Manufacturing Set

NC planning, tooling, and programming in a single, unified system

I-DEAS® Generative Machining™

I-DEAS Generative Machining is the integrated software module for the creation of NC toolpaths on I-DEAS models. This module works directly on solid or surface geometry created within, or imported into, I-DEAS. Working directly on I-DEAS geometry eliminates the errors and work involved in translating geometry to a non-integrated system for NC programming. The integrated nature also means that the NC Programmer can take full advantage of the associativity that is part of all I-DEAS modules. Design changes that occur after the NC toolpath is created will be flagged, and at the discretion of the programmer, are updated to conform to the new geometry. I-DEAS Generative Machining can also use stored rules, or methods, which automate the NC programming task, and ensure the consistent use of best methods. It is possible, using I-DEAS Generative Machining, to model the entire machining environment, including the part, stock, clamps, fixtures, and the machine itself, to optimize tool motion, avoid collisions, and avoid part gouging.

General Capabilities

Feature-Driven Machining

I-DEAS Generative Machining leverages the solid or surface geometry created within the I-DEAS Master Modeler™ software. Guided by the manufacturing engineer, I-DEAS makes intelligent decisions concerning the machining process and strategy necessary to create NC toolpaths. I-DEAS provides an intuitive environment to help plan and verify all of the NC operations required to produce a part.

The feature-based master model provides a complete, unambiguous representation of the part geometry. Some of the benefits of this approach include:

•Robust part / fixture / tool design capabilities enabling tools, fixtures, tables, holders, and other auxiliary geometry to be created during NC programming.

•Standard tooling and fixturing libraries are available.

•Changes in the part, stock, and fixture design are associated to machining. In response to changed design geometry, the system flags the affected toolpaths as “invalid.” In response, you can reprocess individual toolpaths, or the entire job.

•Validity flags inform you of every part, fixture, stock, containment geometry, or setup change.

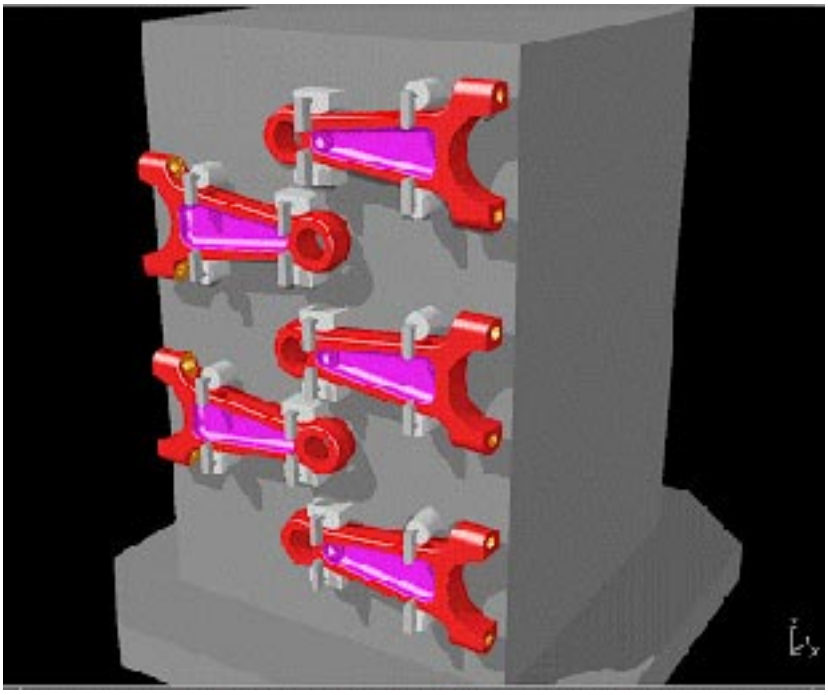
•You avoid gouging the part with the tool.

•You make intelligent use of the solid stock to determine feeds, speeds, entry, and exit locations.

•You avoid interference of the tool with the machine and the work-holding devices.

Team-Oriented Manufacturing

I-DEAS encourages communication across multiple departments or disciplines. Design engineers, structural analysts, and manufacturing engineers all have access to the same master model. Common libraries allow the manufacturing engineer to reference parts instead of copying the part geometry. This enables you to work on the most current version of the part as it evolves, and you can also suggest changes based on manufacturing requirements. I-DEAS enables manufacturing engineering to become an integral part of the design process.



Solids-based machining captures the manufacturing environment.

Job Management
I-DEAS automates the definition of the manufacturing "job." The manufacturing engineer plans the NC job even before the final design is released to production. I-DEAS makes it easy to create the process plan, setup, and operation information. The job captures and organizes manufacturing, including: stock, part, setups, tool selection, and machining operations. With job planning, I-DEAS provides change management capabilities by notifying you of every part, fixture, stock, containment geometry, or setup change. The software can automatically update the toolpath.

Generative Machining
With I-DEAS software, rules and methods files capture and organize your manufacturing knowledge. I-DEAS uses methods to generate specific operations necessary to machine a feature of a part including: feeds, speeds, tools, machining strategy, etc. If any of the feature information changes, such as tolerance, you just reprocess. The method will be reapplied automatically and may generate different operations or machining parameters.

Operation Sequencing
With I-DEAS, you can focus your attention on machining part features, instead of on tool usage and optimization. I-DEAS provides a set of sequence rules that reorder the individual operations to optimize operation order and tool changes to minimize machine time. Optimization of operation ordering is automated through the use of sequence rules. And, because in-process stock is automatically calculated and taken into account in toolpath motion, the correct toolpaths are generated even after the sequence optimization has been used. Examples of the sequence rules provided include volume clear before copy mill, centerdrill before drill, drill before ream, sort by shortest distance between holes, sort by tools, group all operations by index position, and sort by increasing tool diameter.

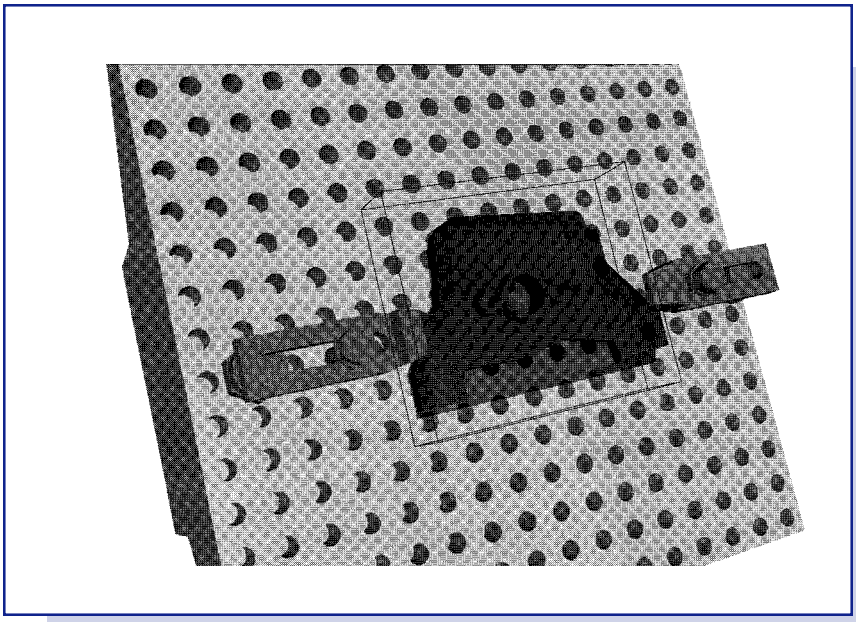
Maximum User Control and Flexibility
Besides offering fully automatic toolpath creation such as cavity roughing, I-DEAS Generative Machining also provides tools to assist you in defining the desired toolpath from being able to drive wireframe geometry on the solid to virtually giving you the ability to create and define any toolpath.

Containment Options
All milling operations can be bounded by containment sections. These limits can be applied to the the whole tool diameter, the tool tip, or the contact point. These boundaries provide very precise control over the cut area.

Transition Moves
Each operation is accompanied by a Transition Move that describes the intermediate motion between discreet operations. The Transition Move is derived from a template, and provides a language-based programming environment for modifying the intermediate motion. The default template can be edited to reflect shop practice and each Transition Move can be edited individually for special requirements.

Standard Tooling
Tool libraries let you save tombstone and fixture configurations for subsequent NC jobs. This eliminates the need to recreate the fixture and mounting arrangements for future jobs. For many NC programming tasks, fixture design and configuration can be as much as 30% of the process time from conceptual design to finished part. With tool libraries eliminating the need to recreate standard tooling and fixturing, the setup planning time is dramatically reduced in some cases.

Standard CL File Output
I-DEAS creates industry standard CL file output in ASCII or binary format so you can use post-processors created with I-DEAS C-Post software as well as other third-party post-processors.



In-process stock model aids in fixturing and job planning.

Milling Capability - 2 1/2 to 3-Axis

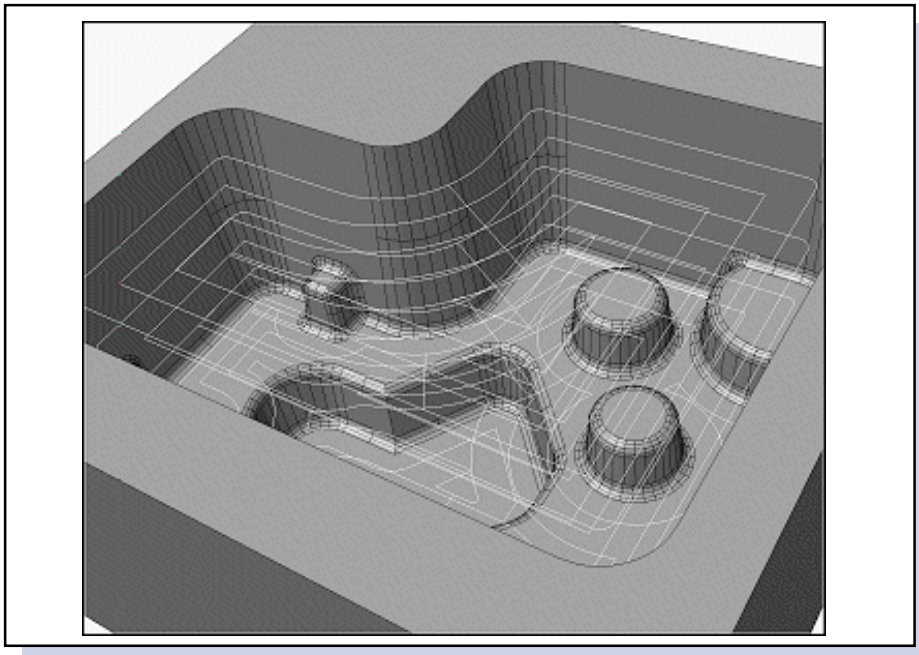
I-DEAS software offers a set of milling functions, including automatic volume clear, cycle operations, profile, drive/part, manual toolpath, and copymill machining. With I-DEAS, you can specify “stock” allowance on each part face or you can specify global avoidance. You can also specify a separate avoidance amount for clamps and fixtures. Gouge avoidance algorithms check around the entire circumference and along the total length of the tool. I-DEAS can also optionally include holder geometry in the check for any tool collisions. Containment curves can be used to isolate areas of the part that need special consideration. With I-DEAS, you have control over the various parameters necessary to achieve your desired machining strategy. Milling capabilities provided include:

Intelligent Feedrate Control
Several feedrate to cut depth options are provided, including rapid, fast, slow, entry, exit, engage, and retract. I-DEAS software monitors the position of the tool relative to the amount of stock being cut and adjusts the feedrates accordingly. For example, I-DEAS evaluates when the tool is cutting with the full diameter of the tool versus the partial diameter, and switches from the “fast” to the “slow” feedrate during the path generation. This minimizes the time to cut a part because the fastest acceptable feedrate is used at every point along the path.

Toolpath Display and Edit Capability
I-DEAS provides a complete set of functions for tool animation and toolpath editing. The animate tool form gives you control over the tool as it moves along the toolpath. At each point in the display the corresponding cutter location data is highlighted. A forms interface allows you to edit cutter location data such as post-processor commands, and X, Y, Z coordinates without worrying about syntax errors.

Features of toolpath display and edit include:
•Graphical differentiation of toolpath segment types such as entry, feed rate changes, exit, rapid, etc.
•Allows quick and easy modification or additions to cutter location data.
•Allows the replacement of cutter location data with manual centerline geometry.
•Allows quick and easy insertion of post-processor commands.
•Dynamic tool display, including:
  •Full tool display.
  •Forward or backward tool animation.
  •Options for playback to include single step or continuous.
  •Option to select tool display start location and end location within the toolpath.
  •User control over speed of display.

Volume Clear
The volume clear capability (planar roughing and finishing) automates roughing of even complex regions and also handles 2 1/2-axis milling and profiling. The algorithms can automatically rough the part, a selected set of faces, or even a subregion of the specified geometry. The algorithms also automatically determine cut depths and assure that horizontal faces have one finish allowance. Volume clear provides:
•Automatic cut depth calculation based on stock allowances, critical faces, multiple islands, tool limitations, and final part geometry.
•Automatic feedrate control (fast and slow settings based on cut monitoring).
•Automatic entry and exit control based on tool type and in-process geometry.
•Ability to specify a finish amount and separate cleanup pass.
•Multiple cutting strategies and user-specified cut direction and cut angle control, including spiral in, spiral out, zig zag, uni-directional cutting, as well as climb (up cutting), conventional (down cutting), and combination.



Volume clear capabilities allow gouge-free material removal.

Profile
Profiling enables you to profile the entire part or selected surfaces. The 2 1/2-axis milling operation can perform multiple functions. You can machine the entire part or selected portions of the part by stepping down in the z direction. Profiling is primarily a wall finishing method-providing everything from finish profiling a single wall to profiling the entire part with multiple passes. You can either input the depth at which to make the desired cut or you can have the software automatically calculate the depths. Profile gives you the ability to perform waterline machining (also known as z-level finishing). The depth of cut is controlled by a user-specified depth of cut or by specifying a constant cusp height which changes the depth of cut, based on the tool and the curvature of the part. Profiling provides:
•Wall finish machining method. Profiling allows you to drive the tool at a given depth around the part geometry, providing the means to finish vertical or near vertical surfaces.

Features are:
•Waterline or z-level finishing capability. This option allows you to machine the entire part using the automatic depth-of-cut calculations.
•Automatic depth-of-cut generation based on a percentage of cutter diameter or an absolute distance.
•Variable depth-of-cut control based on cusp height. Each depth-of-cut is evaluated and adjusted throughout the toolpath to maintain the desired cusp height.
•Multiple pass capability. This option allows you to specify multiple passes to combine both semi-finish and finish machining with the same tool.
•Climb or conventional cutting strategy.
•Automatically controls cutter compensation settings and utilization.

Face Milling
Face Milling automates rough and finish milling of multiple flat surfaces at various Z-levels. Stock can be specified incrementally on each face. Depths of cut can be uniquely specified for each face, or generated automatically for both roughing and finishing. Entries and Exits are positioned clear of the face for best cutter application.

Face Milling provides:
• Automatic filtering for horizontal planar faces.
• Automatic generation of roughing and finishing cut depths.
• Automatic entry and exit control for positioning clear of cut faces.
• Cut patterns including uni-directional, bi-directional, and box cut (stay-down uni-directional).
• Automatic cusp cleanup between passes.
• Options for traversing regions including stay-down rapid, stay-down traverse, and exit/entry.
• Options for crossing gaps including stay-down rapid, stay-down traverse, and exit/entry.
• Options for navigating obstacles including stepover and exit/entry.

Cycle Operations
I-DEAS software provides both manual and automatic capabilities to machine holes. It uses feature information such as hole diameter, depth, tolerances, and desired surface finish. Based on this information, a method selects the appropriate operations and tooling necessary to machine the feature. Cycles supported include drill, center drill, counter bore, countersink, tap, bore, ream, spot face, and plunge mill.

User Cycles
Additionally, any holemaking process can be supported with the User Cycle. The User Cycle can call custom Post Processor sections directly, and provide any required auxiliary paramaters in addition to the position and feedrate information. Any standard or non-standard holemaking process is easily supported with this function, including:
• gun drilling
• thread milling
• back spotfacing
• core drilling

You can also specify cycle-specific parameters such as thread lead, thread pitch, break chip retract distance, minimum peck depth, etc.

The software includes multiple methods for selecting and ordering holes. You can pick holes by their location on a surface, by tool axis, or by their diameter. You can then identify the order in which they're machined. This order also defines the entry and exit points for the toolpath. You can order them by the sequence selected, the shortest distance between them, or by their position along the X and Y axes.

Milling Capability - Surface Machining

I-DEAS provides a complete set of 3-axis surface roughing and finishing capabilities for even the most complex geometry. Multiple surface, gouge-free machining provides variable stepover to give you maximum control over the finish scallop height, while providing the most efficient toolpaths.

Surface machining capabilities and control features are:
•Copymill machining floor finishing method.
•Gouge checking along the entire length of the tool.
•Stepover specified as a percentage of cutter or an absolute distance.
•Variable stepover control based on cusp height, and each stepover is evaluated and adjusted throughout the toolpath to maintain the desired cusp height.
•Constant stepover generated based on desired cusp height.
•Separate cusp height specification for profile and copymill.

Copymill Cutting Patterns
Copymill in the traditional sense is planar motion back and forth across the part at an angle. With I-DEAS Generative Machining software you have your choice of the planar cutting motion across the part, or the ability to spiral in and out. The spiral cutting motion allows you to follow the surface contours more closely.

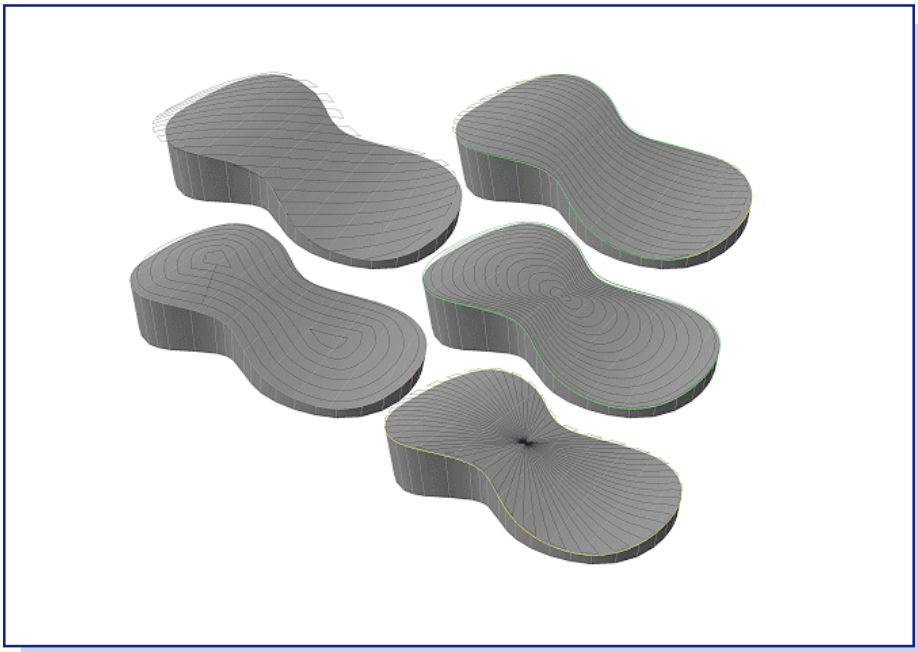
You input a section curve as the reference for the spiral cutting pattern, and the software machines the surfaces you selected. Section curves enable you to contain the tool cutting region. They also enable you to define islands or “keep out” regions in both cutting planar and spiral cutting strategies.

Flowline
Flowline creates cutting passes similar to traditional UV machining, but with more flexibility. This 3-axis milling operation reduces hand finishing and decreases machining time by following the contours of the part. With flowline, the tool drives along the creases or junctions of a part instead of lifting up or over any obstacles. This capability creates cutting paths that more closely match the shape of your part.

You can control the shape and direction of the toolpath by picking edges or curves that form a closed boundary. The cutting passes are then interpolated from curves and edges. You can also create concentric and radial cut patterns, or divide the surface into regions by inserting additional curves.

Features are:
- Control over the shape of the stepover moves between the passes.
- Additional entry and exit options–linear tangential, circular horizontal, circular vertical, circular normal.

Drive/Part
Drive/Part cutting, also known as dual contact cutting, keeps the tool in contact with two surfaces at all times, allowing for valley cleanup cuts across many surfaces. The algorithm sorts through the selected surfaces, allowing many surfaces to be easily navigated. Drive/Part milling includes:
- Support for various tool shapes, including ball-nose and bull-nose cutters.
- Easy surface selection.
- Separate finish allowance for Drive surfaces and Part surfaces.



I-DEAS Generative Machining provides a variety of cutting patterns for surface machining.

Contact Curve Display for Copymill and Drive/Part Machining
I-DEAS Generative Machining lets you display the tool contact curves on the solid. This option allows you to quickly analyze the regions on the solid where the tool was not able to reach or machine. The generate points option automatically creates points and the end of the contact curve. These points can then be used to create constraining sections for subsequent surface machining with a smaller cutting tool or different tool type.

Connection Moves
I-DEAS software allows you to control the connection moves in a toolpath. A connection is an auxiliary motion that takes the tool from the end of one stroke to the start of the next stroke.

For copy mill and flowline operations, you can define the shape of stepover motions between cutting passes. You can choose straight line, projected, and hat connections.

You can also define the rapid move from the end of an exit move to the start of an entry move. You set the height of the retract motions between different regions and different depths of a toolpath, instead of sending the tool to the clearance plane.

Connections also allow you to control milling passes over holes or other gaps on a surface. Based on the size of the hole, you can create either a linear cutting pass over the void, or retract the tool at the edge of the hole and engage on the other side.

Toolpath Entry and Exit Control
I-DEAS provides a variety of methods for entry and exit and control of the entry and exit settings between slices. The start and endpoints of a toolpath are automatically calculated based on part and stock geometry. The entry and exit motions are evaluated to avoid gouging the part or the workholding devices. Entry options include clearance plane specification (absolute or above the stock), plunge, ramp, along path, circular, in plane, and drill. Exit options include retract plane specification (absolute or above the stock), lift, ramp, circular, and in plane.

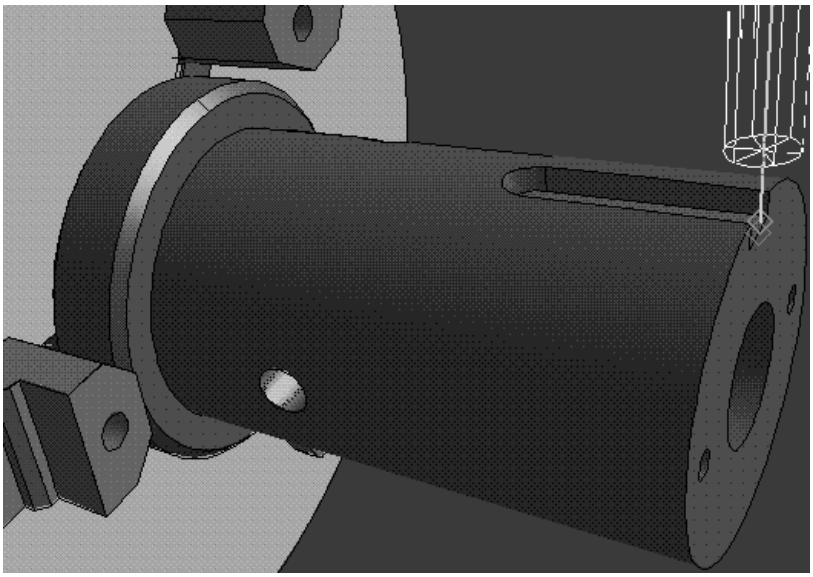
4- and 5-Axis Positioning
Multi-axis machine tools are being used for setup elimination and high precision machining. The need is even more critical for software to represent the complex setups which enable machine tools to machine multiple sides of a part. Along with this need is the ability to program 4- and 5-axis machine tools quickly and easily. I-DEAS Generative Machining lets you design the fixturing, plan the job, and perform NC programming-all in one easy-to-use system. The machine definition form lets you describe the machine type for rotary axis calculation. Based on your machine definition, I-DEAS Generative Machining will check the machine's ability to position to that location.

Turning Capability - 2, 2+2 Axis

I-DEAS Generative Machining software also provides toolpath-generation capabilities for 2-axis and multi-turret lathes. With user-defined bar or cast stock, I-DEAS Generative Machining in-process stock calculations show the evolving stock model as turning operations are performed. The in-process stock is then used for subsequent operations to create optimized tool cutting motion.

Turning capabilities provided include:

General Features
•The application works directly off the solid geometry while maintaining associativity with the design model.



I-DEAS Generative Machining software supports front, rear, and dual-turret turning machines.

•The machine definition form allows you to specify the type of machine: lathe, mill, or mill/turn, and the primary turret-front or rear.
•Front and rear turret support (either 2-axis or 2+2 axis programming).
•Full insert and holder collision detection and gouge avoidance with part geometry, stock, machine, and fixtures.
•Separate finish allowances for diameters and faces.
•Separate X and Z limit specification.
•Separate feedrate specification for turning, facing, and combination cuts.

Mill/Turn Support
This application allows both milling and turning operations within the same setup to take advantage of the milling capabilities of mill/turn machines. You do not have to create separate milling and turning operations and then merge them.

The application uses user-defined coordinate systems to enable the post-processor to generate the appropriate spindle rotations. The mill/turn machine type lets you treat the spindle as a rotary axis for 4-axis positioning.

Rough Turning, Facing, and Boring
•Incremental clean-up capability. The software follows the part profile to the previous cut depth to eliminate troublesome "stair stepping" of the toolpath.
•Critical diameter/face adjustment. Automatically adjusts cutting depths to accommodate different diameters. This provides for more efficient cut utilization and eliminates inconsistent material removal for subsequent semi-finish and finishing operations.

•The software automatically adjusts depths of cut to accommodate those diameters or faces which do not fall into the “single depth” of cut mode when incremental cleanup is off.
•Automatically calculates entry and exits based on cut angle and tool geometry.
•Optimizes tool motion based on the user’s initial stock-either bar or a forged/cast stock. Subsequent operations cut only where stock remains in the in-process stock.
•Face past centerline distance and specification.
•Cut angle and cut direction control.
•Undercut on/off switch. Indicates to the software whether to allow the tool to go into undercut regions (separate face and diameter control).

Finish Facing, Turning, and Boring
•Corner break control-radius or chamfer.
•Undercut on/off switch. Indicates to the software whether to allow the tool to go into undercut regions or to connect regions for finish operations (separate face and diameter control).

Thread and Groove Feature Identification
With I-DEAS Generative Machining, you can identify the thread and groove features of the part. The software will then use the description the manufacturing engineer inputs as the basis for all operations. For subsequent thread and groove operations, you pick only the identified feature rather than the geometry which comprises the feature. Additionally, I-DEAS Generative Machining can calculate the major and minor thread diameters based on thread type and classification.

Threading Operations
The thread operation leverages the thread feature and automatically generates the cut depths. The threading operation provides the ability to define how the thread passes are calculated along with the ability to define both spring and gage passes. Threading features include:
•Multiple starts.
•Internal/external.
•Lead/pitch specification.
•User-specified depth calculation methods decreasing (sometimes referred to as constant volume) or constant depth.
•Multiple spring and gage pass specification.
•User control for gage pass stop conditions and stop location.
•Infeed and pullout distance control.

Rough and Finish Grooving
•Corner control-radius or chamfer.
•Bottom dwell time in seconds or revolutions
.•Finish pass overlap.

Tool Definition and Catalogs
•Insert-standard shapes.
•Touch off, key in offset, or tool nose radius CL file output.
•Holder-definition.
•Combination tool allows mixing and matching inserts with tool holders.
•Support for left- and right-hand tools, front/rear turret mounting.
•Tool catalog support, can use separate catalog for inserts and holders.

I-DEAS C-Post

I-DEAS C-Post software allows you to create standard or custom post-processors for any modern NC machine/control combination. Directly compatible with the standard CL files output from I-DEAS Generative Machining or Camand®, I-DEAS C-Post supports 2 through 4 axes of simultaneous motion for lathes and millturn centers, and 2 1/2 through 5 axes of simultaneous motion for milling machines.

Easy, user friendly, on-site configuration

I-DEAS C-Post uses any standard text editing program to create or modify the desired post- processor. The I-DEAS C-Post language itself is simple and easy to use. There is no need to learn a complicated programming language or to compile a completed post processor with I-DEAS C-Post. Built in run-time error reporting assists in the actual creation of complex post processors, as well as in verifying that the output toolpath will be correct for a given machine.

Powerful and flexible

I-DEAS C-Post gives virtually unlimited control over the format, sequencing, and content of the final toolpath. Automatically updated, I-DEAS C-Post global variables allow access to tool tip, contact point, and tool axis values, as well as all other machining conditions such as feeds, speeds, cutter compensation, and coolant. Formatting rules allow this information to be output for inch or millimeters, absolute or incremental, with control over such details as G and M code formats, toolpath scaling or translation, leading or trailing zeros, machine minimums and maximums, sequence numbers, arc output formats, canned cycle codes, and automatic long file breakup.

Customize to your needs

Additionally, user created variables and custom macros give you ultimate control over special functions, such as automatic tool changes, inverse time feed rate calculations, and special program start and end sequences. Built in math, text, and logic functions allow for even more customization and optimization. For example, trig functions (SIN, COS, TAN, etc.) can be used to calculate 3-D cutter compensation, even if the machine controller does not support it directly with a G code. Text functions can be used to output process sheets, describing such things as tools used and total machining time. And IF..ELSE logic control allows for conditional manipulation and output of any global or user controlled information.

Prerequisite

Core Master Modeler
-or-
I-DEAS Product Design Package
-or-
I-DEAS Artisan™ Package

For More Information

For more information, contact your local SDRC representative or call 1-800-848-7372.