

CAD/CAM Software Benefits for Hole Making

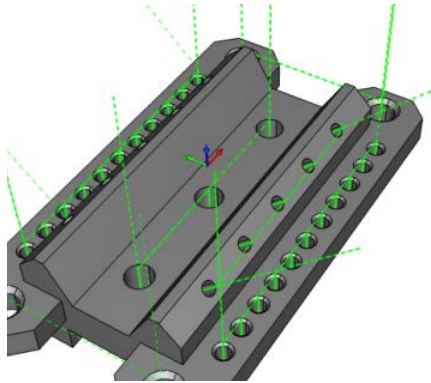


CAD/CAM Software Benefits for Hole Making

Hole Processing Technology

CNC Shops throughout every branch of manufacturing rely on programming automation for hole processing—drilling. While many hole drilling scenarios are simple, some performed on manual machinery and drill presses, CAD/CAM software is still holding strong when it comes to processing and machining holes.

Common CAD/CAM machine cycles for hole making include:



- Center Drill
- Hole
- Tap
- Rolled Tap
- Bore
- Ream
- Counter Bore Hole
- Counter Bore Tap
- Counter Bore Rolled Tap
- Counter Bore Ream

Some CAM systems will list them differently than others. Essentially, by navigating to the machine setup feature, you will have access to all of the 2, 3, 4 and 5 Axis machining strategies or features depending on the software.

These would be provided as hole making features in a CAD/CAM product that should be accessible within a CAM Job Tree. The "Job Tree" will be the place in the software interface that all machining features are located along with posting, tools, materials, machine setup features and more. This is the centralized command center for creating the correct toolpath and NC program that you will send to the machine tool.



What are "Tool Pattern" Hole Processing Cycles in CAD/CAM?

The purpose of CAD/CAM is to automate the machining-part programming phase of manufacturing regardless of how simple the operations. CNC software products will sometimes provide what are known as "tool patterns" for machining features where more than one tool can be used in an operation. For example, a basic hole drilling features can be created using a center drill and a drill, with or without a chamfer as a third action to be completed in the operation. The tool pattern setting should be available on a per-part basis or as a global setting. Once again, "tool pattern" functionality controls the order of operations for all supported milling features that a CAD/CAM system offers. All changes that are made in the pattern control dialog box will generally affect the current part (loaded into the software) only and are not applied to any other parts unless the global settings are used. Then these tool patterns will be affected for all part programs when called for.

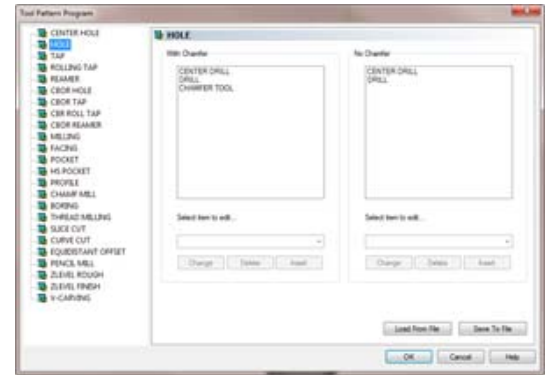




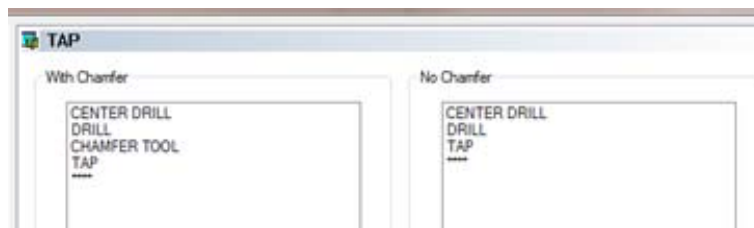
CAD/CAM Software Benefits for Hole Making

CAD/CAM software allows the user to define these tool patterns to create efficient programs way faster than by hand. Typically tool pattern control functionality will allow the user to edit:

- **Operations Lists** - (With Chamfer and No Chamfer) The software will show the order of operations for the hole feature that is chosen (drill, center drill etc.). This is the order in which the operation will call tools and create the program unless otherwise determined by the operator. This is where those changes would be made.
- **Select item to edit** - The software should display a list of all supported milling operations. This way it is easy for the operator to go through machining features (drilling, pocketing, profile milling and other operations) and add or edit tool patterns quickly and easily.
- **Change** - This is more editing type functionality that simply replaces the selected operation with a different one.
- **Delete** - This action removes a selected operation from the Operations list.
- **Insert** - This basically adds a new operation into an existing operation-pattern.
- **Load From File** - You should be able to use a tool pattern configuration from a previously saved file. This saves time by not having to continuously create popular tool patterns.
- **Save To File** - You should be able to save a tool pattern configuration for later use.



Tool patterns provide a nice benefit to programming efficiency as they save time while providing a programmer a lot of control and flexibility in how they want to program their jobs. Example of a Tool Pattern for a Tap cycle that allows for a chamfer or not:



With a chamfer there are 4 individual tools and actions being called up if we were to use that feature. The "Chamfer" option is listed at the top because in this example it is a check box in the Tap feature of BobCAD-CAM. Therefore if we check the box in the machining wizard, all 4 of these tools and parameters will automatically appear in the machining wizard for Tap. If we do not check the chamfer option then only a center drill, drill and tap would be used in the wizard.

Some shops might only need to set up tool patterns once and never have to look back until something new comes along. When evaluating CAD/CAM, make sure the software supports the creation, editing and saving of tool patterns.

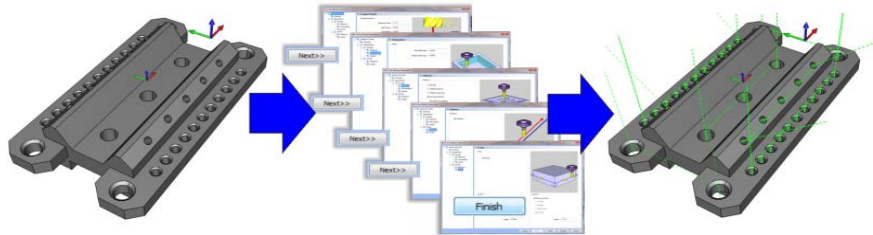


CAD/CAM Software Benefits for Hole Making

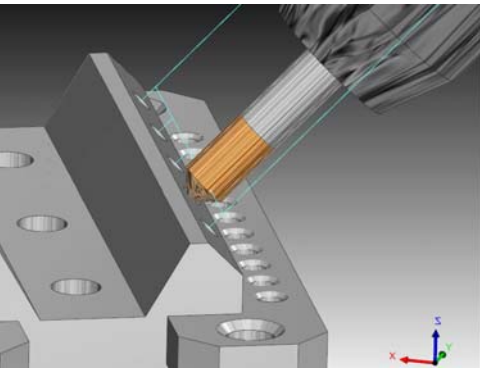
What are "Wizards" in CAD/CAM?

Machining "Wizards" are an important aspect of CAD/CAM functionality as they are basically a series of dialog boxes that properly step an operator through the machining sequence in a CAD/CAM product.

In this paper we are focusing on hole drilling, yet machining wizards are important for all of a CAD/CAM products machining features/operations. Wizards give great benefits as they remove the guesswork from using CAD/CAM to program your parts while still providing the control and flexibility that advanced programmers need to machine their parts.

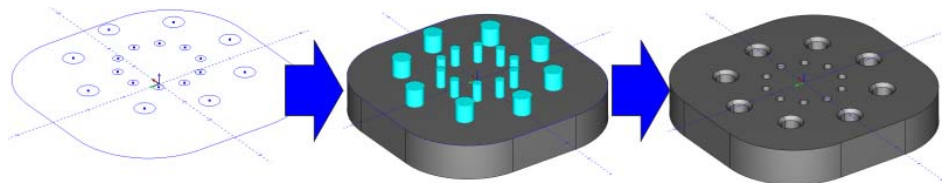


When the machining feature is ready there will be a "Finish" button that allows the operator to create the toolpath for the feature. Machining features can easily be edited by opening the wizard and making changes, then saving the new changes. When evaluating CAD/CAM, make sure the software supports the use of Wizards to program machining features.



Hole Selection & Optimization

The CAM portion of the software will typically look at the part model in the workspace and know where the holes are along with their sizes and depths. The operator will provide the remaining data in the stages of the wizard and produce the toolpath to create an NC program. The software generally will have a hole pattern or grid feature allowing the CAD designer to create various hole patterns. If the pattern or grid type feature is used, the geometry will typically be drawn and grouped for easy selection. Independent holes can be grouped with other holes of the same size to keep similar diameter holes together. This saves time allowing the user to save hole groups on different layers as well. Simple wireframe geometry can be turned into a solid part easily with CAD/CAM.

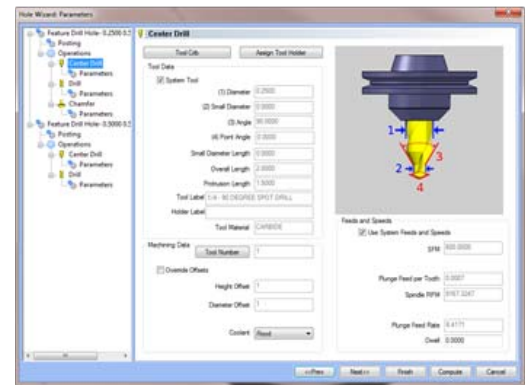


CAM software will allow the operator to pick hole geometry for holes represented by arcs, cylinders or edge/arc geometry of a solid part model to indicate the holes. CAM software should automatically pick up the hole diameter and indicate the correct tool based off of the hole operation type that is being used. In some cases software will offer two types of patterns, Optimized and Selected Sequence. The CAM software should be able to detect the hole geometry and automatically optimize the machining sequence, basically optimizing the route that the tool will take from hole to hole. This can be a valuable time saving benefit in cases where there are many holes to be machined.



CAD/CAM Software Benefits for Hole Making

As mentioned earlier, machining wizards should be used as they step the operator through the process efficiently. If tool patterns are being used, the operation will include each tool operation within the hole feature. The hole feature will generally ask for a clearance plane, rapid plane and the top of the part. Some systems will allow the operator to simply pick the top of the part at this stage. Input parameters will include whether or not there is a chamfer needed, whether it's a blind hole or a through hole and offer machining order settings; Optimized or Pick Order. On 4 axis parts the operator will be asked for a work offset and a rotary output angle, 45, 90, 180 etc. This tells the machine to index the part for the next series of holes to be machined. Next, tool data is input, number, tool holder and tool crib data along with tool offsets, speeds and feeds and even coolant options. The CAM software should acquire the correct tool for the drill cycle based off of the geometry selections that the operator made initially, diameter etc. CAM software uses the tool database, materials selected and input data to optimize speed and feed calculations for the operator. However there should always be the ability to modify these within the software before posting the program to the machine.

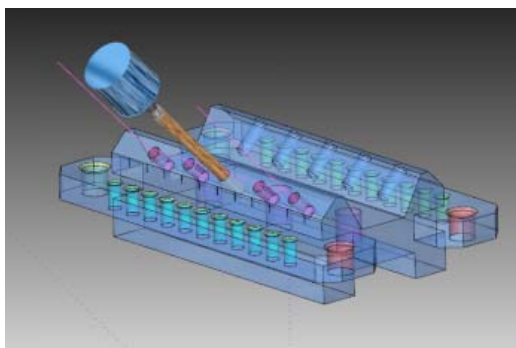


Other operation parameters should include an effective depth, overall depth, hole diameter and the cycle type. For example, whether you are setting the hole to a single depth, peck or fast peck where a first peck depth can be used, a overall peck depth and the number of pecks the operator chooses. The last stage of the machining wizard will be to automatically compute the toolpath for the program.

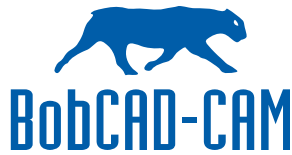
CAM software will allow the operator to visualize the path that the tool will be taking along with rapid moves. This helps the operator to see the pattern that the tool will be taking to perform the drilling operations.

The operator will have already selected a pre-configured post processor for their machine tool. This would be done as a part of setting up the job initially.

Complex Hole Processing



Some jobs require holes to be machined on angled surfaces where a 4th axis index is required. CAM software should allow the operator to set up an index system within a job if a 4th axis is required. This is where an output rotary angle would be used with and angle supplied by the operator. Here is an example where a 225 rotation angle and a 315 rotation angle were used in creating a program that had holes of various sizes on different planes.

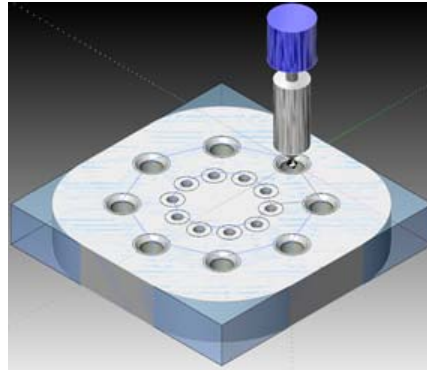


CAD/CAM Software Benefits for Hole Making

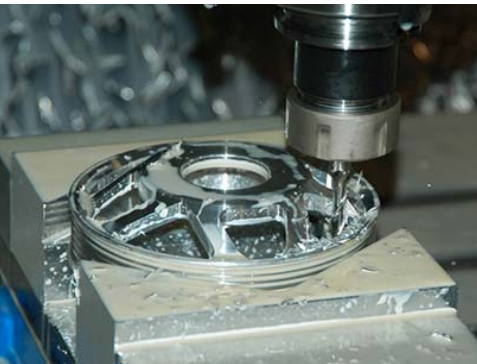
CAD/CAM software benefits can be great when difficult parts require hole patterns to be machined in multi-axis situations. The CAM Job Tree allows the operator to create and control all aspects of the job whether simple or complex.

Simulation & Posting

CAM software should have some form of toolpath simulation which provides full visualization of the machining process from start to finish. Not only should simulation provide the necessary visual, the operator should be able to gather information such as:



- Tool data, tool tip position, tool center position
- Operation data
- Sequence data
- Movement data, rapids, feeds
- Cycle time calculations
- Toolpath lengths
- Deviation analysis
- Higher level machine simulation (the actual machine movements visually)



CAD/CAM software would not be complete without post processing capabilities.

The general sequence of creating a job would be to:

- Import or draw the part geometry
- Create the Stock
- The Job Tree is used for the Machine Setup
- Post Processor selection/Customization
- Machine operation selection
- Toolpath Creation
- Simulation
- G-Code Generation

A "Post Processor" is a translator that converts all of the input data, toolpaths and Job Tree information and converts it into a numeric program (g-code) that the machine controller will accept. Each machine tool controller model is different. Typically a CAD/CAM provider will provide a post processor for their customers. Some providers charge extra for this as a service. However, some do not and if the operator understands the parameters, they can sometimes customize a post themselves. You should not have to pay more for the post processor to work with your machine right out of the box.

NOTE: All of the information in this paper is available as functionality for hole processing in BobCAD-CAM CNC Software. BobCAD-CAM software "Help" files include tutorials and data on hole processing with CAD/CAM.

For more information on CAD/CAM software call BobCAD-CAM, Inc. at 877-262-2231 or 727-442-3554. Visit www.bobcad.com for a free demo.

World Leader In Powerful & Affordable CAD/CAM Software

28200 U.S. Highway 19 N., Suite E

Clearwater, FL 33761



Toll Free: 877.262.2231

Intl: 727.442.3554

www.bobcad.com

