRIT API to help users automate programming through the API or the ESPRIT KnowledgeBase.

- Easy access to custom hole properties in the Property Manager
- Identification of sections as standard hole types: Straight (cylindrical), Taper (chamfer), Blend (radius), Face (flat)
- Exposure of custom hole properties in ESPRIT API

Figure 1. The properties of custom holes are displayed in the Property Manager.



When a hole is too complex to be recognized as a standard hole type (Simple, Countersink, Counterbore), it is categorized as Custom. For example, when a hole has a counterbore as well as a countersink it is recognized as a Custom hole.

A Custom hole is defined by a succession of sections of standard hole types: Straight (cylindrical), Taper (chamfer), Blend (radius), Face (flat). A Custom hole can be defined by as many sections as necessary. The properties of those sections are displayed in the Property Manager.

Custom hole section types and properties include:

1. Straight (type 1)
    - Height
    - Diameter
2. Taper (type 2)
    - Height
    - Start Diameter
    - Taper Angle
3. Blend (type 3)
    - Height
    - Start Diameter
    - End Diameter
    - Blend Radius
    - Direction

4. Face (type 4)
   - Start Diameter
   - End Diameter

Access to all custom hole properties is provided through the ESPRIT API to help users automate programming through the API or the KnowledgeBase.

API Example

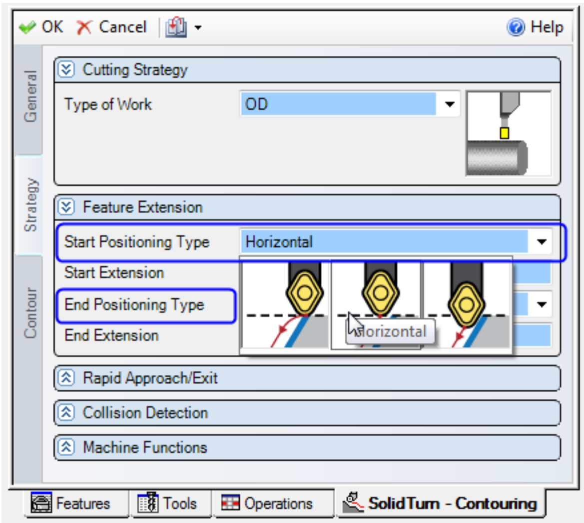
```
Public Sub OutputCustomHoleInformation()

Dim ptop As FeaturePtop
Set ptop = Document.FeaturePtops.Item(1)
Dim cp As EspritProperties.CustomProperties
Dim cp2 As CustomProperties
Dim cpy As CustomProperty
Dim i As Integer
Application.OutputWindow.Visible = True
Application.OutputWindow.Clear
Set cp = ptop.CustomProperties.Item(1)
Application.OutputWindow.Text ("The number of sections is : " & cp.Count)

For i = 1 To cp.Count
  Set cp2 = cp.Item(i)
  Set cpy = cp.Item(i)
  Application.OutputWindow.Text (vbCrLf & "Section " & i & " : is a type " & cp2.Item(1).Value)
Next i

End Sub
```

Enhanced tool positioning for lathe contouring

Several enhancements to SolidTurn Contouring offer better positioning of the tool edge along walls, at the start or end of cutting passes, and inside grooves. These new enhancements are particularly advantageous when contouring with round or grooving inserts.

- Control the touch point of the tool at the start or end of a feature to better accommodate different insert shapes
- Prevent the tool from touching a vertical wall at the end of a horizontal pass with a user-defined safety clearance
- Shift the tool edge along a horizontal cut to ensure precise length compensation output (similar to Grooving)

Figure 1. New Positioning options at feature start and end points



Gain custom control over positioning of the tool at feature start and end points.

Previously, the system would determine the contact point of the tool edge at the start and end of the feature. The default positioning worked well with standard turning inserts, but could cause problems with round inserts.

Figure 2. The default position of Horizontal places the edge of the tool horizontal to the start or end of the feature.

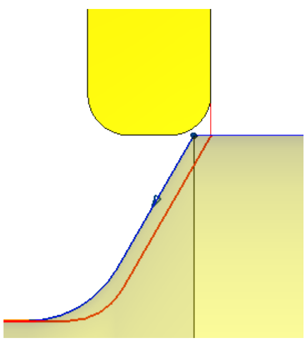


Figure 3. The Normal option places the touch point of the tool at the feature start or end point.

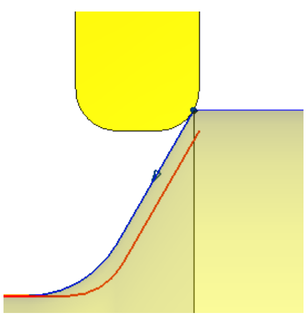


Figure 4. The Blend option adds a blend as the tool enters or exits the feature.

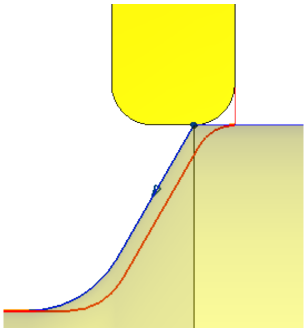
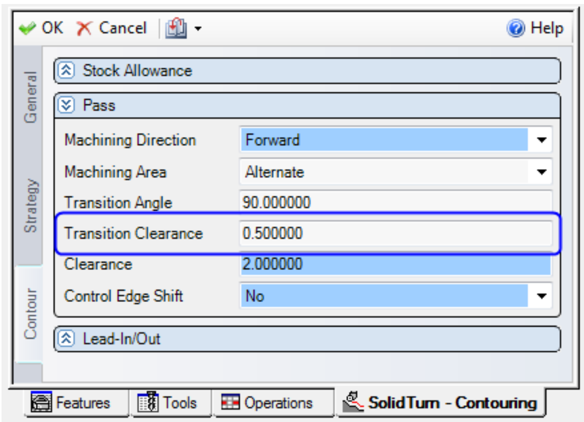


Figure 5. New safety distance for alternating cutting passes.



When Machining Area is set to Alternate, contouring passes alternate between vertical and horizontal passes. A transition angle of 90 degrees is used to cut vertical walls. As the tool approaches a vertical wall at the end of a horizontal pass, it can become enveloped in material and risk damage to the tool.

Figure 6. Without a safety distance, the tool edge comes into direct contact with a vertical wall at the end of a horizontal pass.

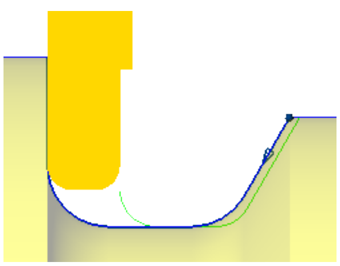


Figure 7. Transition Clearance adds a safety distance to offset the transition point before the tool touches a wall.

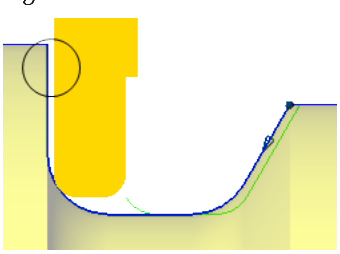
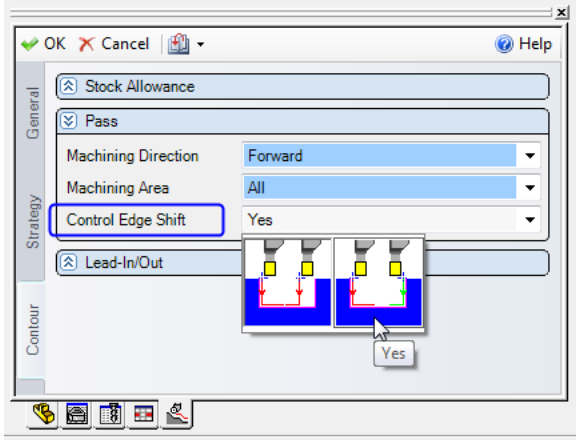


Figure 8. Dynamic change of offset for grooving tools.



Similar to the Grooving cycle, the Control Edge Shift option has been added to the Contouring cycle. Control Edge Shift can be used to change the position of the tool edge on the tool path from one side to the other in the middle of the pass.

Figure 9. Without edge shift, the position on the tool that marks the toolpath (the control edge) remains the same for the entire pass.

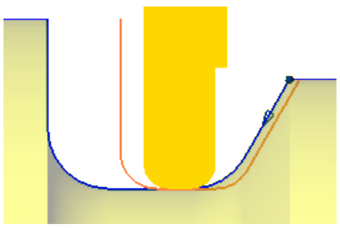
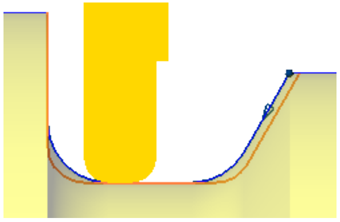


Figure 10. When Control Edge Shift is set to Yes, the control edge is allowed to shift to produce NC code that matches the part drawing. In this illustration, the control edge shifts from right to left.



Upgraded performance in ProfitMilling

The technology behind ProfitMilling has been revisited in ESPRIT 2017 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.

- Larger initial spiral opens closed cavities faster for better chip evacuation and coolant access
- Optimized initial channel width for faster attainment of programmed feed rates and fewer machine decelerations
- Fewer interruptions on initial opening passes

Figure 1. The initial spiral pass is larger in ESPRIT 2017 to open pockets faster.

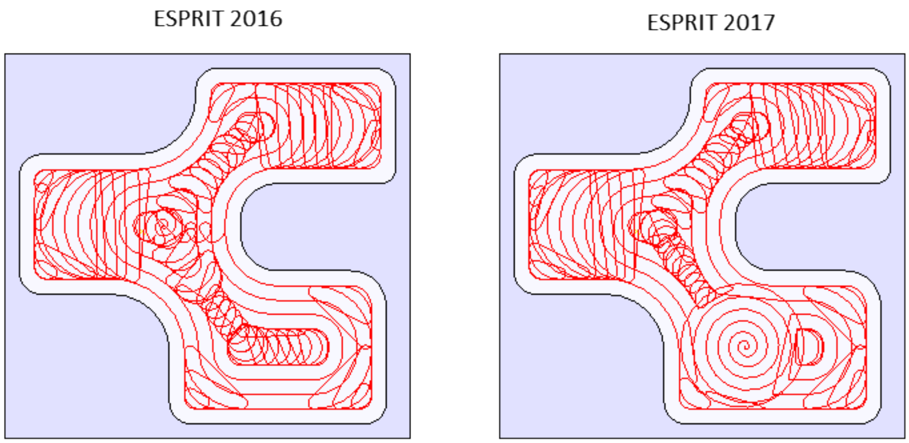
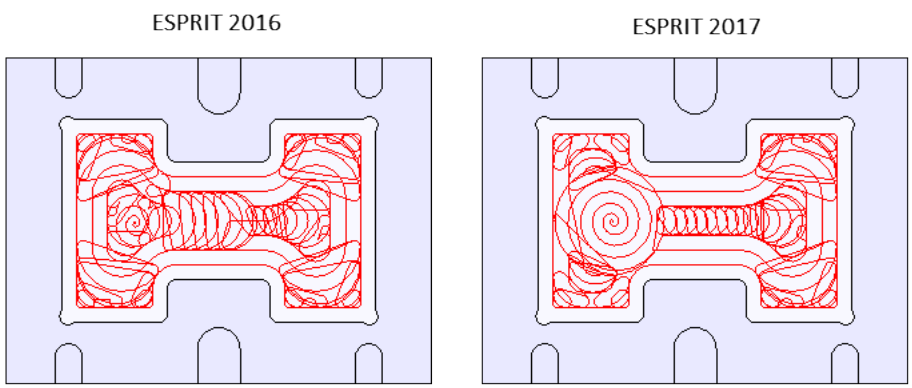


Figure 2. The width of the initial channel is now optimized for the size of the tool for faster cycle times and fewer decelerations.



Faster toolpath calculation times

The code behind the Stock Automation engine in milling operations has been refactored to drastically reduce calculation times. Now, when stock automation is enabled in Facing, Pocketing or Contouring, operations are calculated in seconds rather than minutes. The same refactoring is applied to 3-Axis Mold Roughing to calculate in-process stock models in dramatically less time, even on complex parts.

- Pocketing with stock automation: up to 25 times faster
- Contouring with stock automation: up to 3 times faster
- Mold Roughing: up to 12 times faster

In some circumstances, users had experienced long calculation times for traditional milling cycles like Contouring, Facing and Pocketing when the Stock Automation option was applied. This behavior happened particularly when such cycles were applied after many other cycles, creating a complex reference stock model. This problem was also noted on some Mold Roughing operations, even when applied on simple shapes. Stock management is now more efficient in ESPRIT to calculate in-process stock models in dramatically less time.

Smarter Facing with stock automation

As the first step in most milling processes, facing should be fast and easy. With that in mind, the Facing cycle in ESPRIT has been redesigned to dramatically reduce the number of steps needed to face a part. Enhancements have been programmed directly into the system to now automatically recognize the true shape of in-process stock, set parameters for optimal machining, and create toolpath optimized for that shape. Simply select a face, choose a tool, and let the system do the rest.

- Recognition of the true shape of the stock, even on odd-shaped faces
- Optimized toolpath for a clean finish across the entire face in minimal cutting time
- Selection of a face sets the work plane of the operation, even on tilted faces
- Streamlined interface presents only key parameters for faster programming
- Calculation of optimal cutting angle as the shape of the stock changes

Figure 1. Previously, facing passes relied on the shape of a feature, not the true shape of the stock model.

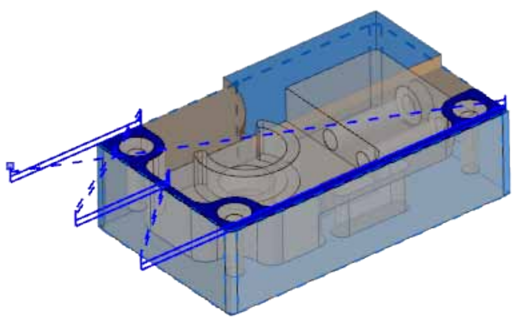
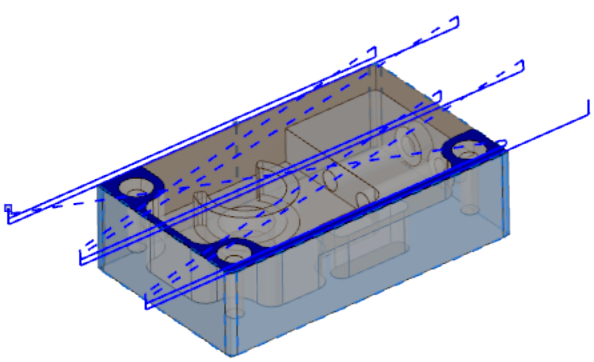


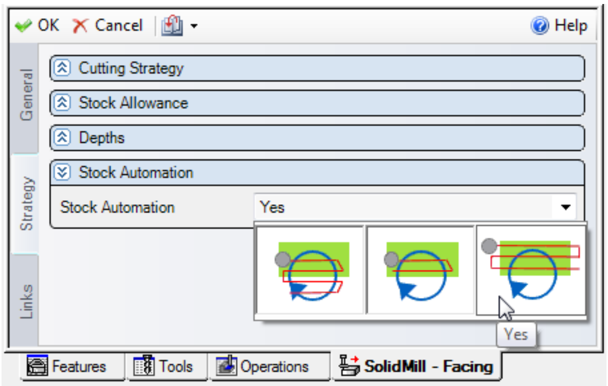
Figure 2. Now, when a face is selected, the system will calculate the ideal toolpath to efficiently remove stock.



In previous versions, if the stock was a standard block and the shape of the top face was smaller or had a different shape, the user had to take extra steps to create a feature that more closely matched the shape of the stock. The user also had to determine the best distance for the tool to sweep past the edges to ensure a smooth surface all around.

In ESPRIT 2017, cutting passes are calculated from the shape of the stock. The selected feature (or face) is used instead to determine the work plane for the operation. The user can simply select the face to be machined and ESPRIT will calculate the boundary of the stock around and above that face.

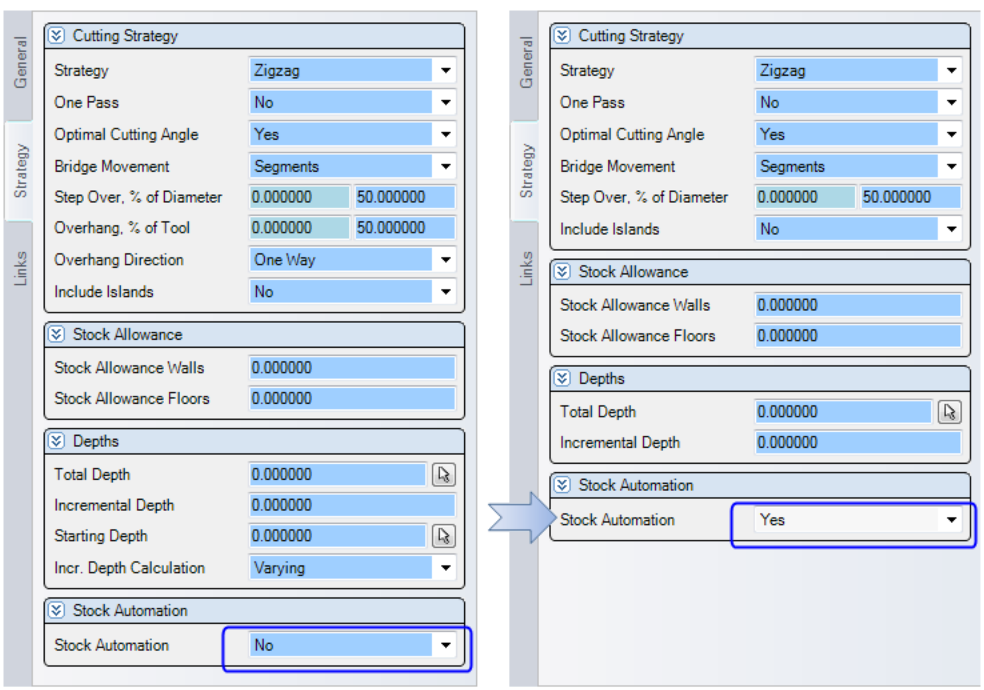
Figure 3. Use new Stock Automation to shorten programming time while optimizing toolpath.



The key to the new optimized facing technology is the completely new Stock Automation engine for milling. Stock Automation now gives the user control over how much automation is applied within the Facing cycle, as well as Pocketing and Contouring:

- **No**: As with previous versions of ESPRIT, a feature defines the boundary for cutting passes. The user must manually set machining parameters.
- **Trim**: A feature defines the boundary for cutting passes and the user sets parameters manually. However, toolpath is trimmed to where stock exists.
- **Yes**: The current state of the stock model defines the boundary for cutting passes. The user simply selects a feature or solid face to define 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.

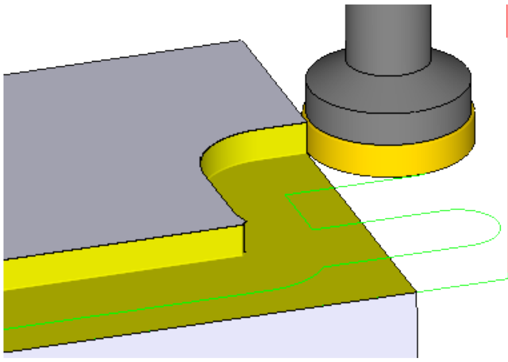
Figure 4. Stock Automation simplifies the interface to show only key parameters.



The following parameters will be calculated automatically by the system and hidden from the user.

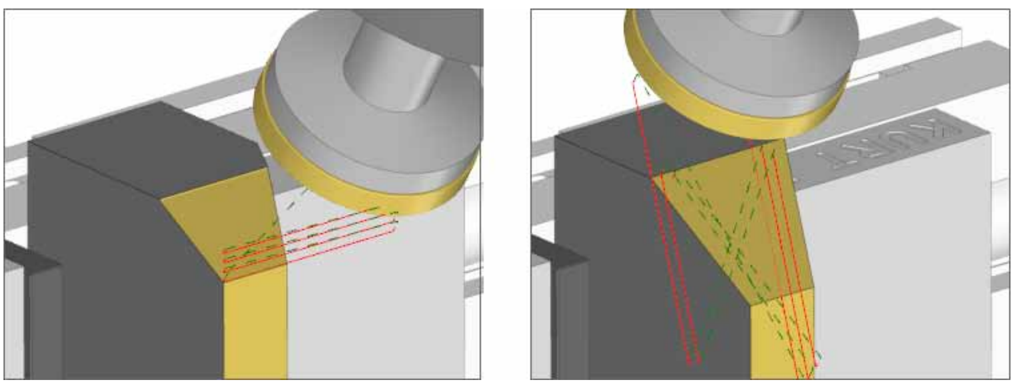
- **Overhang, % of Tool**: This distance will be applied automatically based on the Step Over to ensure a smooth surface all around the edges of the stock.
- **Overhang Direction**: The direction will be set automatically to Two Ways.
- **Starting Depth**: This depth is calculated internally by considering the stock.
- **Lead-In Distance/Lead-Out Distance**: Lead-In will be calculated internally to enter the stock boundary from 0.1 inch or 3 mm; Lead-Out will be ~ 0% of tool diameter. However, the lead-out for the last pass will be ~ 50% of the tool diameter.

Figure 5. Lead-In, Lead-Out, Overhang, and Starting Depth are optimized for cutting time and surface quality.



Stock Automation also adjusts the Optimal Cutting Angle if the shape of the stock changes at incremental cutting depths.

Figure 6. The system adjusts the Optimal Cutting Angle if the shape of stock changes at incremental depths

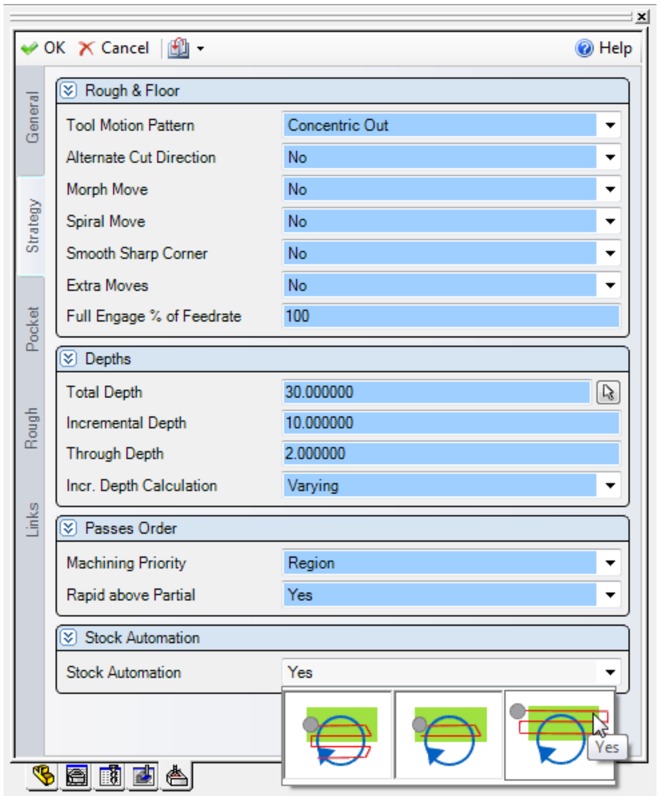


Enhanced stock automation for Pocketing and Contouring

The same enhancements to stock automation in the Facing cycle are available for Pocketing and Contouring. The shape of in-process stock is recognized and toolpath is automatically extended or trimmed based on the current workpiece stock.

- Optimized toolpath for the shape of in-process stock
- Reduction in machining parameters for faster programming

Figure 1. New Stock Automation is available in Pocketing and Contouring.



When Stock Automation is set to Yes in the Pocketing cycle, the following parameters will be set automatically by the system and hidden from the user:

- **Starting Depth**: This depth is automatically applied so that the first cut depth is the same as the specified incremental depth.
- **Retract for IDepth** and **Retract Plane**: Retracts are calculated to avoid stock by the specified Clearance value.
- **Open Pocketing**: Open Edge Offset will be applied automatically to encompass the entire stock at each incremental level.

For Contouring operations, the following parameters will be set automatically by the system and hidden from the user:

- **# of Rough Passes**: A value is automatically applied to encompass the entire stock at all levels.
- **Finish Pass**: Finish passes are disabled.
- **Starting Depth**: This depth is automatically applied so that the first cut depth is the same as the specified incremental depth.
- **Retract for IDepth** and **Retract Plane**: Retracts are calculated to avoid stock by the specified Clearance value.
- **Start/End Overcut**: Automatically applied such that the tool enters from 0.1 x Tool Diameter from the stock boundary (plus any lead in/out applied).
- **Open Contour Trimming**: Automatically set to Collapse.

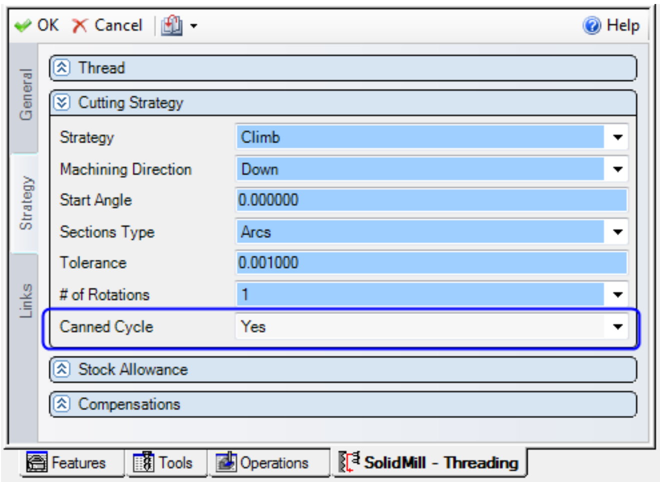
▶ The entire Advanced page is hidden. However, since the setting for Alternate Cut Direction is still needed, the setting has been moved to the Strategy page in ESPRIT 2017.

Canned cycle thread milling

Thread milling now supports the output of canned cycles. Canned cycles allow easy editing of the operation at the machine and shorter NC code. The ESPRIT technology page has been updated to allow users to enable canned cycle output and new keywords have been added to the ESPRIT Post Processor to support the output.

- Enable or disable canned cycle output in SolidMill Threading
- Output NC code with new Ex_ThreadMilling keywords in the post processor
- Behavior is consistent with all other machining cycles with canned cycle support

Figure 1. SolidMill Threading now supports canned cycles.



ESPRIT Post Processor has been updated with new examples for support of canned cycle thread milling:

Ex_ThreadMilling_Start: Example called for first threaded hole of the cycle

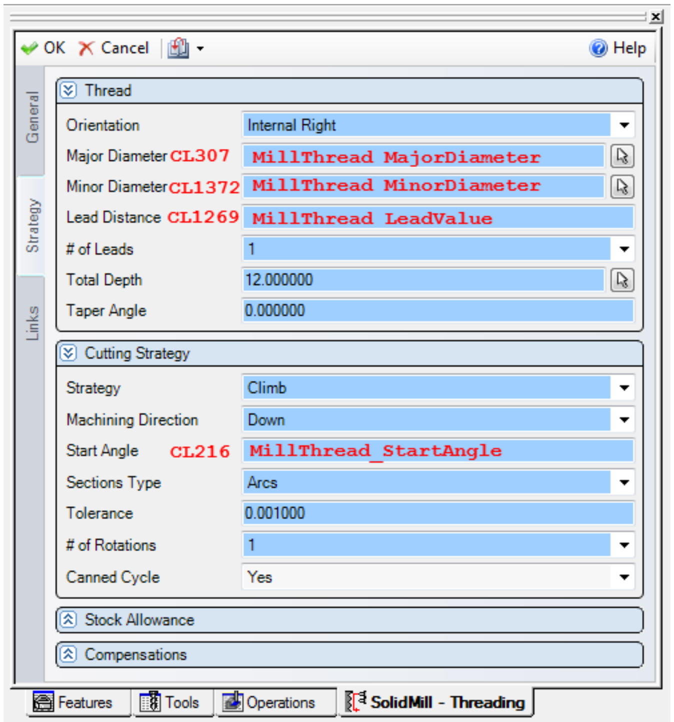
Ex_ThreadMilling_Body: Example called for next threaded holes of the cycle

Ex_ThreadMilling_End: Example called after the last threaded hole, this is the end of the cycle

New system variable keywords have also been added:

- **MillThread_MajorDiameter**: Takes the value of CL307 (Major Diameter)
- **MillThread_MinorDiameter**: Takes the value of CL1372 (Minor Diameter)
- **MillThread_Depth**: Calculated value (Major - Minor)/2
- **MillThread_LeadValue**: If Tool CL2530=0 (Thread Type = Thread Mill) then Lead Value = Tool CL1856. Otherwise, Lead Value = Operation CL1269 (Lead Distance)
- **MillThread_StartAngle**: Takes the value of CL216 (Start Angle)
- **MillThread_ThreadsPerStep**: Distance along tool axes between passes divided by the Lead Value
  - If 1 continuous helical pass then ThreadsPerStep=0
  - If 1 pass (360deg helix) at the bottom, then ThreadsPerStep=1

Figure 2. New system variables are available for threading milling.



Other keyword changes include:

- Symbolic Codes: New ThreadMilling keyword
- Formatable Codes: XAbsolute, YAbsolute, ZAbsolute output bottom end of thread. RPlane outputs altitude of the beginning of the feed motion (top of the hole+ clearance)

Example keyword flow:

```
Ex _ CycleStart

Ex _ StartPoint
Ex _ ThreadMilling _ Start
Ex _ ThreadMilling _ Body
...
Ex _ ThreadMilling _ End

Ex _ CycleEnd
```

For each hole, position(XAbsolute, YAbsolute, ZAbsolute) is updated to the top center of the hole.

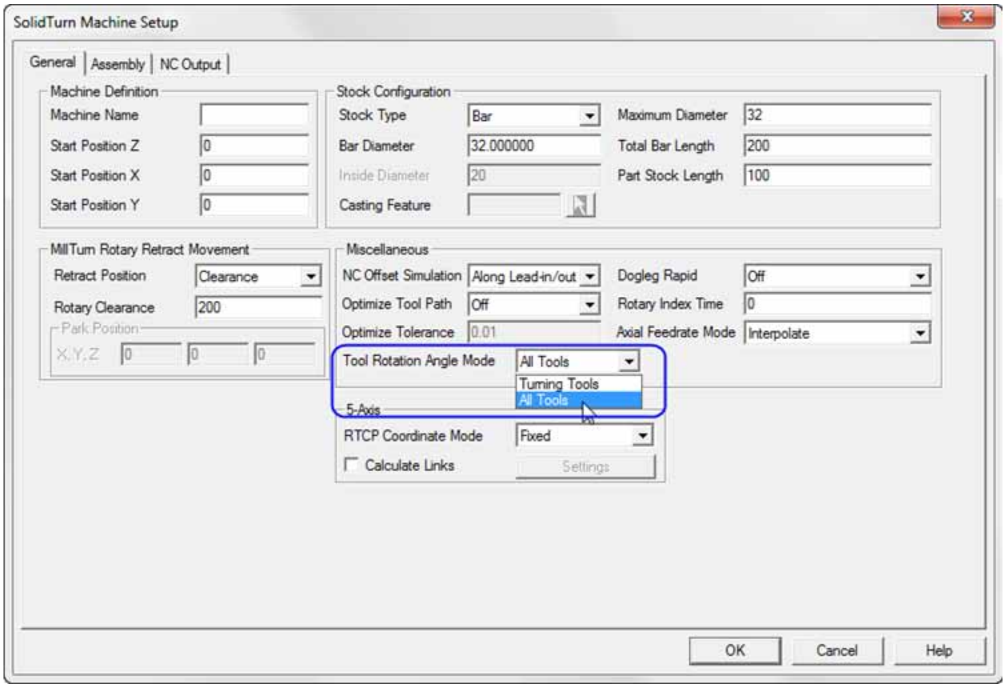
Better support for machines with a tilted turret axis

ESPRIT 2017 extends the support of sophisticated mill-turn machines to machines with a tilted lower turret. In previous versions of ESPRIT, the real tool shift could be input for turning tools on a tilted turret, but milling tools required a manual calculation of the orientation and tool shift by the user. ESPRIT 2017 now supports the input of the real tool shift for both milling and turning tools on tilted turrets.

- Input the true orientation of milling and turning tools on a tilted turret axis
- Input the true tool shift of milling and turning tools on a tilted turret axis

A new setting has been added to SolidTurn Machine Setup to support tilted cutting tools.

Figure 1. The new Tool Rotation Angle Mode in Machine Setup supports all types of cutting tools.



Tool Rotation Angle Mode supports two modes:

- **Turning Tools**: This is the default and produces the same behavior as previous versions of ESPRIT. The true tool shift and orientation of turning tools can be input on the tool page. However, for milling tools, the tool shift and tool orientation must be calculated manually by the user. This option provides support for legacy ESPRIT files.
- **All Tools**: The true tool shift and tool orientation can be set on the tool page for both milling tools and turning tools.

Figure 2. In ESPRIT 2016, the vector and tool shift for a milling tool required manual calculations using cos and sin.

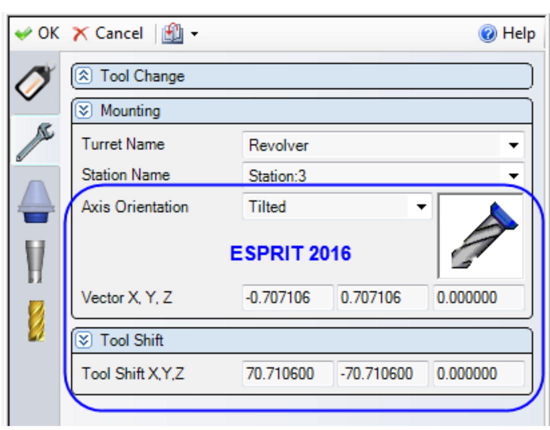


Figure 3. The new option in ESPRIT 2017 makes tool setup straightforward and intuitive. Tilt is handled at the machine, not in the tool.

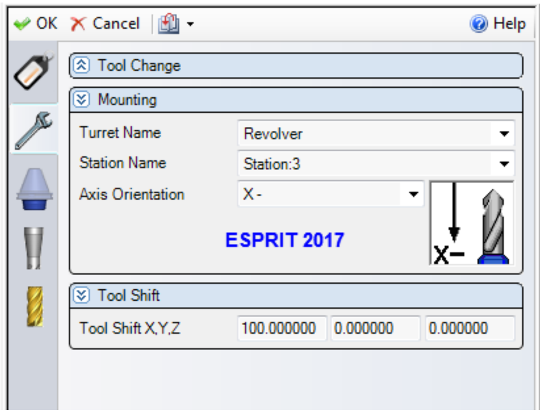
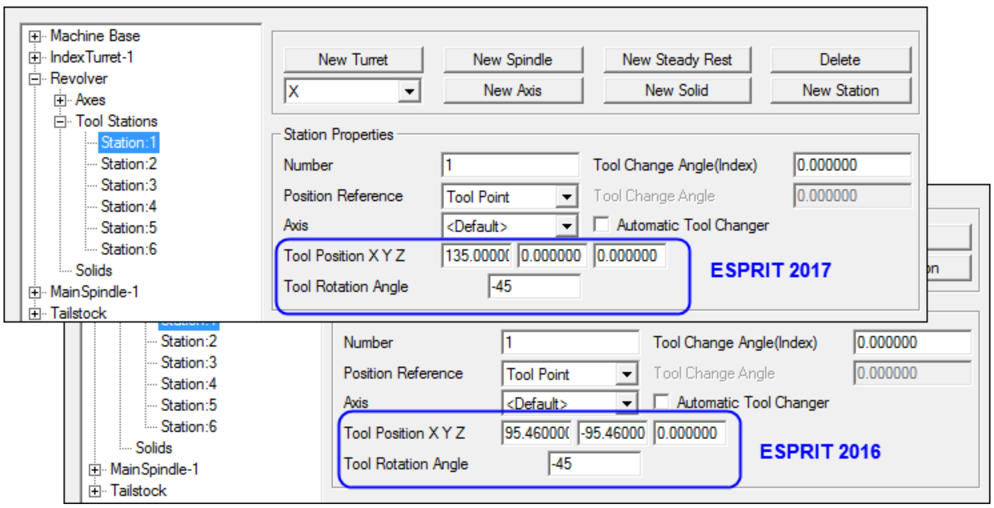


Figure 4. The new option also makes it easier to set up the position of tool stations on the machine.



▶ If you switch Tool Rotation Angle Mode, the system automatically calculates the new tool positon on each station.

NC code output has been improved to output code without the need for computations inside the post processor.

Previous NC code output with Y moves:

```
N35 G54
N40 T3 D1 M3=3 S2255
N45 SPOS[1]=0
N50 G0 C45.
N55 G0 X-45.255 Z-70. M8
N60 X-45.255 Y22.627 C315.
N65 X-45.255 Y22.627 C225.
N70 X-45.255 Y22.627 C135.
N75 G0 Z550.0
N80 X750.0
N85 M5
```

Simpler NC code output in ESPRIT 2017 considers the X-axis move of the turret in the tilted work coordinate. There is no Y move.:

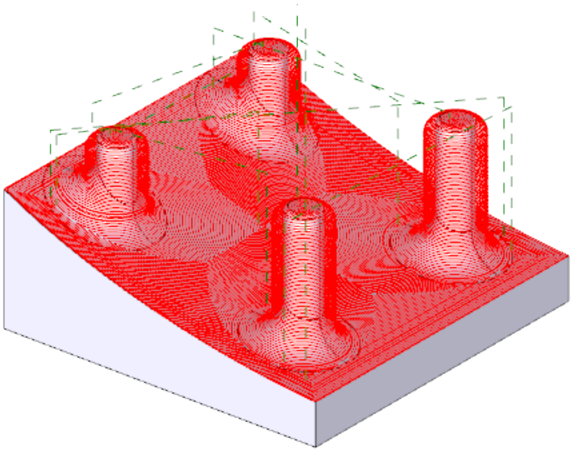
```
N25 MSG("C-AXIS DRILLING OD")
N30 SETMS(1)
N35 G54
N40 T3 D1 M3=3 S2255
N45 SPOS[1]=0
N50 G0 C90.
N55 G0 X-64. Z-70. M8
N60 G0 Z550.0
N65 X750.0
N70 M5
```

Sophisticated new 3-Axis Global Finishing for steep/shallow milling

Finish all zones of a complex part—steep and shallow—in a single operation, in less time to program. The new Global Finishing cycle combines the best of ESPRIT Z-Level Finishing and Floor Finishing in one comprehensive solution. Global Finishing optimizes the machining of complex parts by applying the most appropriate toolpath to steep and shallow areas based on a user-defined slope threshold angle.

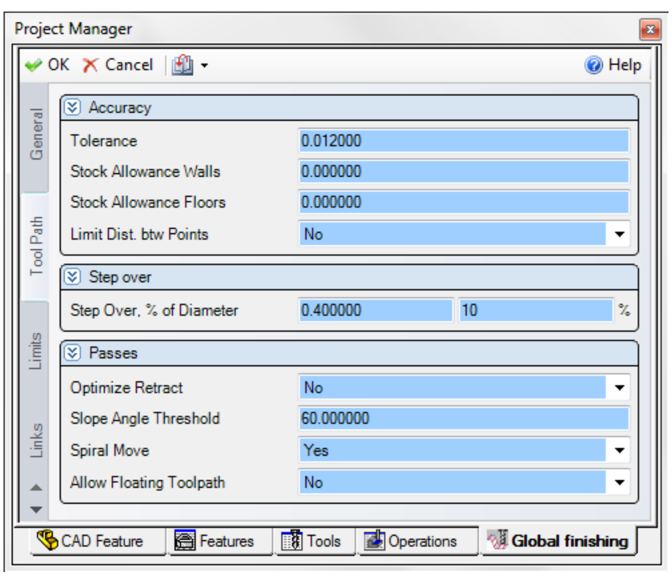
- Streamlined interface presents only key parameters for faster programming
- Sophisticated logic gives priority to continuity of toolpath between steep and shallow areas
- Optimized toolpath applies Z-level cutting passes to steep areas, offset passes to shallow areas
- User-defined slope angle applies appropriate toolpath to steep and shallow areas
- Option to machine over openings in surfaces without stopping or retracting the tool
- Climb mode and spiral mode are enabled by default for smoother finishing

Figure 1. Toolpath is optimized for steep/shallow zones.



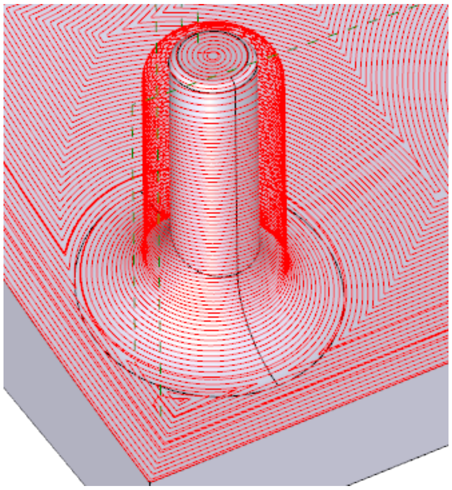
The streamlined interface for the new Global Finish cycle is designed to make programming faster than ever by presenting only key parameters to the user.

Figure 2. Global Finish parameters are easy to set.



Areas of the part having a slope above the **Slope Angle Threshold** are considered steep; areas below the threshold are considered shallow.

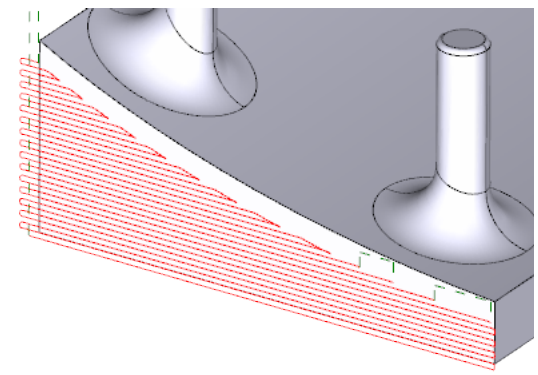
Figure 3. On steep areas, the system applies Z-level toolpath. On shallow areas, offset passes are applied.



The system first calculates Z-level toolpath on the entire part. All toolpath that flows entirely on areas above the slope threshold is maintained. Toolpath in areas below the slope threshold is replaced with concentric passes. This method gives priority to the continuity of the toolpath, and attempts to preserve uninterrupted Z-level passes as much as possible.

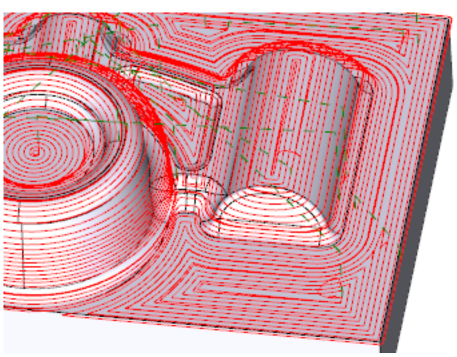
By default, climb cutting is used on all steep areas. However, the **Optimize Retract** option will allow the toolpath to alternate between climb and conventional cutting to minimize retract moves.

Figure 4. Optimize Retract will allow alternating toolpath on steep areas to minimize retracts.



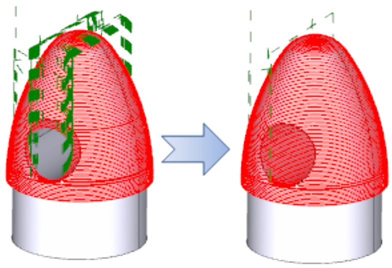
The **Spiral Move** option will apply spiral connections to both Z-level and concentric toolpath. Otherwise, toolpath is connected with an "S" shaped trajectory.

Figure 5. Z-level and concentric passes both take advantage of spiral connections.



In cases where the part has holes, the option for **Allow Floating Toolpath** will generate toolpath over the openings instead of going around them.

Figure 6. Machine over holes with the Allow Floating Toolpath option.

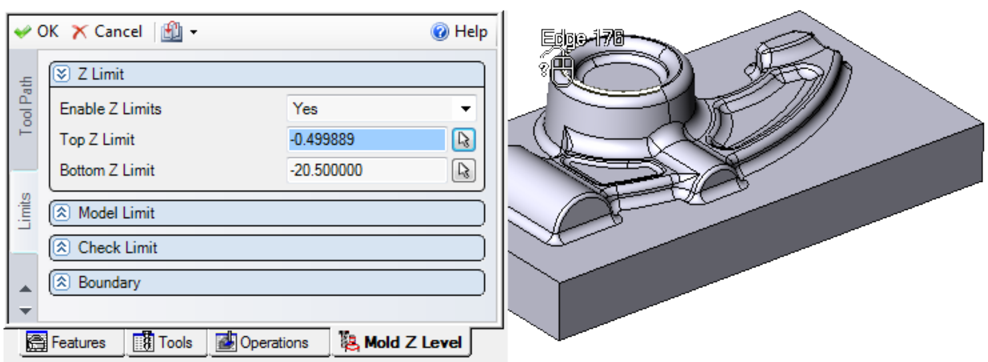


Expanded support for solid digitizing of FreeForm Z limits

Select Z limits for 3-axis FreeForm operations more quickly and easily with new support for any solid element. Select a face, loop, edge or vertex to automatically extract the Z coordinate for the operation.

- Select any solid element for upper and lower Z limits: face, loop, edge, vertex
- Top Z Limit extracts the highest Z coordinate
- Bottom Z Limit extracts the lowest Z coordinate
- Support also includes the selection of any geometry element or SNAP location

Figure 1. Z Limits can be set from the selection of any solid element (face, loop, edge, vertex).



Responsive Swarf toolpath computation

The aerospace industry spoke and we listened. ESPRIT 5-Axis Swarf is now more responsive to a wider range of part shapes, with multiple enhancements that improve the computation of swarf toolpath and automatically compensate for unusual part geometry.

- Automatic machining over gaps in surfaces for uninterrupted toolpath
- Reliable pass extensions that follow the curvature of the part profile
- Smoother tool positioning along walls with irregular or discontinuous borders
- Dependable cutting of small corners with large tools
- Optimized tool axis orientations when toolpath starts or ends on planar faces
- Smoothing of tool trajectories around sharp edges to dampen machine acceleration

Cover gaps automatically with uninterrupted toolpath

The Swarf cycle can now automatically compute toolpath that smoothly machines over gaps and irregularities in the part geometry without the need to create additional geometry, which could be extremely time consuming

Figure 1. Swarf machining will calculate toolpath to automatically cover gaps in part surfaces. In the case of irregular profiles, some air cutting portions are added to create passes without interruptions.

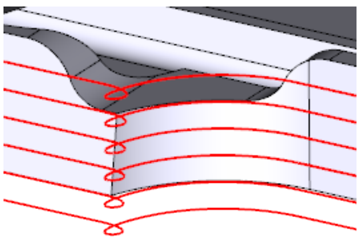
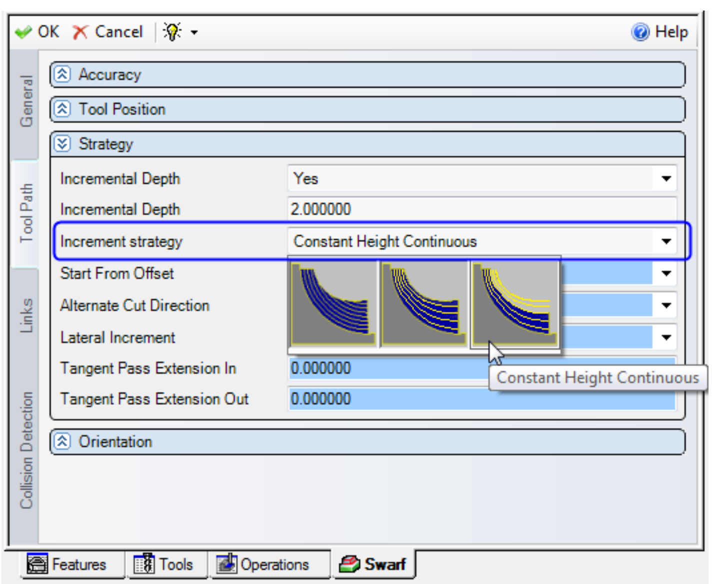


Figure 2. The new Constant Height Continuous option maintains a constant height even when there are gaps in the part geometry.



Follow the ideal surface with true tangent pass extensions

Walls with a sloped leading edge can now be fully machined with pass extensions that adhere to the curvature of the part. Rather than add a simple extension to the entry or exit move, which could compromise the part geometry, the swarf pass extensions are based on an analysis of the profile curvature without the need to actually extend the curve. For machining ribs on aerospace parts, true tangent extensions save programming time and improve the reliability of the toolpath.

Figure 3. When the edge of a wall is sloped, a true tangent extension will follow the ideal surface without compromising the part.

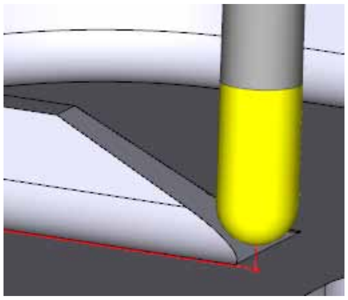
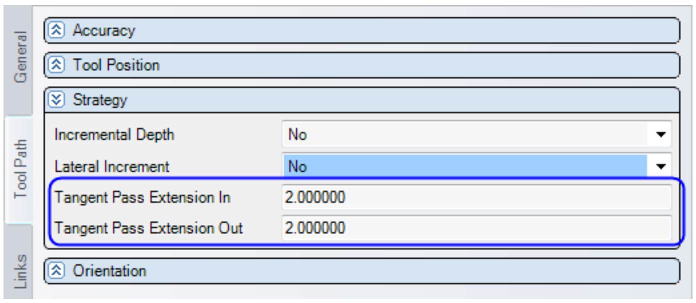


Figure 4. New Tangent Pass Extension settings extend the start or end of the toolpath tangent to the surfaces.



Position the tool smoothly between irregular profiles

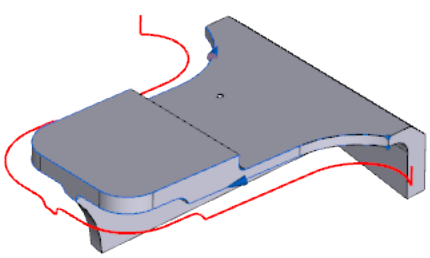
Users now have a range of possibilities to smoothly drive the tool tip along walls with irregular or discontinuous borders. The ability to control the tool within the operation eliminates the necessity for a manual redefinition of continuous profiles.

The user can choose an option that produces the best results for the wall being cut:

- Use either the upper or the lower profile of the wall to drive the tool.
- Have the system calculate either a curve or a plane for the tool to follow. The calculated curve or plane will be positioned so the tool will completely machine the surfaces comprised between the upper and lower profiles. This option is particularly useful when wall profiles have irregularities (gap in the surfaces).
- Project toolpath down until it reaches a floor.

When there are irregularities between the upper and lower profiles of a wall, simply following the profiles will result in an undesirable up-and-down toolpath.

Figure 5. Irregular toolpath when the tool follows the lower profile.



Now the user can control the position of the tool tip to create smooth toolpath even in the presence of irregularities or interruptions of the profiles that define the area to be machined.

Figure 6. Stable toolpath placed on a calculated plane.

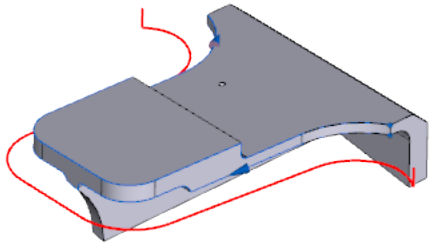


Figure 7. New Tool Reference settings offer a range of options for positioning the tool tip between the upper and lower profiles.

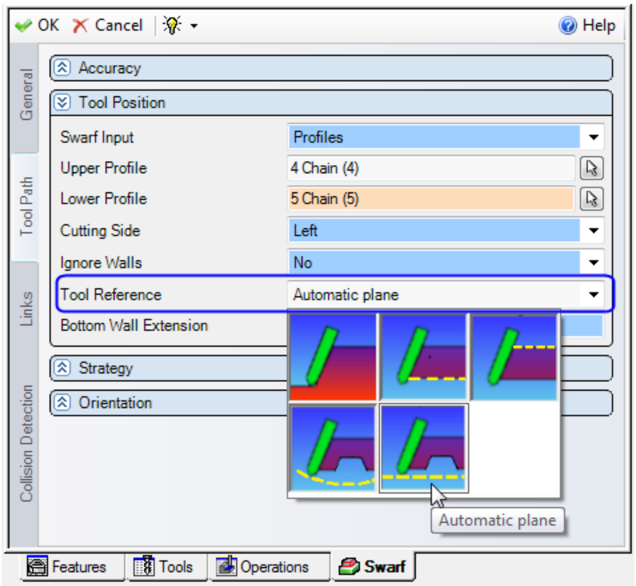


Figure 8. The Lower Profile option positions the tool tip to follow the lower profile.

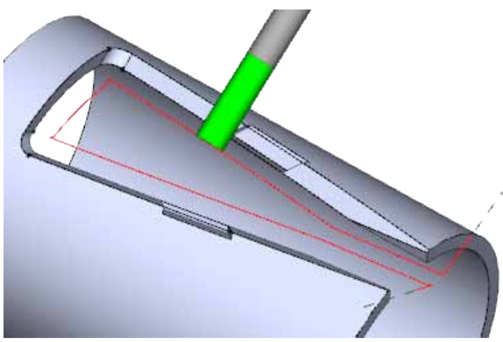


Figure 9. Upper Profile positions the tool to follow the upper profile, automatically shifting the tool along its axis to completely machine the wall.

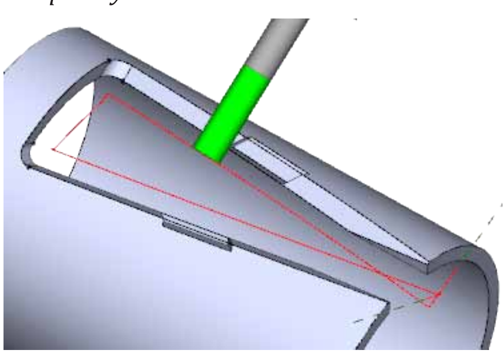


Figure 10. Automatic Curve calculates a curve to keep the tool axis as perpendicular as possible to the feed direction.

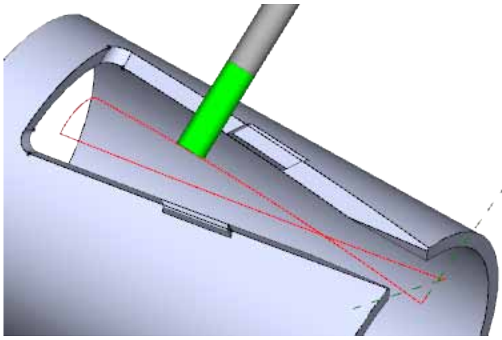


Figure 11. Automatic Plane is similar to Automatic Curve except the curve is planar.

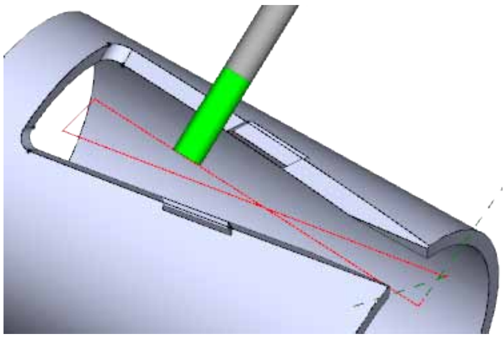
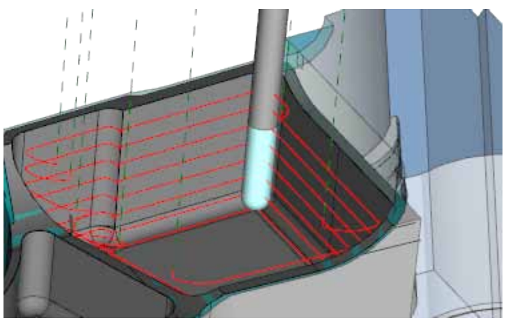


Figure 12. Stop at Floor protects floor surfaces. The tool stops when it reaches the bottom of the wall.



Cut internal corners more reliably with larger tools

In swarf machining, the edge of the tool must maintain constant contact with the part geometry to prevent marks on the surfaces. An enhanced calculation of touch points between the tool and the part will now prevent the tool from losing contact with the wall, even when the tool will not fit into a corner.

Previously, the calculation of swarf toolpath on internal corners caused a bitangency condition when the tool radius was larger than the part radius. The tool would lose contact with the wall, causing interruptions in the cut and extra repositioning moves

Figure 13. Previous toolpath would reposition in corners where the tool could not fit.

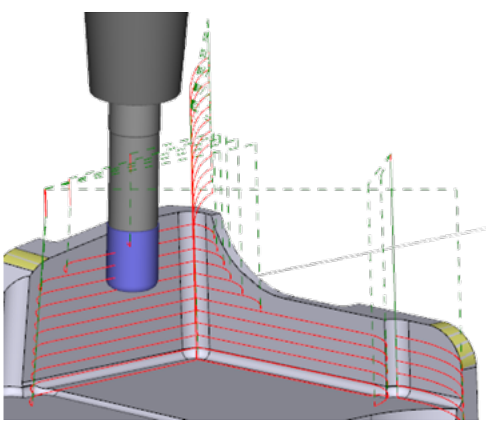
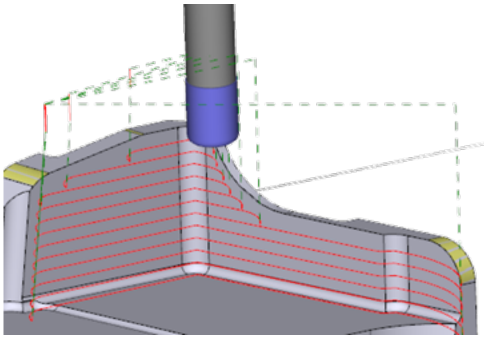


Figure 14. Enhanced toolpath calculation finds tool touch points more accurately.

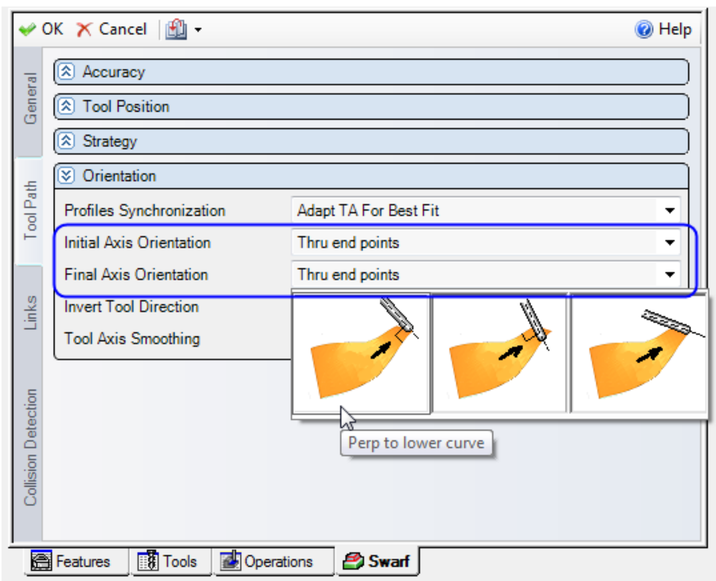


Optimize "Best Fit" tool orientations on planar faces

When Profiles Synchronization is set to Adapt TA For Best Fit, the system finds the best tool axis orientation to minimize the residual material remaining on the machined surface. This method is especially suitable for twisted surfaces, to determine the best orientation of the tool without the necessity of adding extra match lines.

Now, when there are planar faces at the beginning or end of the toolpath, the user can control the orientation of the tool axis even when the upper and lower profiles have different lengths.

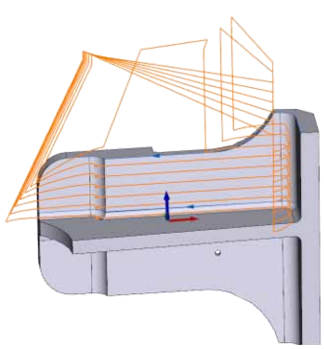
Figure 15. New orientation settings help control the tool axis on planar surfaces at the beginning and end of the cut when Adapt TA For Best Fit is used.



Three options are available:

- Perpendicular to lower curve: Keeps the tool axis perpendicular to the lower profile.
- Perpendicular to upper curve: Keeps the tool axis perpendicular to the upper profile.
- Thru end points: Keeps the tool axis parallel to a segment joining the end points of the profiles.

▶ It is only necessary to specify this option when the faces at the beginning or at the end of the toolpath are PLANAR. If the faces are not planar, the tool orientation is automatically optimized, without the necessity of user intervention.



Smooth the trajectory of the tool axis between convex and concave surfaces

Sometimes during Swarf machining, the tool makes marks on the wall surfaces when surfaces change from concave to convex and vice versa. In most cases, the marks are due to excessive acceleration of the machine axes.

Figure 16. The new Tool Axis Smoothing setting helps control acceleration between convex and concave surfaces for a smoother toolpath.

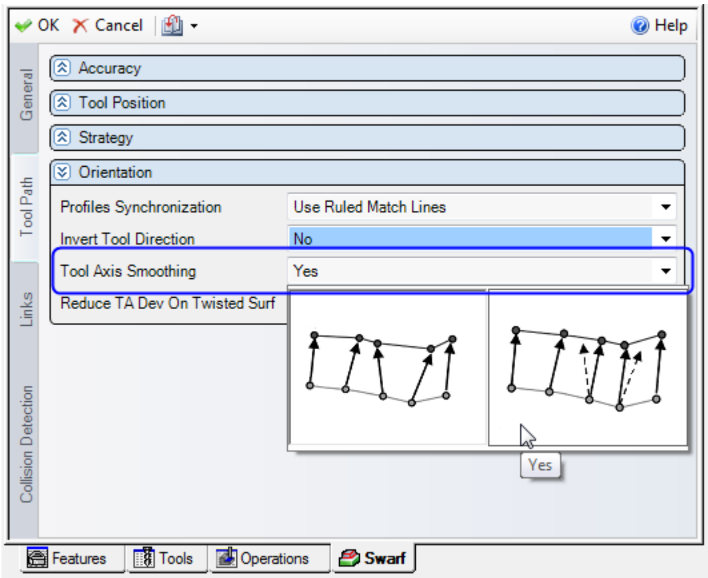
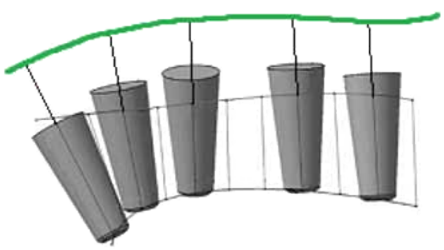


Figure 17. Tool Axis Smoothing acts to smooth the trajectory of the tool axis, reducing the machine axes acceleration and, as a consequence, marks are eliminated.

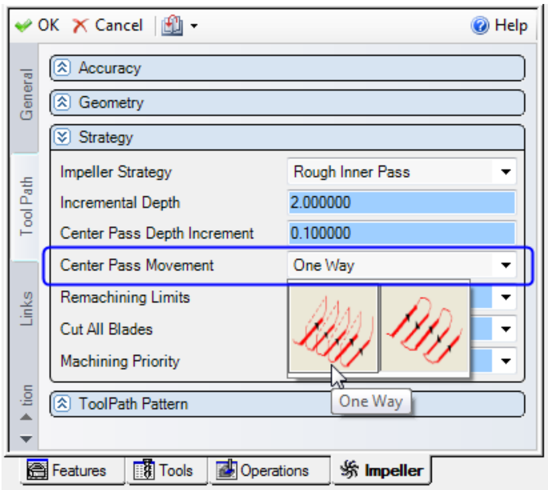


▶ The smoothing function is executed at the expense of a small loss in precision (a fraction of the Machining Tolerance). When the Part Geometry must be absolutely respected, this option should be disabled.

Take heavier rough cuts on impellers

The initial slotting passes in Impeller roughing can now be set to cut in a single direction to handle heavier material removal. When an impeller roughing strategy includes center passes (Rough Single Pass or Rough Inner Pass), the user can now choose whether to create the center passes in a single direction or allow the tool to alternate directions.

Figure 1. A new option controls the direction of center passes for impeller roughing.



EDM Pocketing on open profiles

In response to requests from the field, it is now possible to apply an EDM pocketing operation to open features. Open pocketing makes it easier and faster to cut small openings along a contour. Simply cut the contour first with look-ahead enabled to stop the wire from entering small openings. Then follow up with pocketing operations that burn away only the small amount of material that remains.

- Apply no-core pocketing on open profiles
- Save time by contouring larger areas first, then pocketing only where needed
- Quickly customize the operation order to rough the contour and pockets before skimming the entire profile

Figure 1. Combine contouring and pocketing for more efficient wire path.

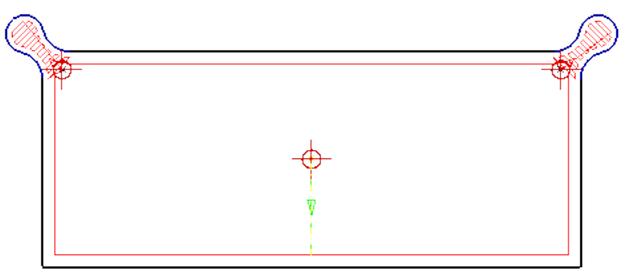


Figure 2. Contouring inside small internal openings can create self-intersecting wire path, possibly violating the profile.

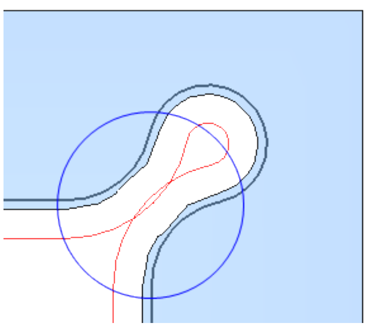


Figure 3. No-core pocketing on the entire profile is inefficient and time consuming.

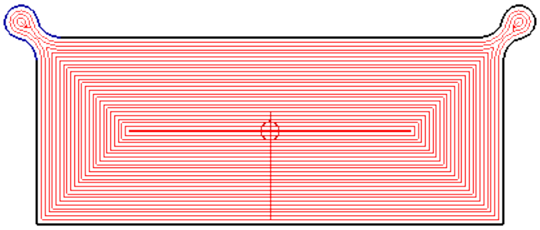


Figure 4. Enabling the Look Ahead function in a contouring operation will avoid small areas where the wire will not fit.

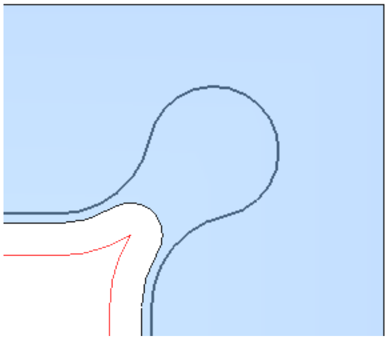


Figure 5. Pocketing on an open profile will rough only the remaining material.

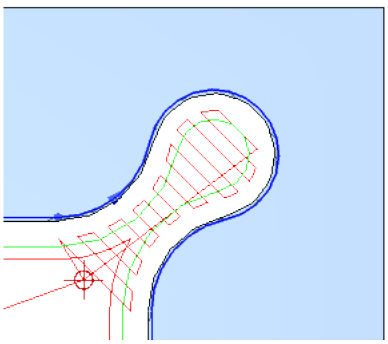
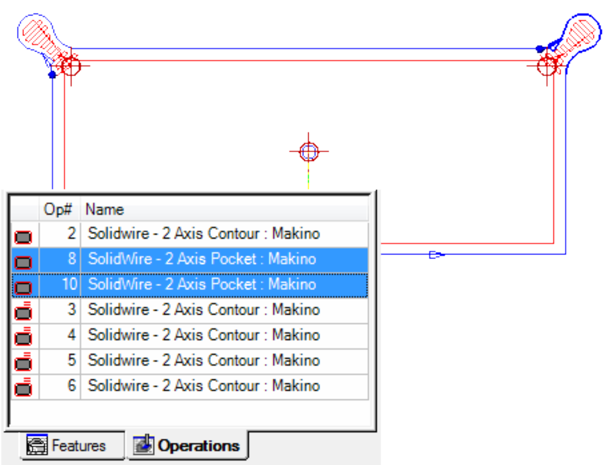


Figure 6. Changing the operation order to perform pocketing between rough and skim cuts results in more efficient wire path.

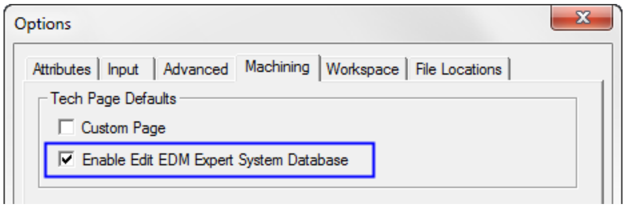


Edit Expert System data directly from ESPRIT

The data in an Expert System file can now be edited directly in ESPRIT from the database browser.

- Easily modify a wire EDM database from inside ESPRIT
- Quickly create a custom database
- Add, edit, move and rename list items
- Add new fields to the database
- Add, edit and delete cut data

Figure 1. A new option in the Options dialog enables or disables the editing of databases in the EDM Expert System.



When database editing is enabled, values can be edited in the database browser and extra icons are displayed that allow the user to edit items in the database, add items, and to save changes to the current database or save the changes to a new file.

Figure 2. Edit values under Series Technology to create a custom database.

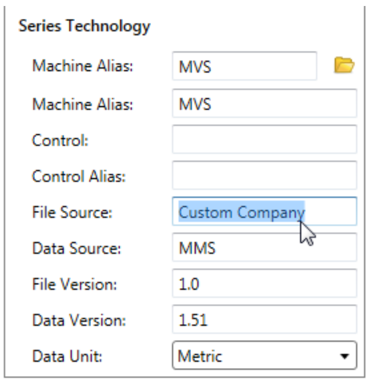


Figure 3. To add, edit, rename, and move items in a list, click Edit Items.

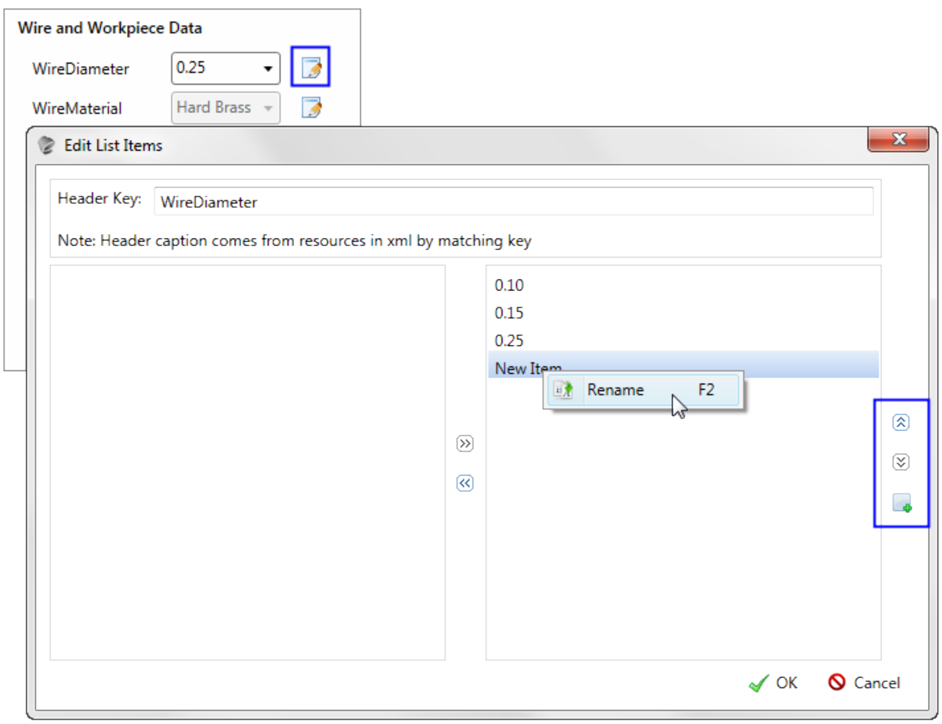


Figure 4. Click Add SubItem to add a new field to the database.

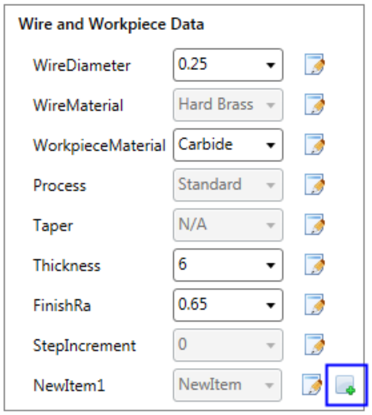


Figure 5. Edit the Cut Strategy by adding new rows, editing values in a row, or deleting a row.

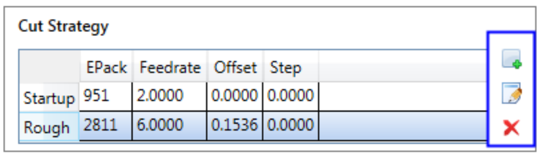
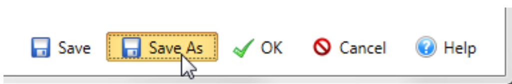


Figure 6. Save changes by either overwriting the current database (Save) or saving to a new file (Save As).



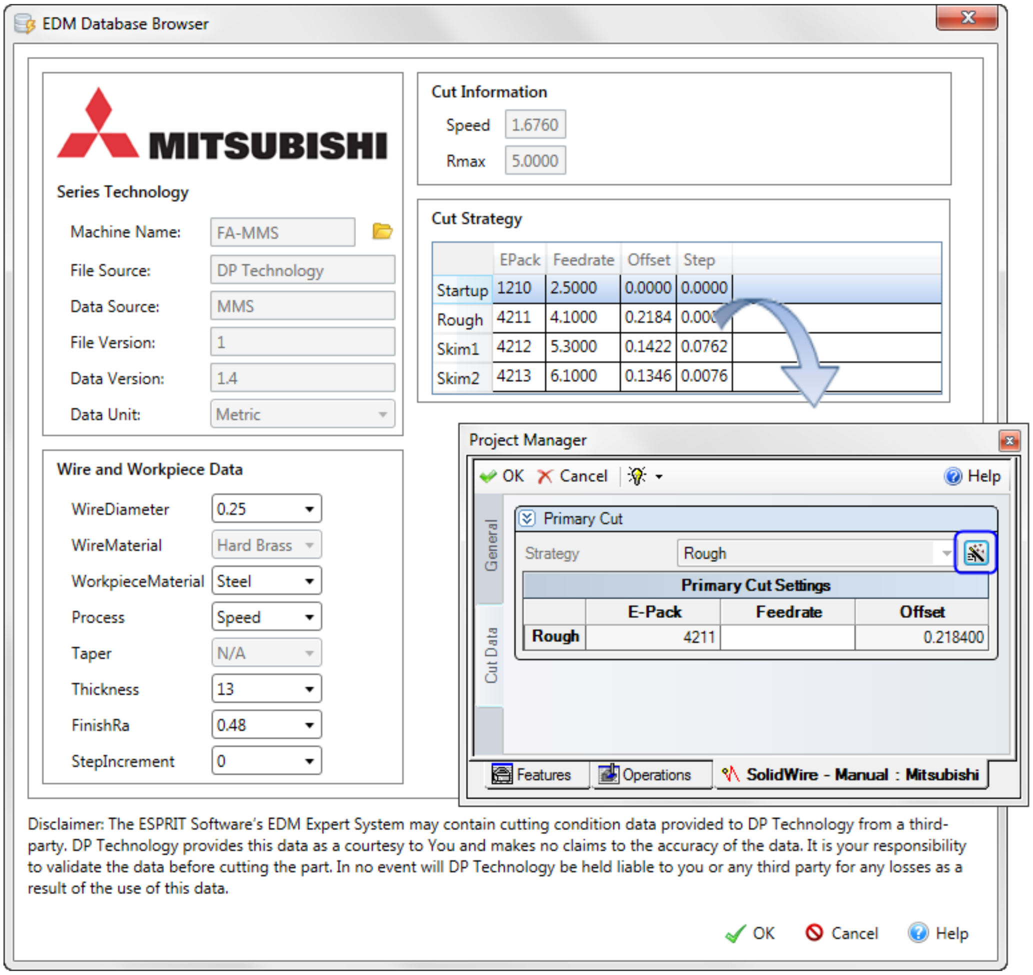
▶ Values in the current EDM operation will update when OK is clicked. However, values in other operations will not be updated automatically. To update existing EDM operations with updates from the database, you must edit the operation and reopen the Expert System to load the new values on the technology page.

EDM Expert System now available for Manual EDMing/Agie123 machines

ESPRIT users can now access and utilize the Expert System database for any Manual EDM operation, as well as use the Expert System for Agie123 machines.

- Access the Expert System for all Manual EDM operations
- Access an Agie123 database directly from ESPRIT

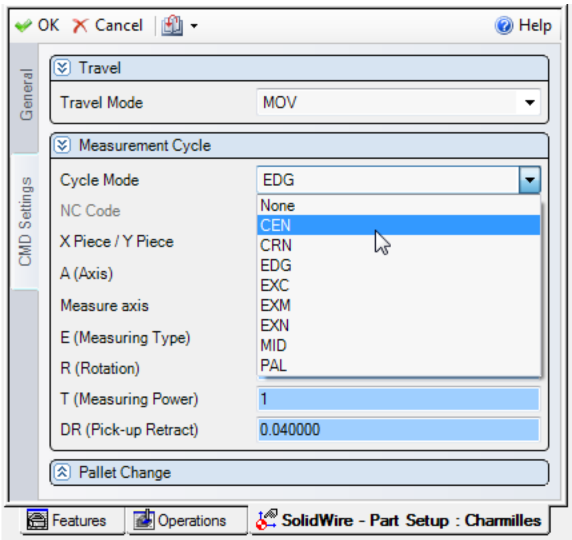
Figure 1. The EDM Expert System is now more widely available for SolidWire operations in ESPRIT.



New EDM Measurement Cycles for Charmilles HMI

New Measurement Cycle Modes have been added for Charmilles HMI machines.

Figure 1. Part Setup has new measurement cycle modes.

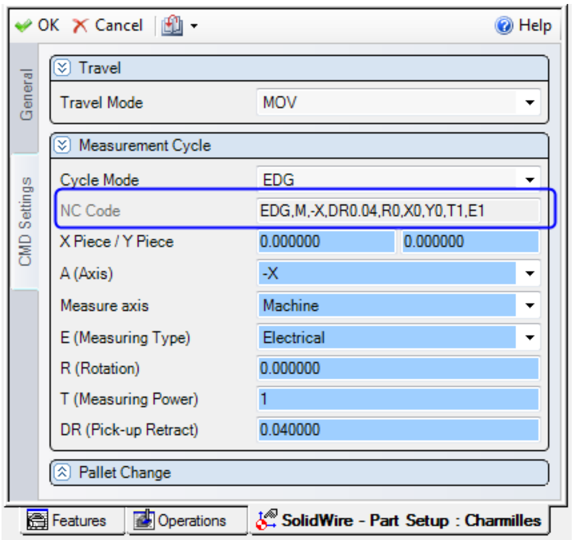


Supported measurement cycles for Charmilles HMI include:

- CEN – centering on the internal of a radius or diameter
- CRN – corner cycle
- EDG – edge search cycle
- EXC – center of an external cylinder, distributed according to rotation angle
- EXM – search cycle for the middle between 2 parallel faces
- EXN – middle between 2 parallel faces by bypassing part in XY plane
- MID – internal centering cycle between two parallel faces
- PAL – part alignment cycle according to the machine X axis

The ESPRIT Post has been updated to support these cycles and a preview of the output NC code is also displayed in Part Setup.

Figure 2. Example NC code




```
    <Program>
      <ObjectId>10002</ObjectId>
      <Name> Measurement Cycle </Name>
      <OPLanguage>CMD</OPLanguage>
EDG,M,X,DR0.04,R0,X10,Y10,T1,E1
      </Operations>
    </Program>
```

Rendering of rapid tool motion

Now both feed and rapid tool motion is displayed on the screen to better visualize movement of the tool without the need to run a simulation. This enhancement is especially useful for evaluating rapid tool motion in rotary and multi-axis machining operations.

- Graphic display of tool motion for milling and turning
- Rendering of rapid motion as dashed lines
- Rendering of feed motion as solid lines

However, some operations generate rapid connections at run-time (drilling, for example). In those cases, rapid motion is not rendered on the screen.

Figure 1. Easily evaluate retract and clearance moves in milling operations.

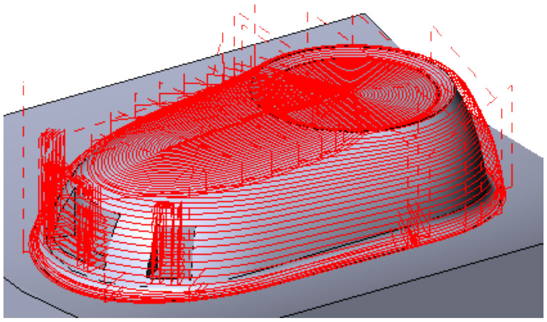


Figure 2. Visualize rotary clearance.

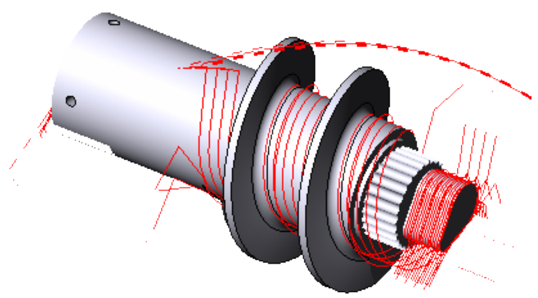
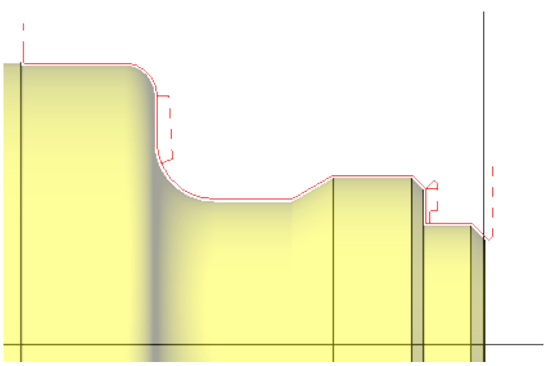


Figure 3. View entry and exit moves in turning operations.



Enhanced CAD to CAM interchange

ESPRIT now supports the import of sketch data from Inventor files, 3D PDF files, and cloud-based Onshape documents directly into ESPRIT.

- Support for Inventor sketch data with the latest Inventor FX Add-in
- Import of 3D PDF file types
- Import of cloud-based Onshape documents directly into ESPRIT with the new Onshape Connect Addin
- Import of STEP AP 242 file types, a STEP standard that is a convergence of STEP 203 and STEP 214 to allow companies to integrate all phases of the product lifecycle

3D PDF is a step in the direction of "drawingless design" allowing for 3D documents that are interactive, compact, and easy to share. This 3D PDF file support allows users to import the BREP (boundary representation) data contained within the PDF into ESPRIT.

Onshape is a new generation of CAD designed specifically for agile design teams, offering built-in collaboration and access to the same CAD system and the same CAD data on any device.

Figure 1. ESPRIT Onshape Connect Addin