



# The Path to a Smarter, Simpler, Faster ESPRIT®

## A Technical Overview

### Contents

|  |           |
|--|-----------|
| <b>Finding the Right Balance Between Power and Simplicity .....</b>              | <b>2</b>  |
| <b>Smarter Toolpath .....</b>  | <b>2</b>  |
| Applying sophisticated logic to steep/shallow milling .....                      | 2         |
| Reducing the reliance on features for facing operations .....                    | 4         |
| Opening more strategies for wire EDM pocketing .....                             | 9         |
| <b>Simpler Interaction .....</b>   | <b>12</b> |
| Automatically calculating optimal tool positions within milling operations ..... | 12        |
| Automatically adjusting the feature path when elements are removed .....         | 13        |
| Integrating the newest generation of 3D CAD models .....                         | 14        |
| User-guided wire EDM technology input .....                                      | 15        |
| <b>Faster Programming .....</b>  | <b>16</b> |
| Accelerating the calculation of in-process stock models .....                    | 16        |
| Continuously improving the performance of ProfitMilling .....                    | 16        |
| <b>Conclusion .....</b>  | <b>19</b> |

## Finding the Right Balance Between Power and Simplicity

*The answer to better productivity proves to be “less is more.”*

In the race to add technical features to CAM software in support of rapid advancements in machine tool technology, the CAM software industry tends to overlook the importance of the customer experience. Although an ever-expanding array of toolpath choices is good for selling software, what customers experience after the sale can be daunting as they attempt to carve out the time to learn dozens of functions, while maintaining productivity in the machine shop.

A well-designed CAM software should make a CNC programmer more productive, not less. Long-term gains in productivity require a three-pronged approach to CAM software development. First, build the CAM system on a strong foundation of machining intelligence garnered from decades of real-world experience with modern machining methods. Second, simplify the way users interact with the software to get reliable results with less effort. And third, harness the raw processing power of modern computers to compute toolpath in significantly less time.

ESPRIT® 2017 brings the customer experience to the forefront by focusing on creating a CAM software that is smarter, simpler, and faster to use.

## Smarter Toolpath

*Developing intelligent, optimized toolpath that handles routine tasks automatically.*

A long-term focus has put ESPRIT® in an industry-leading position since 1985. The machining intelligence learned over the years by ESPRIT® engineers is built into the software to streamline CNC programming without taking control away from the CNC programmer. Powerful new algorithms inside ESPRIT® 2017 deliver exceptional toolpath, more reliably, with far fewer clicks.

### Applying sophisticated logic to steep/shallow milling

Achieving the proper finish on complex surfaces often requires a combination of finishing strategies and experience in choosing the best strategy for each type of surface. ESPRIT® offers nine different finishing and semi-finishing cycles.

However, for many part models, Z-level toolpath is the first choice for steep or vertical surfaces and offset toolpath is often chosen for shallow or flat areas.

Global Finishing is a new strategy in ESPRIT® 2017 that combines the best of Z-level finishing and floor finishing in a single comprehensive solution for steep/shallow milling.

Global Finishing is designed to be fast, reliable, and easy to use. The user enters a threshold angle and a few other key parameters, like step over and tolerance. The system performs an internal analysis of the part model, using the threshold angle, to identify zones that are either steep or shallow.

The system initially calculates Z-level toolpath on the entire part. All toolpath that flows entirely on areas above the slope threshold is maintained, as these steep areas are the most critical for surface quality. Toolpath in areas below the slope threshold is replaced with offset passes.

This sophisticated method gives priority to the continuity of the toolpath and attempts to preserve uninterrupted Z-level passes as much as possible.

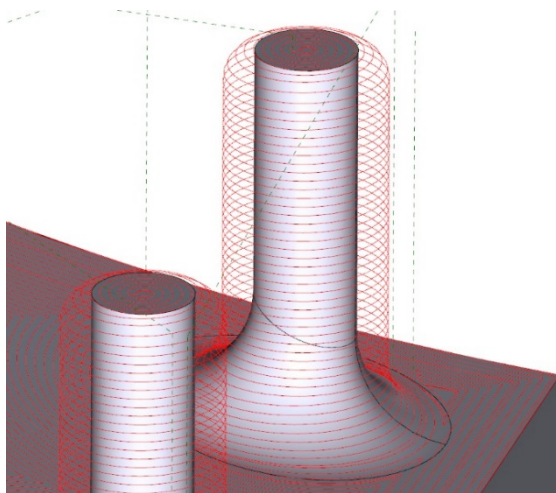


Figure 1. The system identifies steep areas above a threshold angle to machine them automatically with Z-level toolpath. Climb cutting is always maintained on steep areas.

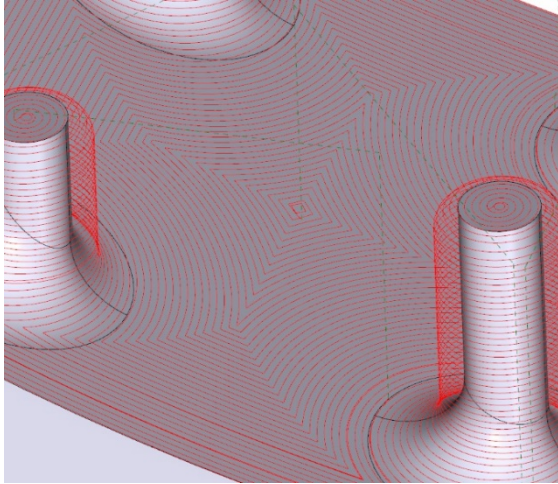


Figure 2. Shallow areas are identified below the threshold angle and machined automatically with offset toolpath.

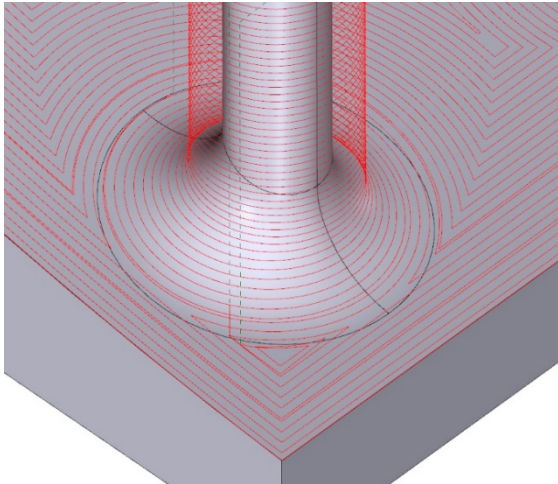


Figure 3. Priority is given to continuity of toolpath as the tool transitions between steep and shallow areas.

For parts of medium complexity, Global Finishing offers maximum time savings and optimized toolpath. For parts of high complexity, traditional ESPRIT® finishing strategies offer precise control and superior surface finish.

The initial release of Global Finishing is designed for optimized machining. Development is underway to enhance the range of toolpath types for shallow machining and offering the user the option of emphasizing optimization or surface quality.

## Reducing the reliance on features for facing operations

Most machining operations in ESPRIT® rely on the selection of a feature to determine where and how to calculate toolpath. But, for facing operations, the shape of a top face on the target part does not always correspond to the shape of the stock face to be cut. The user often wastes time creating extra geometry to represent the stock boundary.

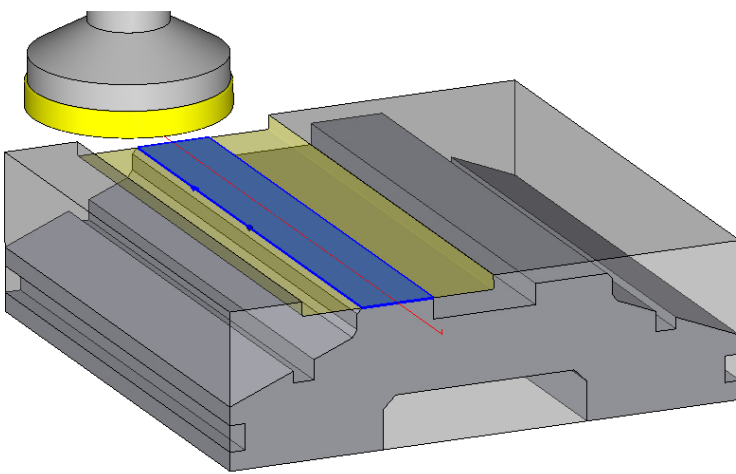


Figure 4. Machining a face without knowledge of the surrounding stock can result in incomplete or inefficient toolpath, wasting time when extra geometry must be created.

Relying on the shape of a feature for facing operations also ignores the fact that the shape of the stock changes during machining. Since facing operations are often the first step in shaping a piece of raw material into its final form, knowledge of the stock shape takes precedence over knowledge of the target shape.

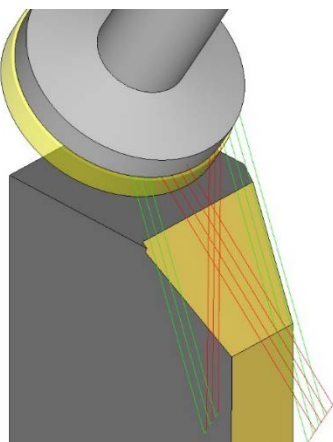


Figure 5. Facing toolpath that has no knowledge of changes to the in-process stock cannot optimize the angle and length of cutting passes as the shape of the stock face changes.

The facing cycle in ESPRIT® 2017 is redesigned to rely less on the shape of a feature and more on the actual shape of the stock model to calculate optimized toolpath. With up-to-date knowledge of the current state of the stock, a CAM system gains the ability to internally calculate optimized toolpath for facing operations.

Knowledge of in-process stock offers several benefits.

- The CAM system can extract accurate boundaries from the shape of the stock that encompasses the part model.
- A single selected face on the solid model, regardless of shape or tilt, can be used to set the work plane and final depth (plus stock allowance) for the machining operation.
- The CAM system can use knowledge of the cutter in relation to the in-process stock to calculate an ideal toolpath more quickly and accurately than manual programming.
  - Automatic calculation of safe start position to avoid collisions
  - Optimal overhang along edges for a clean finish
  - Optimal lead-in and lead-out distances to save cycle time
  - Optimal cutting angle at all incremental depths

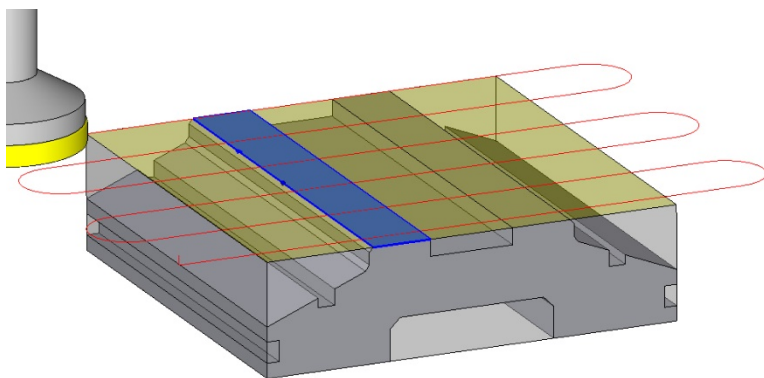


Figure 6. Knowledge of the stock model and the cutting tool allows ESPRIT® to internally calculate optimized toolpath with minimal intervention by the user.

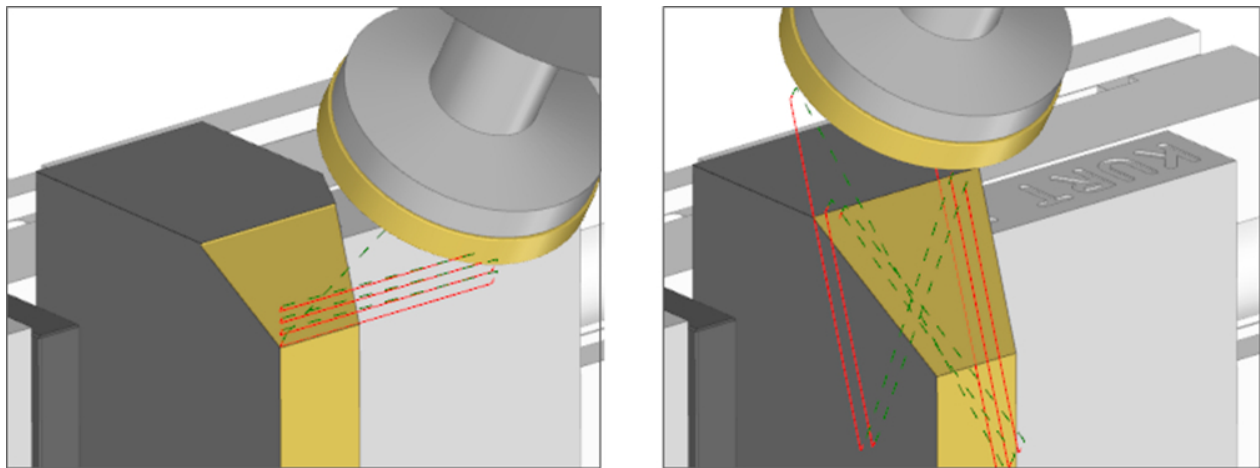


Figure 7. Facing toolpath that continuously computes the stock can adjust optimal cutting angles as the shape of the stock face changes.

Smarter facing in ESPRIT® 2017 gives CNC programmers the option to select a solid face, choose a cutting tool, and let the system do the rest. All the familiar manual controls remain untouched.

### **Automatically compensating for irregular part geometry in swarf machining**

Unusual part geometry is the norm and the challenge for aerospace suppliers. When a CAM system cannot compensate for irregularities in a solid model to be machined, the CNC programmer is tasked with creating additional geometry or surfaces to produce an acceptable toolpath.

An advanced CAM system can compensate for the challenges of unusual part geometry through analysis of the solid model to support smooth, reliable 5-axis swarf machining without the need for extraneous geometry.

5-Axis swarf machining in ESPRIT® 2017 creates a closer bond between the part model and the computation of touch points on the cutter, making the toolpath more responsive to a wider range of part shapes.



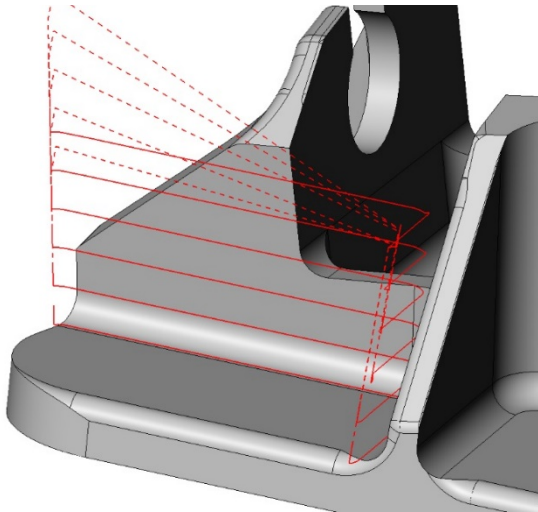


Figure 8. Continuous machining over gaps in walls is achieved by automatically calculating toolpath trajectories when the tool loses contact with a wall.

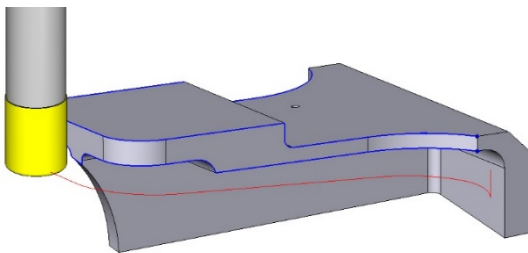


Figure 9. Tool positioning along walls remains smooth regardless of whether top or bottom profiles are irregular or discontinuous.

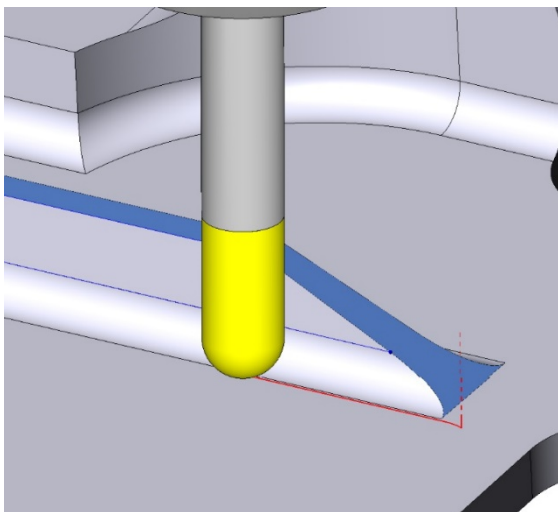


Figure 10. Toolpath extensions adhere to the curvature of the part to reliably and completely machine ribs that have angled leading edges.



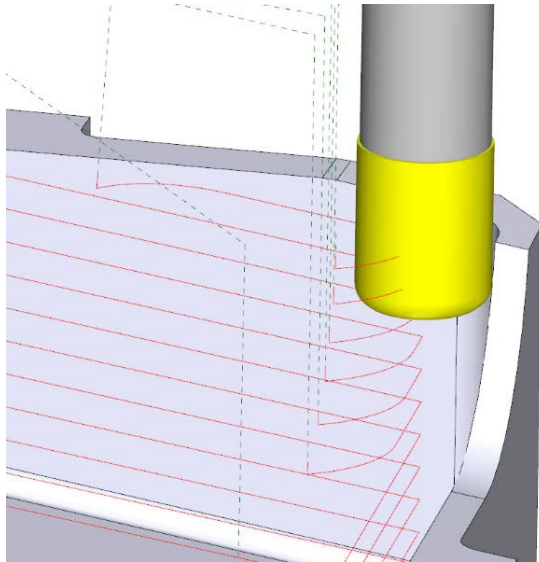


Figure 11. Accurate calculation of touch points between the tool and the part prevents the tool from losing contact with the wall, even when the tool diameter will not fit into a corner.

Swarf toolpath uses advanced algorithms within the operation to eliminate the need to manually create extra geometry to save programming time and improve the reliability of the toolpath.

### Opening more strategies for wire EDM pocketing

Producing intricate profiles is the domain of wire EDM, capable of creating ultra-precise shapes and superior finishes that are near-impossible to machine by other means. However, just because a shape is hard to machine does not mean it should be hard to program.

As a leader in CAM for wire EDM, ESPRIT® consistently supports innovations in wire EDM programming. Innovation begins by challenging traditional thinking. Innovation also begins by applying traditional methods in untraditional ways. For example, ESPRIT® has long supported open pocketing and morphed toolpath for milling operations. What if that technology were applied to wire EDM as well? It would free EDM programmers to cut more types of profiles more easily.

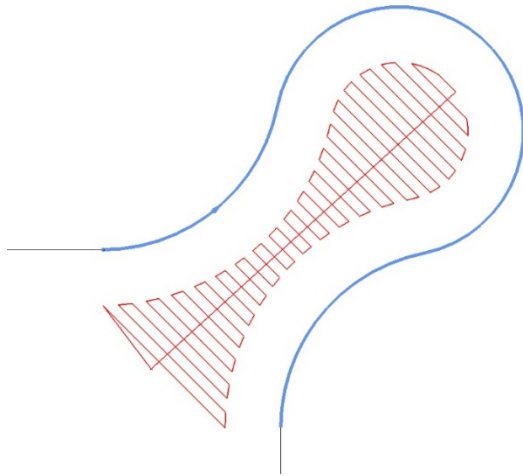


Figure 12. Efficiently burn small openings with no-core pocketing passes, regardless of whether the profile has a closed or open shape.

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.

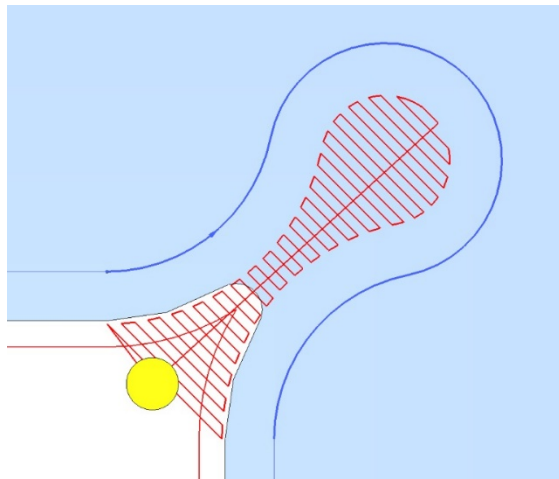


Figure 13. Combine contouring and open pocketing to efficiently cut awkward profiles.

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.

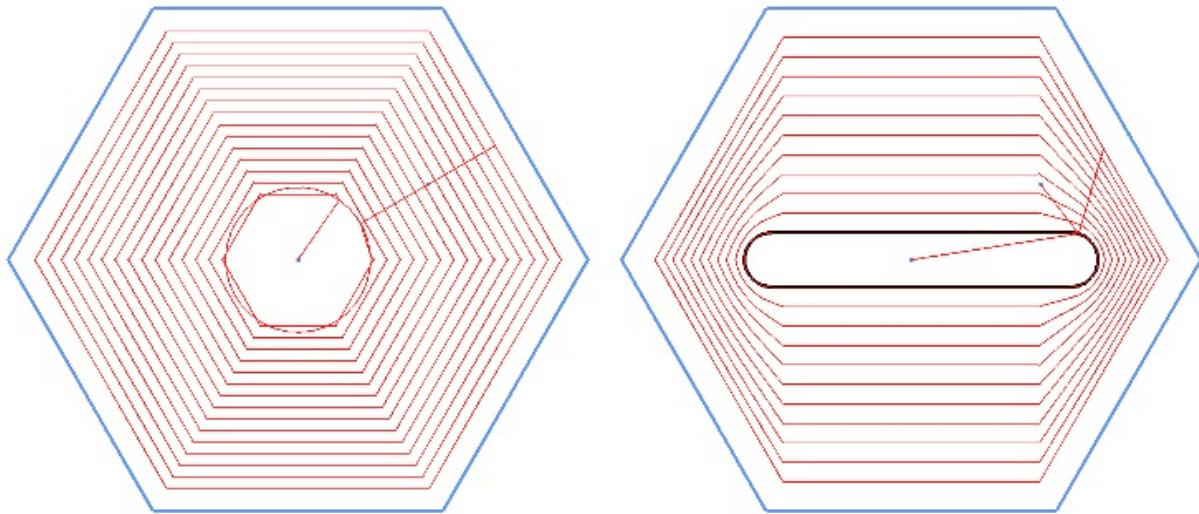


Figure 14. No-core pocketing offers the option of a constant offset around a pre-drilled hole or a smooth transition between the shape of an internal cavity and an outer profile.

ESPRIT® Wire EDM has a long tradition with wire EDM partners, customers, and experts to consistently understand the needs of the field and provide innovative solutions.

## Simpler Interaction

*Using less effort to get results faster, more reliably.*

A challenge with simplifying a CAM system is to give the user control over the machining process while reducing time and effort. Getting fast, reliable results with less effort is achieved by shifting time-consuming tasks that have little added value to a computer system that is more adept at performing routine tasks.

### Automatically calculating optimal tool positions within milling operations

Accurate knowledge of the stock model allows ESPRIT® 2017 to automatically calculate optimal tool positions in roughing operations by knowing the exact position of the cutter in relation to the stock model.

To make programming faster and more reliable, the user can let ESPRIT® calculate certain parameters internally for facing, rough pocketing, and rough contouring operations. When automation is enabled, the interface presented to the user is simplified to show only key parameters to speed the decision-making process.

With accurate knowledge of in-process stock, the system knows the perfect start height for the cutter, knows where to position the bottom of the cutter along open edges for a clean finish, and can calculate the optimal distance for tool lead-in and lead-out at each incremental depth of cut.

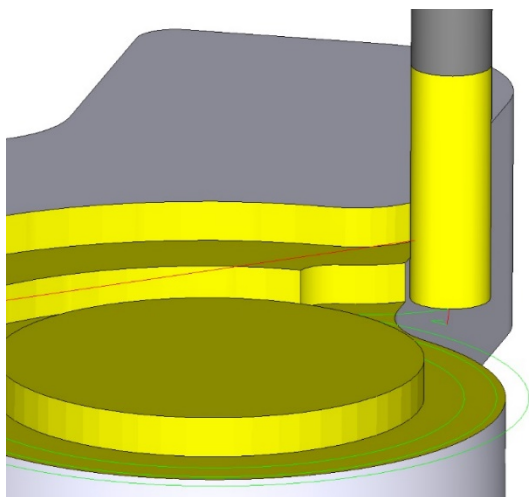


Figure 15. Tool positioning is optimized by the system to reduce cutting time and safely avoid stock at all incremental depths.

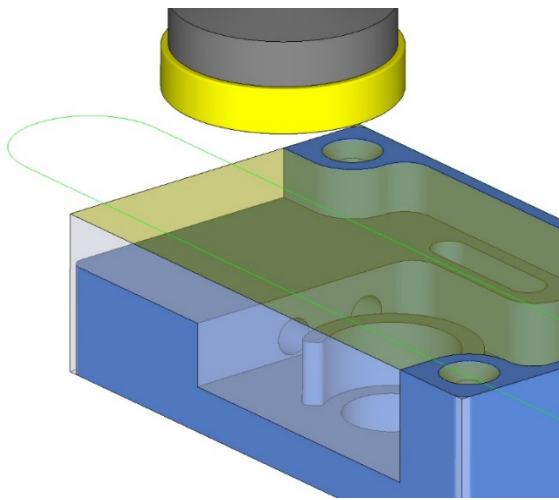


Figure 16. The bottom of the cutter is positioned on open edges to always produce a clean finish.

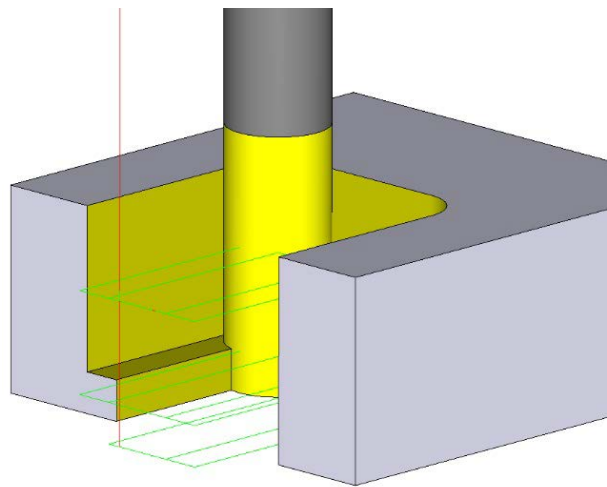


Figure 17. Contouring toolpath is calculated to rough all stock at all levels.

Automation within ESPRIT® takes the guesswork out of rough milling without taking control of the process.

### **Automatically adjusting the feature path when elements are removed**

Feature recognition in a CAM system must preserve the design intent of the original CAD model. However, the features created by a design engineer often need editing to make them more functional for machining operations.

Feature recognition in ESPRIT® is fast and easy, and so is the task of editing those features.

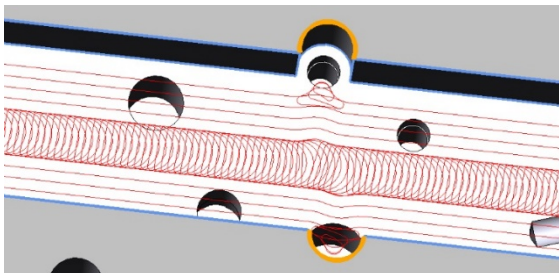


Figure 18. Direct edits on a feature make it easy to select individual elements to remove from machining.

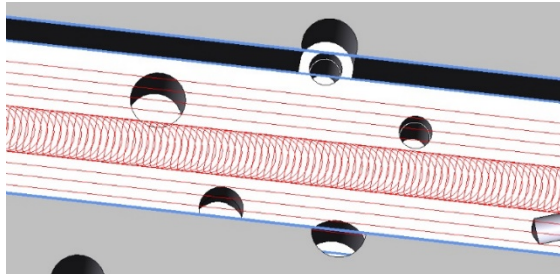


Figure 19. The feature is instantly updated to close the gaps. An update of the toolpath removes the holes from machining

The way features are edited in ESPRIT® 2017 is straightforward to more easily bypass geometry you do not want to machine.

### **Integrating the newest generation of 3D CAD models**

The first step in smoothing the path from design to NC code is intelligent integration of 3D CAD models into the CAM system. The architecture of ESPRIT® is built to provide a natural path between the newest generation of CAD models, including full-cloud CAD, and ESPRIT®.

As the CAD industry evolves, so does ESPRIT®.

- Onshape Full-Cloud CAD is a new generation of CAD designed specifically for agile design teams, offering built-in collaboration and full-cloud access to the same CAD system and the same CAD data on any device. ESPRIT® offers a free Onshape Connect add-in to import cloud-based Onshape documents directly into ESPRIT®.
- 3D PDF is a step in the direction of "drawingless design," allowing for 3D documents that are interactive, compact, and easy to share. 3D PDF file support allows users to import the BREP (boundary representation) data contained within the PDF into ESPRIT®.
- STEP AP 242 is a new STEP standard that is a convergence of STEP 203 and STEP 214. STEP support in ESPRIT® allows companies to integrate all phases of the product lifecycle.

A seamless CAD to CAM path gives CNC programmers complete and accurate part data to simplify the path to accurate NC code.

## User-guided wire EDM technology input

The ESPRIT® EDM Expert System is a universal data reader that gives users access to manufacturer-recommended cut data. The Expert System is an integrated component of ESPRIT® wire EDM. The user accesses the Expert System with a single click, chooses a wire, and enters workpiece data to display an optimal cutting strategy, then loads the manufacturer-recommended values into their operation with a single click.

The EDM cut data is stored in external XML files that are simple to use and easy to update. However, instead of exiting ESPRIT® and working with XML code directly, the Expert System lets CNC programmers edit their database directly in the ESPRIT® work environment.

Using the databases provided by ESPRIT® as a baseline, wire EDM users have the option to modify the data while they work to create a custom database more finely tuned to the way they cut parts.

Directly inside the Expert System data reader, users can easily add new items to lists, add new fields for workpiece data, and create customized cut strategies. Changes to the database can be saved to a new file, or the current database can be overwritten.

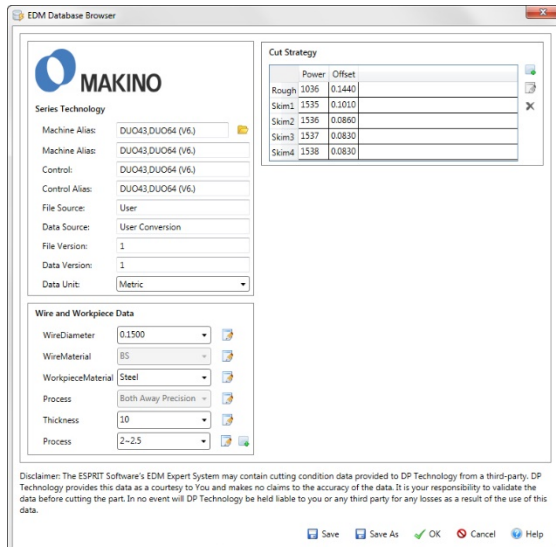


Figure 20. The ESPRIT® EDM Expert System makes it easy to access and customize manufacturer-recommended EDM cut data.

This simplified method of editing data in a familiar environment lets EDM programmers test and fine-tune cut strategies quickly while they work, without the need to learn XML coding.



## Faster Programming

### *Putting machining processes on the fast track.*

Refinement of the raw processing power in modern computers has significantly reduced the time needed to compute in-process stock models, thus accelerating programming efficiency.

### **Accelerating the calculation of in-process stock models**

CAM systems that utilize a digital representation of the stock model allow users to work as close to real life as possible. ESPRIT® utilizes an internal stock automation engine to calculate the condition of the material during the calculation of toolpath. These dynamic stock models are updated the same as if the material is being cut on the machine.

Since the actual stock condition is known during all phases of roughing, ESPRIT® can calculate reliable toolpath that does not waste movement on areas where material has already been removed.

However, as more operations are applied to the stock, the in-process stock model becomes more and more complex until adding even a simple operation can result in long calculation times.

The Stock Automation Engine inside ESPRIT® underwent a refactoring in ESPRIT® 2017 to drastically reduce calculation times. Now, when the stock automation option is applied in facing, pocketing and contouring, these milling operations are calculated in seconds rather than minutes.

- Pocketing: up to 25 times faster
- Contouring: up to 3 times faster
- 3-Axis mold roughing: up to 12 times faster

The same refactoring is applied to 3-axis mold roughing to compute in-process stock in dramatically less time, even on complex parts.

### **Continuously improving the performance of ProfitMilling™**

ProfitMilling™ technology developed by ESPRIT® is a high-speed roughing strategy that allows a significant increase in cutting feed while taking a deeper cut of the tool. ProfitMilling™ technology manages chip load and side-cutter force in the calculation of toolpath while keeping the engagement angle and material removal rate within a specific range.

The technology behind ProfitMilling™ was 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.

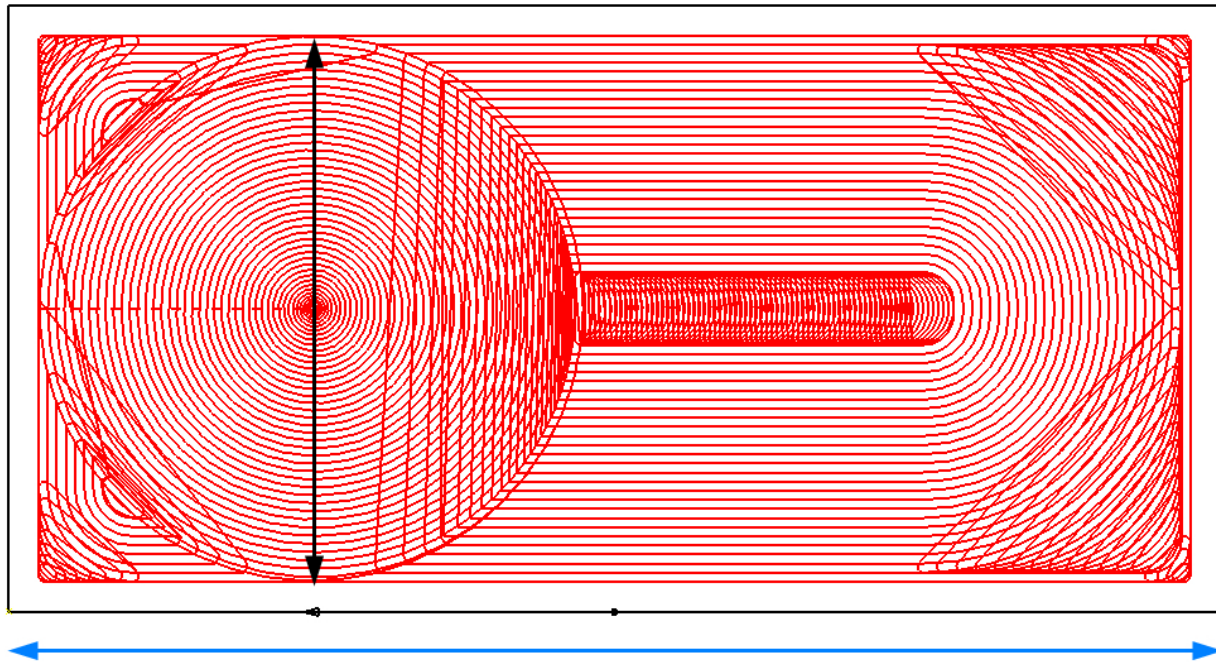


Figure 21. A larger initial spiral opens closed cavities faster for better chip evacuation and coolant access.

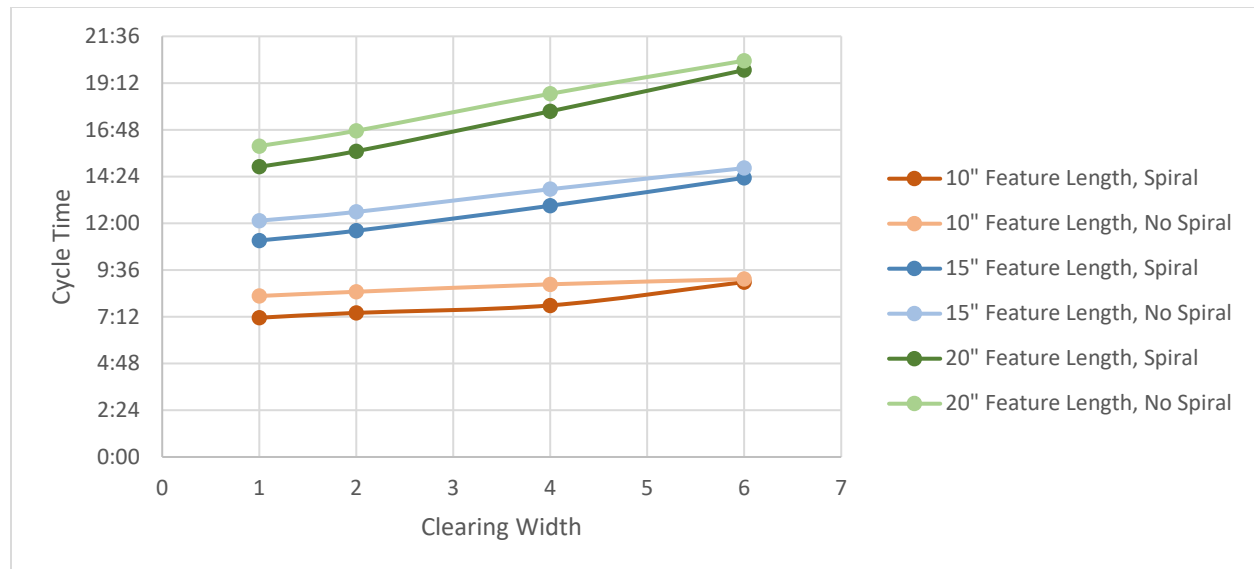


Figure 22. Tests showed that an initial spiral out resulted in the fastest cycle times.

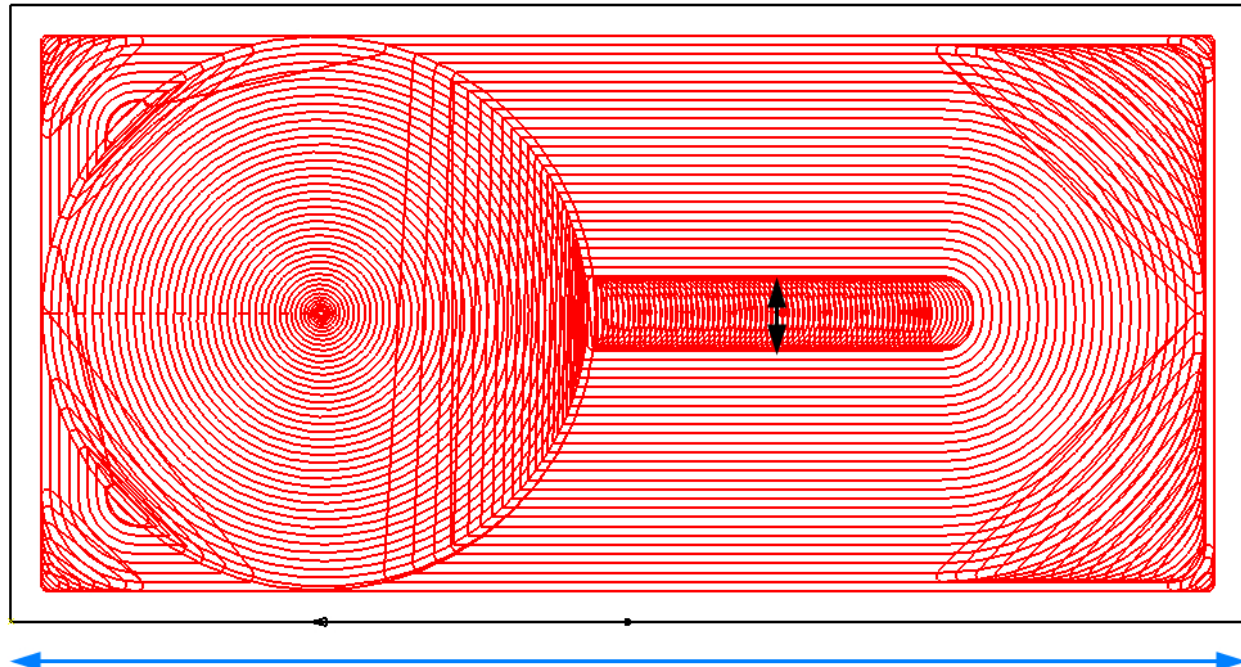


Figure 23. The width of the initial channel is optimized for faster attainment of programmed feed rates and fewer machine decelerations.

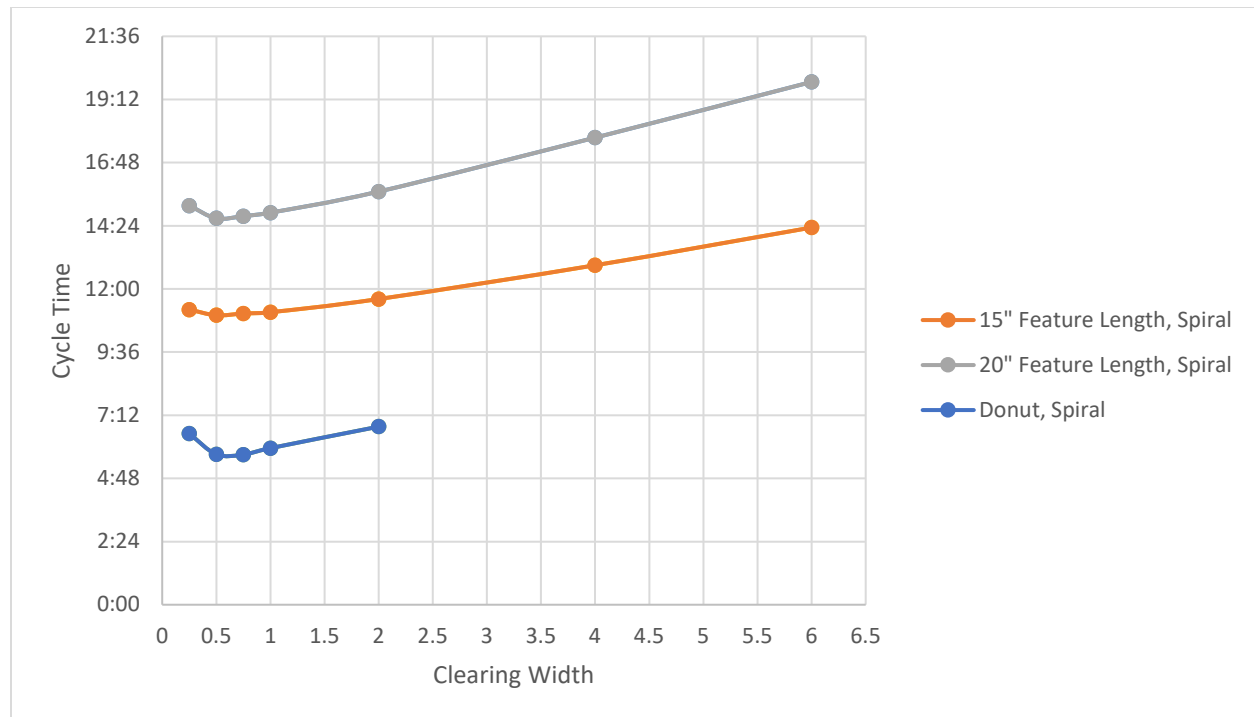


Figure 24. Tests showed that the fastest cycle time is achieved with a clearing width of 0.5 – 1.0 x tool radius.

## Conclusion

ESPRIT® remains a leader in the CAM industry by designing software that delivers long-term gains in productivity through consistently smarter, simpler, and faster toolpath. Smarter toolpath is built upon decades of machining knowledge, engineering creativity, industry partnerships, and field testing. Simpler interaction is achieved by transferring routine or time-consuming tasks from the user to the computer. Faster programming is accomplished through better mathematical processes that take full advantage of modern computing power to solve machining problems. ESPRIT® has everything you need to make complex machining simple.

## About ESPRIT®

ESPRIT® is a global leader in high-performance computer aided manufacturing (CAM) Software. ESPRIT® is a cloud-enabled CAD/CAM software compatible with a full range of machine tool applications in a wide array of industries. ESPRIT® delivers powerful full-spectrum programming for 2-5 axis milling, 2-22 axis turning, wire EDM, multitasking, mill-turn, Swiss-turn and B-axis machine tools, and high-speed 3- and 5-axis machining. Offering factory certified post processors, accurate G-code, and world-class customer support ESPRIT® is the ultimate machine tool solution.

## About DP Technology Corp.

DP Technology Corp., maker of ESPRIT® CAD/CAM software, maintains its worldwide headquarters in Camarillo, California, and product development teams in California, Florence, Italy, and Berlin, Germany. Sales and support operations are located in Europe, Asia, and North and South America.

For additional information about DP Technology and ESPRIT®, call +1 805 388 6000, send an email to [ESPRIT@dptechnology.com](mailto:ESPRIT@dptechnology.com), or visit the company website at [www.ESPRITcam.com](http://www.ESPRITcam.com).

DP Technology Corp.  
1150 Avenida Acaso  
Camarillo, CA  
93012 USA  
Tel: + 1 800 627 8479  
Outside the US: + 1 805 388 6000

[esprit@dptechnology.com](mailto:esprit@dptechnology.com)  
[espritcam.com](http://espritcam.com)

