

## The most powerful CAM software ever.



ESPRIT SolidMillTurn is a comprehensive CAM system specifically designed for programming parts for multi-function lathes and mill-turn machines. SolidMillTurn features an extensive suite of machining cycles for 2-5 axis milling, including full simultaneous 3-axis and 5-axis 3D multi-surface/solid machining. SolidMillTurn provides the freedom to use milling and turning cutting cycles in any combination and provides control over the Z, X, "C", "Y", and "B" axes common to these machine tools.

w w w . M T T S . n e t

### General Specifications

Each ESPRIT system is built upon a set of core functionality, as summarized in the Product Overview brochure. Beyond this you will add one or more components depending on your particular machining needs. Choose from among several milling, turning and Wire EDM components. No matter how you put it together, ESPRIT is "The Right Choice!".

ESPRIT for Milling, **SolidMill**, Turning, **SolidTurn**, Wire EDM, **SolidWire**, and the ESPRIT **KnowledgeBase™** are covered in detail in separate brochures.

### SolidMillTurn Traditional\*

SolidMillTurn Traditional is a CAM system for 2½-axis milling utilizing a "C" axis on a mill-turn machine tool.

- CAM for "C" 2½-axis milling on a mill-turn machine tool
- Feature Based Machining from any solid, surface, 3D wire frame model, or 2D drawing
- Adaptive machining processes
- Wrap Pocketing – any regular or irregular profile
- Wrap Contouring - profile cutting for roughing, slotting, semi-finishing, and finishing
- Wrap Drilling – hole making with up to 13 other point to point machining cycles
- Library of standard tools - flat, ball nose, tapered, corner round, chamfered, and dovetail end mills. Face mill, thread mill, drill, center drill, reamer, tap, boring bar
- Unlimited custom tools and holders with any geometry
- Library for materials, speeds and feeds
- The ESPRIT KnowledgeBase™ for milling:
  - Project Manager Feature Tab, organizing your work piece
  - Project Manager Tool Tab, managing your cutting tools
  - Project Manager Operation Tab, managing your machining cycles
  - Process Manager, machine one or more part features with one or more machining cycle(s) in one simple step.
  - Speed & feed calculator and material library
  - Property Browser providing extensive control over all cutting conditions
- Universal post processing with an extensive library and custom post generator
- Solid simulation and verification with comprehensive collision detection

### SolidMillTurn Advanced\*

SolidMillTurn Advanced includes added support for off-center "Y"-axis milling. The Advanced package also includes a large number of advanced machining cycles.

- CAM for "C" & "Y" 2½-axis milling on a mill-turn machine tool
- Includes SolidMillTurn Traditional plus:
- Support for "Y"-axis off center milling utilizing the following machining cycles:
  - Facing – any regular or irregular profile with or without islands and pockets using climb, conventional, and alternating zigzag machining patterns
  - Pocketing – any regular or irregular profile with an unlimited number of sub-pockets and islands. Use concentric-in/out and uni/bi-directional machining patterns to perform climb or conventional cutting with roughing, wall finishing, and floor finishing options
  - Contouring – profile cutting for roughing, slotting, semi-finishing, and finishing straight or tapered walls with constant or variable cutting depths
  - Rest Machining – automatic Pocketing and Contouring of the uncut areas of remaining material
  - Hole Making - drilling, peck drilling, boring, tapping and up to 13 other point to point machining cycles
  - Spiraling – machining circular pockets and bores, concentric-in/out cutting as a true spiral or tangent arcs
  - Threading – thread milling cycle for internal/external right/left hand threads
  - Manual Milling - operates from user-defined locations and manually selected geometry inside the work area.
  - Custom Cycle - machine specific instructions, comments, and in-line G-Code
  - Park – pause the cutting process and retract the cutting tool for part/tool inspection
  - Insert – change machining parameters at any point during a machining cycle

### SolidMillTurn Production\*

SolidMillTurn Production includes the additional support for 4th, "C", and 5th, "B" -axis, giving you the ability to program milling on any style "C", "Y", and "B"-axis mill-turn machine tool.

- CAM for "C", "Y", and "B" 5-axis milling (indexing and rotary cutting) on a mill-turn machine tool
- Includes SolidMillTurn Traditional and Advanced plus:
- Extends the capabilities of SolidMillTurn Advanced machining cycles to support "B"-axis
- 4th and 5th axis index positioning used in conjunction with any SolidMillTurn Advanced machining cycle
- Wire Frame Milling – 3D machining of simpler surfaces/solids using surface/face edges as drive and base curves

### SolidMillTurn FreeForm\*

3D free-form multi-surface/solid machining for your mill-turn machine tool. SolidMill FreeForm adds the power to program full 3-axis machining of any complex 3D free-form shape.

- CAM for simultaneous 3-axis multi-surface/solid 3D milling on a mill-turn machine tool
- Includes SolidMillTurn Traditional, Advanced, and Production plus:
- Solids - NURBS based 3-axis free-form machining
- Surfaces - NURBS based 3-axis free-form machining
- STL\*\* - NURBS based 3-axis free-form machining
- 4th and 5th axis index positioning used in conjunction with any 3-axis SolidMillTurn FreeForm machining cycle
- Roughing - zigzag or offset from any shape stock using boundaries and check surfaces
- Finishing - Planar, Translation, Normal, Rotation, Offset, Parametric, Parametric Spiral machining across any/all sections of the work piece
- Z-Level Machining – uni/bi-directional climb/conventional machining optimized for near horizontal and near vertical zones
- Re-Machining – automatic re-machining of un-cut areas with pencil tracing, lace cutting, flowline, flowline spiral patterns
- Remaining Stock Machining – cutting cycles are optimized for the actual remaining material based on the original stock and previous machining operations
- Project-Contour – Profile cutting projected onto any set of surfaces/solids
- High Speed Machining (HSM) support and G-code optimization
- High-speed solid simulation and verification including "as designed" versus "as machined" comparisons
- Universal post processing for NURBS and 3D NC cutter compensation

### SolidMillTurn FreeForm 5-axis\*

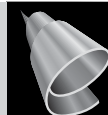
Full 5-axis free-form multi-surface/solid machining for your mill-turn machine tool. SolidMillTurn FreeForm 5-axis extends the SolidMill FreeForm Finishing and Project-Contour machining cycles to support full simultaneous 5-axis machining.

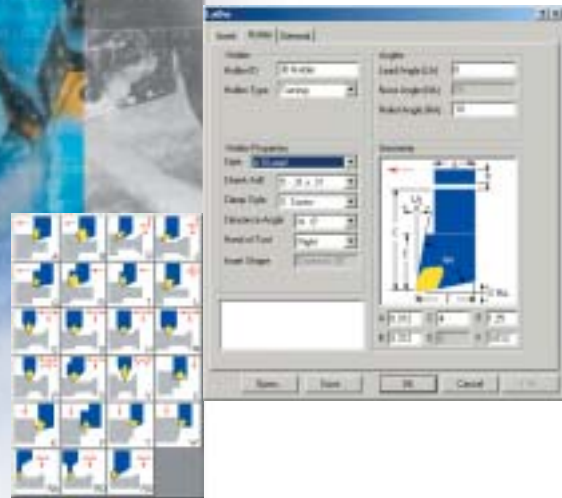
- CAM for simultaneous 5-axis multi-surface/solid milling on a mill-turn machine tool
- Includes SolidMillTurn Traditional, Advanced, Production, and FreeForm plus:
- Solids - NURBS based 5-axis free-form machining
- Surfaces - NURBS based 5-axis free-form machining
- STL\*\* - NURBS based 5-axis free-form machining
- FreeForm Finishing and Project-Contour 5-axis cycles

\*Note: SolidMillTurn is designed to program mill-turn machine tools and requires either SolidTurn Traditional or SolidTurn Production. \*\*Optional component

Machine Tool Technology & Sales  
2000 E. Randol Mill Rd. Ste 612  
Arlington, TX 76011  
USA

Tel: 817 265 6983  
Fax: 817 861 9749  
sales@MTTS.net  
www.MTTS.net





### Roughing and Pocketing

ESPRIT's comprehensive pocket machining cycle handles straight wall pockets, tapered pockets, and through or blind pockets. Whether they are simple squares, or pockets with complex irregular boundaries and a large number of multi-level islands and bosses or sub-pockets, ESPRIT will drastically simplify your programming. The ESPRIT Pocketing cycle provides full tool control and extensive automation for roughing and finishing pockets in one simple step.

The Pocketing cycle supports concentric outside-in and inside-out cutting with a constant overlap, bi-directional zigzag, and one way zigzag movements. Finishing options provide cutting tool and condition choices for finishing the walls and/or the floors with the same or different tools, with individual stock allowances and step over control for each cut.

### Contouring

You'll find ESPRIT's contour machining cycle an important tool with universal applications for a wide variety of machining tasks. This flexible cycle is ideal for profile cutting, roughing, slotting, semi-finishing, and finishing straight or tapered walls with constant or variable cutting depths. You simply set the machining parameters and ESPRIT reads the various feature properties to create tool path to cut any shape. When detailed tool control is needed, you can rely on the Contouring cycle to manage the individual tool movements, while allowing you to control the machining parameters at every point during a cut.

### Rest Machining

This intelligent cutting cycle recognizes the machining operations and cutting tools associated with a given feature of the part, and automatically compares the work piece "as machined" to the original part "as designed". Using this knowledge the Rest Machining cycle automatically performs machining (Pocketing and Contouring) of the uncut areas of remaining material. Use this cycle to easily perform roughing and pre-finishing work with optimal cutting tool choices.

### KnowledgeBase™

The array of knowledge base tools in ESPRIT enables you to automatically recognize and analyze part features (Feature Recognition), and easily create, store, and re-use machining processes (Process Manager). In addition, you can easily synchronize and optimize machining cycles through a combined display of cutting operations and cycle times (Sync List). ESPRIT then dry-runs simulations of the resulting machining processes in dynamic 3D images that show the whole part on screen, including each spindle and turret, and the cutting tools. ESPRIT also includes a comprehensive feeds, speeds and material database and an ANSI/ISO coded tooling library, providing users with optimum feedrates and tools for machining parts.

### Drilling and Hole Making

Choose between ESPRIT's machining cycles, your machine tool's canned cycles, or use a combination of both. You have the freedom of choice. Drilling, peck drilling, boring, tapping and up to thirteen other point-to-point machining cycles are available. Use one simple step in the Process Manager to cut complex holes that require multiple cycles and several tools. As an example: spot drill with chamfer, through drill, counter bore, and ream a set of holes, all in one easy step. Machine internal or external right or left hand threads using ESPRIT's Thread Milling machining cycle. The Spiraling cycle is well suited to milling and counterboring holes. In each case ESPRIT will automatically generate optimized tool path minimizing the cycle time.

### Custom Machining Cycles

ESPRIT's collection of Custom machining cycles lets you control individual cutting tool movements. The Insert technology function is available to adjust machining parameters within a machining cycle at any point during the cut. Or use the Custom cycle to insert commands, in-line G-code, and special cutting tool movements at any point in the CNC program. Take advantage of the Park instruction to pause the cutting process and retract the cutting tool for part/tool inspection. Use the Tool Path Editor to edit the tool path as if it were geometry to make special modifications to the tool path.

### FreeForm 3D Surface and Solid Machining

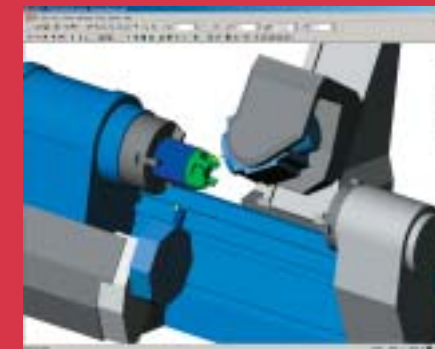
SolidMill FreeForm uses the ESPRIT solid machining engine to cut complex 3D free-form shapes. Specifically designed to handle the most challenging requirements of the mold, die, and tool making markets, ESPRIT provides you with a powerful set of tools and a large selection of machining cycles for roughing, semi-finishing, finishing, and re-machining.

### FreeForm Roughing

The FreeForm 3D Roughing cycle rough machines any complex 3D shape from any shape stock. Similar to the SolidMill Traditional Pocketing cycle, this 3D machining cycle performs zigzag and offset style rough machining on even the most complex set of 3D solids/surfaces.

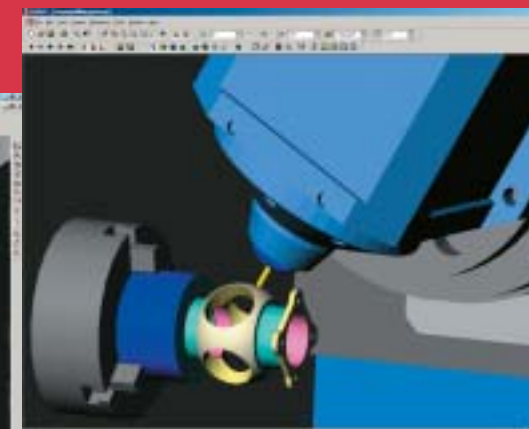
### FreeForm Finishing

With the choice of seven different semi-finishing and finishing cycles you can easily create all the tool path necessary to machine your part. Use one cycle for your entire part, or optimize your machining by defining containment boundaries. Machine different zones of your part using the cycle and cutting style most appropriate.



### Rest Material Re-Machining

Re-machining of uncut areas of your part is automated through the Rest Material cycle. Select a cutter and ESPRIT will automatically remove all possible remaining material using any one of a number of cutting styles. Choose from pencil tracing, lace cutting, parallel flowline, and spiral movements. This maximizes part quality and minimizes bench work.



### Remaining Stock Machining

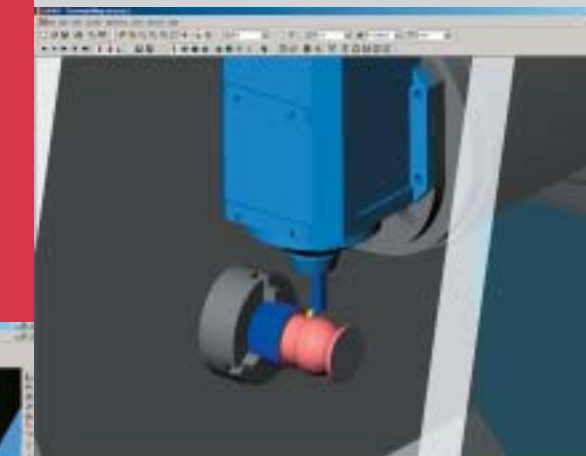
All of the FreeForm machining cycles are optimized for machining from regular and irregular shaped stock/workpiece. Surface, Solid and STL models may be used as stock, as well as the results from previous machining operations. All cutting is optimized and contained, or limited to the volume of material between the stock and the final part, resulting in significantly reduced machining times and extended tool life.

### Z-Level Machining

This cycle identifies near vertical and near horizontal areas and allows the operator to choose different cutting strategies for each area. Z-Level Machining offers both roughing and finishing options using one or two tools in a single cycle. It is an excellent cycle for High Speed Machining, providing very smooth, continuous tool path.

### High Speed Machining

Every ESPRIT machining cycle includes a number of High Speed Machining options to insure the creation of smooth, continuous movements while maintaining constant loads on the cutting tools. Whether it's HSM to cut hardened materials, minimize cycle times, or both, ESPRIT supplies the tools to get the results you need. Choose from sharp corner smoothing, loop style bridge movements, ramp and helical entry into material, spiral style cutting with constant overlaps, tangency entry, constant Z-Level cutting, and many more. Included within the Universal Post Processor is HSM optimization for NURBS and Spline curve G-code output.



### Simulation and Verification

The Solid Simulation and Verification tool allows you to quickly and easily dry run your programs on the computer to verify accuracy, and assures your CNC programs are of the highest quality, all before ever cutting a chip. The entire machining environment including spindle(s), turret(s), B-axis heads, and all your tooling is clearly represented in dynamic 3D solid images. Meanwhile, all the machine's movements, including sliding head stock, are accurately represented to promote complete confidence in the quality of your G-code program.

