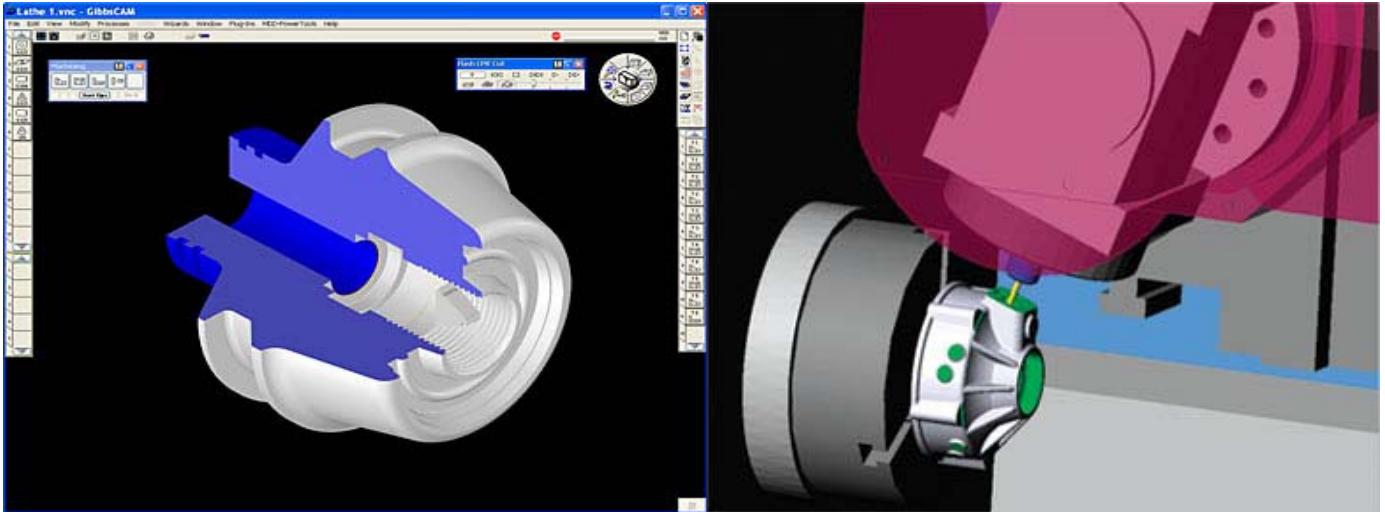


CAD-CAM 3D PROGRAMMING SOFTWARE FOR LATHE, MILLING AND EDM CNC MACHINES



As manufacturing technology continues to move forward exponentially, the dependence on CAD/CAM software becomes ever more critical. CAD-CAM software is increasingly so challenged to better support more complex processes and NC program creation for more complex machines. Worldwide competitive pressures appear to be driving this trend toward higher automation and consolidation of previously separate setups and machining operations on a single machine tool. Some of the new machines developed to support this requirement, are complex to set up, program, and simulate.

CAD-CAM software today may face with all the main aspects involved in manufacturing processes, the software potentialities and evolutions are in great growing more and more and for that reason this technology is now in a full expansion phase all over the Industry. As a consequence the training on these technologies is essential to guarantee the Industry highly skilled technicians.

From the above premises it originates the **AUTOCAM-2000**: our proposal which integrates a series of software packages with high level of features to satisfy the most sophisticated Industrial needs but fitting well also to the educational purposes at the same time.

AUTOCAM-2000 is an advanced software package for unparalleled automation in computer-aided manufacturing. Autocam is a feature-based, or knowledge-based, manufacturing system. It knows about common features of machined parts and uses that knowledge to automate the manufacturing details.

Nowadays **AUTOCAM-2000** represents surely the most complete solution available on the market since it includes 5 different CAD-CAM packages allow satisfying any needs in the mechanical area:

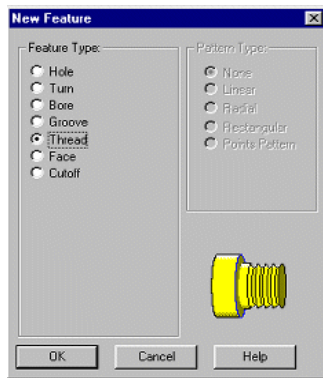
- TURN package
- TURN with motorized tools package
- MILL 2,5D package
- MILL 3D package
- EDM wire package

In the following pages they are the main features of each package

TURN PACKAGE FEATURES

Feature Based

Just describe your part to TURN in the same shop terms you are already familiar with, and it will automatically generate the proper tool paths, feeds and speeds, and accurate NC code. The feature graphic will dynamically update as you scroll through the different options to help you visualizing the different features available.



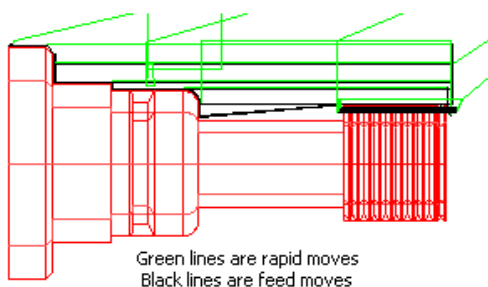
Modelling

Your parts are easily drawn with the integrated CAD drawing tools. Your parts can also be imported from IGES, or AutoCAD DXF and DWG files. The drawing tools are fully integrated with MILL and MILL3D, once you know how to draw in one program, you can draw in any of them.

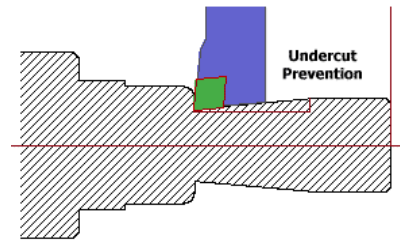
Tool Path Generation

TURN offers 3 choices of simulation to help you visualizing your part and see an accurate tool path before you begin cutting.

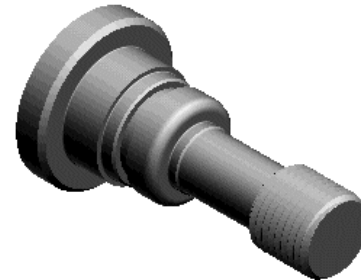
Centerline Simulation



2D Simulation



3D Simulation



Canned Cycles

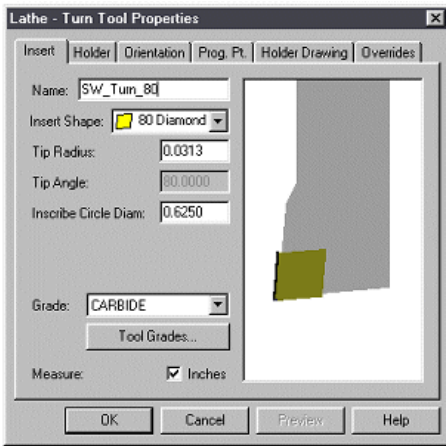
TURN allows you to take advantage of canned cycles to minimize the size of NC programs.

Canned cycles support includes:

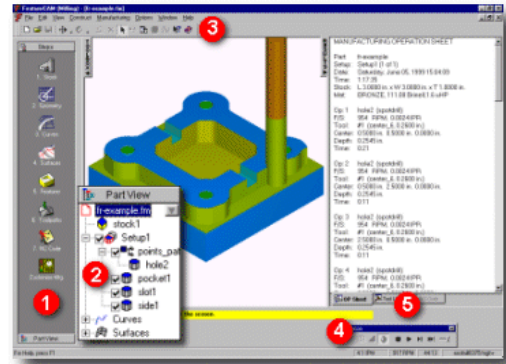
- Rough Profiling
- Finish Profiling
- Threading
- Drilling
- Grooving
- Tapping

Tooling Database

Tooling becomes very important in turning, and TURN simplifies this task. Tool selection is automatic based on a user customizable tool crib, while leaving you the flexibility to easily override any chosen tool. TURN comes standard with hundreds of standard lathe tools to choose from. You can also easily create custom tool cribs and tooling to represent your collection of tooling. The feed/speed tables can also be customized or extended to reflect your preferences.

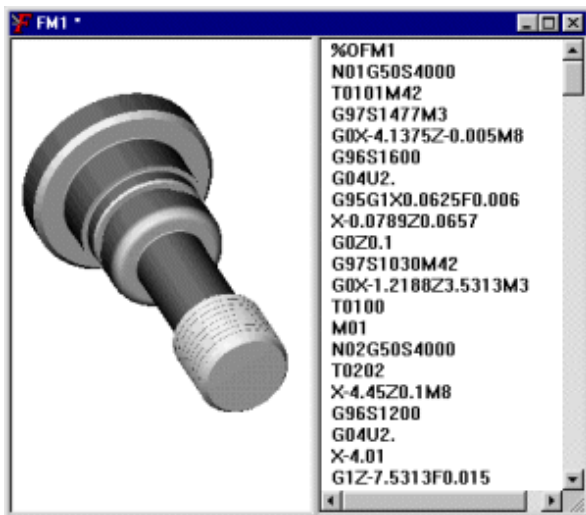


1. Steps Toolbox that contains wizards for the entire part creation process.
2. Part view that provides a hierarchical view of the part.
3. Only a single toolbar is required.
4. Other toolbars are displayed as required.
5. Tab interface to viewing documentation.



Documentation

TURN will automatically generate a process plan for your part drawing including all the documentation you will need to produce the part (NC codes list, manufacturing operation sheet and tooling list)



Post Processing

TURN comes with many post processors which allow you to run your NC program on virtually any machine. These post processors can be customized, and you have the ability to modify a post to suit your needs.

On-Line Help

TURN has a comprehensive and detailed on-line help system. It will assist you through many aspects of running TURN - from "How do I draw a line?" to "How do I post the code to my machine?" It even includes a "What's this?" point and click help for getting answers on-the-fly while using TURN.

MILL 2,5D PACKAGE FEATURES

Graphical User Interface

The CAM interface has many productive features including:

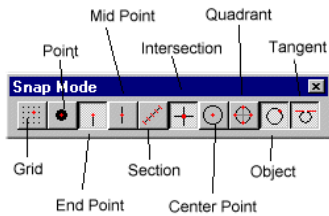
Steps Toolbox

<p>1. Stock</p>	<p>Step 1 - Stock. The New Stock Wizard steps you through entering the stock dimensions and material and part program zero.</p>
<p>2. Geometry</p>	<p>Step 2 - Geometry. A single dialog box provides you with all the tools for points, lines, arcs, fillets, dimensions and editing.</p>
<p>3. Curves</p>	<p>Step 3 - Curves & Surfaces. The Curve and Surface Wizards provide step-by-step instructions for creating 2D and 3D curves and 3D NURBS surfaces.</p>
<p>4. Features</p>	<p>Step 4 - Features. A New Feature Wizard has been added to Version 6 to walk you through the creation of manufacturing features.</p>
<p>5. Toolpaths</p>	<p>Step 5 - Toolpaths. This step displays the new VCR controls for centerline, 2D and 3D solid toolpath simulation.</p>
<p>6. NC Code</p>	<p>Step 6 - NC Code. NC code commands are now organized into a single dialog box for convenience.</p>
<p>Customize Mfg.</p>	<p>Customize Manufacturing. Edit manufacturing defaults, feed/speed tables or tooling databases with this step.</p>

Drawing Tools

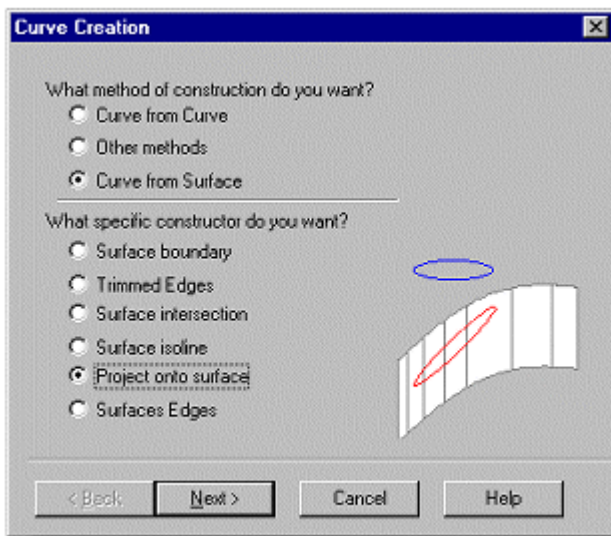


The geometry construction tools make drawing in MILL a snap. MILL also includes many useful snap modes to make drawing quick and accurate



Curve Wizard

MILL provides a unique step-by-step interface to 3D curve creation. Just select the Curve Wizard from the toolbar and walk through the graphical instructions. From the Curve Wizard you can access a wide variety of curve creation routines such as creating curves from a Windows font, intersecting 3D surfaces, or offsetting a curve. Click "Next" below to see a complete list of curve creation methods.



Curve Constructors

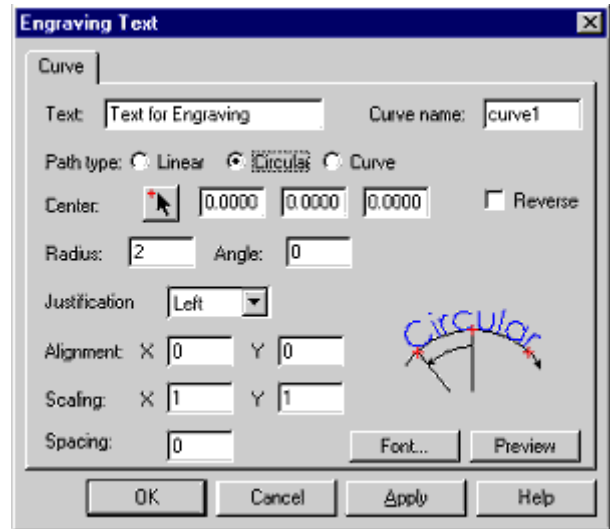
The following list shows the options for creating 2D curves:

- Join
- Curve start/reverse
- Offset
- Project to UCS
- Extract Font Curve
- Smooth/Reduce Curve
- Functions
- Cams
- Splines
- Text

AUTOCAM-200 offers several more options for creating smooth 3D curves.

Engraving on an Arc or Curve

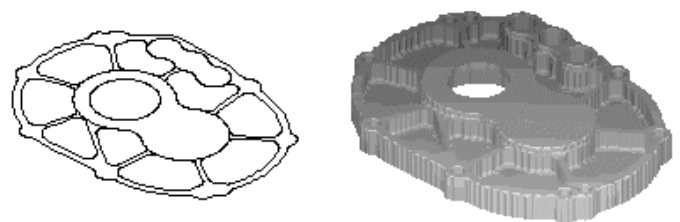
Create text along a line at any angle, along an arc or along a curve. Easily control scaling and spacing. Use any Windows font or the two single-line fonts that are supplied with MILL. Use the resulting curves in a groove feature for engraving or in a boss feature to create raised letters.



Importing

MILL allows for easy and robust importation of IGES or Autocad files. This allows almost any part drawing to be brought into MILL to take advantage of its powerful automation.

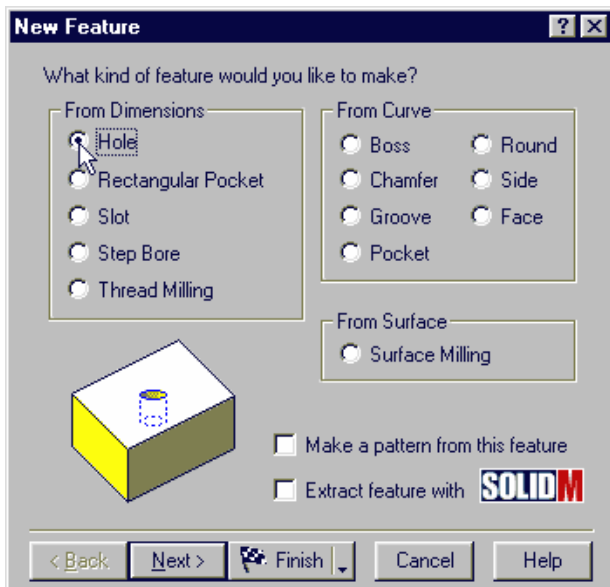
In this example, these curves were imported into MILL. From here, just tell MILL which curves will be pockets, bosses, holes, etc. Once the features are defined for depth, chamfer, and bottom radius, you are ready to manufacture your part.



This part uses only 20 features to control 85 manufacturing operations. Think of the time and money you will save if all you have to do is specify which curves will become pockets, bosses, sides, etc. and let MILL to calculate all the roughing and finishing tool paths, feeds, and speeds. MILL will even select the proper tool for the job from the tools you have available in your workshop!

Feature-Based

Just describe your part to MILL in the same shop terms you are already familiar with, and it will automatically generate the proper tool paths, feeds and speeds, and accurate NC code. The feature graphic dynamically updates as you scroll through the different options to help you visualize the different features available.



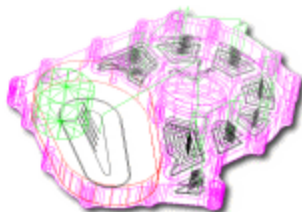
Process Planning

After you create the part features, MILL automatically handles the details of process planning and tool path generation. MILL automatically:

- Translates features into manufacturing operations
- Selects tools for each operation
- Calculates feeds and speeds
- Generates toolpaths.

Toolpath Simulation

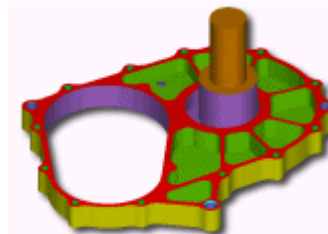
MILL provides 3 different modes of integrated tool path simulation.



Centerline simulation displays the movement of the center of the tool with a 3D line drawing.



2D simulation shows the area covered by the tool paths in a 2D color mode from the top view.



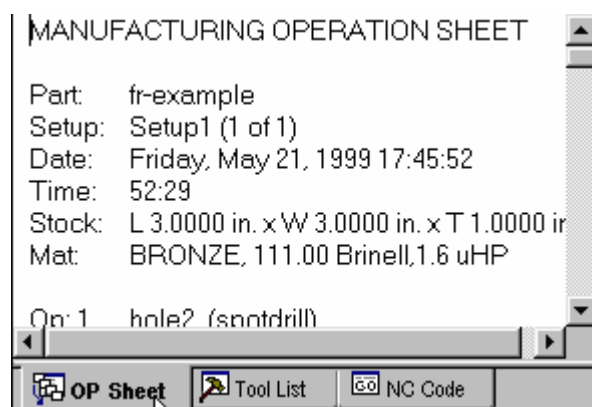
3D simulation shows a three dimensional solid simulation of the material being removed. Gouges with the tool holder or the tool during a rapid move can be visually detected. The movement of each tool can be optionally rendered in a separate color.

Documentation

MILL will automatically generate a process plan for your part drawing including all the documentation your shop floor will need to produce the part. With a simple click of the mouse you can generate:

- Operation sheets
- Tooling lists
- NC Code

You can also use dimensioning to create part drawings.



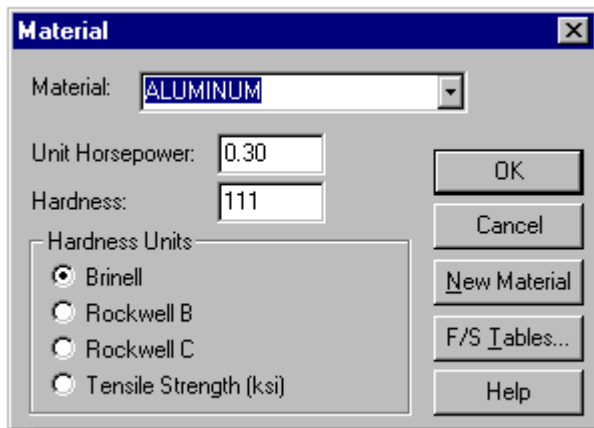
Post Processing

MILL comes with over 100 proven post processors which allow you to run your NC program on virtually any machine. These post processors can be customized, and **you** have the ability to modify a post to suit your needs. If you cannot find a post processor

for your particular machine tool, our in house manufacturing specialists can create one for you.

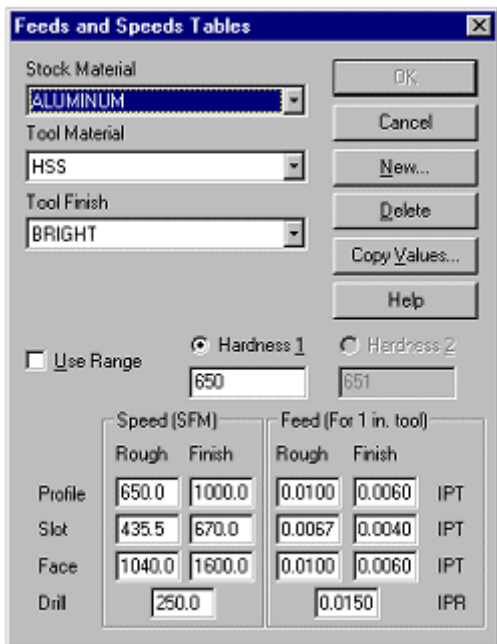
Materials Database

MILL allows you to specify your stock material. Your material type will be used in the calculations for tool feeds and speeds. The standard database includes 33 steel types, 40 stainless types, cast iron, brass, bronze, copper, tool steels, and many others. You also have the flexibility to create your own materials to include with MILL automation.



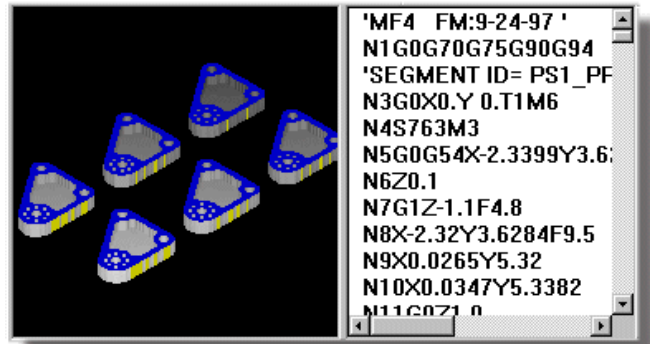
Feeds and Speeds Database

The extensive built-in databases are what provide the automation behind MILL. The feeds and speeds tables are fully customizable to give you the freedom to machine your parts the way you want to.



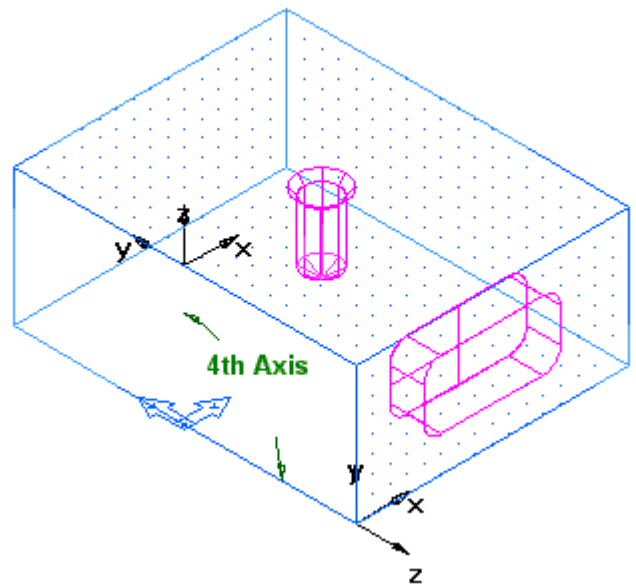
Multiple Parts

MILL will allow you to easily convert any single part program to a multiple part program with as few as 10 additional keystrokes. Multiple part programs can be formatted to include such advanced NC programming techniques as macros and fixture offset changes to minimize program length. The process is as simple as telling MILL how many parts you want and which face you are working on.



Indexing

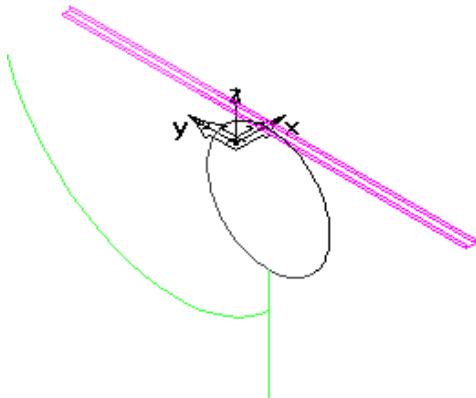
Indexing uses the fourth axis to rotate the part between machining operations so the machining takes place on different faces of the stock. You simply check a box to tell MILL that you want an indexed part around the X or Y axis. The NC code will now contain the B-axis rotation to take advantage of a 4th axis rotary table that will allow you to make this part from a single program.



Wrapping

4th axis wrapping allows you to wrap a 2 1/2 D feature around either the X or Y axis. You must have a machine tool with 4th axis capabilities to take advantage of this feature. This picture shows a 2D groove that has been wrapped around the x axis.

The magenta object is the groove feature, the green line is the tool in rapid motion and the black line shows the tool path relative to the feature. Imagine bending the groove around a cylinder, and you can visualize 4th axis wrapping.

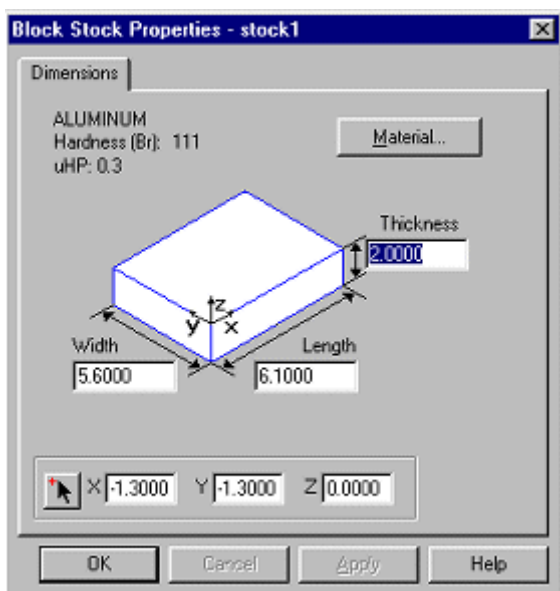


On-Line Help

MILL has a comprehensive and detailed on-line help system. It will assist you through many aspects of running MILL - from "How do I draw a circle?" to "How do I post the code to my machine?" It even includes a "What's this?" point and click help for getting answers on-the-fly while using MILL.

Define the Stock Parameters

1. Start by selecting your material type.
2. Then enter the dimensions of the block with which you are working. Your block of material is displayed, along with an X,Y,Z origin to help you to visualize the orientation.



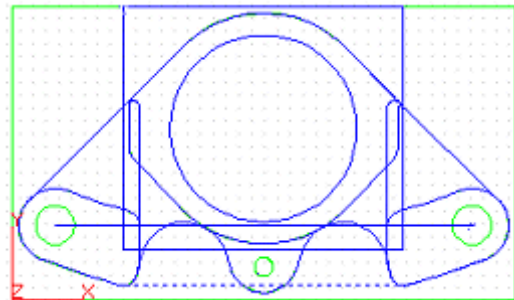
3. Now define your geometry using the flyout toolbars.

Draw Geometry

Use the flyouts from this toolbar to quickly and easily create any part geometry with a complete suite of geometry primitives including circles, arcs, lines, etc.



Curves and lines are combined to create profiles. These profiles define shapes for bosses, pockets, or grooves.



You can even import your existing IGES or AutoCAD DXF files.

Use your drawn geometry to tell MILL exactly what features you want to mill.

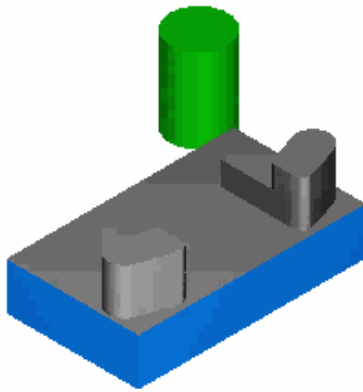
Add Features

MILL use of "features" gives quick and easy manipulation of a model's details. MILL full automation will automatically provide:

- Complete tool selection
- All feeds and speeds including ramping, stepovers & depth of cuts
- Optimized tool paths
- Complete documentation for operations, tool list, and NC code

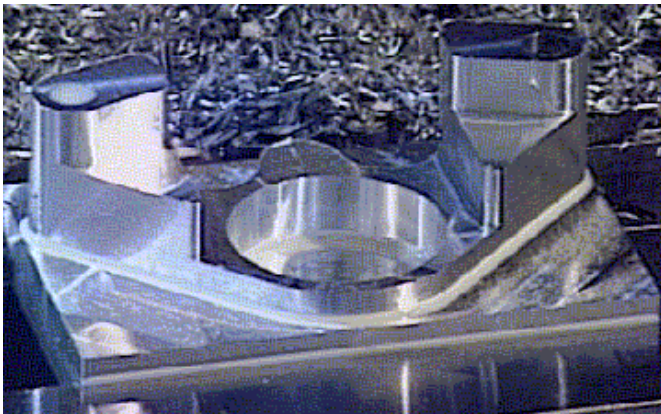
Show Toolpath

MILL will show you a tool centerline simulation, 2D simulation, or shaded 3D simulation. With the mouse, you can instantly display the NC code, the Operations List, or the Tool List. MILL automatically selects the best tool for each milling operation, yet gives you control to override any tool selected by the system. Each of your part features has been broken down into manufacturing operations including roughing passes, finishing passes, drilling cycles, and tapping cycles. The entire manufacturing process plan is instantly generated and documented.



Cut Metal

Posting your part program for a specific machine is as simple as a single mouse click. Virtually every CNC controller is supported by MILL. Select your CNC controller from the list, or customize your own post processor using the general post processor and editor provided with the system.



Click the NC button with the mouse and you have code that is ready to run.

Manufacturing documentation including a tooling list and setup sheets are also created so that the operator is fully informed on how to manufacture the part.

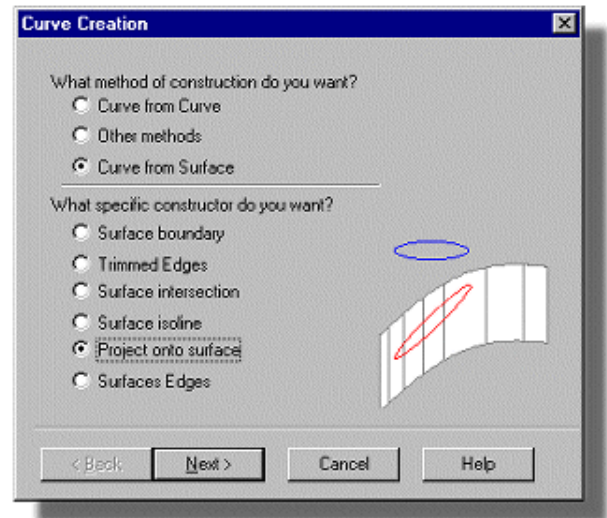
MILL 3D PACKAGE FEATURES

MILL3D is the premier 3D computer aided Manufacturing package. It offers unparalleled ease of use combined with the power expected from a world-class CAM system. It has been designed to be easy to learn and easy to use. It has a complete set of 3D modelling tools and robust 3D IGES import. Creating 3-axis tool paths for complex parts is stream-lined for making the most of your time. Click on a topic to the right to learn more about this state-of-the-art 3D CAM solution.

Curve Wizard

MILL3D provides a unique step-by-step interface to 3D curve creation. Just select the Curve Wizard from the toolbar and walk through the graphical

instructions. From the Curve Wizard you can access a wide variety of curve creation routines such as creating curves from a Windows font, intersecting 3D surfaces, or offsetting a curve.



Curve Constructors

MILL3D offers all the same methods as our 2 1/2D product plus these advanced 3D curve creation methods.

- Curve
- Join
- Curve start/reverse
- Offset
- Project to UCS
- Extract Font Curve
- Smooth Curve

Other Methods

- Functions
- Cams
- Splines
- Text
- Surface Isoline
- Project Onto Surface
- Surface Edges

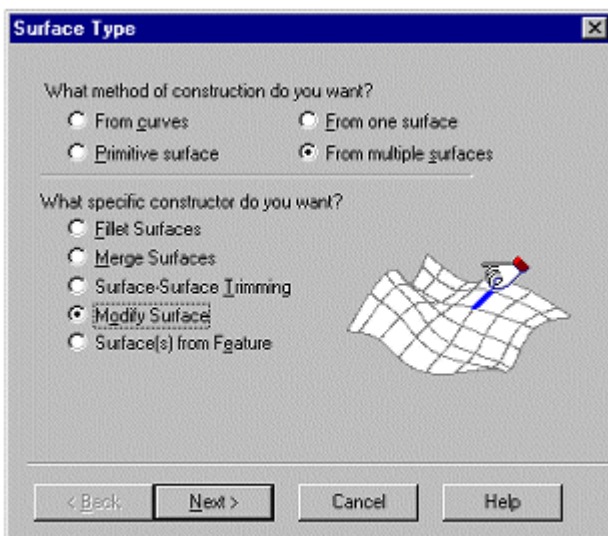
Curve From Surface

- Surface Boundary

- Trimmed Edges
- Surface Intersection
- Surface Isoline
- Project Onto Surface
- Surface Edges

Surface Wizard

The easy to use Surface Wizard walks you through the steps to creating complex 3D surfaces. MILL3D offers unparalleled ease of use and dynamic manipulation including revolve, sweep, fillet, surfaces between multiple curves, and more. You will next see the Surface Constructors which shows a complete list of the various ways MILL3D will get you on your way to machining in 3 dimensions quickly and accurately.



Surface Constructors

MILL3D offers you the following methods for creating 2D and 3D surfaces:

From Curves:

- Extrude
- Surface of Revolution
- Swept
- Ruled
- Coons
- Curve Mesh
- Lofted
- Cap

Primitive surfaces:

- Sphere
- Cylinder
- Flat

From one surfaces

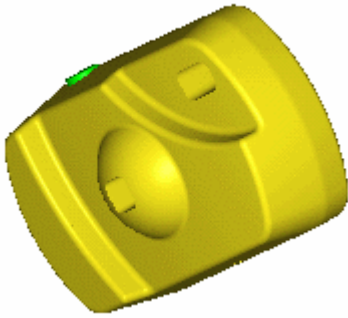
- Region From Surface
- Reverse Surface
- Offset Surface
- Extend Surface
- Trim Surface
- Untrim Surface
- Split Surface

Fillet Surfaces

- Merge Surfaces
- Surface-Surface Trimming
- Modify Surface
- Surfaces from Feature
- Corner Blend

An optional Solid Modelling module is available to add to all CAM products. The solid modelling option features the following:

- Add or subtract extruded, revolved, lofted or swept volumes during modelling.
- Work in line drawing, hidden line drawing or shaded mode.
- Create constant or variable radius fillets across multiple faces with automatic corner blending.
- Create thin-walled solids from surface models.
- And more.



taking advantage of MILL3D's automation and ease of use.

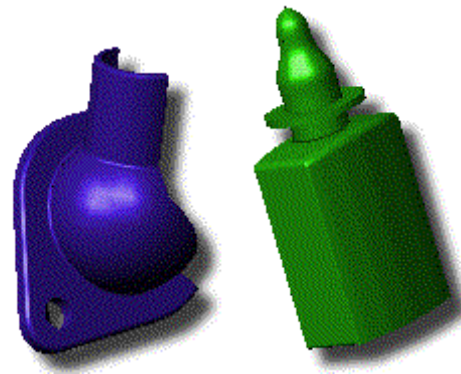
Solid Model Import

Import solid models directly from industry-leading CAD packages. Import Parasolid XMT files from systems including SolidWorks, SolidEdge and Unigraphics. Import ACIS SAT files from systems including CADKEY and AutoDesk's Mechanical Desktop.

Solids-based Manufacturing Tools

MILL3D contains innovative tools for manufacturing solid models. Creating complex 3D tool paths has never been easier. These tools include:

- A variety of parting surface creation tools.
- The ability to cut a solid with a parting surface.
- Calculation of parting curves.
- Calculate core and cavity portions of a mould.



Solid models imported directly from CAD systems

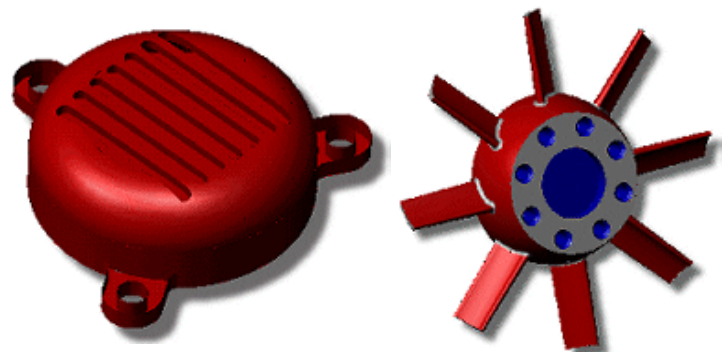


3D Graphics

MILL3D takes advantage of the latest in OpenGL photo quality shading. This helps you to visualize your part before it is manufactured.

Importing

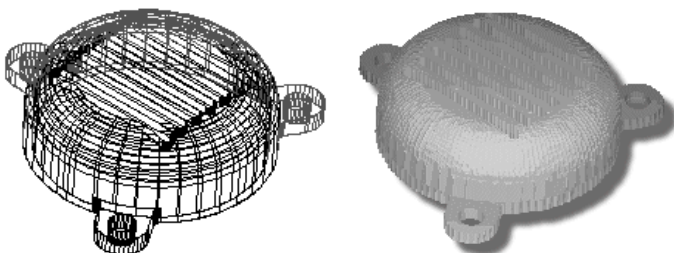
MILL3D allows easy and robust importation of IGES or Autocad files. Any part drawing can be brought into MILL3D to take advantage of its powerful automation. MILL3D allows the user to combine both 2^{1/2}D features with the full palette of 3D tools. When you buy MILL3D, you get the complete 2 1/2D product seamlessly integrated.



This is an example of a part drawing that was imported as an IGES file. The geometry is used to describe the features, and from here, MILL3D does the rest. This part uses 2 side features, 3 holes, 1 surface milling feature, and 7 pockets.

Process Planning

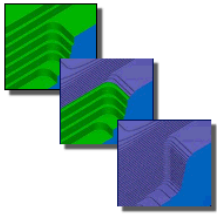
MILL3D looks at the features that have been described and breaks them down into the required operations including roughing passes, finishing passes, drilling cycles, etc. All the needed tools, feeds, speeds, and tool paths are generated automatically. This is what makes MILL3D the fastest and most advanced CAM system on the market today!



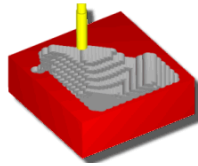
Here is an example to see just how easy it is to get your operations list ready to go with all the correct information to manufacture your part.

A part that would take hours to describe with other CAM software packages is reduced to minutes by

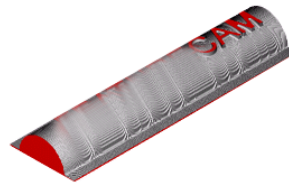
Summary of 3D Milling Techniques



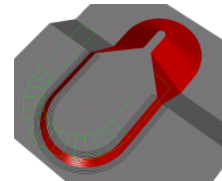
NEW Z-level Semi Finish



NEW Plunge Rough

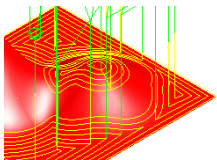


NEW Multiple Drive Curve Finish

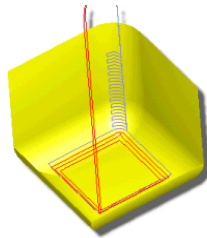


Flowline Finish

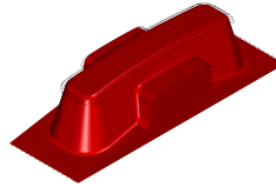
Summary of 3D Milling Techniques



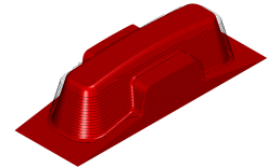
NEW Spiral Boundary



NEW Horiz+Vert Remachining

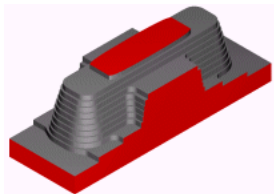


Horizontal-only Finishing

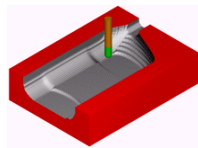


Vertical-only Finishing

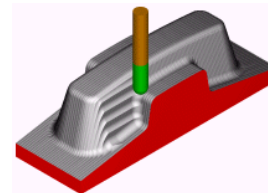
Summary of 3D Milling Techniques



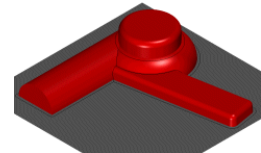
Z-level Rough



Parallel Rough



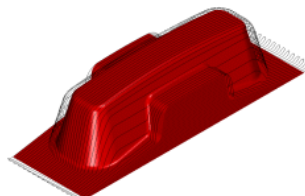
Steep Slope Remachining



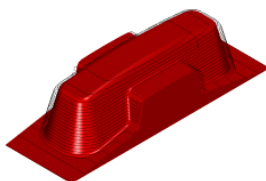
Pencil Milling

Post Processing

MILL3D comes with over 100 proven post processors which allow you to run your NC program on virtually any machine. These post processors can be customized to suit your needs, and you have the ability to modify a post to suit your needs. If you cannot find a post processor for your particular machine tool, our in house manufacturing specialists can create one for you.



Parallel Semi-Rough



Z-level Finish

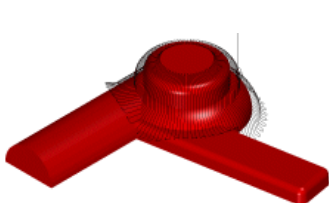
EDM (WIRE) PACKAGE FEATURES

WIRE

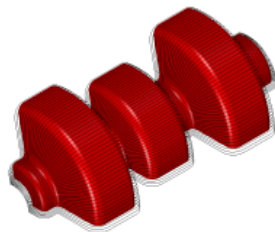
Once the part dimensions or geometry is created, the feature wizard walks you through the creation of features including:

- 2 or 4 axis dies
- 2 or 4 axis punches
- 2 or 4 axis sides

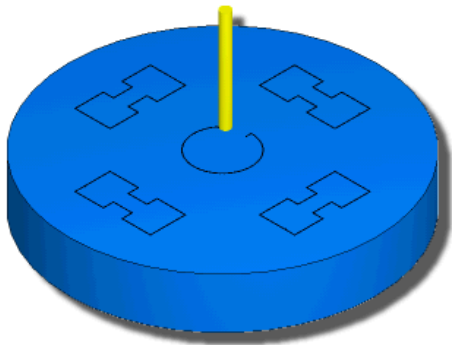
Summary of 3D Milling Techniques



Radial Finish



Isoline Finish



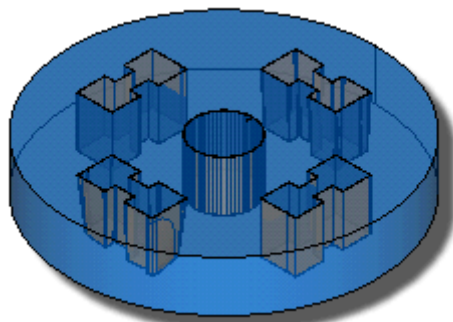
2axis Wire EDM

Once you create the die, punch or side feature, automatically create one or more of the following wire EDM operations:

- Retract operation that creates a tab by retracting the wire when it reaches the stop length
- Cutoff operation to cut off the tab.
- Glue stop operation that stops before the slug is cut loose
- Contour operation that follows the shape of the curve
- Pocketing and zigzag operations for no-core cutting

Additional details include

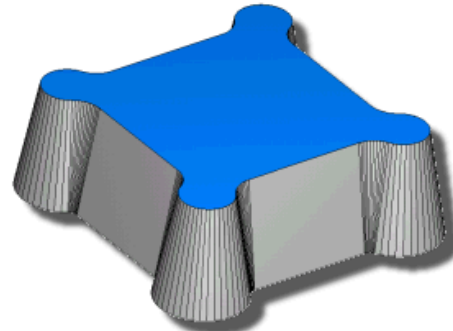
- Cutter compensated tool path or CAM offset
- Corner rounding options
- Creating one feature from multiple curves
- Control of the cut direction, stop length and overlap distance



2axis Taper

Create 2axis tapered part quickly and easily:

- Constant taper using conical corners or cylindrical (ISO) corners
- Create variable tapers by conveniently specifying a separate taper angle for each segment of the feature



4axis Wire EDM

WIRE support of 4axis cutting includes:

- Full XYUV 4-axis contouring
- Automatic synchronization of upper and lower curves
- Graphical picking of upper and lower curve synchronization points for manual specification

