Version 14 : September 2020

Multi-Task Machining (MTM)

Proprietary Notice

Contents

Introduction

The Multi-Task Machining option allows for the definition and creation of multi-turret, multi-spindle parts. There is no limit to the number of spindles, turrets, or axes defined by the system. This includes turning machines that are multi-axis (4 or more axes), machines that can work on multiple parts simultaneously, and Swiss-type machines.

Why the term “Multi-Task Machining”? Traditional 2-axis lathes and 3-axis mills do one thing at a time. In order to perform more than one task the setup needs to be changed on these machines. Multi-Task machines can perform more than one machining operation at a time or sequentially without a setup change or user intervention.

Multi-Task Machining (MTM) is an add-on option to the GibbsCAM Turning module. Other GibbsCAM modules and options can add functionality, reinforcing the Multi-Task Machining concept. The Mill module is required for Mill/Turn functionality, the Polar & Cylindrical Milling option is required for 4th-axis simultaneous milling, and Advanced CS is required for 5-axis positioning.

Before using MTM, you should be familiar with the basic GibbsCAM Turning or Mill modules. If you have not already read the *Geometry Creation* and *Turning* guides, please do so before continuing with this product. The documentation for MTM assumes familiarity with many items that are covered in the documentation for other product options.

Overview of Features

Actual Multi-Task Machining features include detailed machine definition, accurate cycle time calculation, better support for multiple spindles, multi-flow program creation, Cut Part Rendering, and program optimization, Utility Operations, and custom post processing. Additional functions and features include:

- Each Multi-Task Machine is fully defined (see “MDD Support” on page 23)
  - Each machine has its own Machine Definition Document (MDD) that fully describes the machine’s layout and capabilities.
- Program parts easily using the standard Turning interface first (see “MTM Workflow ” on page 9)
- All spindles, coordinate systems and accompanying geometry are displayed or can be hidden (see “The Part Station list controls the display of spindles (part stations) on the screen. There is one entry in the list for each part station on the machine. Spindles (part stations) are shown and hidden by clicking the eye icon. The contents of this dialog cannot be modified, only marked as hidden or seen. The contents are defined in the MDD and are machine-specific. ” on page 17)
- Drag and drop associative tool tiles with turret and position assignment (see “Tool Dialog” on page 14)
- Utility operations for full time awareness and simulation (see “ Utility Processes ” on page 47)

- Synchronization of operations (see “Operation Synchronization” on page 69)
  - Real time display of a multi-flow synchronous program with zoomable time-line and gaps for tool change time that includes positioning, tool and part rotations as well as the tool change.
  - Associative with tool and operation changes:
    - Interactive feed and speed changes
    - Interactive optimization of program run-time, operation / spindle order, tool turret and position assignment
  - Point and click Sync insertion/removal
  - Point and click RPM spindle control assignment
  - Real time update for all changes
  - Very accurate time calculations: rapid moves, CSS run-times, dependent operation RPM CSS run-times, independent axes rapid rates, turret, unlock, index, and lock times
- Real-time synchronized Cut Part Rendering with multiple spindles and tools (see “Stock Display” on page 79)
- Mill support for Y axis and B axis (see “What Can Be Done With MTM?” on page 30)
- Post processors that do not require editing (see “Posts and Posted Output ” on page 83)

MTM Workflow

The Multi-Task Machining option works no differently from any other GibbsCAM product, it simply has additional functionality and features. As such, there is not a set way to create parts but there are some guidelines. A file needs to be open before creating a part; there must be geometry to machine before machining processes can be created; tools must be defined in order to make processes; finally, machining operations must be post processed to be sent to the control.

Turning and Mill	Multi-Task Machining
The method most often recommended to create a part in the GibbsCAM Turning or Mill modules (the method used in the manuals), follows the order shown below.	The sequence shown to the left is basically still true for MTM, but working with multiple spindles and multiple Tool Groups makes things a little more difficult. We recommend that while learning MTM, parts should be programmed as a simple 2-axis lathe first, and then: (a) duplicate and move the geometry to non-primary spindles, (b) sort tools to different Tool Groups, and (c) reprocess the part. The modified workflow would be like this:
New File / Open existing file	New File / Open existing file
Create/modify geometry	Create geometry on primary spindle
Create tool list	Create tool list, all tools in one turret
Create operations	Create operations

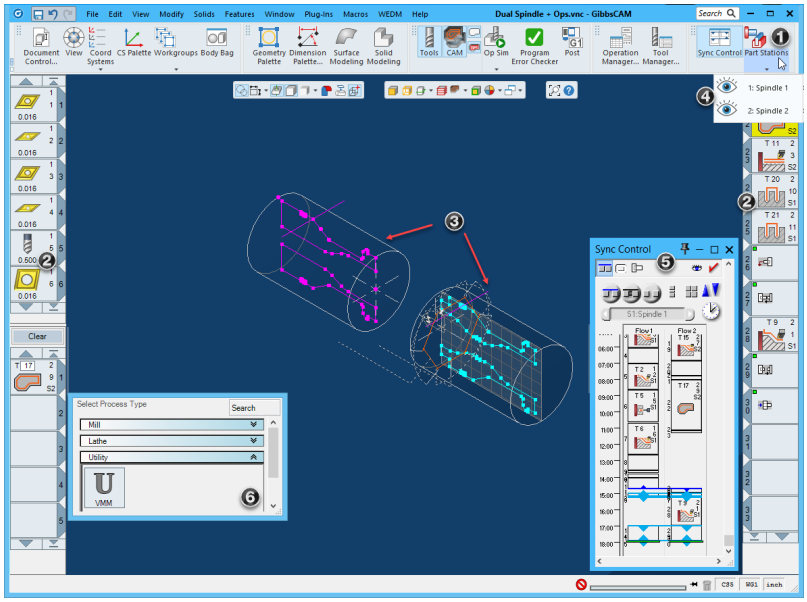
Turning and Mill	Multi-Task Machining
Verify and Render	Create / Modify geometry to be used on other spindles
Post the file	Sort tools to various turrets for optimal workflow
	Synchronize the operations and run the Sync Checker
	Verify and Render
	Post the file

Interface

Standard Interface

New users and those who have only used the GibbsCAM Turning module might not be familiar with the Level 2 interface. It is suggested that you review this topic in the GibbsCAM *Getting Started* guide before continuing. The Multi-Task Machining module functions only from the Level 2 interface. Any elements of the interface that are not thoroughly covered in a prerequisite guide (*Getting Started* and *Geometry Creation*) are discussed in this guide.

Items that are unique to the Multi-Task Machining interface include two extra buttons in the Command palette. Additional data is shown on tiles and multiple spindles are displayed in the Workspace. A Part Station list (listing the spindles), and a Sync Control dialog are available. There are also modifications to the Document Control and Tool dialogs, a new Utility process type appears in the Processes Menu, and additional options appear in the Cut Part Rendering palette.

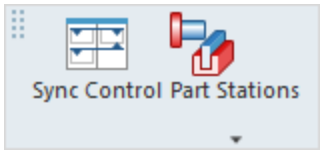


1. Command palette items
2. Additional tile data
3. Multiple spindles
4. Spindle list
5. Synchronization dialog
6. Additional process tile

Modifications to the GibbsCAM interface for MTM.

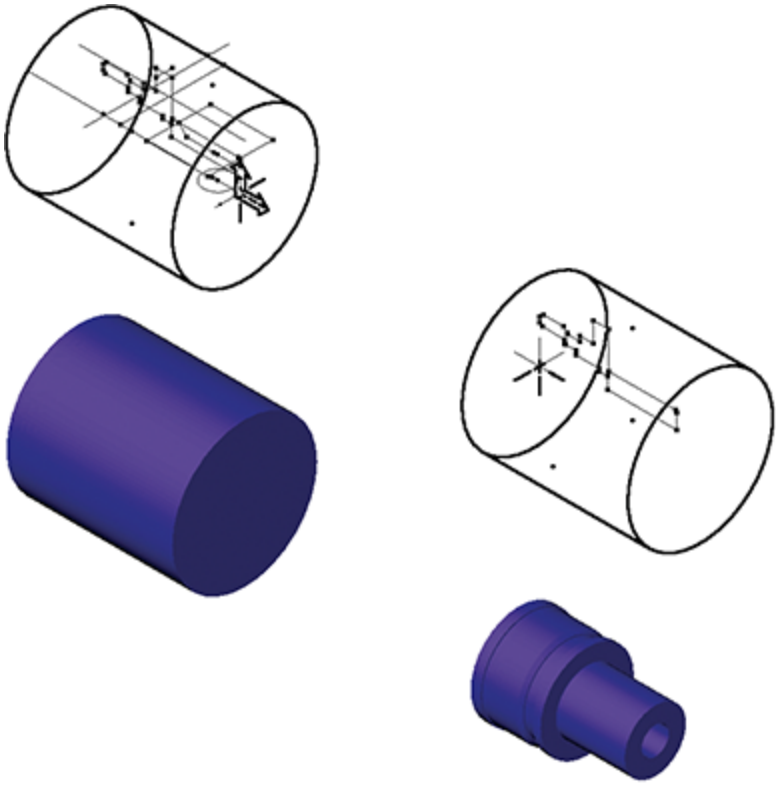
Main Palette

The Command palette has two additional buttons when creating a Multi-Task Machining part. The buttons are located at the right of the palette.



Multiple Spindles

The Multi-Task Machining option enhances the standard GibbsCAM interface and Workspace to allow for an unlimited number of spindles. Each spindle has a number, a type, and other parameters needed to fully describe the spindle, based on its type. Once loaded with a part, any machining operation can be performed on a spindle (within the Tool Group and tool constraints defined). The system will manage multiple work pieces being machined on different spindles through a part load-move-unload set of utility processes. The number of spindles available is defined by your machine and the MDD. The spindle setup is accessed through the Document Control dialog. More information on this may be found in “The Part Station list controls the display of spindles (part stations) on the screen. There is one entry in the list for each part station on the machine. Spindles (part stations) are shown and hidden by clicking the eye icon. The contents of this dialog cannot be modified, only marked as hidden or seen. The contents are defined in the MDD and are machine-specific. ” on page 17 and also in “Stock and Tool Change” on page 19.



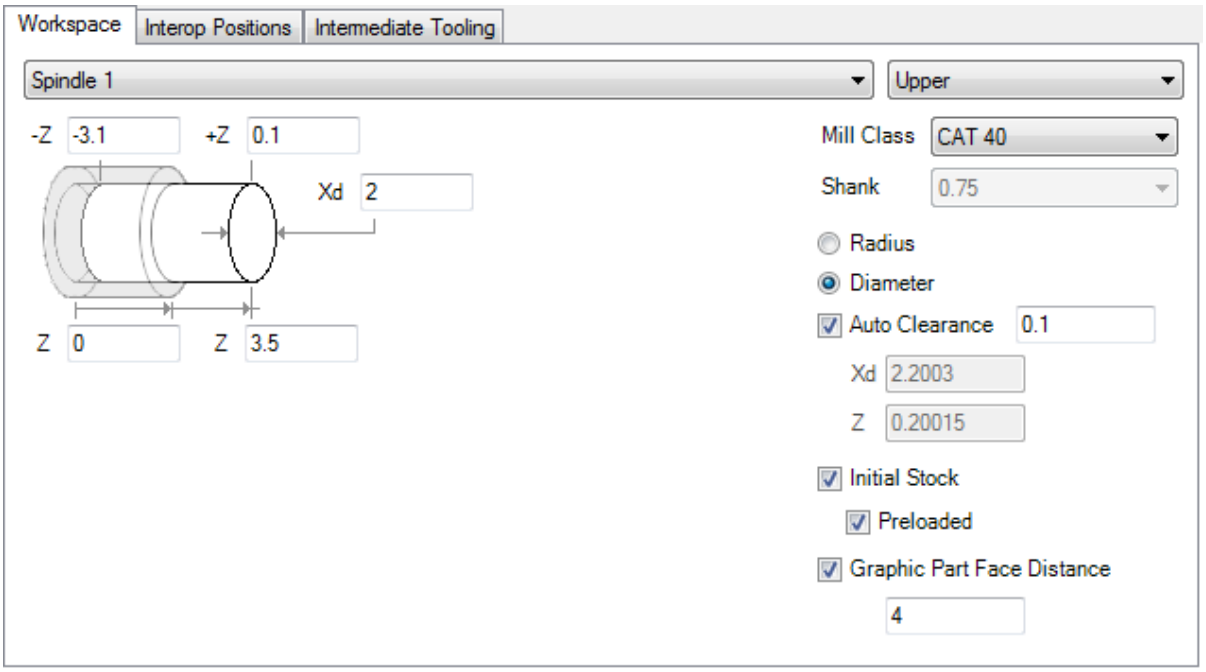
2 spindles in workspace and rendering.

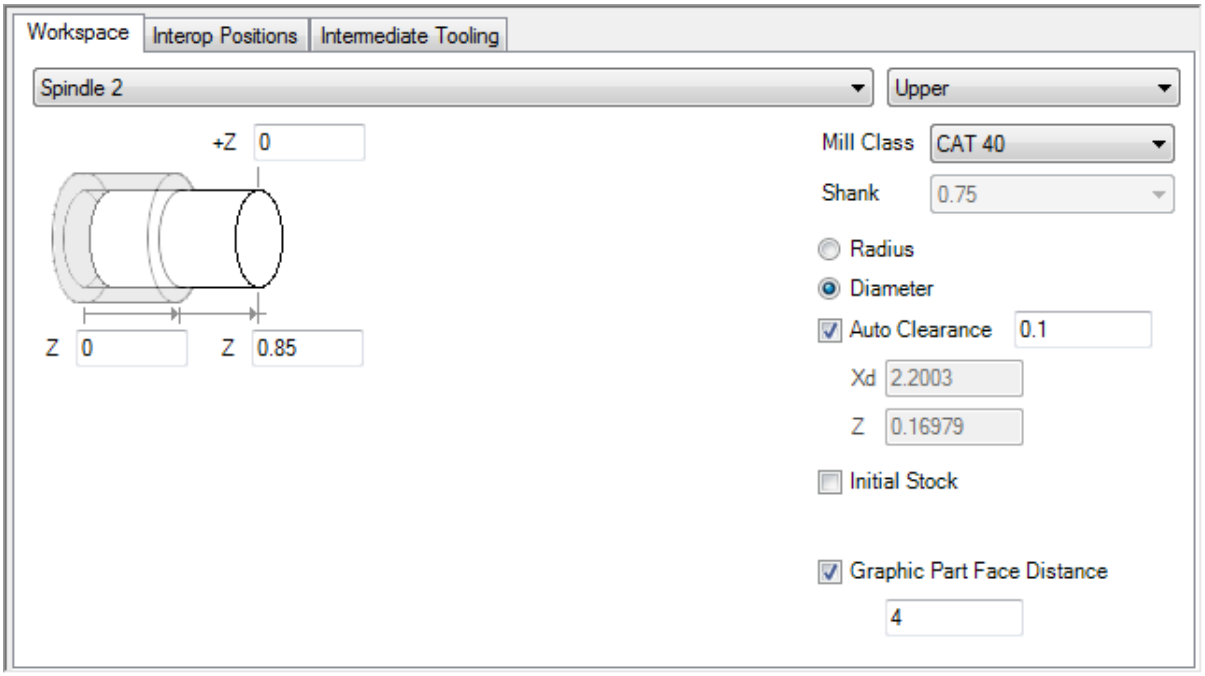
Multiple Tool Groups

“Tool Group” is the term used to refer to a turret, gang or slide. Multi-Task Machining (MTM) extends the basic functionality of GibbsCAM to provide the ability to program multiple Tool Groups machining parts on the same or different spindles. Each machining operation is programmed for a specific spindle with a tool group and parallel operations (operations that happen concurrently on the machine) are controlled through the Sync Control function. The number of Tool Groups available is defined by the MDD and set up in the Document Control dialog. More information on this can be found in “Tool Change Position” on page 22, “Tool Group ” on page 32, and “Tool Dialogs” on page 41.

Document Dialog

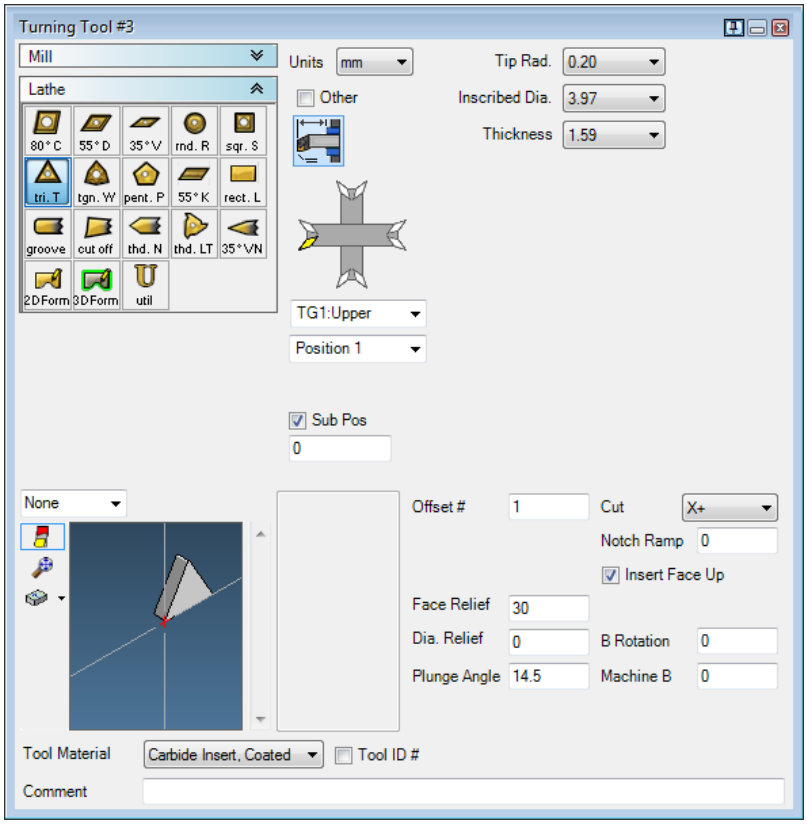
When creating a Multi-Task Machining part the Document dialog is modified to allow for the setup of multiple spindles and Tool Groups. Each spindle setup is accessed through a button that cycles through the available spindles. Each available spindle has settings for the initial stock size and condition and the available Tool Groups. More information on the Document Control dialog may be found in “Part Setup ” on page 19.





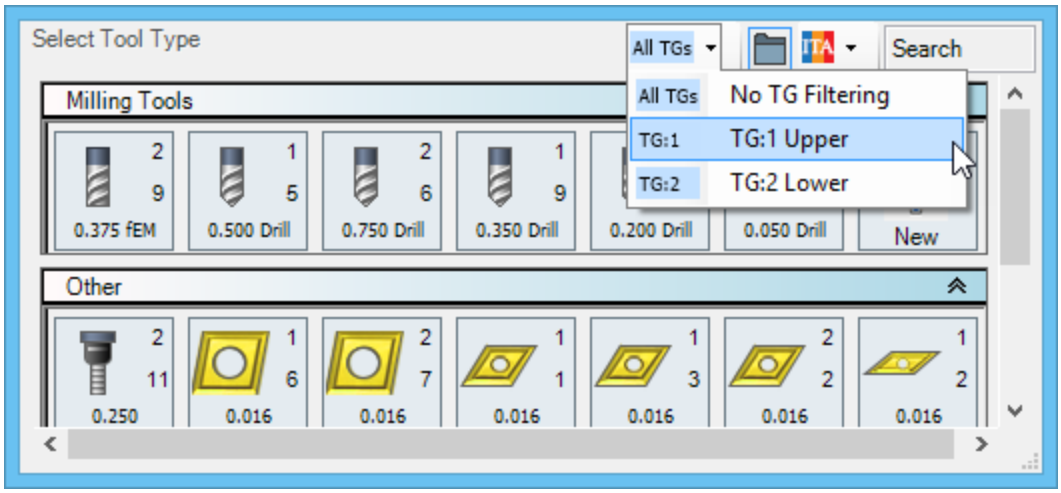
Tool Dialog

Tool dialogs have been modified for Multi-Task Machining parts. The tool dialogs include pop-up menus for selecting the Tool Group that holds the tool and the position within the Tool Group. Additionally, there is a new tool type, the Utility tool. For more information on the Tool dialog, see “Tool Creation ” on page 40.



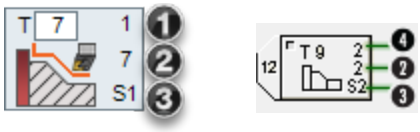
Additional Tool Type Selection

When creating a Process, the Tool selection page has an additional item to enable selection of tools within a particular toolgroup.



Additional Tile Data

Items in Tile lists have been expanded to include additional data. The Tool tile displays the Tool Group the tool is in and the tool's position in the Tool Group. The Process tile displays the Tool Group and position of the tool as well as the spindle being used. The Operation tile displays the Flow number, position of the tool as well as the spindle being used.

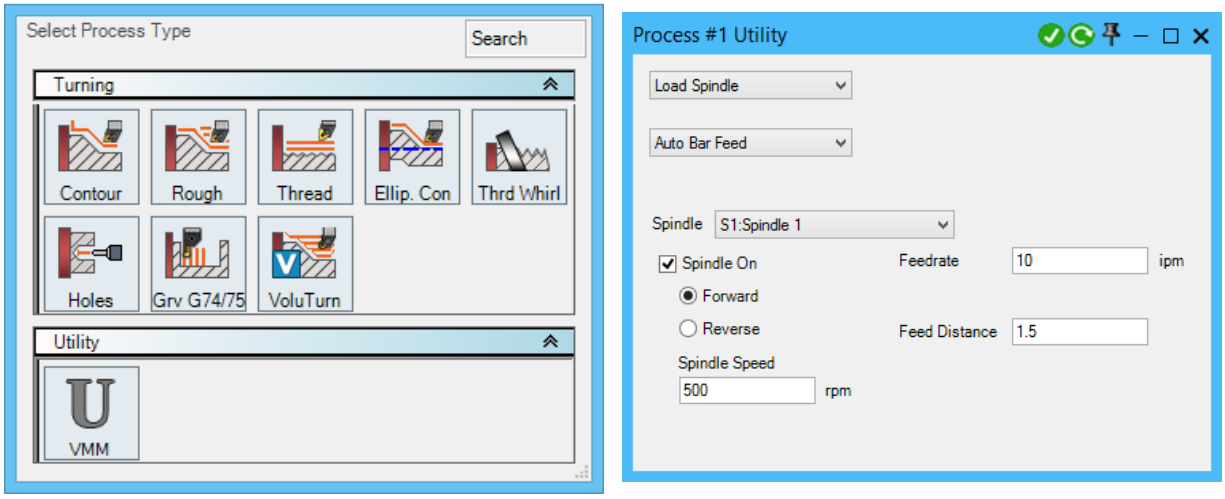


Tool Tile Process Tile Operation Tile

1. Tool Group Assigned
2. Tool's Position in TG
3. Spindle the Op is on
4. Flow the Op is in

Machining Palette Processes

The Machining palette has an additional item when working with Multi-Task Machining parts, the Utility process. Utility processes create non-cutting operations such as loading and unloading spindles, returning sub spindles and moving tailstock in and out. Some Utility processes need a tool associated with the process, such as a Utility tool that represents a bar stop, but most do not. More information on Utility processes may be found in the section "Processes " on page 47.

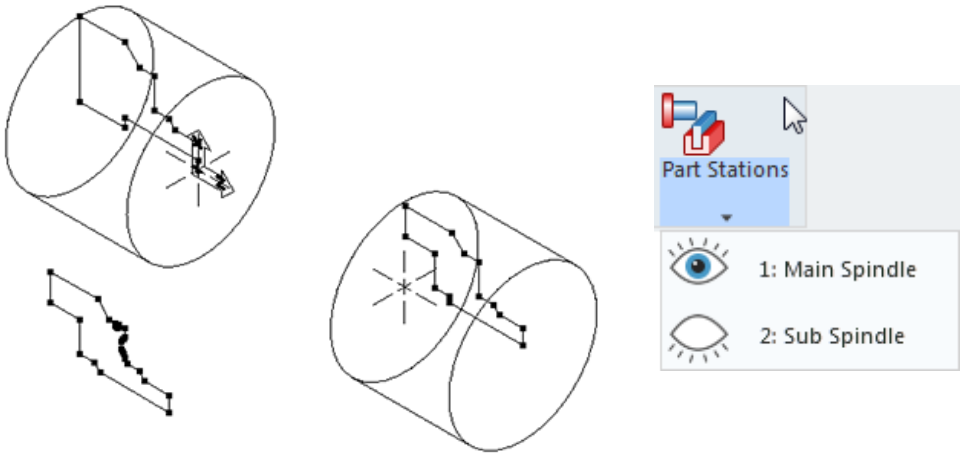


The Utility Process Tiles and a Utility Process dialog box

Part Station List (Spindle List)

The Part Station list (formerly called the "Spindle List") is accessed from the Command palette.

The Part Station list controls the display of spindles (part stations) on the screen. There is one entry in the list for each part station on the machine. Spindles (part stations) are shown and hidden by clicking the eye icon. The contents of this dialog cannot be modified, only marked as hidden or seen. The contents are defined in the MDD and are machine-specific.



Two of three spindles are displayed

When viewing the Workspace, the control in the Part Station list toggles the wireframe display of the stock space, but not of the geometry. (Geometry can be hidden or shown by using the Workgroup list.)

When rendering using Op Sim, you can use the Part Station list to hide one or more spindle and all operations performed on them. (However, rendering using Machine Sim hides suppressed spindles but renders all operations.)

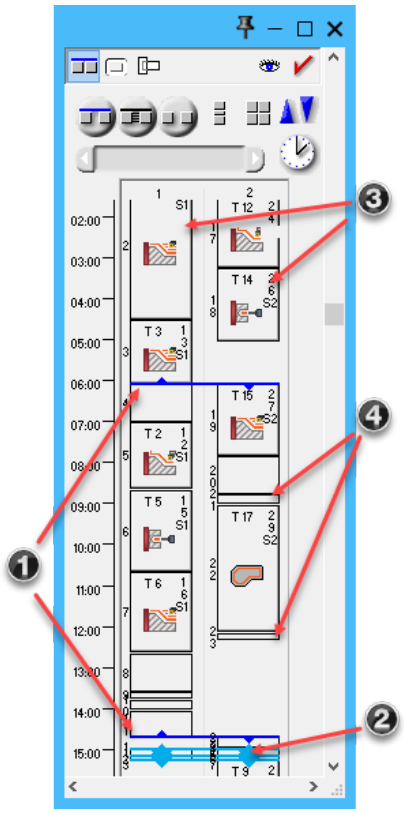
The Op Sim hiding of operations and spindles can be dangerous, because it fails to calculate collisions in hidden operations or collisions with items not included in the simulation. Therefore, when checking for collisions, always show all part stations.

The Op Sim hiding of operations and spindles is most useful in combination with view control. When an unzoom is done and all spindles are displayed, the system will zoom out far enough to show all spindles. However, if only one spindle is displayed, the unzoom will fill the Workspace with that one spindle. This is very convenient when creating geometry and operations by limiting the area being worked in, providing the largest image possible.

Sync Control

The Sync Control dialog is accessible by clicking on the Sync Control button in the Command palette. It displays all operations in a run-time format, including time for tool changes and allows the user to synchronize machining by tool turrets on a specific spindle. This will allow the user to specify that OD and ID operations are done simultaneously or to ensure that two turrets will not collide when moving into position. Greater detail on the use of the Sync Control dialog may be found in the

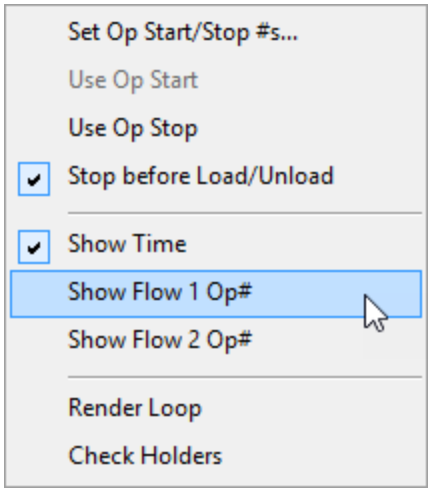
section “Operations ” on page 68. A Fit View menu item is available by right clicking on the Sync Control title bar, this selection will fit all operation tiles into the MTM Sync Control Dialog box.



1. Op Syncs
2. System Syncs
3. Actual run-time of the Operation Shown as Scaled Operation Tiles (If Fit View is chosen)
4. Time needed for the Tool Change

Render Control

The Render Control (CPR) palette can display the cut time as well as the operation number of specific flows. See “Render Control” on page 79 for more information.



Part Setup

About MTM Setups

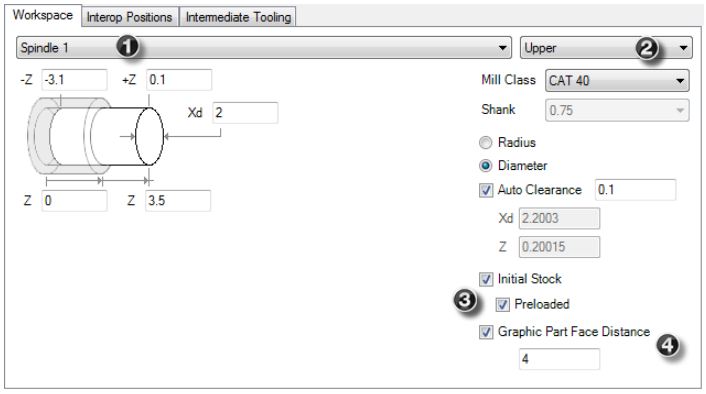
The setup of a part is more crucial in the Multi-Task Machining module than in other GibbsCAM products. The setup in MTM more completely defines what actions may be performed and the capabilities of the machine being used. In other GibbsCAM products, such as the Turning and Mill modules, the setup is relatively generic – all that is needed is the type of machine (e.g. a horizontal lathe or a 4-axis vertical mill), the size of the stock, and clearance data. In the MTM module, more machine information must be provided. This includes the machine's workpieces' orientation and axes, as well as the machine's Toolgroups' orientation and axes. Each and every machining center needs its own parameters. As such, the setup is more detailed. Fortunately this is made simpler, because all of the machine's data is stored in an MDD (Machine Definition Document) file and is used to define the part. The MDD is selected in the Document Control dialog from the Machine list.

 Document Control Dialog

The top part of the dialog remains unchanged from the other GibbsCAM modules, the only difference being the choices in the Machine list. When a MTM Machine is chosen, the second part of this dialog changes significantly. When creating a Multi-Task Machining part, an MTM Machine Definition Document (MDD) must be selected. Each MTM MDD is made for a specific machining center's parameters and manufacturer's recommendations – this includes the number of spindles and turrets in each Machining Center. The MDD controls many things in the software, including the quantity, position and display of spindles, the availability of turrets and which spindles they can access and the number of tool positions in a turret, drill cycles and coolant. You will receive an MDD with each post processor for each different machine you have. For more information on MDDs and Multi-Task Machining see the section "MDD Support" on page 23.

Stock and Tool Change

The second part of the Document Control dialog has been modified for the MTM module to accommodate multiple spindles and multiple turrets. The components of the Document Control dialog are described on the following pages.



1. Spindle Selection
2. Tool Group Selector
3. Initial machine condition when starting a program
4. Visual part distance

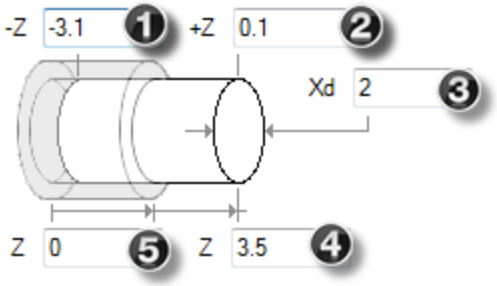
Elements of the lower part of the Document Control dialog

Spindle Selection

This item is used to select a spindle so its stock size, location and condition may be configured and the tool change position may be set for each spindle. Click one of the blue scroll arrow buttons to cycle forward or backward through the available spindles. The stock size and condition of each spindle in the machine must be defined.

Stock Diagram

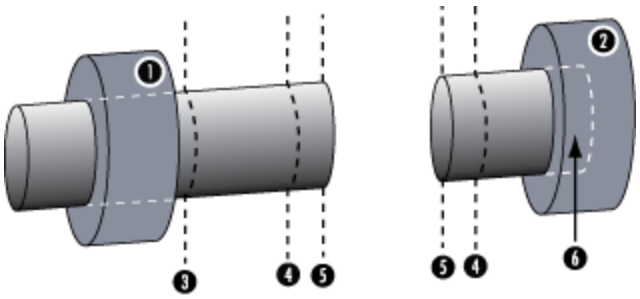
The Stock diagram is significantly different for Multi-Task Machining parts. The spindle is represented by the light cylinder around the stock. In addition to the standard Xradius or Xdiameter value and the -Z and +Z values, there is a text box marked *Z*. This item specifies the distance from the spindle's face to the edge of the stock. If the part is being cut from bar stock and fully sticks out of the spindle this value will be greater than the total length of the part. The value may also be smaller than the total length of the part. This can be true if only the front of the part extends out of the spindle face because the part is to be transferred to a sub spindle to cut the back end of the part. The setup in the Document Control dialog should reflect the actual setup on your machine. The Back of Stock Z (shown as *-Z* in the following image) is not the part length. This value only controls the length of the displayed stock. The Chuck Face Thickness is an optional item. Enter the Z thickness of the chuck or "0" if no chuck is used.



1. Back of Stock Z
2. Front of Stock Z (Facing Allowance)
3. Diameter or Radius Measurement
4. Part Distance From Spindle
5. Chuck Face Thickness

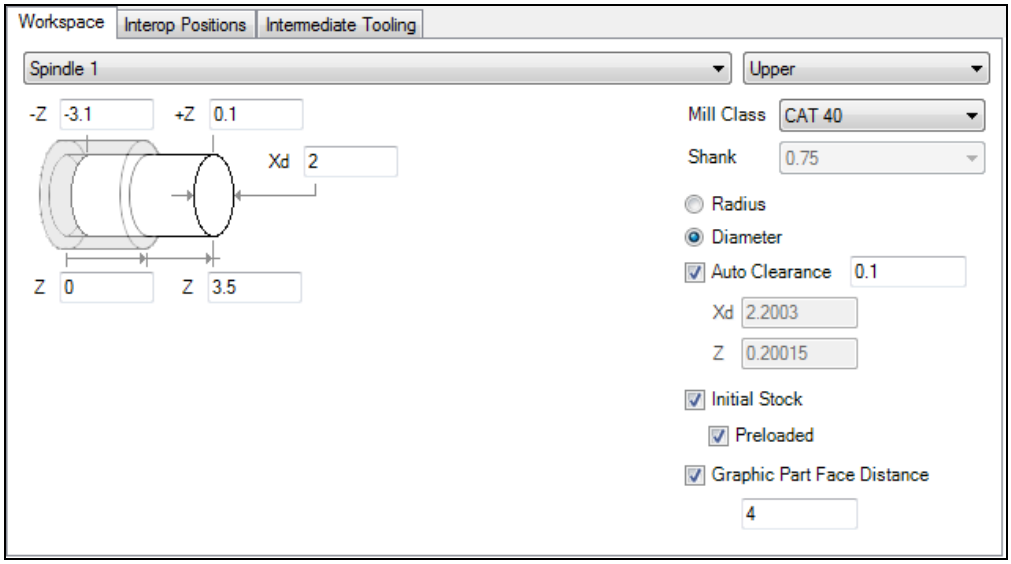
The orientation of the stock diagram does not change with each spindle, the view is always the "Home" view of the spindle's ZX coordinate system. Basically, all spindles will be shown on the left with the stock extending to the right, regardless of where the spindle really is in the machine. When you are first learning to set up a part file this may not be immediately obvious or helpful. Just imagine that for each spindle you move to the same place relative to the spindle.

Let's look at this another way because setting up your part correctly is critical. Let's use a dual-spindle machine as an example and use a different graphic to explain the elements of the stock diagram. In all likelihood, the stock conditions of the spindles will be different, one spindle will have bar stock and the other will have a partially cut piece of stock that has been transferred to the spindle. The system needs different information for these conditions. Spindles with uncut stock (a spindle designated as having Initial Stock) is used to define the size of the part and how far it sticks out from the face of the spindle. Spindles that will hold transferred stock (a spindle set with Initial Stock off) need to have the part's origin and distance from the spindle face designated. The remaining length of the part is assumed to be used for the pick-off and is in the spindle's chuck or collet.

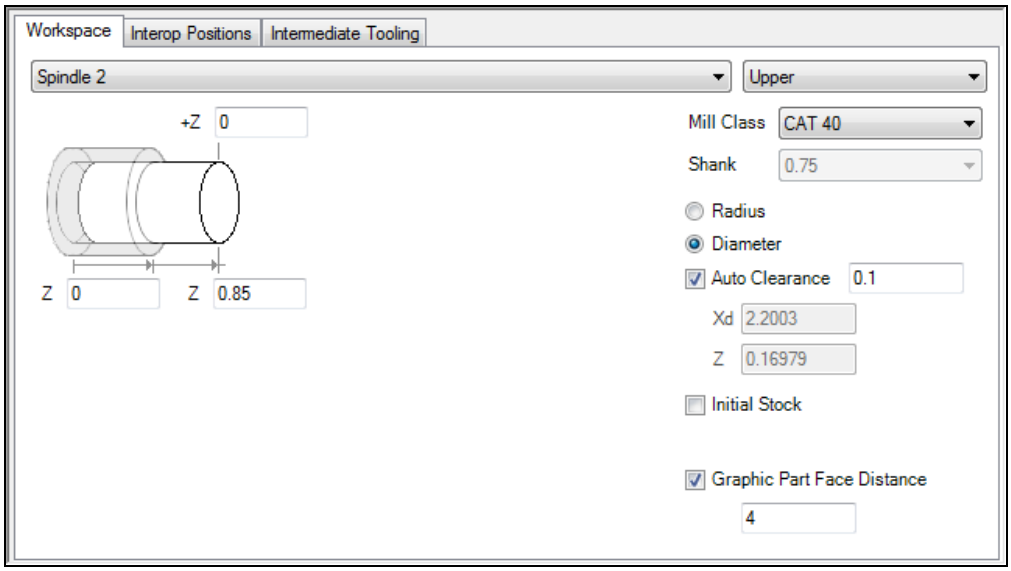


1. Spindle 1
2. Spindle 2
3. Z Depth 1
4. Part Origin
5. Part Distance From Spindle
6. Pick Off

Let's use an example with real numbers. We have a part stock that is 52mm long with 1mm on the Z+ side of the origin. The part sticks out of the spindle by 55mm. We will face off the 1mm, making the part stock 51mm long, sticking out of the spindle by 54mm and the face is now at the origin.



When we transfer the part we will pick-off the part at Z-25mm. That will leave 26mm sticking out of the sub spindle face, 1mm of which will be faced off later, leaving Z0 at the face. The stock diagram for such a part would look like the following image.



If the part is held in a chuck, add the Z-thickness of the chuck to the total part length.

Graphic Part Face Distance

This setting allows you to set the distance of the display of available spindles. This is the graphic distance between the spindles. The actual distance between spindles is fixed by the machine and defined in the MDD. However, for ease of viewing the spindles may be brought closer together. This way when unzoomed, the parts are not difficult to see because of their size. Enter the distance you want between opposing part faces on the screen. This has no effect on the actual program. The setting is a value in part units, inches or millimeters.

Clearance Data

This section of the dialog has two states, with and without Auto Clearance selected. When Auto Clearance is on, the system will calculate positioning moves between operations. The positions will be changed as the part is machined. The value entered is an offset amount from the current part stock that the system will use to always keep an adequate clearance from the material.

When Auto Clearance is off, a fixed clearance position may be entered in the X and Z text boxes for each spindle. The text boxes specify the X and Z location that a tool will rapid to and from during a tool change. This position will also be used when moving from one approach type to another.

X Dimension Style

This selection determines whether X values for the part are input as radii or diameters. Some text boxes in particular dialogs specify that the value entered is either a radius or a diameter value, regardless of the selection made here. X values entered in text boxes are usually based on the radius unless specifically labelled Xd.

Tool Change Position

The Tool change position is generally handled by the MDD. Some MDDs provide text boxes for specifying the X and Z values for the tool change position. In this case, a different value may be entered for each turret. Each available turret is accessed by clicking through the blue scroll arrow buttons. You will need to set all Tool Groups for each spindle the TG can be used on. The values are absolute values from the part origin to the Tool Group/Turret datum. Most MTM MDDs do not

have an option of specifying where a tool change is performed. These MDDs will move the Tool Group to a pre-defined, hard-coded position that may not be overridden.

Initial Condition of the Machine When Starting a Program

The starting conditions specify whether stock is or is not loaded into the machine and whether the stock is ready to be machined or has to be fed into position. This is controlled by two checkboxes, Initial Stock and Preloaded.

Initial Stock

This item tells the system which spindle or spindles start with unmachined material in this program. If a part is going to be transferred to the currently active spindle then this item should remain unchecked. When Initial Stock is off, the system expects that the plug will be moved from another spindle to the spindle being defined, (such as from the main spindle to the sub spindle). When Initial Stock is off, the Stock diagram values to be entered are the Z value that specifies the part's distance from the spindle face (the Z value is the length of the part sticking out from the spindle after being picked off) and any material beyond the part origin, such as material to be faced.

Preloaded

This item is only available when there is Initial Stock on a spindle. The Preloaded checkbox specifies how the machine is currently set up. When Preloaded is unchecked the system is told that there is no material in the chuck and a Load Spindle will need to be programmed by the user before machining can be programmed. When checked, the system expects that the first bar feed has been done and that the material is ready to be machined as soon as the program is started. At some point in generating operations a load spindle command will need to be generated, usually after all the operations are made. With Preloaded checked you are telling the system to assume that you will take care of this detail later and to start with the material in place.

MDD Support

A Machine Definition Document (MDD) controls many things in the software, including the number of, position of, and display of spindles, the availability of turrets, the number of tool positions in a turret, drill cycles and coolants available. Because MDDs for Multi-Task Machining parts are machine-specific and the programming of a part is also dependent on a particular machine's capabilities, a copy of the MDD data is stored with the part file. If the part file is moved to another computer and the MDD associated with the part is not stored on that computer, the system will automatically generate the required MDD. MDDs are kept in a folder labelled "mdd" in the application folder.

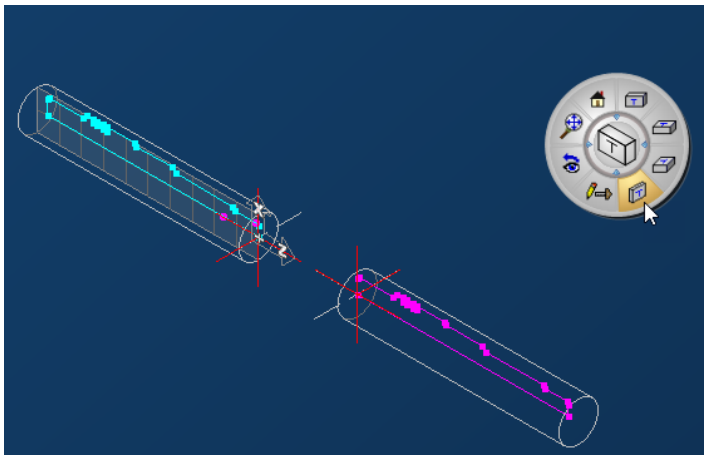
An MDD must be available for each machine that will be used with the MTM product. This is because each machine has different parameters and abilities. A Gildemeister 3+1 spindle machine is very different from a Hardinge Twin Turn which is different from a Miyano 6 spindle machine. That is not to say that a part originally programmed for a Star SV-series machine could not be easily modified to be machined on a Eurotech Elite. In fact, to make the change would likely only take a new MDD, a double check of tool assignments and redoing the Utility operations. You will probably receive a finished MDD from the same person that provides your post processor.

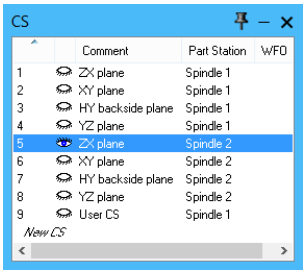
Geometry Creation

Setting Up Geometry

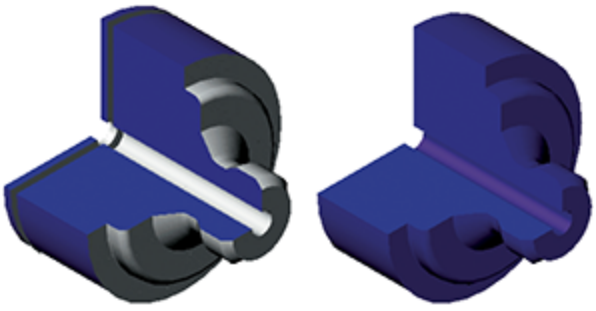
When working with multiple-spindle parts, geometry must be set up for each spindle. Each spindle has its own set of coordinate systems that define the primary planes you may work in, based on the options installed. There are several different methods to prepare geometry for Multi-Task Machining. The rule is that the geometry must be properly positioned relative to the spindle it will be used on. This is best achieved by using the coordinate systems associated with a specific spindle. The methods include:

- Create one finished shape at Spindle 1. Duplicate and move the geometry to the other spindles. Just as in the Turning module, you can usually define your roughing operations from the final shape geometry.
- Create different shapes for each spindle at Spindle 1, and then move the geometry to the other spindles.
- Create the shapes at the spindle you intend to use the shape on.
- Any combination of the above.



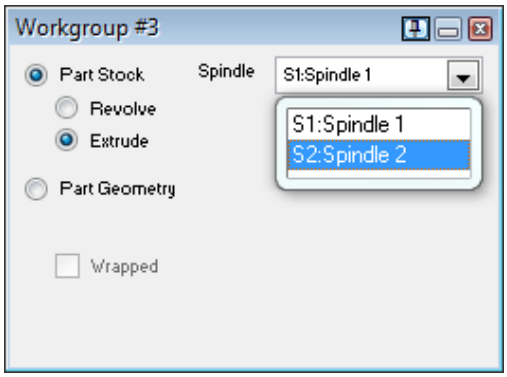


Geometry as set up for the main and sub spindles



One of the things you do not need to be concerned about is defining a stock condition for each spindle with geometry. While you can make a non-cylindrical stock body (as documented in the *Turning* guide), such as a casting for the initial stock on the primary spindle, all subspindles calculate the stock remaining from previous operations and generate the condition of the stock as it exists when transferred to a spindle in Cut Part Rendering. This is illustrated above. The image shows the condition of the stock on Spindle 1 after being machined and how Spindle 2 will look when Cut Part Rendering begins on it. This includes milling operations. This feature is automatic and fully associative. It establishes the Material Only starting condition for machining on a spindle.

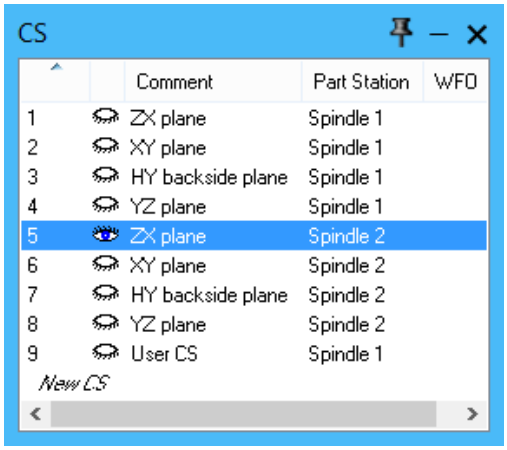
WorkGroups



From the *Geometry Creation* guide, you should already be familiar with Workgroups. Workgroups are a means of storing and separating geometry or defining custom stock, such as a revolved or extruded shape. The Part Stock setting in the Workgroup dialog can be used to define the original stock as being a custom shape, such as hexagonal bar stock or a casting.

When defining custom stock in MTM, you must designate which spindle is to receive the stock definition. Also, Initial Stock must be activated in the Document Control dialog. For more information on setting custom Part Stock, see the guides for *Geometry Creation*, *Turning*, and *Mill*.

Coordinate Systems



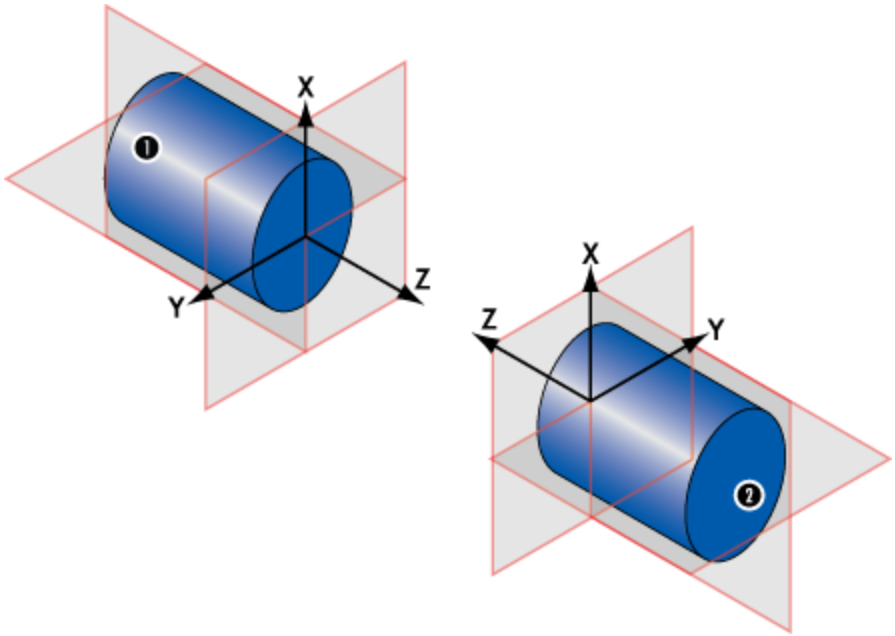
If you have used the Mill/Turn or Advanced CS options, you should already be familiar with Coordinate Systems. A coordinate system (CS) is the plane in which geometry lies. Depending on the options installed, you will have different coordinate systems available. All MTM systems will have at least one coordinate system per spindle. These coordinate systems represent the primary planes on which a part can be machined. Some of the various combinations of default Coordinate Systems follows. The actual coordinate systems you have available may differ, depending upon your machine configuration.

Turning and MTM

1 ZX plane per spindle

Mill/Turn and MTM
4 planes per spindle, the ZX, XY front face, XY back face and YZ.

Mill/Turn, MTM and Advanced CS
4 planes per spindle, the ZX, XY front face, XY back face and YZ plus the capability to make an unlimited number of additional coordinate systems.



1. Spindle 1
2. Spindle 2

Geometry Colors

Geometry in the current coordinate system is displayed in light blue. Geometry that is visible but that is in a coordinate system other than the currently active CS is displayed in magenta (a pinkish color). Geometry in a non-active CS may be edited and modified but all actions are performed relative to the current CS.

Changing Coordinate Systems

Moving geometry to another spindle can be achieved by XYZ translation or by the Change CS command. Keeping shapes assigned to the target spindle's CS is the recommended method.

However, unlike workgroups, simply getting geometry into a coordinate system may be just the beginning. Geometry may still need to be reoriented within the new target CS. The geometry may need to be translated, mirrored and/or rotated.

There are two commands in the Modify menu that allow you to easily move geometry to another CS. These commands are Change CS XYZ and Change CS HVD. While similar, these items modify geometry in different ways.

XYZ

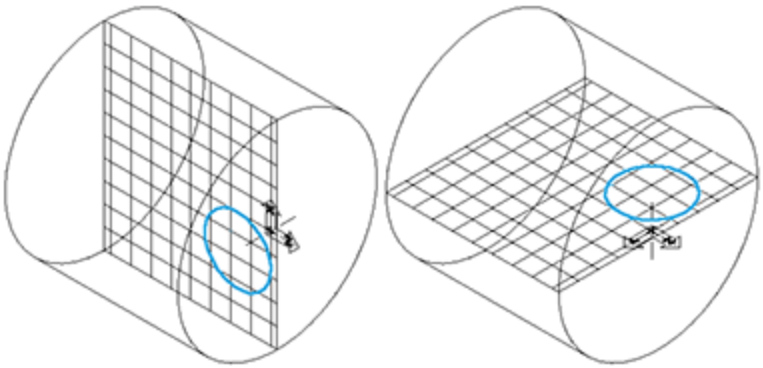
Change CS XYZ

This command will change the geometry from its currently assigned CS to the currently active CS, such as from the ZX to the YZ coordinate systems. The geometry will remain in the same XYZ location, it will not move.

HVD

Change CS HVD

(`Ctrl-\`) This menu item is available when geometry is selected. Choosing this command will change the CS to which the geometry belongs, and will move the geometry relative to the new coordinate system's origin and orientation HVD stands for Horizontal, Vertical and Depth. The arrows at the CS origin indicate the H+ and V+ directions. There is also a small + or - at the origin showing the side the D+ axis arises from. Using the Change CS HVD command keeps the HVD values the same in the new CS. Thus if a circle was in the ZX plane at Z-1, X-1, Y0 it would also be at H-1, V-1, D0. When moved to the YZ plane the circle would now be at Y-1, Z-1, X0 or still at H-1, V-1, D0 but oriented differently. Using this command will change the geometry to lie in the currently selected CS.



Note that the Home view of the shape in the first CS before the Change CS HVD is identical to the Home view of the shape in the second CS after the change.

Moving geometry between coordinate systems is similar to moving geometry between Workgroups. Geometry may be cut (`Ctrl-X`), copied (`Ctrl-C`) and pasted (`Ctrl-V`) between Coordinate Systems (CS's) by switching from the original CS to the target CS and pasting. Using the paste method replicates the geometry at the same point in space in the original coordinate system. Once the geometry is pasted the Change CS command can be used to switch to the current CS. For detailed information on coordinate systems, see the *Advanced CS* guide.

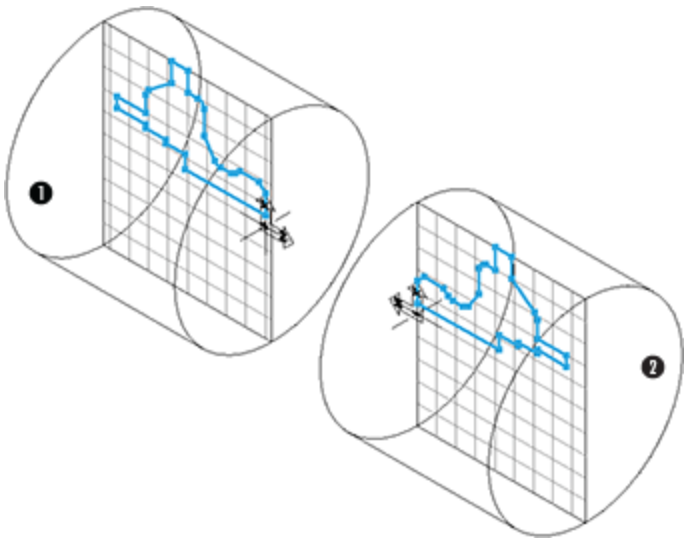
Setting Geometry Between Spindles

The coordinate systems and the orientation of the coordinate systems are identical for each spindle. The ZX plane is the primary turning plane. The spindle is always to the negative side of the Z axis, thus the spindle rotates about the Z axis with the positive end of the Z axis projecting out from the spindle face. Again, all spindles are oriented this way.

How do I get geometry from one spindle to another?

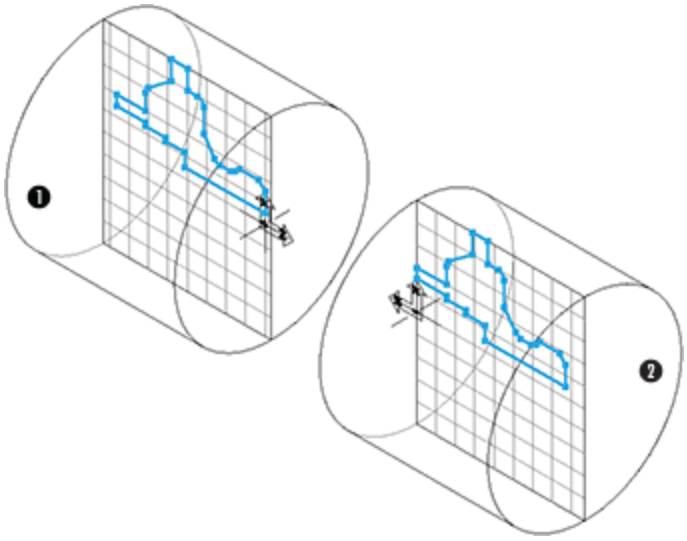
There are several ways to accomplish this. The method detailed below involves duplicating the geometry to be moved, changing the geometry to a CS on the target spindle, mirroring the geometry and then translating the geometry. While this may seem to be a cumbersome method, it is actually very powerful. Using this method you may specify what feature will be at the origin of the target spindle.

1. Select and Duplicate the geometry to be transferred (`double click` and `Ctrl-D`).
2. Select the Coordinate System for the target spindle in the CS list (typically this is the ZX plane for the target spindle, such as CS 5: ZX plane - S2: Spindle 2).
3. Select Change CS (HVD) from the Modify menu (`Ctrl-\`).



1. Spindle 1
2. Spindle 2

3. Open the Mirror dialog from the Modify menu and mirror the geometry about Z0
4. Open the Translate dialog from the Modify menu.
5. Place the cursor in the Z text box. `Alt-Click` the point that is to be at the origin of the target spindle. This will load the Z coordinate of the point.
6. Change the interrogated coordinate point to be a negative value.



7. Click Do It.

The part must be mirrored because the depth axes of the spindles face each other. When the geometry is moved to the target spindle it is automatically flipped to match the depth orientation of the target CS.

If the part origins are at the opposite ends of the part shape, you may change step 4 to mirror about a -Z value equal to half the parts length. Thus if the geometry is 40mm long, it should be mirrored about Z-20. This allows you to skip steps 5, 6 and 7.

Using Part Shift

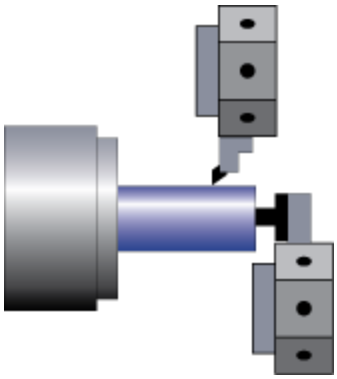
Part Shift is a Utility Process that allows you to pull or push a part in and out of a spindle, (see “Part Shift” on page 52). When Part Shift is used, new geometry must be created that represents the new part position. The new geometry should be offset in Z by the same amount that the part will be shifted.

Machining Overview

This chapter discusses general information on how machining with the Multi-Task Machining module is different from the rest of the GibbsCAM products. The methods used are not that different from the basic Turning or Mill/Turn products: a tool and a process are defined, geometry is selected and an operation is created. In fact, the processes are identical. However, with MTM there is more detail in what can be done, just as is true when comparing a simple lathe to a Multi-Task Machine.

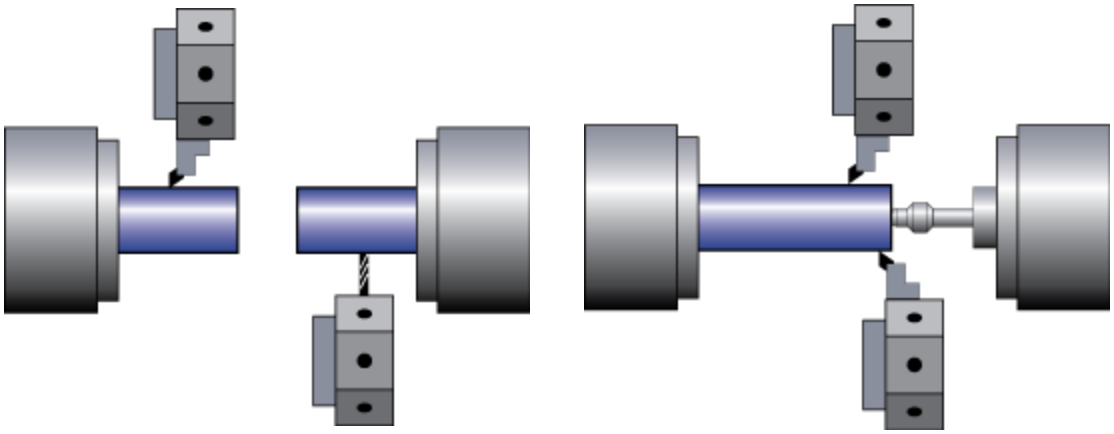
What Can Be Done With MTM?

Many things can be done with MTM. This includes turning, milling, Mill/Turn functions (Y-axis), and Polar & Cylindrical Milling (A-, B- and C-axis functions). It all depends on the capabilities of your machine. Chances are that if your machine supports a function and there is an MDD for your machine, then MTM supports the function. The base enhanced capabilities brought to the system by the Multi-Task Machining product include multiple Tool Groups simultaneously cutting on a single spindle.

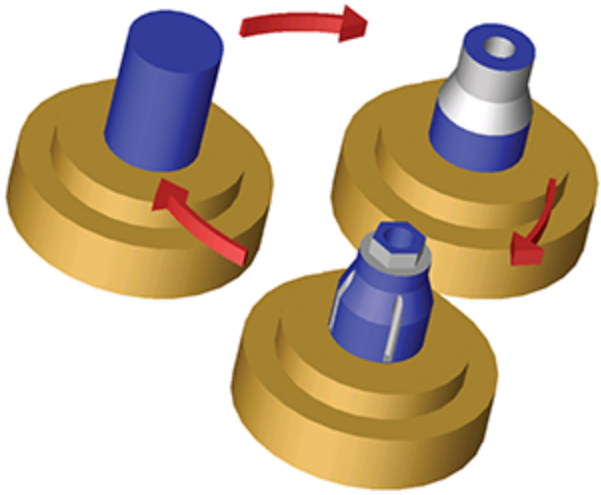


Example of simultaneous cutting

MTM functionality also includes support for multiple spindles and multiple Tool Groups. This includes Swiss and Multi-Tool Slave process cuts.



MTM is also capable of supporting more advanced machines, such as 3-spindle indexing machines.



Three-spindle setup

Machining With MTM

Creating operations with the Multi-Task Machining option is no different than working with the Turning or Mill/Turn options. What makes the MTM option different is the ability to control multiple turrets working on multiple spindles. This is accomplished through tool definition, spindle setup and operation synching.

It is recommended that when making operations, the actual setup of which tool is on what turret and the synching of Tool Groups be ignored until the operations are complete. This will aid in visualizing and organizing the part.

There are several items and terms you need to become familiar with to fully understand the Multi-Task Machining option. This includes the concept of a “Flow,” what a “Tool Group” is, and the different types of Syncs.

Important Terminology

Several terms are vital to understanding MTM: Flow , Tool Group and Sync .

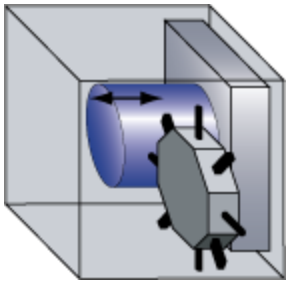
Flow

Multi-Task Machines have multiple flows. Typically there is one flow per turret. Some machine manufacturers may refer to this as a “channel” or “program.” A flow represents the sequence of moves you want a CNC machine to make. All actions by a turret are a part of its flow. Some machines require different files for each flow. Others combine all flows into a single file.

A flow is equivalent to a simple CNC program. A mill is a single-flow program. A simple lathe is a single-flow program. Only one tool moves or cuts at a time. One thing at a time is the hallmark of a single flow. When two tools can move and cut at the same time, the program will have two flows. For example, if a turning machine has two independently programmable turrets that can cut at the same time, the program will need one flow for each, or two flows. In general, there is one flow for each Tool Group, although there are exceptions. Specifically, there is one flow for each group of tools that can cut at the same time.

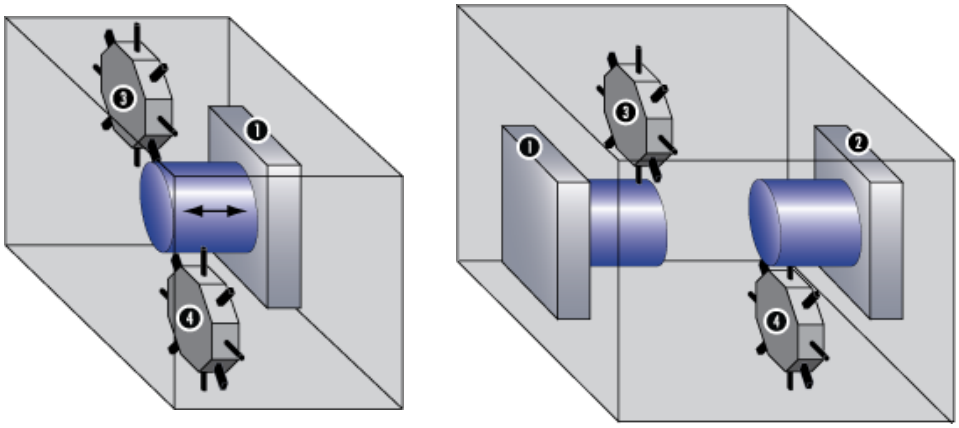
Tool Group

A Tool Group is a turret, gang, slide or mill-style tool changer. A Tool Group is any collection of tools that are dependent upon each other. These tools do not need to be in the same gang to be dependent. The key is that only one tool from the group may be used at a time. The number of Tool Groups is a determining factor in the number of flows a multi-task machine has. The following image represents a machine with a single flow.



Example of a single Tool group, single Flow setup.

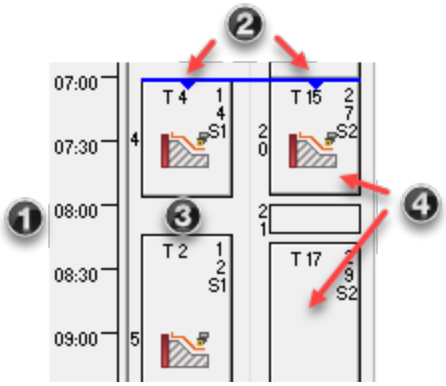
The following figure illustrates two examples of machines with two flows. These are just two of the many possibilities.



1. Spindle 1
2. Spindle 2
3. Tool Group 1
4. Tool Group 2

Sync

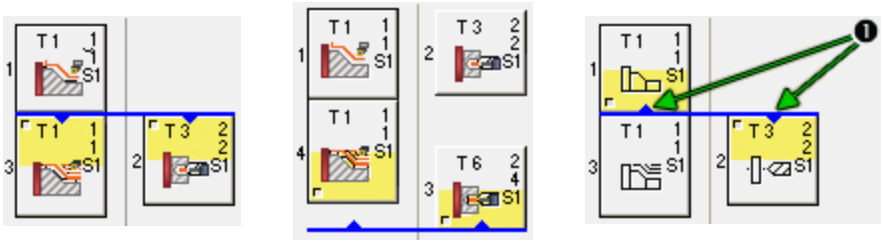
A sync is where one flow waits for another. Once the synced Tool Groups are in position the machining may begin again. There are three types of syncs, the Operation Sync (Op Sync), the Stroke Sync and the System Sync. Op Syncs and Stroke Syncs are set by users in the Sync Control dialog. The Sync Control dialog is detailed later in this chapter.



1. Real time scale
2. One column per Flow
3. Gap for inter-op moves
4. Proportional tile sizes

Op Sync

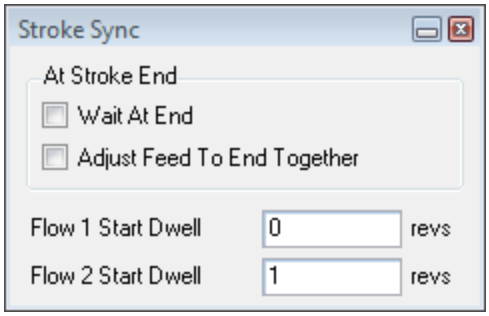
Op Syncs are synchronizations entered to control the interaction of operations. An Op Sync is applied to operations in different flows to cause one to wait until another operation either starts or finishes. The Op Sync is shown in blue. The pointer shows which op is being synced to which. You can sync the start or end of an op to the start or end of another op.



1. Pointer to the ops being synced.

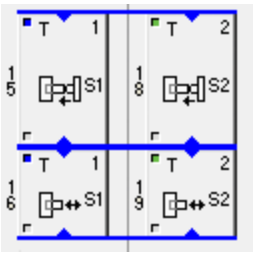
Stroke Sync

With a Stroke Sync, every stroke of a tool is to be matched with another tool in another Flow. This can be used to match OD and ID turning at the same time. Stroke Syncs are shown in black. There are three ways the operations can be synced. All three ways are controlled through the Stroke Sync dialog. The Stroke Sync dialog is accessed when a Stroke Sync is applied.



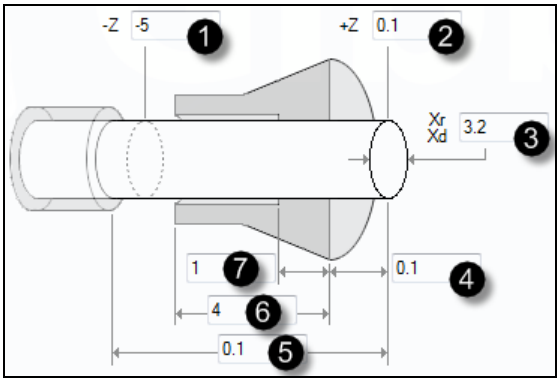
System Sync

The System Sync is applied to a set of operations and can be seen in the Sync dialog, shown in light blue. System syncs are created by the program in situations where the flows must work together or stay out of each other's way. This is automatically done by the system to make things easier for you. The following image shows a series of Utility Operations that are automatically synced.



Swiss-Style Parts

If the part station has a guide bushing (called "Swiss-style machining"), then the stock diagram provides additional choices and options, shown below.



1. Negative depth
2. Positive Depth
3. X Dimension (Radial or Diametral)
4. Stickout length, measured from the front of the guide bushing to the front face of the part
5. Distance between the face of the stock and the chuck (or part station).
6. Depth of the guide bushing, measured from the guide bushing's back to its front
7. Pullback distance of the guide bushing, measured from the back of the pullback distance to the front of the guide bushing

Other controls:

Outer Diameter of Guide Bushing
Check Guide Bushing

Sample part:`Swiss part + Ops_Guidebushing.vnc`, referenced in sample video `Swiss Data Setup plugin, v2.mp4`

Clearance

MTM uses the same clearance concepts as the standard Turning module. Part Clearance refers to clearance data that is used when moving around the part. The Part Clearance is determined either by the Auto Clearance setting (in the Document Control dialog) or by the part's CP1 setting. We need a quick review of GibbsCAM Clearance terminology before getting further into MTM Clearance information. There are three clearance positions (CP) used by the system.

CP1
The Master clearance plane defined in the Document Control dialog. This is the position the tool will rapid to and from during a tool change and when changing approach types, such as when moving from an OD to Face operation.

CP2
The Entry clearance plane set in the process dialog.

CP3
The Exit clearance plane set in the process dialog.

When Auto Clearance is active, the system will use the Auto Clearance value to calculate a constant distance above and past the current stock condition. Auto Clearance completely replaces CP1, CP2, and CP3.

Operation Moves

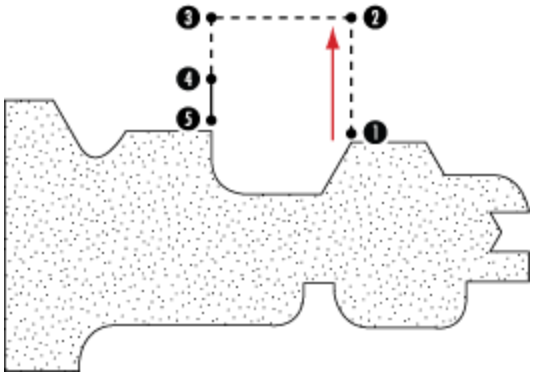
Operations begin with the tool positioned over the start point, at a clearance position (CP1 or the prior CP3) referred to as the Op Start position. The first move is from the Op Start to CP2 then to the Start Point (SP) of the toolpath. The toolpath continues to the toolpath's End Point (EP). Once at the End Point the tool moves to a clearance position (CP3). Let's call this the Op End position.

Inter-Op Moves

There are two types of inter-op moves, tool change, and non-tool change. The movement as detailed on the following pages are generalizations. They do not include possible part rotations and clearance moves in two or three linear axes.

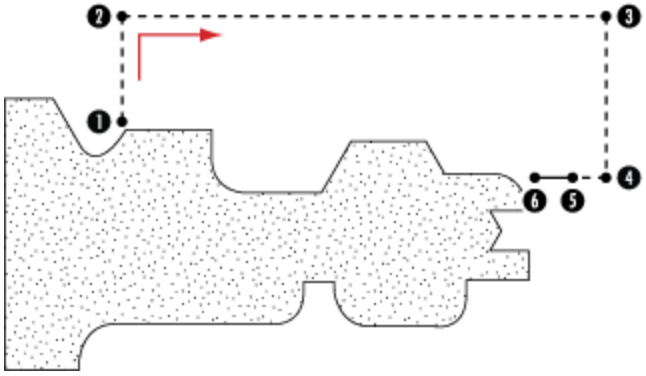
Same Tool Inter-Op Moves

If the last op and next op are on the same part side (X+ OD to X+ OD), the tool will rapid to the last operation's CP3, into position over the new SP, then rapid to CP2.



1. Op 1 End Point
2. Op 1 Clearance Plane 3
3. Op 2, Op 1 Clearance Plane 3
4. Op 2 Clearance Plane 2
5. Op 2 Start Point

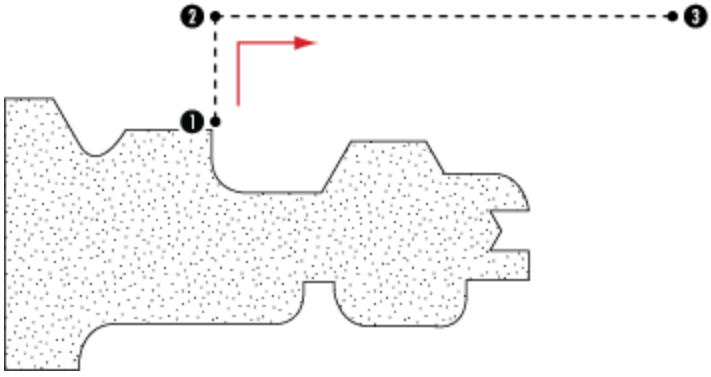
If the ops are not on the same side, the tool will exit to the Part Clearance (CP1), move all the way around the Part Clearance to the next operation's SP Xd value, then rapid in to the new operation's clearance (CP2) and finally to the new operation's SP.



1. Op 1 End Point
2. Clearance Plane 1 Xd
3. Clearance Plane 1
4. Clearance Plane 1 Z
5. Op 2 Clearance Plane 2
6. Op 2 Start Point

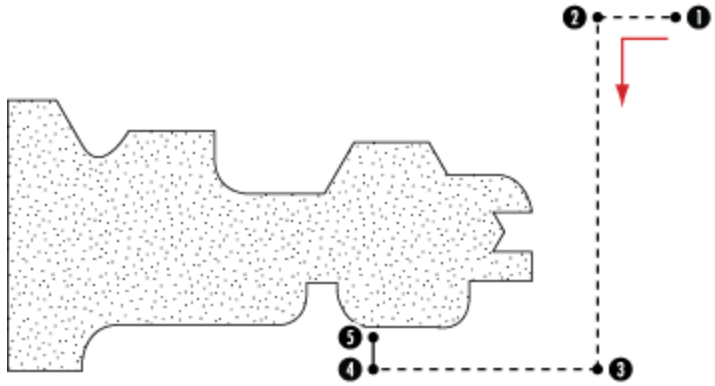
Tool Change Inter-Op Moves

When the next op uses a different tool, the tool will rapid out to the Part Clearance, and then around the Part Clearance as needed until it reaches a suitable exit point. Rapids from the exit point are in X and then Z for maximum safety due to the possibility of a sub spindle being on the part. When the tool arrives at the tool change position, the tool change will take place. The move logic is reversed as the tool moves to the clearance over SP position for the next op.



1. Op 1 End Point
2. Clearance Plane 1 Xd Value
3. Tool Change Position

Moving from operation 1 to the Tool Change Position



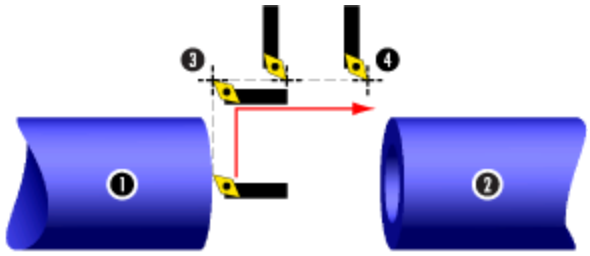
1. Tool Change Position
2. Clearance Plane 1
3. Clearance Plane 1 Z value, Op 2 Clearance Plane 2 Xd value
4. Op 2 Clearance Plane 2
5. Op 2 Start Point

Moving from the Tool Change Position to operation 2

Utility operations have exceptions to this rule as many Utility ops don't move a tool. Utility ops without tools begin before their first effect, and end after their last effect. Utility ops that do move tools (e.g. Load Spindle with a Stop or Position Turret), may have their inter-operation moves checked against the Part Clearance. Other than moving around the part in the event of an intersection, the inter-op moves will simply be “out” (X+) then “over” (Z).

Switching Spindles

When a Tool Group moves from one spindle to another it passes through both the standard tool change location of both spindles. Thus, a tool will retract to the standard tool change location, rapid to the other spindle and then move to the other spindle. If a tool change is required, the tool will change at the first spindle’s tool change location.



1. Spindle 1
2. Spindle 2
3. Standard TC position - tool change occurs here
4. Standard TC position

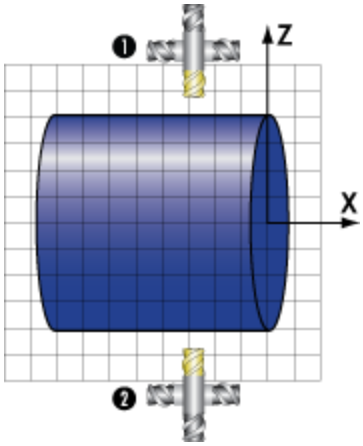
Coordinate Systems and Input Values

The system has two types of coordinate systems. There are part coordinate systems (a set for each spindle) and spindle coordinate systems. The part Coordinate Systems are visible in the Workspace. The only difference between the coordinate systems is the Z position of the origins. MDD values are entered in spindle CS values by the post creators, which never changes with a new part. Part file values are entered in the appropriate part CS, with one notable exception detailed in “Utility Process Data ” on page 38. When entering data into dialogs it is important to understand the orientation of the data to the coordinate system, or the approach axis. All geometry is input in Part Coordinate System values, meaning X+ for top of centerline and X- for below centerline.

Process Dialog Data

Process dialog X values are not entered in part coordinate system polarity, but rather in “tool side” polarity. Since most cutting is done from the X side the tool is on, most processes use X+ values. An X+ location will be on the X+ side for an X+ tool, and on the X- side for an X- side tool.

This convention allows you to change a tool’s cut side or switch the tool to a different Tool Group and have the operation flip X sides without reprocessing. For more details, see “Setting Cut Side” on page 44. Essentially, you can move a tool from the plus side to the minus side and re-post immediately. With “tool side” X polarity for OD work, X+ moves the tool away from the part, X- into the part.



1. + Side Approach
2. - Side Approach

Turning and Milling operations are basically the same.

- Turning processes are input in Tool Side X values. This means X+ is interpreted as being away from centerline towards the tool, regardless of whether the op is being created on the top or bottom side of the part.
- Mill processes interpret depth values with + being towards the tool side.

Utility Process Data

The X values in Utility processes are in Part coordinate system values, not Tool Side, where X- is entered for an X side position. The exceptions are flagged with a small icon (seen to the right) showing a plus arrow to the tool side. This indicates that the specific X value needs to be entered in Tool Side polarity. This only occurs in the Move Tool Group Utility operation and is so that the tool can be moved without reprocessing. Utility operations do not automatically flip with tool redefinition, with one exception. That exception is the Move Tool Group *User Position* X value which is flagged with a “Tool Side” icon. The X value is interpreted as a Tool Side value, and will automatically flip with a tool redefinition.

Item	Spindle Side
Geometry Creation	Part Coordinate System

Item	Spindle Side
Turning Processes	Tool Side
Milling Processes	Tool Side
Utility operations that do not use a tool	Part Coordinate System
Utility operations (except Move Tool Group) that use a tool	Part Coordinate System
Move Tool Group Utility Op, Standard Tool Change	Part Coordinate System
Move Tool Group Utility Op, Machine Home	Part Coordinate System
Move Tool Group Utility Op, User Position	Tool Side

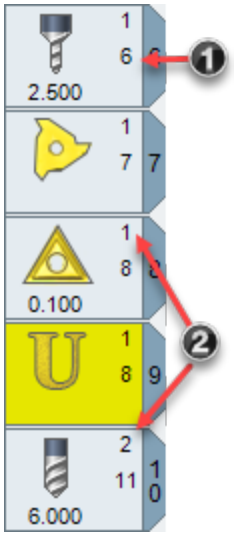
Tool Creation

About Tool Creation

Creating a tool in MTM is almost identical to the Mill or Turning modules. In fact, this chapter will only cover data specific to MTM, because the method and the tools are identical. With the Multi-Task Machining module, however, it is very important to have greater tool definition than in other GibbsCAM products – more information about the tool is needed. The information needed includes which Tool Group (a turret, gang or slide) the tool belongs to, the tool's position within the Tool Group and which spindle the tool is oriented to. These changes are reflected in the Tool list as well as the Tool dialog.

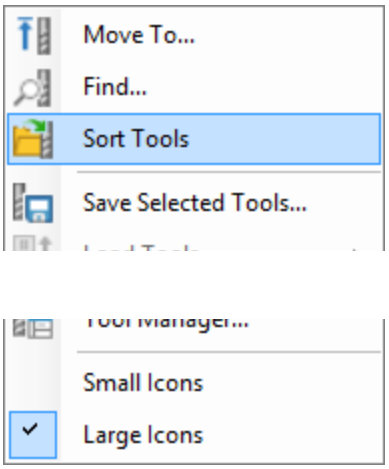
Tool List

The additional data displayed includes the Tool Group number and the tool's position in the Tool Group. Both of these items are set within the Tool dialog and will show up in other tile lists.



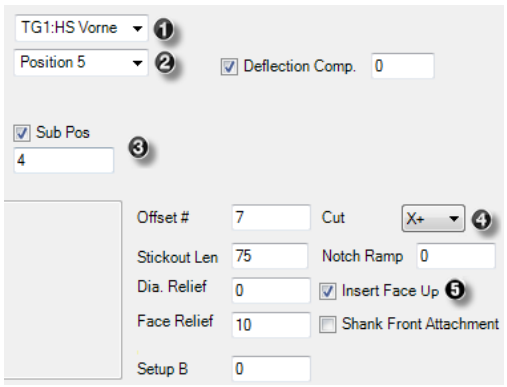
1. Tool Position
2. Tool Group #

It is recommended that tools in the same Tool Group be put together in the tool list and that there be a space separating the Tool Groups. As there will generally be more tools used in MTM parts sorting through tools can be a chore. To aid this, use the Sort Tools command. When selected, tools will be sorted by Tool Group and Tool Groups will be separated by spaces between tiles.



Tool Dialogs

The Milling and Turning Tool dialogs are modified in MTM to include items specific to the Multi-Task Machining option. These items allow the definition of which Tool Group the tool belongs to, the tool's position in the Tool Group, the side of the part the tool will cut on and how the insert is aligned to the spindle.



1. Assigned Tool Group
2. Tool Group Position
3. Sub Position
4. Approach Side
5. Insert Direction

Turning Tool dialog and the MTM specific options.

Assigned Tool Group

This pop-up menu may be found in both Turning and Mill Tool dialogs. There is an entry for each available Tool Group as defined by the current MDD (Machine Definition Document). Select the Tool Group the current tool will be assigned to. The actual name of each Tool Group will depend upon your MDD.

Tool Group Position

This pop-up menu may be found in both Turning and Mill Tool dialogs. There is an entry for each available position in the Tool Group as defined by the current MDD. Select the position in the gang, slide or turret that the tool occupies or will be placed in. Each position may hold more than one tool. This way, you can define a mini-gang of tools at one turret position. Each tool would have a different offset number.

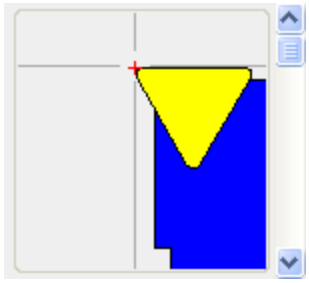
A "Mini Gang" may be created by creating multiple tools that are in the same position, each with different offsets. This can be one tool used with four different offsets or four different tools.

Sub Position
If the MDD allows subpositions to be specified manually for the current toolgroup, then this checkbox and textbox will appear in the Tool dialog.

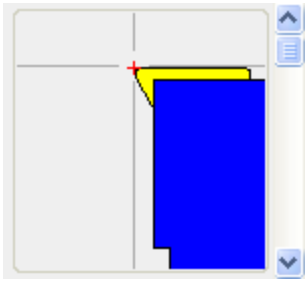
Cut Side
This item is found in Turning tool dialogs. If you are familiar with the GibbsCAM Turning module, you are accustomed to seeing a Clockwise/Counter Clockwise (CW/CCW) option in the Turning Tool dialog. For the MTM option, turning tools do not have this setting but instead have an Approach Side setting that works with the Insert Face Up setting. Together these items specify from which side of the spindle the tool will approach and cut the part, either the X+ or X- side. From this data the system automatically calculates the spindle direction. The actual cut side may be overridden in the Process dialog. See "Insert Orientation" on page 43 for clarification.

Tool ID#
Tool ID # is used to specify a Tool Group position that is serviced by an ancillary mill style tool changer. Enter the tool number you want loaded into a specific Tool Group position. Note that a Tool ID greater than 999 will display on tiles as ##, because tiles are not big enough to display four-digit tool IDs.

Insert Face Up
This item is found in the Turning Tool dialogs. This setting specifies whether an insert is Face Up or Face Down in a tool holder. If this item is unchecked the system assumes the insert is Face Down. The selection made here will toggle the tool display picture to show the insert as Face Up or Face Down. See "Insert Orientation" on page 43 for clarification on insert direction.



Insert Face Up



Insert Face Down

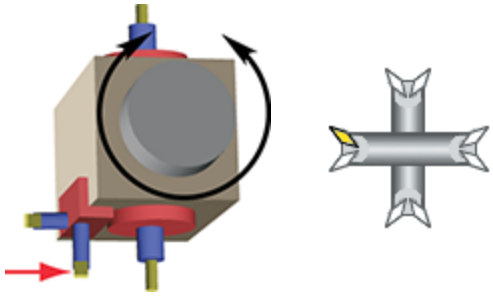
You should visualize a machine setup as if you are standing in front of it. We call this view the Spindle 1 ZX Coordinate System Home View. This view is the same as the trackball "T" or top view for MTM parts. Use this view to determine Cut Side, Insert Face Up and Orientation for all spindles. Do not use the specific spindles' ZX CS Home View.*

*Unless you have an Orientation Spindle selection in the Tool dialog, which is offered for some multi-spindle machines (as determined by their MDDs)

B Rotation
This item allows you to set the B-axis rotation needed to move the current tool to the orientation specified in the tool dialog. This is only available in MDDs for machines that have a B-axis capable

turret with unusual tool arrangements.

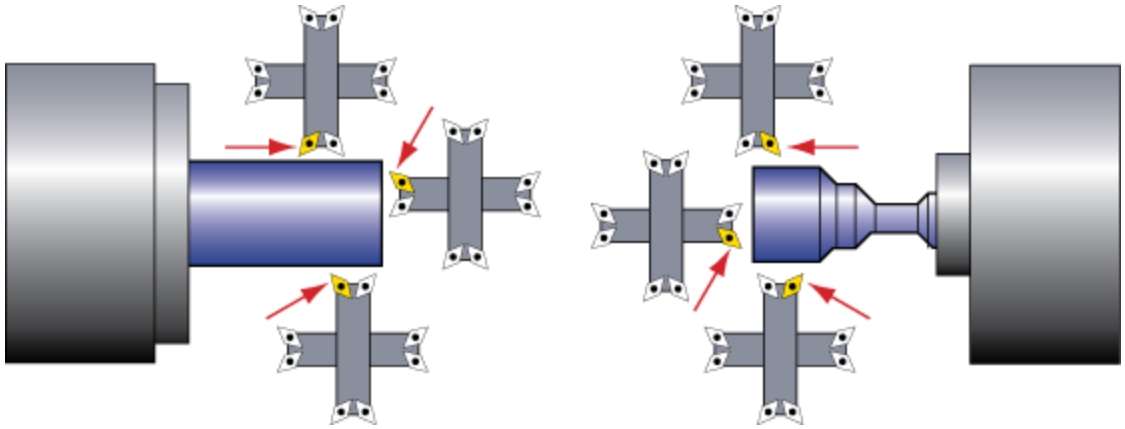
Milling Tools at B0 are pointing at the OD of a part. The B Rotation for a milling tool should be determined solely from its orientation in the machine at the GibbsCAM B0 position. For example, the tool arrangement shown below includes an endmill at B0 and a boring bar that is mounted parallel to the endmill. These tools use the same ID# but different offsets. The boring bar must have a B Rotation of -90 to match the orientation designated in the tool dialog. The Super HiCell has an arrangement similar to this.



A turret set at B0 is the standard turning orientation. Turning tools should be defined as they will be used. First set the tool orientation as it is to be used (not as it is arranged in the turret) and then set the B Rotation required to get the tool into position. Turning tools may be at any arbitrary angle entered in the B Rotation text box. If both tool arrangements shown below were to be used as boring bars, the second tool would need to be rotated by -90° to get into position.

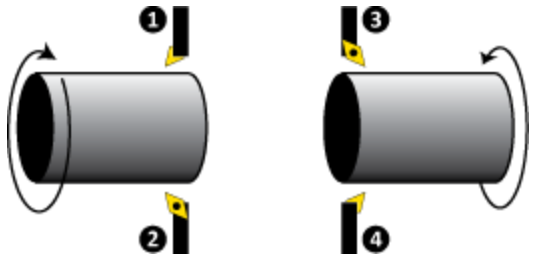
Insert Orientation

The orientation of a tool is very important to properly set up a tool. By default, all tools are oriented to the main or primary spindle. That means that the tool orientation in the tool dialog should be set to match the tool's actual position when looking at the spindles from the front of your machine.



Main Spindle Sub Spindle

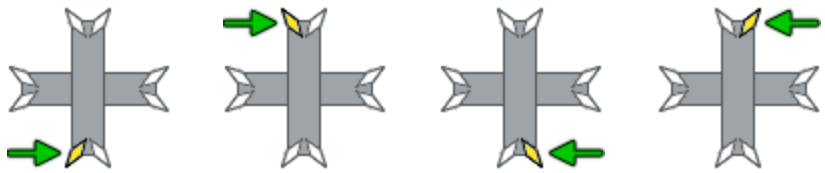
This image shows tools that are oriented to the Main Spindle. The tool's selection in the Tool dialog insert orientation diagram is as shown below, either Face Down or Face Up. The tool orientations are set as they actually appear.



1. Down
2. Up
3. Up
4. Down

Main Spindle Sub Spindle

Tools 1 and 4 are designated as Insert Face Down. Tools 2 and 3 are designated as Insert Face Up.



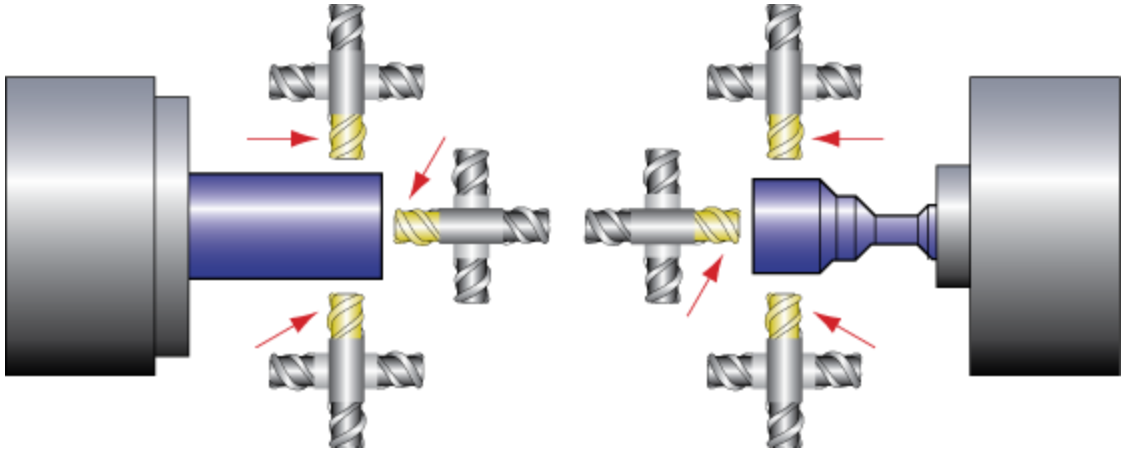
Tool 1 Tool 2 Tool 3 Tool 4

Setting Cut Side

When creating your part, it is possible to set up all of your tools without worrying about the cut side of the tool. This is because toolpath is associative to the tool's definition. If a tool is first set to approach from and cut on the X+ side of the part but then switched to approach and cut from the X- side, the toolpath will automatically update to reflect this change. Changing the tool's Cut Side, Insert Face Up and Insert Orientation settings (as well as a likely change in Tool Group) are required to flip the toolpath. This is illustrated in "Auto-Flip" in the MTM tutorial.

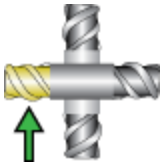
Mill Tool Orientation

The orientation of a mill tool is a very important aspect in properly setting up a tool. By default, all tools are oriented to the main or primary spindle. That means that the tool orientation in the tool dialog should be set to match the tool's actual position when looking at the spindles from the front of your machine.

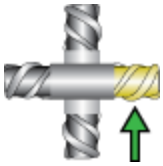


Main Spindle	Sub Spindle

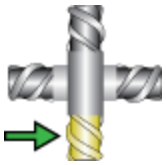
Milling tools are slightly easier to set up than turning tools because there are fewer variations. There are only four positions – two each for horizontal and vertical. Please note that the orientation specified is when the B-axis is zero. For machines with B-axis support, the tool may be used in a range of orientations, and this is controlled by the Machining CS setting found in Process dialogs.



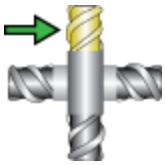
Approach towards the face of the main spindle.



Approach away from the face of the main spindle or back side milling.



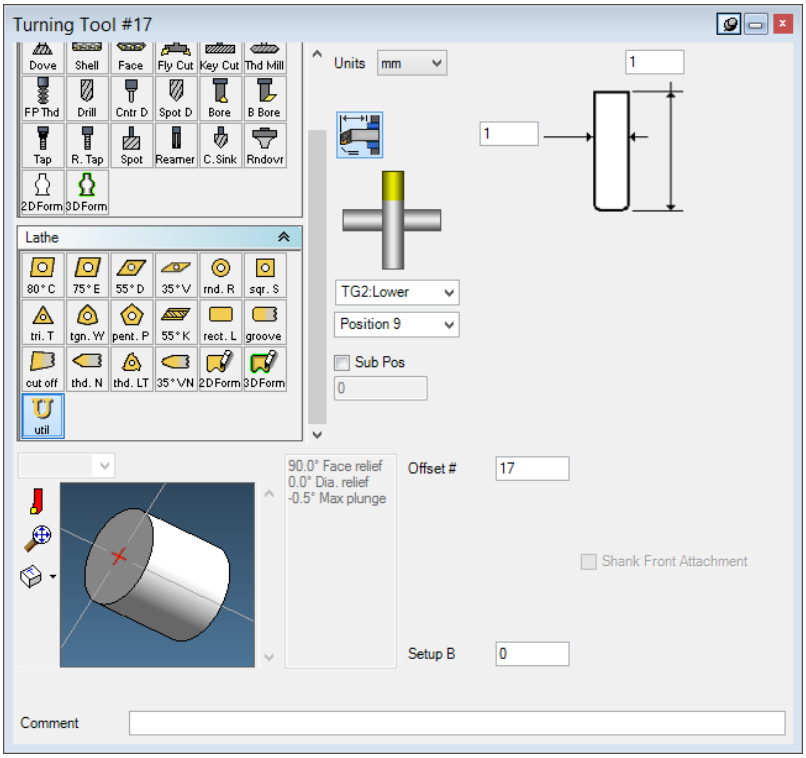
Approach from the X+ side.



Approach from the X- side.

Turning Utility Tools

A special tool type, called a *utility tool*, is sometimes used for utility processes such as move part and move tailstock in/out, which may require a tool position. A utility tool is a non-cutting placeholder. It can represent a stop, probe, part gripper, or dummy tool to help set up the part. Utility tools are used with certain utility processes, depending on your setup. Such a setup might be a bar feed with a stop.


Turning Tool #17
Dove
Shell
Face
Fly Cut
Key Cut
Thd Mill
FP Thd
Drill
Cntr D
Spot D
Bore
B Bore
Tap
R. Tap
Spot
Reamer
C.Sink
Rndovr
2D Form
3D Form
Lathe
80° C
75° E
55° D
35° V
rnd. R
sqr. S
tri. T
tgn. W
pent. P
55° K
rect. L
groove
cut off
thd. N
thd. LT
35° VN
2D Form
3D Form
util
Units
mm
1
1
TG2:Lower
Position 9
Sub Pos
0
90.0° Face relief
0.0° Dia. relief
-0.5° Max plunge
Offset #
17
Shank Front Attachment
Setup B
0
Comment

Processes

This chapter discusses creating processes for parts made with Multi-Task Machining and modifications made to the existing process dialogs for MTM support. As in other modules of the GibbsCAM product line, toolpath is generated by setting up a process with a tool and specifying where on the part the toolpath should be created. All of this is covered in the *Turning* and *Mill* guides, and there is no need to discuss it here as the processes are identical. The major topic of this chapter is Utility processes, which are quite different from regular, toolpath-generating processes. Utility processes generate operations that allow you to control other, non-cutting motions on your machine.

Machining Process Dialogs

Generating MTM processes is no different than defining processes for any other module. Process parameters need to be defined including clearances, speeds, the type of cut to be made and where to perform the cut. Turning operations are always performed in the ZX plane and a machining coordinate system must be selected for milling processes. For MTM there is simply a little more that needs to be defined, such as the spindle that a process will be created on and the side of the part that the tool approaches from.

All Processes

All processes (turning and milling) in the MTM module have a spindle selection pull-down menu. This pull-down menu lets you set which spindle this process will be machined on.

Turning Processes in MTM

Modifications made to turning process dialogs for the MTM module include the display of what side of a spindle the process is to be performed on. There is also an override checkbox to Cut Other Side.

The Cut Side is determined by the Cut Side selected in the Tool dialog. Toolpaths are created on the cut side, unless the Cut Other Side box is checked. We will use an X+ OD tool as an example. Selecting the Cut Other Side item would allow you to use this tool to cut the same OD but on the X- side. The spindle direction will automatically reverse. The Process Cut Side is associative with the Tool Cut Side: if you change a tool's Cut Side, the toolpath changes sides automatically.

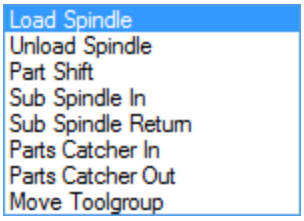
Utility Processes

A Utility process is a process found in the process palette for MTM parts. The Utility process provides the ability to program a number of new operations for loading/unloading spindles, turning spindles on/off, controlling the tail stock, bar feed, and other miscellaneous operations on advanced

lathes. Whether or not a Utility Op needs a tool is defined by the MDD. The system will inform you if the Utility operation requires a tool.

Utility processes may create operations in many or all flows but no more than one operation per flow. These are a multi-op Utility processes and they create post output in all flows to which they belong.

Utility Process Settings



The options found in the Utility process dialog will depend on your machine and MDD. The basic functions include the ability to load and unload a spindle, shift a part in Z, move the sub spindle on and off the part and to control a parts catcher. These items are discussed in detail on the following pages. When a process type (i.e. Load Spindle) has been selected, the icon in the process list will change from a "U" to an icon that reflects the process. Please note that items with the notation ***No Toolpath*** do not generate toolpath and cause the Tool Group to retract to home.

Utility Process Common Items

Spindle

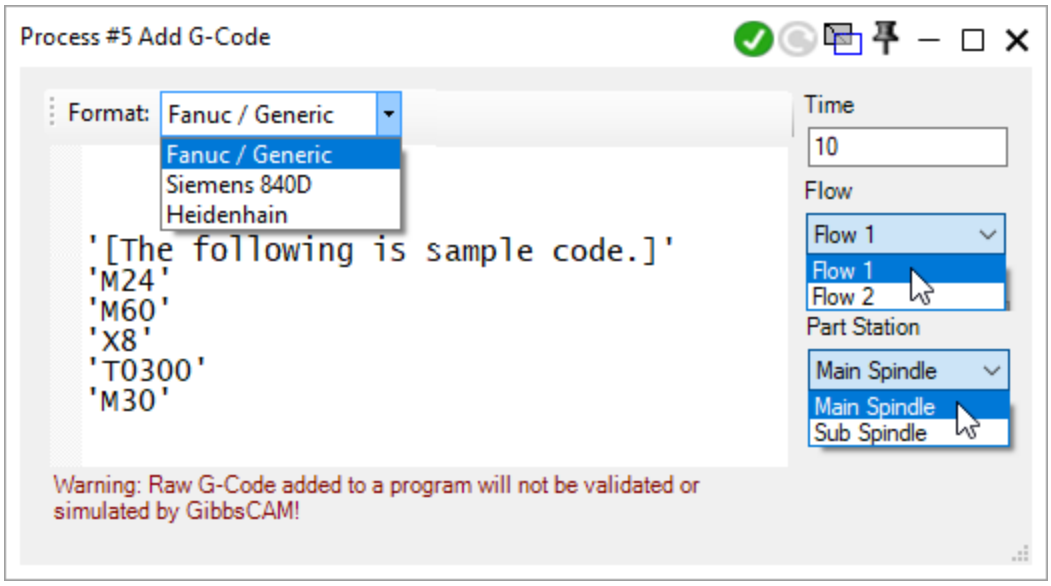
Each process type has a selection for either Spindle or Main Spindle. The actual available choices will depend upon your machine and the MDD. This setting lets you select which spindle the utility process will be performed on.

Main Spindle / Sub Spindle

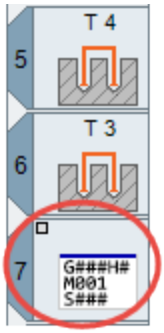
Each process type has a selection for Spindle or Main Spindle. The actual available choices will depend upon your machine and the MDD. This setting is available in utility processes that involve multiple spindles. This setting lets you select which spindle holds the part and which spindle will move to hold the part. The Main Spindle setting is used for the spindle that holds the part, the Sub Spindle setting is used for the spindle that will be moving.

Add G-Code

The Add G-Code utility process allows you to add custom code to your posted output. It can be used with any machine whose MDD supports utility operations. The post processor must also support utility operations.



The data added can either be text or actual code. Select the Custom... option (which is the default option) from the Presets menu and then enter the G-Code you wish to add in the text field. Select the Flow and Part Station to which the G-Code will apply, and specify the estimated Time (in seconds) that the G-Code will take to run.



The Add G-Code utility process creates an operation at the first available slot in the Operation List. You can then reposition the G-Code operation tile as needed to ensure that the code will be placed in the correct position in your posted output. If you need to make changes, you can double-click the process tile to re-open the Add G-Code dialog.

Limitations of this utility include the following:

- The G-Code can be applied to only one flow at a time.
- The G-Code operation will not show up during rendering.

Load Spindle

The Load Spindle process should be created at least once in every part program, either at the beginning or at the end of the operations. The dialog allows you to set which spindle is being loaded, which Tool Group is assisting (if any), how far from the stop should the stock begin feeding, and how fast the stock will be moved. The Type setting specifies the Load method to be used. Only types that are available on your machine will be displayed. See “Part Shift” on page 52 for another Load option.

Load Spindle Type Descriptions

Manual Chuck
This item will pause the program to allow a new piece of material to be loaded into the chuck. Opening and closing the chuck is done by hand. ***No Toolpath***.

Auto Chuck or Collet
This item will pause the program and open the chuck to allow a new piece of material to be loaded into the chuck. Opening and closing the chuck is programmed. ***No Toolpath***.

Bar Feed
This will use a bar stop, (typically a defined utility tool) with a bar feeder to advance the bar stock for a new part. The stop will rapid close to the part face prior to feeding, the collet will open, the bar stop will make a feed move out in Z+ to the new position, and the collet will close. This process requires a Utility tool.

Auto Bar Feed
Stock feeds to a specified Z for a new part. ***No Toolpath***.

Bar Pull
A bar puller, (typically a defined utility tool) works with a bar feeder to feed the bar a specified distance for a new part. The spindle will be stopped. ***No Toolpath***.

Sub Spindle Pull
This operation uses a sub spindle and a bar feeder to advance the bar for a new part. The sub spindle must already be in, and the sub spindle must be unloaded.

Robot
This will use a robot to load a new piece of material into the spindle.

Load Spindle Settings

Feed Distance
Enter the incremental Z distance to feed the stock (part off width + stock)

Feedrate
This is the rate at which the bar puller feeds on and pulls back or the rate at which the bar stop or sub spindle retracts.

Grip Z
This is the Z position the bar puller will feed onto or the position at which the sub spindle will grip the bar. The sub spindle will rapid or feed to this position depending on the MDD.

Initial Face Z
This is the bar face position (in Z) before the bar pull or feed, left by the prior cut off.

Spindle On
This checkbox allows you to specify whether the spindle is spinning ("On") or is stopped. If the spindle is on, you must specify whether the spindle is turning Forward or in Reverse.

Spindle Speed
This is the speed in RPMs at which the spindle is spinning.

Time
This is the amount of time (in seconds) required for the Utility operation.

X Position
Enter the X value for the bar puller or bar stop

Z Clearance
Enter the Z position to which the sub spindle, bar puller or bar stop will rapid. This setting is in part coordinates.

It is important to be sure that your Load Spindle data creates the initial spindle conditions described in the Document Control dialog. Specifically, the distance of the stock face from the spindle face is extremely important.

Unload Spindle

The Unload Spindle process sets a post command to clear a spindle and dictates how to do so. This is typically set once per program. For most setups, a flow and the spindle to unload is selected. Additionally, the Type setting must be selected as appropriate for your machine as well as the rate of travel and Feed Distance. This process is typically used with a Utility tool and may also control a part catcher. See "Sub Spindle In" on page 54 for another Unload option.

Unload Spindle Type Descriptions

Manual Chuck
This option pauses the program to allow for manual removal of a finished part from the chuck or collet. Opening and closing the chuck is done by hand. *No Toolpath*.

Auto Chuck
This option pauses the program and opens the chuck to allow for manual removal of a finished part from the chuck. Opening and Closing the chuck or collet is programmed. *No Toolpath*.

Part Catcher
This option will position a Part Catcher, unload the finished part into the catcher and withdraw the Part Catcher. This option is used when the part drops from the spindle into a Part Catcher. *No Toolpath*.

Part Gripper
This option uses a part gripper to unload the part from the specified spindle. This option will advance the Part Gripper (typically a defined utility tool), stop the spindle, grip the finished part, unload the spindle, and release the part in the specified location. This is frequently used in conjunction with a Part Catcher, which is also programmed by this Utility Op.

Robot
This option will use a robot to unload a finished part from the spindle.

Unload Spindle Settings

Grip Z
This setting is the depth at which the part gripper will hold the part. The gripper will feed from the Z Clearance to this location.

Time
This is the amount of time (in seconds) required for the Utility operation.

X Drop
This setting is the X position from the origin, for the part gripper to rapid to for dropping the part.

X Position
This item is the X setting for the gripper to move to when gripping the part.

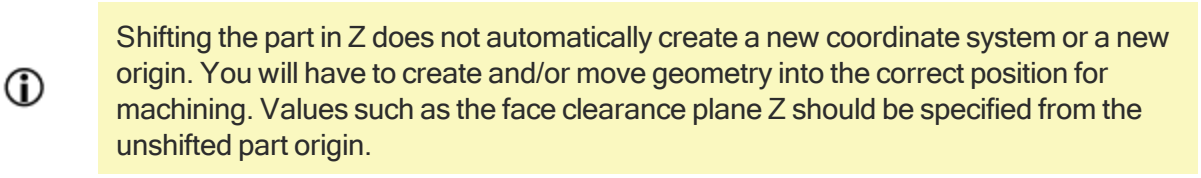
Z Clearance
This setting is the distance from the part origin at which the gripper will stop rapiding and will begin feeding to the part.

Z Drop
This setting is the Z position from the origin, for the part gripper to rapid to for dropping the part.

Z Retract
This setting is the depth to which the gripper will retract when moving the part from the spindle.

Part Shift

The Part Shift process is used to pull the stock out of the chuck by a specified amount to perform work on areas that were inside or blocked by the chuck. Specify the spindle to work from to determine the flow the process will reside in. Additional settings will depend on the method used to perform the part shift.



Shifting the part in Z does not automatically create a new coordinate system or a new origin. You will have to create and/or move geometry into the correct position for machining. Values such as the face clearance plane Z should be specified from the unshifted part origin.

Part Shift Type Descriptions

Manual Chuck
This option will pause the program for the operator to manually shift the part. Opening and closing the chuck is done manually. ***No Toolpath***.

Auto Chuck
This option will pause the program and open the chuck while the operator shifts the part. Opening and Closing the chuck or collet is programmed. ***No Toolpath***.

Bar Feed
This will use a bar stop, (typically a defined utility tool) with a bar feeder to shift the part by the specified amount.

Auto Bar Feed
This item will use the automatic bar feed function to advance the part by the specified amount. ***No Toolpath***.

Bar Pull
This item will use a bar puller, (typically a defined utility tool) and bar feeder to shift the part by the specified amount.

Sub Spindle Pull
The sub spindle will be used to pull the stock out of the chuck and will support the extended stock. The sub spindle will automatically match the Spindle On and Forward/Reverse settings made for this process. The sub spindle should already be in when programming this. See "Part Shift" on page 52 for information on this. ***No Toolpath***.

Robot
This choice will use a robot to shift the part.

Part Shift Settings

Feedrate
If using Bar Feed, it is the rate at which the bar stop will pull back. If using Bar Pull this is the rate at which the puller will feed on and pull back. If using Auto Bar Feed, this is the rate at which the bar will be advanced. The rate is typically set at the auto bar feeder, not in the G-code. This setting will be used to calculate the operation run-time. If using Sub Spindle Pull this is the rate at which the sub spindle will pull back.

Grip Z
This setting is the depth at which the bar puller or sub spindle will hold the part. The bar puller or sub spindle will feed from the Z Clearance to this location. The sub spindle will rapid or feed to this position depending on the MDD customization.

Initial Face Z
This is the bar face position in Z from the origin left by the prior cut off, before the part shift or pull.

Load
If the part is being shifted far enough so that there will be a new length of bar ready to be machined after the part cut off, check the Load box.

Shift Distance
Enter the distance to shift the part in Z.

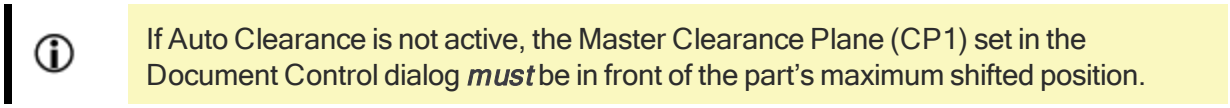
Spindle On
This checkbox allows you to specify whether the spindle is spinning, ("On") or is stopped. If the spindle is on, you must specify whether the spindle is turning Forward or in Reverse.

Spindle Speed
This is the speed in RPMs at which the spindles are spinning.

Time
Enter the time the part shift will take.

X Position
This item is the X setting for the bar stop or bar puller to rapid to.

Z Clearance
This setting is the distance from the part origin at which the bar puller, bar stop or sub spindle will stop rapiding and will begin feeding to the part.



If Auto Clearance is not active, the Master Clearance Plane (CP1) set in the Document Control dialog ***must*** be in front of the part's maximum shifted position.

Sub Spindle In

The Sub Spindle On Part process can be used in several ways including as a support for the primary spindle when machining. Sub Spindle In may also be used as the first step in transferring the part to a sub spindle, typically after a cut off. Specify the Main and Sub spindles (these may be pre-specified by your MDD), the speed of the sub spindle, the Z location to grab the part as well as the distance from the part face that the sub spindle will begin feeding. ***No Toolpath***.

Sub Spindle In Settings

Part in Main
If there is a part in the main spindle prior to moving the sub spindle in, check this box. This is normally activated when moving a part from the main to the sub spindle.

Part in Sub
If there is a part in the sub spindle prior to moving the sub spindle in, check this box. This is only activated with a reverse transfer or if your machine has a "push out" unload.

To perform a "push out," both spindles would be checked as having the part and the Unloads Spindle item should be checked. We assume the part is being unloaded from the sub spindle. The tile will have a special red arrow signifying a spindle unload is part of this Sub Spindle In. This should only be used for machines with this capability.

Spindle On
This checkbox allows you to specify whether the spindle is spinning, ("On") or is stopped. If the spindle is on, you must specify whether the spindle is turning Forward or in Reverse.

C Synced
Activating this checkbox will cause the spindles to synchronize their C-axis prior to a transfer. This will allow for precise positioning of the part for milling operations. This is used in situations such as drilling holes on the Main Spindle then transferring the part and tapping the same holes on the sub spindle. If milling operations are not used or are not related between flows, you may turn this option off.

Unloads Spindle
Selecting this item tells the system that you will be performing a push out unload. If this item is not checked the part will not be unloaded.

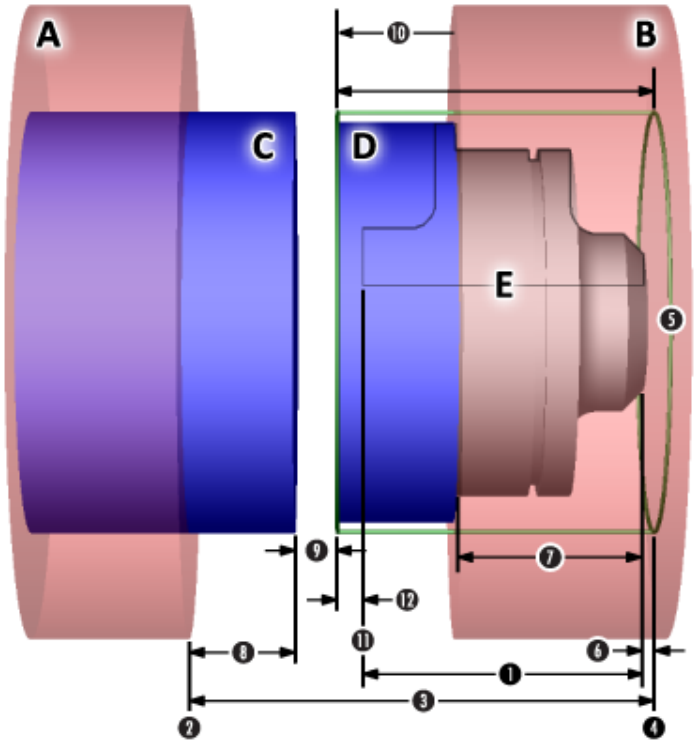
Performing a Push Out Unload requires that Part In Main, Part In Sub and Unloads Spindle are all active. If Unloads Spindle is not selected there is the chance of interference.

Spindle Speed
This is the speed in RPM at which the spindle is spinning.

Z Clearance
This setting is the distance from the part origin at which the sub spindle will stop rapiding and will begin feeding to the main spindle.

Feedrate
This is the sub spindle's feed rate towards the main spindle from the Z Clearance to the Grip Z.

Grip Z
This setting is the depth at which the sub spindle will grip the part. Data entered for Grip Z must match data set in the Document Control dialog. One way to check this is to display the sub spindle geometry on the rendered image of the sub spindle being machined. Any mismatch indicates that Grip Z Data does not match Document Control dialog data.



A. Spindle 1

B. Spindle 2

C. Stock in the Chuck

D. Part - cut and uncut

E. Geometry

1. Part Length
2. Spindle 1 Face
3. Spindle 1 Stock Face
4. Spindle 1 Z 0
5. Stock Face Z
6. Spindle 1 Face Stock Allowance
7. Grip Z
8. Cut Off Clearance
9. Cut Off Width
10. Spindle 2 Stock Face
11. Spindle 2 Z0
12. Spindle 2 Face Stock Allowance

Check the Extras folder of your GibbsCAM product media for an interactive PDF of the above graphic that will help you set up your part's values.

Sub Spindle Return

The Sub Spindle Return process can be used to simply return the sub spindle after supporting a part or it may be used to move the part to the sub spindle. Specify the Main and Sub spindles as well as

the rate of travel. If the spindle is returning with the part be sure to select that option. This will allow rendering to display the stock condition on the sub spindle. ***No Toolpath***.

Sub Spindle Return Settings

Spindle On
This checkbox allows you to specify whether the spindle is spinning, ("On") or is stopped. If the spindle is on, you must specify whether the spindle is turning Forward or in Reverse.

With Part
Check With Part if you want the sub spindle to take the part with it on returning.

Open Main Collet
Check Open Collet if you need to open the main spindle to move or remove the part.

Main Loaded
Check Main Loaded if the main spindle is loaded with stock when the sub spindle returns with the part.

The Double Pull. If you have programmed a Sub Spindle In, a Part Shift, a Cut Off and a Sub Spindle Return, you can have the system easily generate stock in the Main Spindle. Simply activate the Main Loaded option. Be sure that the Part Shift is twice the length of the part. This can be seen in the part file Double Pull.vnc, found with the MTM sample parts.

Spindle Speed
This is the speed in RPMs at which the spindle is spinning.

Feedrate
This is the sub spindle's feed rate when moving away from the main spindle after passing the Clearance Z plane specified in the Utility process that moved the sub spindle in.

Parts Catcher In

The Parts Catcher In process lets you set the command to bring a part catcher in during an unload process. Specify the flow and which spindle the catcher is moving to. This is typically used in conjunction with an Unload Part and a Parts Catcher In process. ***No Toolpath***.

Parts Catcher In Settings

X In Position
This item is the X position from the origin for the part catcher to move to for unloading the part.

Z In Position
This item is the Z position from the origin for the part catcher to move to for unloading the part.

Parts Catcher Out

The Parts Catcher Out process returns the parts catcher. Specify the flow and which spindle the catcher is moving from. This is typically used in conjunction with an Unload Part and a Parts Catcher In process. *No Toolpath*.

Part Catcher Out Settings

X In Position
This item is the X position from the origin for the part catcher to move to for unloading the part.

Z In Position
This item is the Z position from the origin for the part catcher to move to for unloading the part.

Move Tool Group (MTG)

Move Tool Group (MTG) is a Utility process that provides non-cutting positioning control over a tool and its Tool Group. Typical uses are to move the tool to a location at which you plan to wait (with a sync), temporarily change the location at which you want to change tools, or create a sequence of moves to avoid an obstacle. A Move Tool Group operation (MTG operation) overrides a following tool change location with its position. *No Toolpath*.

Concepts

Tool Change Locations
All tool change locations may be categorized as either a “standard tool change location” or an “MTG tool change location”. The standard tool change position is defined by the MDD. This is potentially either Machine Home, a fixed position, or a user-defined position in the Document Control dialog, all of which are typically somewhere “off part”. Conditions that trigger the automatic move to a Tool Change Location are:

- Tool position change (example: indexing a turret)
- Tool offset change, or causing offsets to be activated
- Op Program Stop
- Program Start at Home, before first operation
- Spindle change (tool goes from S1 to S2)
- Tool ID# change in the target position (tool ID#s are used with Mill style tool changers)
- B-Axis rotations

Users should make the standard tool change position the location that will be the most useful – in other words, the location that is used most of the time. The MTG utility process will be used for special conditions or overrides.

Move Tool Group Locations

A tool will go to a Move Tool Group (MTG) tool change location if the operation occurs after an MTG operation. MTG operations simply move a tool to a specified location. The user may specify this location to be with or without tool offsets, depending on the settings used in creating the operation. MTG will typically be used to take the tool to an off part location. The term “off part location” refers to a tool position outside the normal clearances around the part being machined.

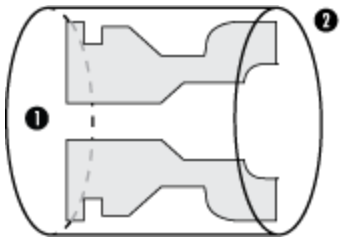
The MTG Utility operation allows you to select from three location options:

1. Machine Home location
2. The Standard Tool Change Location for this part
3. X and Z location of your choice

What the Move Tool Group Operation Can Do

Off Part

An MTG operation can move a tool to a location “off part.” Any location inside of the stock boundary is referred to as “On Part.” If Auto Clearance is not used, On Part is considered anywhere within the Clearance settings in the Document Control dialog.



1. On Part
2. Off Part

MTM will automatically move a tool to an Off Part Location when any one of several conditions occurs, such as a tool change. This automatic positioning move occurs between operations. MTM will move the tool to the Standard Tool Change Location unless the preceding operation is a Move Tool Group operation. If this is the case, the tool will not move to the standard tool change position but will remain at the MTG location. MTM does not need to move the tool “off part” if the programmer has already done so with an MTG op. All tool change location events will cancel tool offsets on their way to the tool change location.

A tool is moved off part when any of the following items occur:

• Tool position change (example: indexing a turret)	• Tool offset change, or causing offsets to be activated
• Spindle change (tool goes from Spindle 1 to Spindle 2)	• Program Start at Home (before the first operation)
• No Path Utility operations	• B-Axis rotations
• Op Program Stop	• Tool ID# change in the target position

Override Next Tool Change Location

MTG operations override the next tool change location, including the starting position of a Tool Group if an MTG operation is the last operation in a flow.

> ⓘ A very important detail to remember is the condition of Tool Groups at the start and end of a program. At the start of a program, Tool Groups are expected to be at their tool change location. At the end of a program, all first tools are put into play and the Tool Group goes to the tool change position.

Eliminate Subsequent "No Toolpath" Utility Operation

An MTG operation will eliminate a subsequent "No Toolpath" Utility operation that would send a tool to the Tool Change Position. For example, a Utility operation of type Sub Spindle In sends the Tool Group home.

If you are unsure whether a particular Utility operation, or subtype within a Utility operation, does or does not generate toolpath, refer to the documentation for that Utility process.

Move Tool Group Settings

Standard Tool Change Location
Selecting this item will move the Tool Group to its standard position for a tool change.

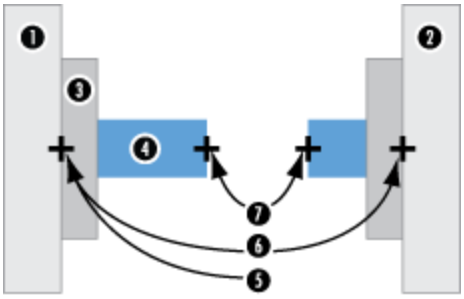
New Location
This item allows for moving the Tool Group to a position other than its standard tool change location.

Xr / Z
For both the Xr and Z positions, you can send the Tool Group to its Machine Home position or a position specified by User. Once a position has been selected the reference coordinate system (CS) and Control Point settings must then be selected.

- Machine Home is defined as the origin of the Tool Group coordinate system, (which is defined relative to the Spindle 1 Machine Home) and each turret has a single home position. This preset is a different location for every Tool Group on the machine. The position is defined by your machine and is known to the MDD.
- User can be set any value relative to part or spindle coordinates.

CS Selection
These options are available only when you choose Xr option User. You can specify whether the new location is based upon the ZX axis of the part or upon the ZX coordinate system of the spindle.

- The Part ZX choice refers to values measured from the part origin as defined in the Document Control dialog.
- The Spindle ZX choice refers to values measured from the Machine Home, which is the spindle origin (the face) of chosen spindle.



1. Spindle 1
2. Spindle 2
3. Chuck or Collet
4. Stock
5. Machine Origin
6. Machine Home
7. User

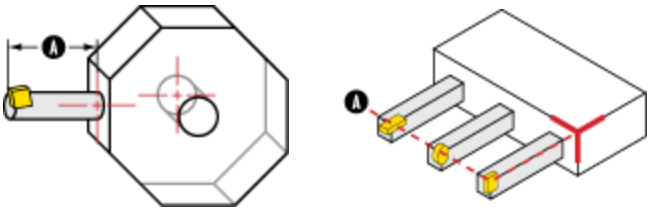
Control Point Settings

These options are available only when you choose Z option User. You can specify whether the Tool Group control point (the TG Datum choice), or whether the tool tip goes to the specified location (the Tool Tip choice).

Tool Group Datum

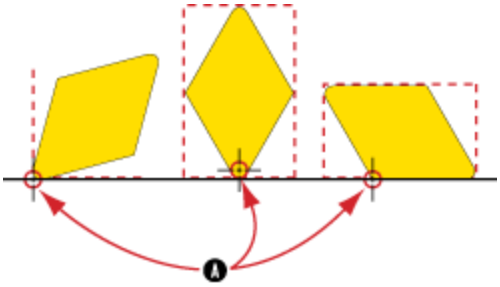
A single point per Tool Group that can be represented or specified regardless of tool position. The specified point is what will go to the tool change position. Offsets are measured from this point.

The following images illustrate possible Tool Group Datum positions. The Tool Group datum point for a turret might be on either side of the center of the turret, or at the base of the tool holder. The datum for a gang might be at a specific corner. This option is recommended when the Tool Group will be sent far from the part. The offset (labelled as “A”) for each tool is measured from this point. Be sure that the Tool Group is far enough from the part to ensure that there is no interference between the holders and the part. Offsets are canceled when using this option.



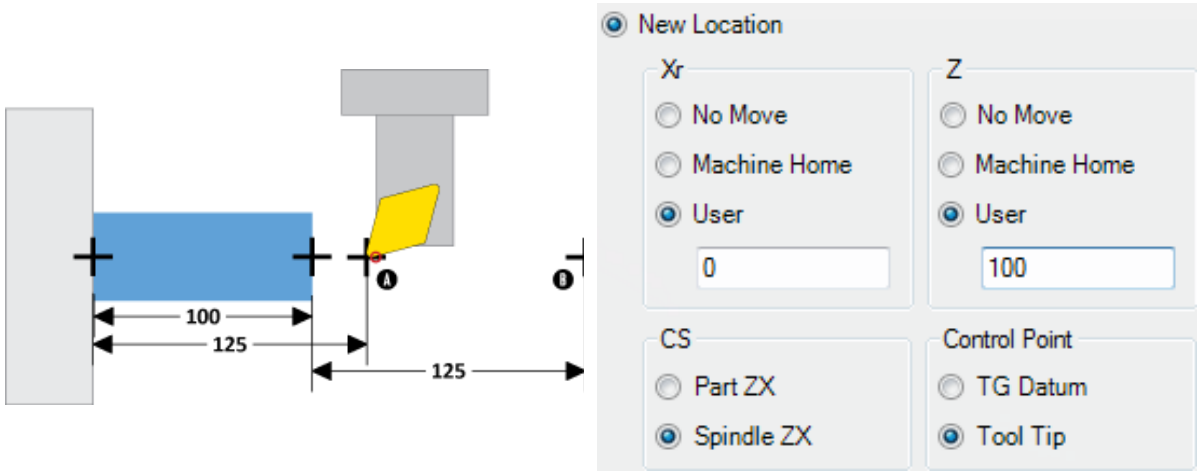
Tool Tip

If Tool Tip is selected, the Tool Group will retract so the current tool’s touch off point (labeled as “A”) is at the specified location. This is the recommended option if you are working near the part as there is little chance of interference. If the location you have selected is very near the part you should be aware of what tool is being used as this setting can over-travel.



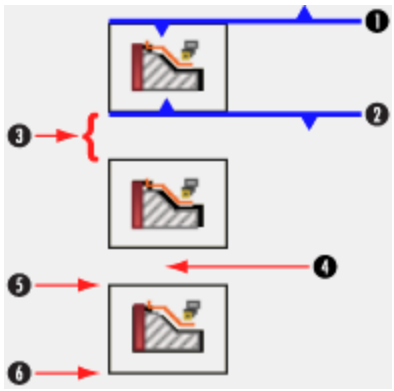
The following image is an example of the difference between selecting Spindle ZX or Part ZX when Control Point is set to Tool Tip. There is a part that is 100mm long with the origin at the part face. The Move Tool Group process is set to retract by 125mm in Z along the Spindle ZX axis and the control point is the tool tip. The net result is that the tool will be 25mm from the part face

at the point labeled “A”. If Part ZX was selected as the reference coordinate system the tool would have retracted further in Z, to the point labeled “B”.



Standard Interactions Around Operations

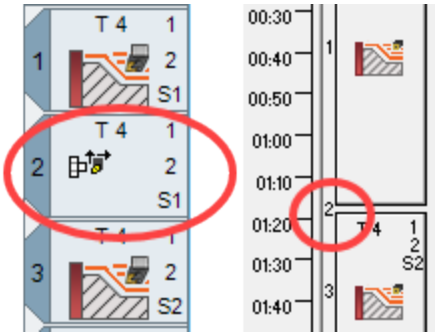
Using the full potential of the Move Tool Group process requires an understanding of the typical interactions of processes and tool movements. The following graphic shows the normal interactions of processes and other movements on operations.



1. Sync at Start
2. Sync at End
3. Inter Operation Moves
4. Tool Change
5. Op Start = CP 1 Start Point or CP3 for ops that are the same Tool on the same Side
6. Op End = CP 1 End Point or CP3 for ops that are the same Tool on the same Side

- Syncs are at the start or end of an operation, at the maximum clear position. The maximum clear position is the position farthest away from the part given the operation. This may be Machine Home, the Master Clearance plane, or the Exit Clearance plane.
- At the start and end of operations, tools are located over the start or end point at a clearance value, typically CP1. If two consecutive operations are on the same side of the part using the same tool, the tool will be at CP3.
- Inter-operation moves (such as tool moves) occupy the time between ops and are represented by gaps in the Sync Control dialog.
- Tool Changes take place between operations as well. The Move Tool Group Process allows you to create an operation that will provide you with alternatives to this.

Using the Move Tool Group Process



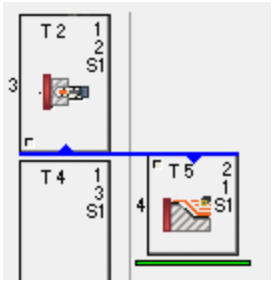
The Move Tool Group (MTG) process creates an operation that is zero seconds in length and does not generate toolpath. It can be thought of as a synching operation that lets you tell the Tool Group to move. When the Sync Control dialog is in Uniform View, you will clearly see the operation, but when in normal view (time-based), an operation tile will not be visible, but the operation number will be.

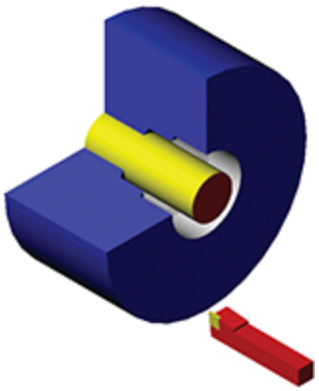
MTG is a “No Toolpath” operation. Like other “No Toolpath” operations trigger a Tool Group retract. When a No Toolpath operation follows an MTG operation, the MTG process overrides the retract. If you are unsure whether a particular Utility operation, or subtype within a Utility operation, does or does not generate toolpath, refer to the documentation for that Utility process.

Examples of MTG Use

- *I want to have a tool wait close to the part*

  You can have a tool wait close to the part for another tool to begin or finish cutting. This is the default behavior in MTM when a sync is set. Other than setting the sync, there is no work needed to accomplish this. In the following image, a tool that will face the part is waiting for a set of drilling operations to finish.

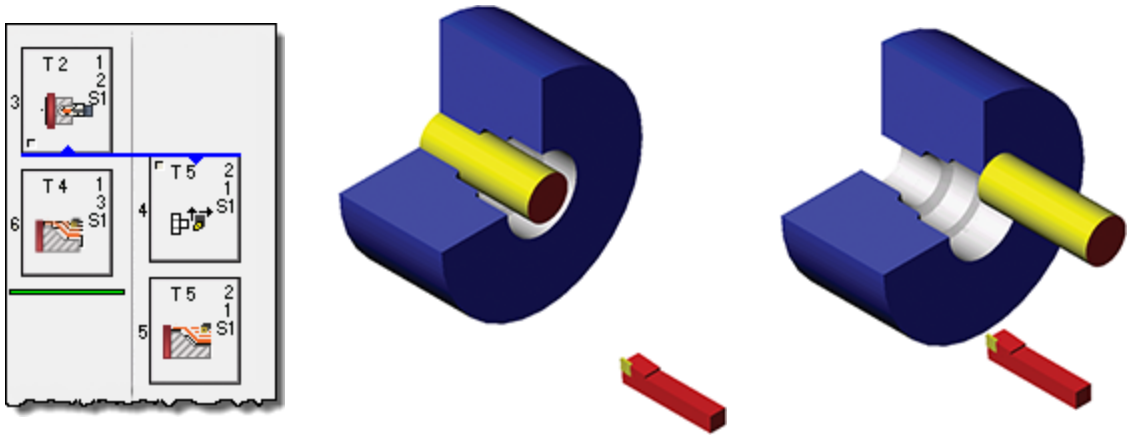




- *I want to have a tool wait away from the part*

  You can have a tool wait away from the part for another tool to begin or finish cutting, providing a greater amount of clearance and safety. This requires a Move Tool Group operation. Create a MTG process using the tool you wish to have wait and enter the position at which you want the tool to wait. Place the MTG operation before the operation that is to wait. Sync the start of the MTG operation to the end of the prior operation. You will need to have the Sync Control dialog in Uniform View mode to sync the MTG operation as it takes up no time and will only appear as a number in the normal, time-based view. The following example shows a synced MTG operation

and the tool waiting away from the part. Once the drilling operation has finished and the tool is retracting the insert will move in to face the part.

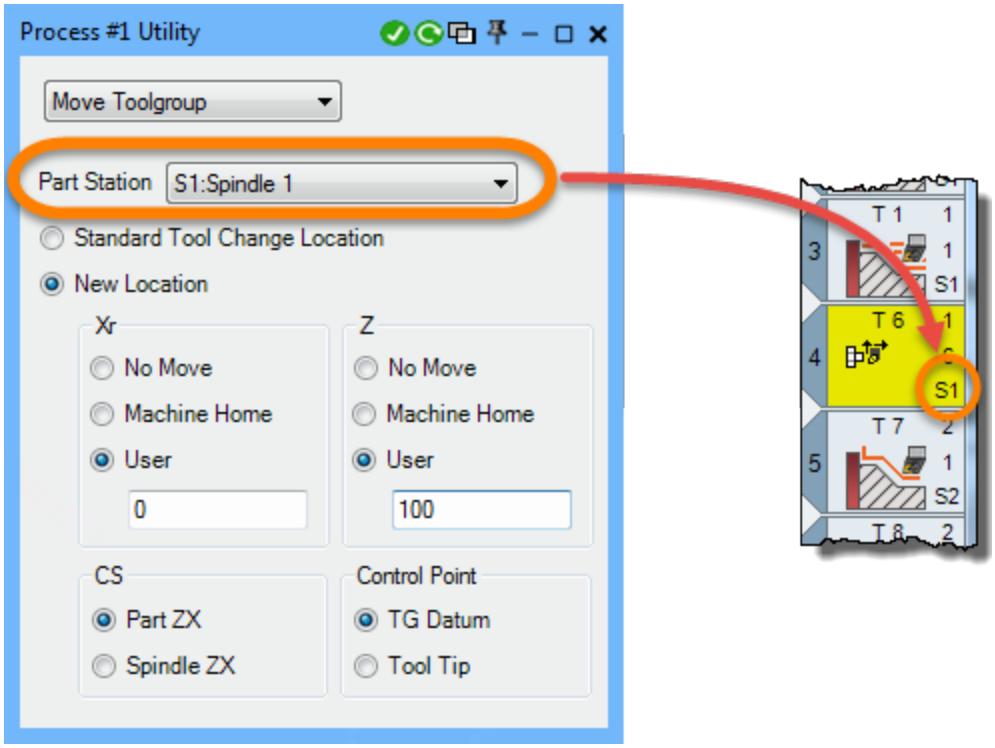


- *I want to override a tool change position*

  Normally, you do not need to concern yourself with a tool change. The system handles the retract to the tool change position, which is specific to your machine and is known to the system. But if you want to override the system behavior, the MTG Utility operation allows you to do so.

  Create the operations. Then, add an MTG operation that is assigned the tool that will be deselected, not the tool that will be used next. The MTG operation should be placed *before* the operation that requires a tool change – remember that a MTG operation affects the operation that *follows* it.

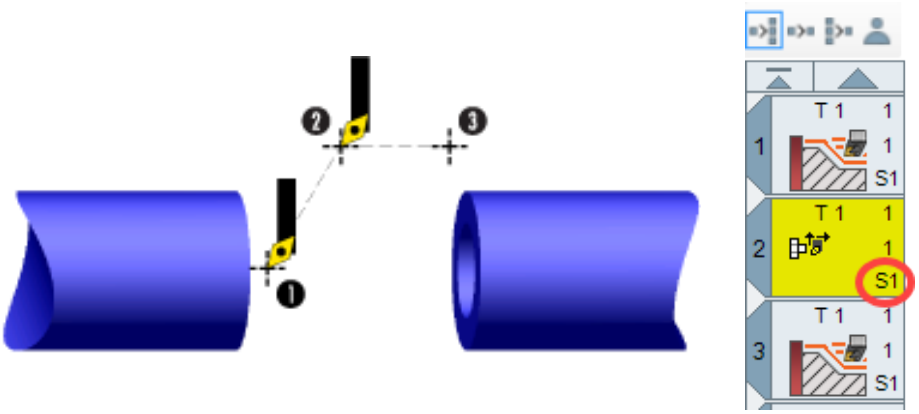
  In the example below, a drilling operation is followed by a roughing operation. Rather than have the drill retract to the normal tool change position, there are User values for `Xr` and `Z` – in other words, the tool will move out by about 100mm, to `Z100 Xr0`, and perform the tool change there.



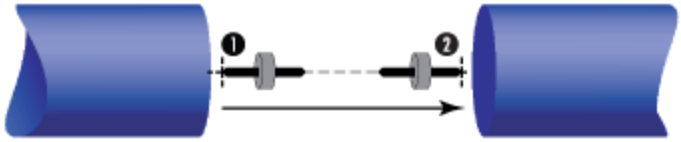
If you were to use Tool 2 in the MTG operation you would get a very different result. The drill would retract to the tool change position, the insert would be put into play, the Tool Group would then move to Z100 Xr0 and then move in to the part.

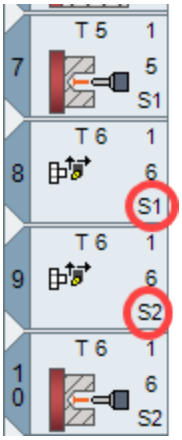
- *I want to change the locations a tool passes through when moving to a different spindle*

  As stated in "Machining Overview " on page 30, when a turret performs a spindle change, the tool will retract and pass through the standard tool change position of both spindles. You can override this using two MTG operations. One MTG operation must be created for each spindle. The MTG ops must specify the points the tool will pass through. If only one MTG operation is made, such as a custom location for Spindle 1, the tool will move to the standard tool change location, and then move in to cut on spindle 2. See the part file "MTG Comparison.vnc" for examples of this functionality.



1. Spindle 1 Custom TC position
2. Standard TC location
3. Spindle 2 Entry Clearance

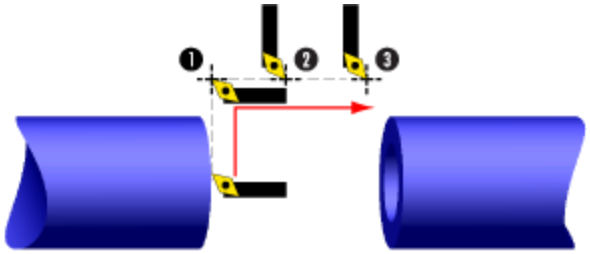




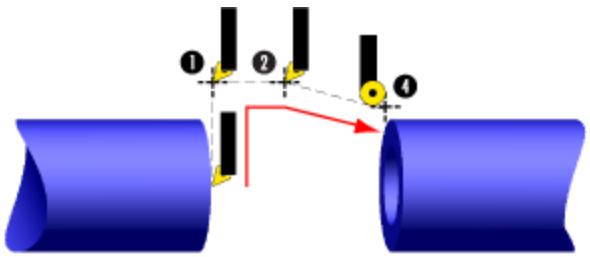
1. Spindle 1 Custom TC position
2. Spindle 2 Custom TC position

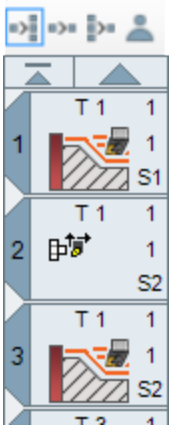
- *I want to delay the tool change when moving the turret to a different spindle*

  If a tool change occurs when moving a turret to a different spindle, the tool change normally occurs at the standard tool change location. To create an override that will change tools at the second spindle, create an MTG operation. The MTG operation should position the first tool somewhere around the second spindle. This will cause the tool to retract from the first spindle, move to the standard tool change position, then move to the position you designate in the MTG process.



1. Spindle 1 Exit Position
2. Standard Tool Change Position
3. Spindle 2 Entry Position
4. Spindle 2 Custom Tool Change Position





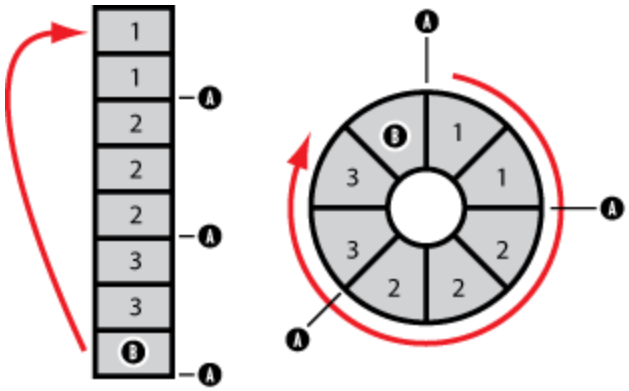
Normal tool change at top, Custom shown at the bottom.

- *I want to start and end at Machine Home*

At Program Start all Tool Groups go to the standard tool change position as defined by the MDD. This is potentially either Machine Home, a fixed position, or a user defined position in the Document Control dialog. At Program End, all Tool Groups go back to the standard tool change position and the first tool is made active. By doing this, the machine is prepared for the next run of the program as programs are essentially loops.

Please note that G28s are supported, the tool goes to the standard position. However, G30s are not supported.

Because programs are loops and are generally made to cut more than one part, programs can be thought of as being circular. Instead of starting and ending your program at the standard TC location, you can have MTG start the program at Machine Home. Simply create an MTG operation set to Machine Home at the end of your operation list. Be sure that the MTG operation is assigned the same tool that is used in the last operation.

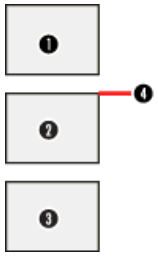


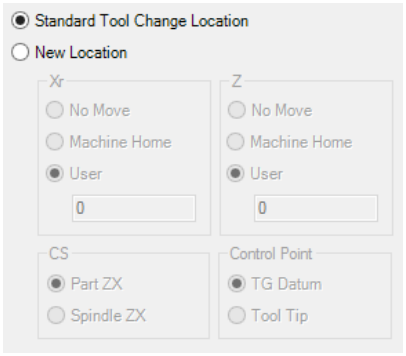
You will not get the same result if you place the MTG operation at Operation 1, the beginning of the operations list. Instead of starting and ending the program at Machine Home, the program will start at the standard tool change position, move to the MTG position (Machine Home), and then move to the first cutting operation.

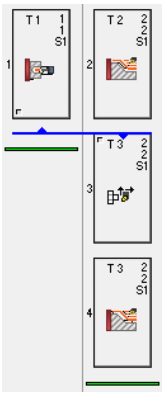
Various MTG Results

Here we look at various results when making an MTG operation.

- Move a tool to the standard tool change position, perform a tool change and wait at the tool change position until the next operation. This may be used to force a tool to wait away from the part until an operation in another flow finishes.

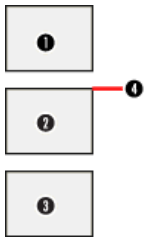


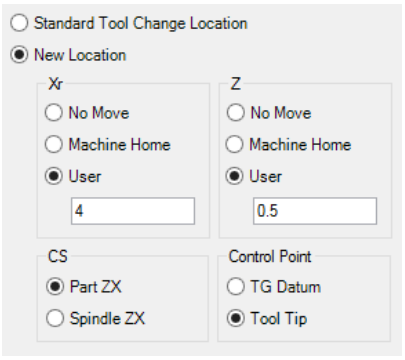


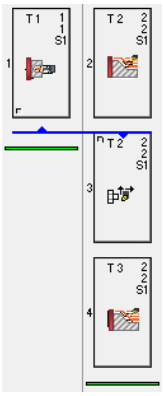


1. 1st Op, Tool 1
2. MTG, Tool 2
3. 2nd Op, Tool 2
4. Sync start to other flow

- Move a tool to a specified tool change position, perform a tool change and wait at the specified tool change position until the next operation. Similar to above but the tool changes and waits near the part.

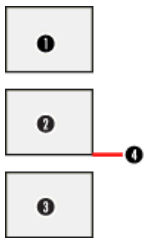


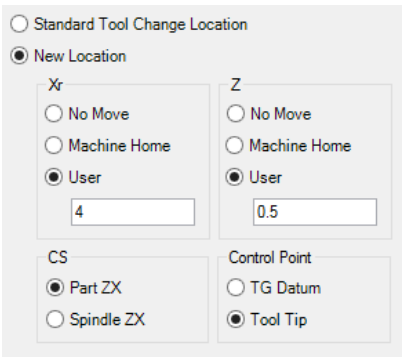


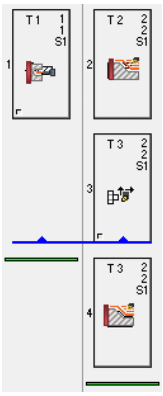


1. 1st Op, Tool 1
2. MTG, Tool 1
3. 2nd Op, Tool 2
4. Sync start to other flow

- Move a tool to a specified tool change position, perform a tool change, move the tool to wait near the part for the following operation to begin. Similar to above except that the second tool will move into its entry clearance position as soon as the tool change is finished.







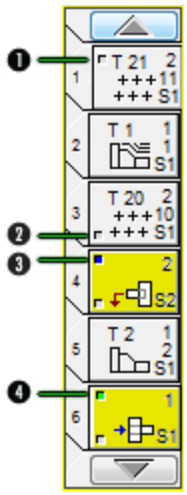
1. 1st Op, Tool 1
2. MTG, Tool 2
3. 2nd Op, Tool 2
4. Sync end to other flow

Operations

As in other modules in the GibbsCAM product line, MTM operations, or toolpath, are created by generating processes. A turning operation in the Turning module is really no different from a turning operation in the MTM module except that the turning operation in MTM may be cutting on the -X side of the part or on another spindle. Additionally, MTM operations may be synched to another operation. Standard Turning parts have only one flow of tools to worry about. The order in which you see the operations in the Operations list is the order in which they will be used. This is basically still true with MTM but there are more flows to be concerned with and the timing of the operations is essential. The main focus of this chapter is to discuss synching operations to help set the timing of an MTM part.

Operations List

Operation tiles in Multi-Task machining are different from the base GibbsCAM products. In addition to displaying the Flow Number (typically the same as the TG#), Tool Number, Tool Group Position and Spindle being used in the operation, any and all Syncs are shown on the tiles.



1. Sync at start of the op
2. Sync at end of the op
3. Master Utility operation
4. Slave Utility operation

Syncs are displayed as boxes in the top left or bottom left corners of tiles and they may be colored.

Uncolored Boxes

Boxes that are uncolored are Op syncs, Stroke syncs, or Utility Data. Uncolored syncs at the top left corner are syncs at the start of the operation while uncolored syncs at the bottom left of a tile are syncs at the end of the operation.

Utility Ops

Utility operations may have syncs in both corners. This is because many Utility processes create two syncs, one for the start of the op and one for the end of the op. These are System Syncs. The box at the top of a system sync is either blue or green. Blue boxes state that the operation is a Master while a green box signifies that the operation is a Slave.

Utility processes often create multiple operations, signifying that the process spans multiple flows, one operation being in each flow. Whenever a process creates operations in more than one flow, one of the operations is flagged as the master. The master operation contains information such as speeds and clearances for all of its slave operations. For more information see "Utility Data" on page 78.

Sort Ops

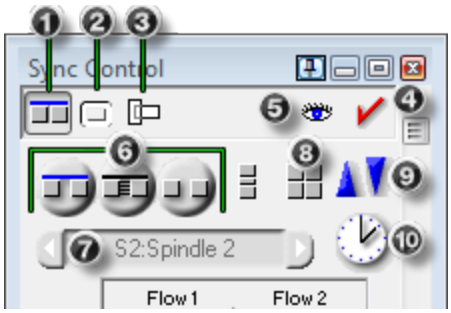
The Sort Ops function in MTM sorts operations by flow and separates individual flows with a space in the Operations list. The operations within the flows are also sorted by their order within the flow.

Operation Synchronization

Multi-Task Machining lets you manage the run-time ordering of operations on multiple turrets and multiple spindles by setting constraints on the start or end of operations. This includes functions for creating, modifying and erasing synchronization constraints between any numbers of operations from different Tool Groups. This is accomplished through the Sync Control dialog. All sync information set in the Sync Control dialog will be used, displayed and stored by the cut part rendering, post and optional Machine Simulation to provide the correct run-time information about the program to the user.

Sync Control Dialog

When it comes to synching and managing operations, the Sync Control dialog is your best friend. This simple dialog is very powerful. It displays operation tiles sized according to their individual run-times in the order in which they will be executed. The Sync Control dialog allows you to add Op and Stroke syncs to your operations, edit the syncs, recalculate cut times on one spindle to match another, and easily modify when the operation will take place. Selecting an operation in the Sync Control Dialog also selects the appropriate tile in the op list and op manager, and updates the on-screen toolpath appropriately. In addition, the op data and utility markers dialogs are also updated any time the selected operation changes, regardless of how it was changed. The dialog lists all available flows and the operations that are in the flows. The operations may be selected and modified depending on the mode you are currently in. The dialog also displays the current run-time calculation. The run-time will automatically be updated with all modifications you make. .



1. Sync Mode
2. Op Mode
3. Spindle Mode
4. Sync Controls
5. Spindle Select
6. Checker
7. Uniform Tile
8. Zoom Time
9. Recalculate

Elements of the Sync Control dialog

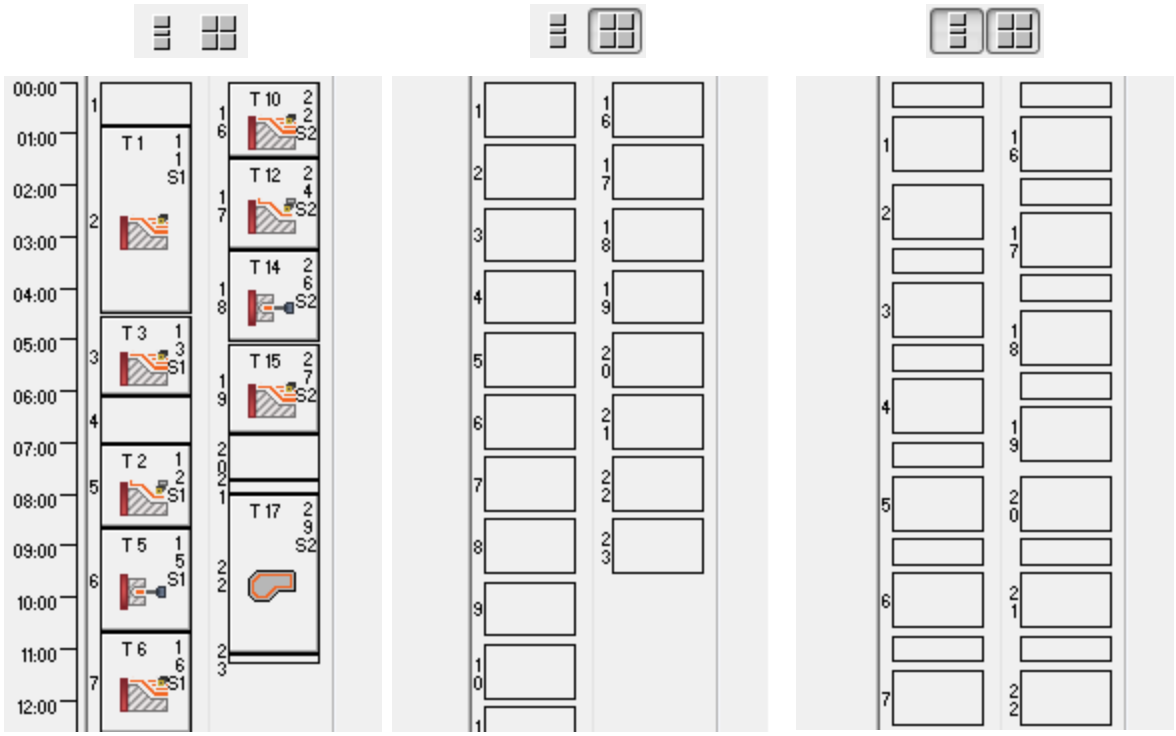
One of the more complex aspects of some MTM machines is that they can do more than one thing at the same time. These machines are literally running multiple G-Code programs at the same time. The sync dialog graphically displays this for you. A Flow is usually one Tool Group; more specifically, it is the program for one turret. The length of an operation is its run-time. The gap between operation tiles is the time of the inter-operation moves.

An operation begins with the tool over its cutting start point, at a clearance position. An operation ends with its tool over its end point at a clearance position. The clearance position can be specified, or you can have it automatically calculated by the Auto Clearance option in the Document Control dialog, just as in the standard Turning module. The inter-operation moves will include the moves to a tool change position and tool change tile as well as all rapid moves.

Uniform Tile View and Interop Blocks

The Uniform Tile view button switches the view between a real time scale and equal sized tiles. This view can be useful when you have very small or fast operations.

When Uniform Tile view is in effect, the Show Interop Blocks button switches on and off the display of additional blocks when events occur between operations. Events of this kind, such as tool changes, are called Interop Events.



Modes

At the top of the Sync Control dialog are three buttons, the Sync Mode, Op Mode and Spindle Mode buttons. These buttons toggle the dialog between three states. The content of the dialog does not change but the appearance and functions of the dialog change with the mode. Each mode offers a

different tile selection and functionality. Sync mode allows you to create and delete syncs, causing ops to wait for other ops. Op Mode provides a real-time view of your Operation list for program editing and reorganization. Spindle mode lets you select which ops control the spindles. Each mode is detailed below.

Sync Mode

A “sync” is an instruction in the G-code program telling one flow to wait for another. In practice, all flows synced together will wait for the last flow to arrive. For example, a twin turn machine has two turrets/Tool Groups and one flow for each of them. If we sync two operations together, the first to arrive at the sync instruction in the G-code will wait for the other TG to arrive at its corresponding sync point in its G-code flow. This is graphically displayed for you in the Sync dialog.

Tile Selection

In Sync Mode, the top or bottom halves of a tile are selectable. You may only make one selection per flow. Multiple selection in the same flow is disabled. The top half of a tile selection represents the start of the operation. The bottom half of a tile selection represents the end of the operation.

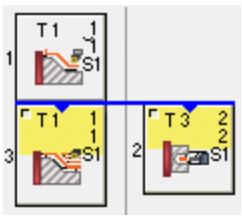
In Sync mode the Sync Control dialog is used to add and edit Op Syncs and Stroke Syncs. Syncs are set using the Sync Controls. The Sync Controls consist of three buttons, one for adding Op Syncs, one for adding Stroke Syncs and one for clearing syncs.

1. Add Op Sync
2. Add Stroke Sync
3. Clear Sync

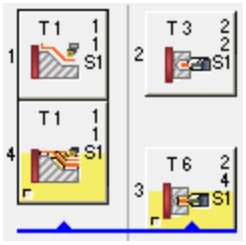
Op Sync

Operation Syncs (Op Syncs) let you synchronize operations in multiple flows. You may “sync” the start or end of any operation to the start or end of an operation in another flow. In fact, you may sync as many operations as you have flows.

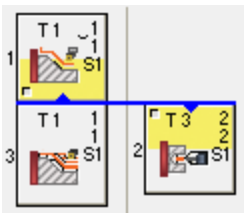
To set a sync, select the ops and click the Op Sync button. When you have made your selection, a blue line with pointers will be drawn, linking the operations. The small triangle pointers on the blue line indicate whether the sync is to the start of the following op or the end of the prior op. All run-time calculations are automatically updated, and the tiles will shift position. It is that simple.



Op Start / Op Start



Op End / Op End

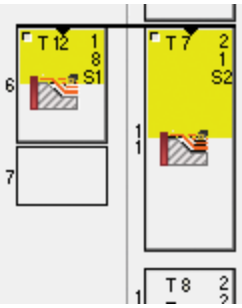


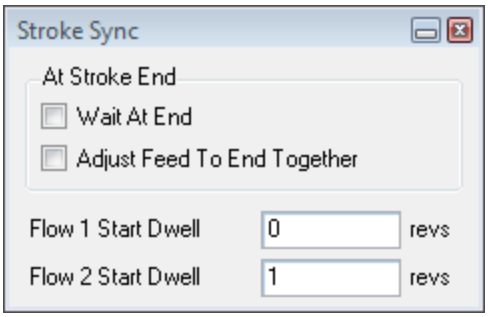
Op End / Op Start

To clear an Op Sync, select the side of the operation that it is synced to and click the Clear Sync button. You do not have to select all the ops involved to clear a sync.

Stroke Sync

Stroke Syncs let you synchronize the individual strokes of operations in multiple flows. When the operations are selected and the Stroke sync button is clicked, a black line is drawn, linking the operations and the Stroke Sync dialog opens. Stroke Syncs are always applied to the start of operations no matter how you select them. This causes the operations to start each stroke together. Additionally, you can set the behavior of the operations through the following options. To open the Stroke Sync dialog after it has been set, right-click one of the black arrows.





Wait at End

When selected, whichever tool gets to the end of a stroke first will wait at the end of a stroke for the other tool to get to its end position. The tools will retract and position to the start of the next stroke together.

Adjust Feed To End Together

The Feed rate of the operations will be adjusted so that the tools maintain the dwells specified.

Neither Wait at End nor Adjust To Feed Together

The tools will simply start each stroke at the same time.

Both Wait at End and Adjust Feed To End Together

The tools will start and end each stroke at the same time and will feed at the same rate. While this may seem redundant, it is not if a dwell was applied to one or both operations or one operation is simply much longer than the other. This way the operations will always end together, even if one must wait for the other to “catch up.”

Flow 1 and Flow 2 Start Dwell

Each operation may be given a specific dwell (in revolutions) to wait before an operation starts.

> ⓘ Creating a stroke sync can have several effects on posted output. If a stroke synced operation is created with Canned Cycles, the Canned Cycle will not be output in the G-Code. CSS is not deactivated if used, but one of the operations will have to be given control over setting the CSS in the Spindle Mode setting.

Op Mode

In Op mode, the Sync Control dialog behaves just like the Operations list, but in a time-based format that shows Tool Groups and relationships between flows.

Op mode allows you to drag operations to another location on the list:

- Normal dragging tries to preserve syncs. When syncs are broken after a normal drag, a warning dialog tells you of all problems and the Sync Control provides visual feedback showing the problems.
- ALT+dragging makes no effort to preserve syncs. Error messages and visual feedback are suppressed when syncs are broken.

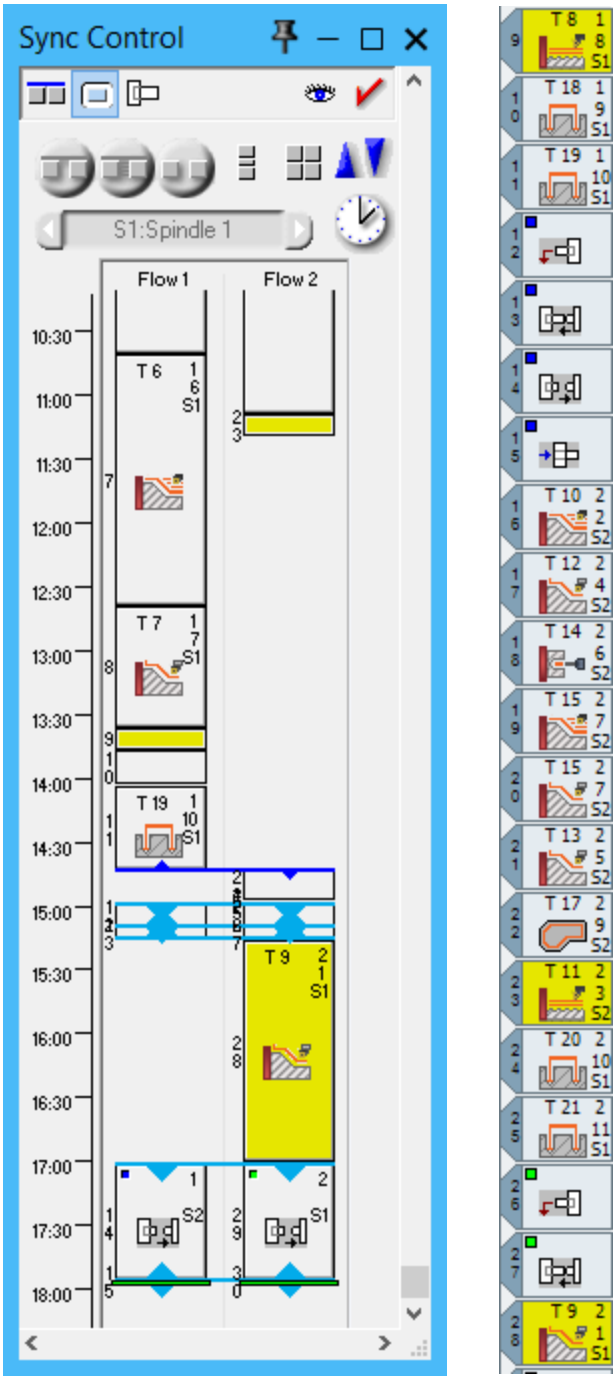
`Double-clicking` an operation loads the process. `Right-clicking` an operation displays the context menu for the Operations list. Everything you do in the Sync Control while in Op mode will be reflected in the Operations list.

Operations cannot be dragged across flows. To change the flow that contains an operation, you must recreate the operation or change the tool group containing the operation.

Tile Selection

In Op Mode, you select and deselect operation tiles using standard Windows modifiers: `click` = toggle selection, `Ctrl+click` = multiple selection, `Shift+click` = range selection. Selected tiles are shown highlighted in both the Operation List and the Sync Control dialog. You can use both lists at the same time, interchangeably.

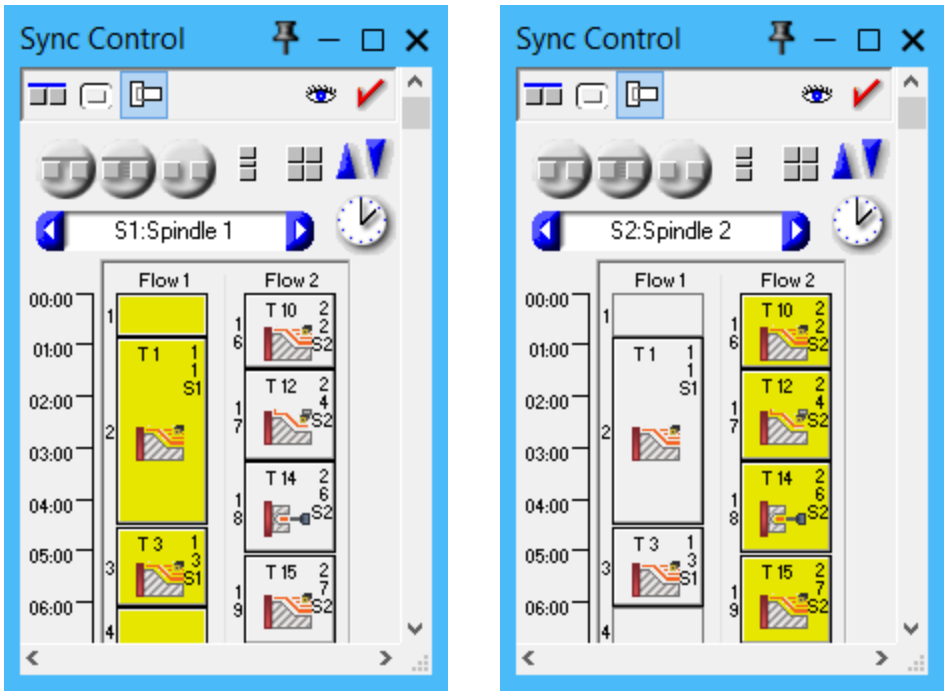
The following image illustrates operations selected in the Sync Control and thus have become selected in the Operations list.



Spindle Mode

Spindle mode allows you to specify which Flow has control over a spindle. This is used when more than one Tool Group will be cutting on a spindle at the same time. This includes RPM, direction and C-axis for milling. When in Spindle Mode the Spindle Selection button becomes active. This button cycles through the available spindles, letting you set each spindle individually.

When in Spindle Mode, any operation being performed on the spindle is available for selection, while operations that are not being performed on the spindle are grayed out. In Spindle Mode, you select and deselect an entire operation tile using standard Windows modifiers: `click` = toggle selection, `Ctrl+click` = multiple selection, `Shift+click` = range selection.



Comparison of setting controlling operations in Spindle Mode

When operations in two separate flows are cutting on the same spindle there may very well be a conflict in RPM or direction set by the operation. If nothing is set there will be no spindle commands output. To give an operation in a flow control over a spindle's RPM, simply select the operation and ensure that any operation cutting at the same time is deselected. Then click the Recalculate button.

You could give an entire flow the priority over setting RPM this way. If overlapping operations are selected, both flows will output spindle commands with random results. This is not recommended.

> ⓘ You should pay careful attention to which flow has control over a spindle's speed. If a tool is cutting deep and spindle control is given to a different flow, the sudden change in RPM could damage the tool and/or part.

Checker

The Checker is a means of manually running a check on operations in MTM parts. The checker will look for conflicts in operations, (trying to turn and mill at the same time for example), bad syncs and conflicts in RPM settings. Conflicts in RPM settings can occur when more than one Tool Group or Flow is machining a spindle at the same time. If the operations were created with different RPM settings, as is common with roughing and drilling operations, the system needs to know which operation has control over the RPM of the spindle. A check is done every time you open the Sync

Control dialog. This button allows you to check your work. Dragging operations that are already synced can result in crossed syncs. This is one type of error the Checker looks for.

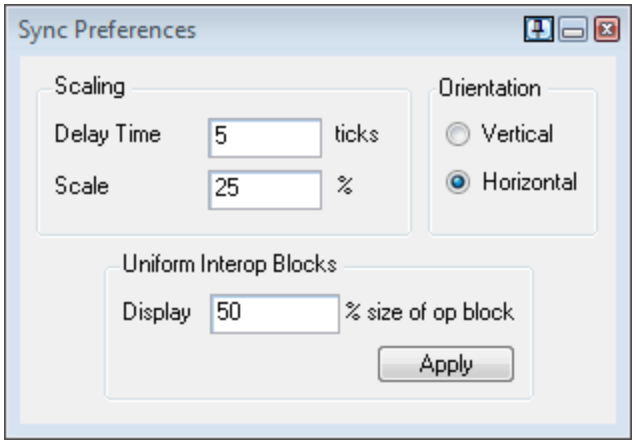
Time Scale

The Time Scale is a set of blue arrows, one to expand the flow time line, the other to compress the time line. All operations within each flow will be proportionately stretched to make viewing the time line easier. When the Sync Control dialog is closed and re-opened, the time line will automatically scale to fit within the dialog.

> Do not close the Sync Control dialog if you wish to maintain the current Time Scale. Doing so will recalculate the Time Scale to fit the current operations the next time the dialog is opened. Try minimizing the window to minimize onscreen clutter and maintain your current Time Scale.

Sync Preferences

Clicking the (View Preferences) icon, or right-clicking the title bar of Sync Control dialog allows you to open the **Sync Preferences** dialog, which offers the following types of settings.



Scaling Preferences

Delay Time

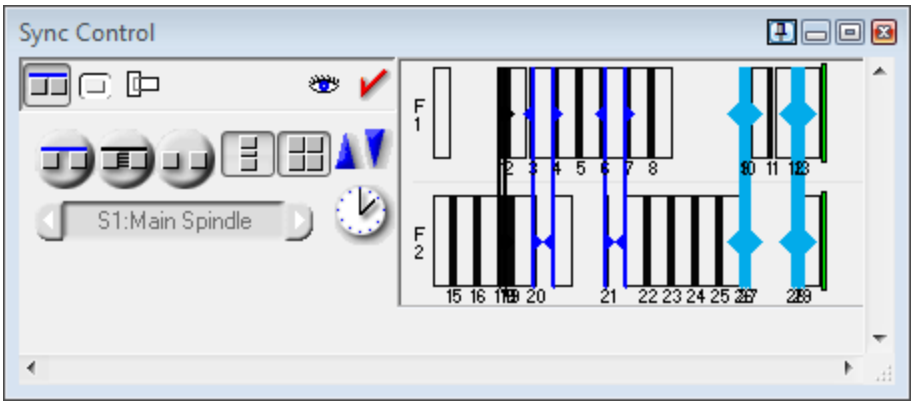
This item controls how fast the zoom action works. A “tick” is 1/60th of a second so the default value of “5” will cause the scaling function to occur 12 times each second.

Scale

This item controls how much of a change occurs to each operation with each click.

Orientation Preferences

To display the **Sync Control** dialog in vertical layout, choose Vertical. For horizontal layout, as shown below, choose Horizontal.



Uniform Interop Blocks

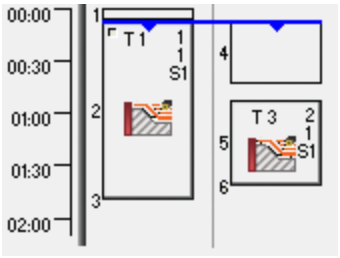
You can specify the relative size of the interop block display as a percentage of the size of a regular op block. This display setting applies only when the Sync Control dialog buttons and are both pressed – in other words, when Show Interop Blocks and Uniform View are both in effect.

Recalculate

The Recalculate button looks like a clock. This button is used to apply any changes you have made that may affect cut times. The system will automatically recalculate many things but not everything.

Flow Display

The flows are represented with all of the operations within each flow with a time line to the left of the flows. Items within the flows are selectable and, depending upon the mode you are in, have different functions. Each tile represents an operation and the amount of time that operation takes, including all movement between CP1 and the start or end points. The space between tiles represents the time required for a tool change and inter-operation moves. The operation tiles will then account for the time for all movement from CP2, all the feeds and rapids needed to cut the part and the move out to CP3.



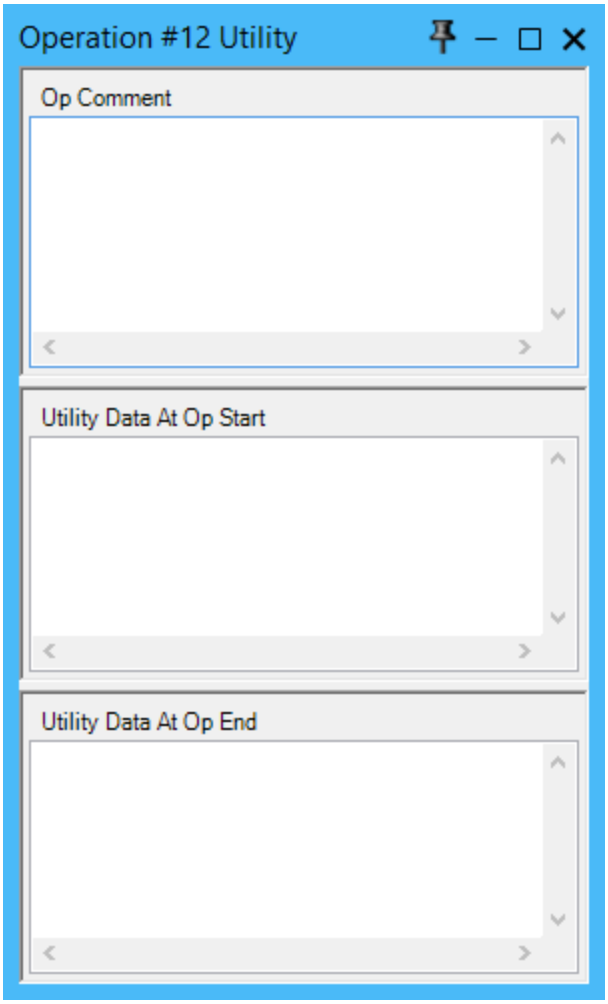
Flipping Operations

A very powerful feature of MTM is that there is no “right” way to make your part. Different people will set up their part geometry for a multi-spindle part in different ways; some may make all of the geometry on one spindle and then move it to a sub-spindle, others may create the geometry from scratch on each spindle. Even moving the geometry from spindle to spindle can be performed numerous ways. The same is true of operations, that is, the part can be programmed in different ways. One of the functions that allows this flexibility is the associativity between tools and operations that allows you to “flip” the operation. The associativity will automatically update the cut side of an operation’s toolpath depending on the tool setup. If you change a tool’s definition to cut on

the X- side instead of cutting on the X+ side, the operation will automatically reflect this. For more information on flipping an operation, see “Setting Cut Side” on page 44.

Utility Data

All operations can have Utility Data manually added that will be output in the post, such as a stop command. All data in these Utility Data dialogs is customized for each MDD. The data shown below might not appear in your MDD.



Op Comment

You can enter a comment about the operation that will display in the posted output. For example, you might enter “Start of Sub Spindle In” at the beginning of a Sub Spindle In Utility operation.

Information entered in the Op Comment text box appears in the posted output before the selected operation it refers to. You can also enter Utility Data in this dialog.

At Op Start and At Op End:

You can use these text boxes to pass custom commands that trigger actions inside the post processor. The list of custom commands and their actions should be supplied to you with the documentation for your custom post.

Also, in these text boxes, any text string enclosed between simple straight quotes (either "..." or '...') will appear as a text string in the posted output. Other types of quotemarks, such as “ ” ‘ ’ « » 「 」 『 』, do NOT act as text delimiters. In the posted output, each such quoted text string is inserted on a new line.

If utility data is entered in the Utility Data At Op Start text box, then a small square appears on the Operation tile in the upper left hand corner. If utility data is entered in the Utility Data At Op End text box, then a small square appears in the lower left hand corner of the Operation tile. If operations contain locked values, then a small lock symbol appears on the Operation tile.

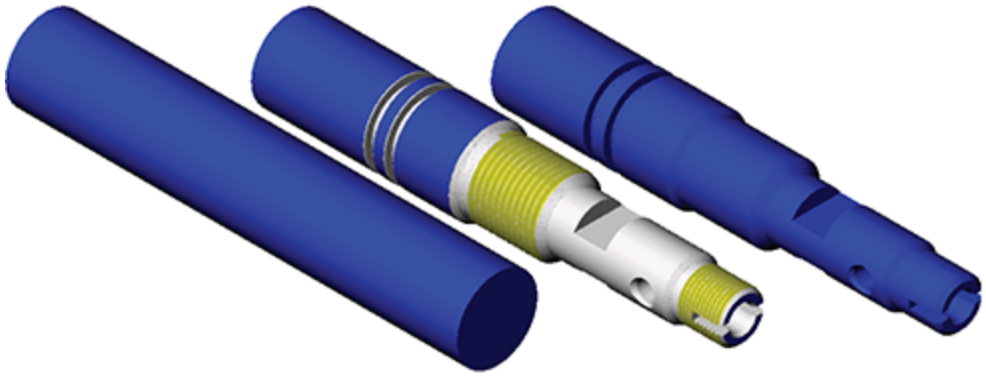
Render Control

Visually verifying a part and its operations is critical, especially in MTM. This is done through Cut Part Rendering. Rendering fully supports Multi-Task Machining functions, from displaying multiple spindles and Tool Groups to accurate tool rendering and retracts. Additionally, with Multi-Task Machining, the Render function has been expanded to include additional features including modifications to the Render Control palette.

Rendering displays the stock condition on all spindles as it currently exists in run-time. If a spindle has stock, the stock condition will be displayed. If there is no stock on a spindle, such as before a load or after an unload, stock will not be displayed. Because the system is part-centric, the actual movement of a part from spindle to spindle is not shown, simply the stock's presence and condition.

Stock Display

Multi-Task Machining parts accurately render the stock conditions of all spindles. When stock is moved from the primary to a sub spindle, the stock shown on the sub spindle will accurately display the end condition of the stock from the primary spindle. This includes any milling operations performed on the part. What is not rendered are the results of any tapping or threading operations.

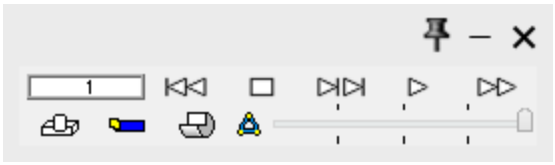


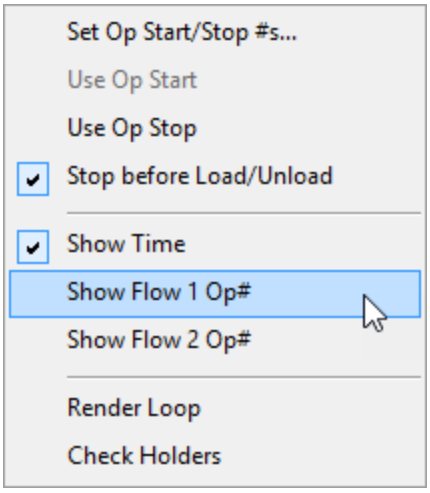
Example of initial stock on the primary spindle (1), the final condition of the stock on the primary spindle (2) and the initial condition of the sub spindle stock after a part transfer (3).

To show stock, a spindle must have some sort of a load command, in the form of a Utility operation. The one exception is if a spindle is designated as "Preloaded" in the Document Control dialog, (see "Initial Condition of the Machine When Starting a Program" on page 23 for more information). To create operations, or even to run operations on a spindle, the spindle does not need to have stock explicitly loaded. This may seem odd at first, to have tools running on a spindle with no stock, but on a first run of a part, a machine likely does not have any stock on the Subs Spindle. If you think there should be stock displayed and none is displayed, it is likely to be because of a lack of a Utility operation loading or unloading a spindle, an incorrect Utility operation or a lack of checking "Initial Stock" or "Pre-Loaded."

Render Control Palette

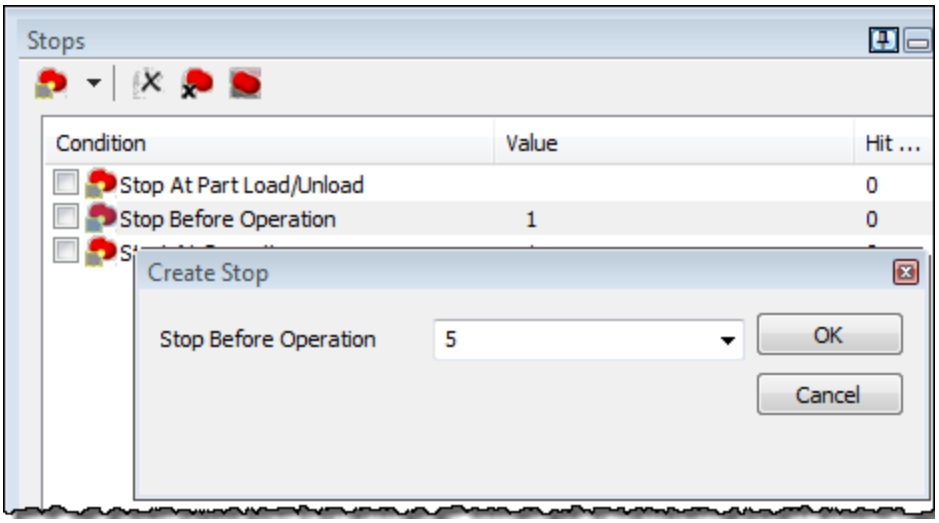
The Render Control context menus offer several ways to display information on the part currently being rendered. The palette can display the current run-time or the number of the current operation in the selected flow. Additionally, the rendering can automatically be stopped before a Load or Unload Utility operation occurs. For more information, see the *Common Reference* Guide.





Set Op Stop

The guess work can be taken out of stopping cut part rendering at a specific location. Selecting the Set Op Stop # command allows you to set a point before which the rendering should stop. Rendering will stop at the end of the operation before the number specified in the dialog. This is activated and deactivated by the Use Op Stop option.



Use Op Stop

Selecting the Use Op Stop option will cause Cut Part Rendering to stop before the operation specified in the Set Op Stop # dialog. If this option is not active, the Set Op Stop # is ignored.

Stop before Load/Unload

Selecting this item will cause Cut Part Rendering to stop rendering the part before a Load or Unload Utility operation as well as before any part transfer. This function is very useful for seeing the end condition of the stock on a spindle before the final unload.

Show Time

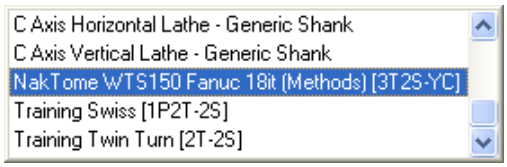
When active, the Render Control palette will display the current program run-time location instead of the current operation number.

Show Flow

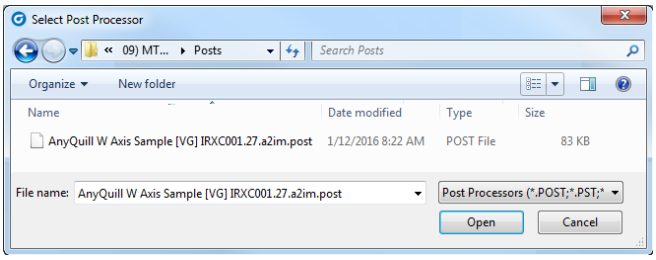
This function may be used instead of Show Time. The user may choose to show operation numbers within a specific flow instead of the program run-time. Because operations on different spindles can overlap, the user must select which Flow's operation numbers are to be displayed.

Post Processing

Post Processing with the Multi-Task Machining option is more specific than the standard GibbsCAM system. Instead of being able to post a Mill VNC part with any mill post processor, MTM parts are very specific to their MDD and matching post processor. The post for each machine is custom made to match a particular machine and its capabilities. When re-posting to a different MTM machine, it is recommended that you first change the MDD to the appropriate machine and check the VNC carefully.



Machine Type from the Document Control dialog



Post Selection from the Post Processing dialog

Examples of a machine-specific MDD and Post Processor file.

Concepts

GibbsCAM, in general, offers a standard interface for a machine type that is unconcerned with the specifics of one model of CNC. Instead, the user learns and programs in GibbsCAM standard axes, axis polarities, coordinate systems, orientations, and terms. With a single standard interface, the post processor is responsible for translating from GibbsCAM standards into specific machine formats. This provides a high level of interchangeability between machines. A mill or turning part can be posted for a wide variety of machines without reprogramming. A new programmer needs to be taught only one standard to be able to program a variety of machines. An experienced GibbsCAM programmer can program a new machine without learning a new standard. This is the GibbsCAM interface philosophy, develop a single standard to support an entire class of machines.

Multi-Task Machining causes a new set of problems with an enormous new level of machine variation and detail to be handled. Time is an extremely important factor in MTM. In MTM, time requires an accurate definition of things that move. These things are machine specific and time must be exposed in the interface. Additionally, there are machine-specific details which must be supported. These details are typically supported with Utility operations. MTM adds new MDD capabilities to accommodate these complexities. While this new format greatly complicates file interchangeability, the fundamental GibbsCAM interface philosophy is unchanged. MTM represents a single interface in GibbsCAM standard terms. Every XYZABC value is entered in

GibbsCAM standard orientation and polarity. While they will correspond to a specific machines physical orientation, they will ignore machine specific orientations and polarities. A user is always entering GibbsCAM standard values.

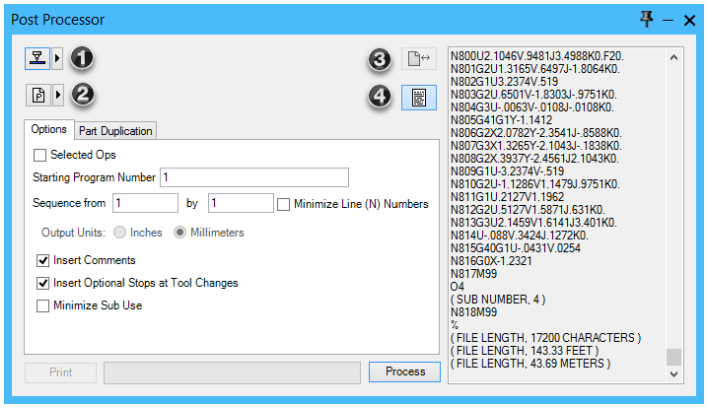
Posts and Posted Output

MTM posts require no editing from the user. All features found in a machine are fully supported by the MDD. Minor changes made to a post for a specific machine by the 3D Systems GibbsCAM Post Department may be made for individual clients (such as adding polar interpolation), but this should be set in the parameters when a post is first made. When MTM is purchased, all of the information on the machine and many of the user's post style preferences are collected and implemented.

MTM programs typically have multiple flows. How the flows are output will depend on your machine. G-code programs for each flow may be contained in one file or separate files, depending on the machine.

Post Processor Dialog

The MTM Post Processor dialog is essentially the same as a Turning or Mill/Turn Post Processor dialog. For details, see the *Common Reference* guide (chapter "Commands Toolbar", section "Post Processor dialog").



1. Post Processor selection
2. NC file name
3. CNC communication
4. Post text window

Post Names

Multi Task Machining post names use letters to signify their capabilities. The designation specifies the post's capability. Following the letter designation is a unique number for this post.

The general format of a post can be described as shown below. A metric post will end with an "m".

```
<CONTROL NAME><MACHINE NAME>[CLIENT INITIALS]<LETTER>###.##.PST
```

A post for a Fanuc 18i running a Hardinge Conquest Twin Turn may look like the following example.

```
FANUC 18I HARDINGE CONQUEST TT 65 (VG) NIAML.1150.22.6.PST
```

The following is a description of how Multi Task Machining Posts are named and what they do. Also included are brief explanations of code issues that might be encountered in Multi Task Machining Posts.

A Multi Task Machining post supports both milling and turning operations in the same part. A 2-axis lathe post is no longer needed if a Mill/Turn post is available.

Label Definitions:

A description of each label is presented here, followed by an example of the use of most labels.

AML	This designates a Multi Task Machining post.
S	This designates a Multi Task Machining post that segments rotary arcs into linear moves.
	Fanuc 16T [VG] ***S***ML800.19.pst
I	This designates a Multi Task Machining post that supports Polar and Cylindrical Interpolation. A Polar and Cylindrical Interpolation Mill/Turn post will output a G2 or G3 with rotary moves.
	Fanuc 16T [VG] ***I***ML800.19.pst
Y	This designation is for a 4-axis Multi Task Machining machine that has a linear Y-axis.
	Fanuc 16T [VG] ***Y***IML800.19.pst Fanuc 16T [VG] ***Y***SML800.19.pst
P	This designates a C-axis positioning post. A Multi Task Machining positioning post will rotate the part and then move in X and Z. It will not rotate and cut simultaneously.
	Fanuc 16T [VG] ***P***ML800.19.pst
N	This designates a Multi Task Machining post that does not use subprograms. This is known as a “Long Hand post”. Subprograms are frequently used for multi-process drilling, C-repeat drilling, Z-repeat milling, C-repeat milling, Patterns (OD only), etcetera.
	Fanuc 16T [VG] ***N***SML800.19.pst Fanuc 16T [VG] ***N***IML800.19.pst
B	This designates a B-axis rotation post. This supports the creation of coordinate systems that has the tool rotate about the B-axis.
	Super Hicell 250 HS [JMC] ***B***SML1082.19.7.pst

Code Issues

Tool Orientation

- When using a mill tool on the Face or OD, it is important to define the orientation of that tool correctly. When Milling or Drilling on the face, make sure the orientation of the tool is perpendicular to the face. When Milling or Drilling on the OD, make sure the orientation of the tool is perpendicular to the OD. If the tool is not oriented properly, the output will not be correct.

C-Axis And Y-Axis Output

- In the Rotate tab, the option buttons Position and Polar & Cylindrical Milling determine whether C-axis moves or Y-axis moves are output during Polar & Cylindrical Milling operations. If the Position option button is selected, the system calculates Y-axis moves. If the Polar & Cylindrical Milling option button is selected, the system calculates C-axis moves.
  - If your machine does not have a Y-axis, then you need to select the Polar & Cylindrical Milling option button.
  - If your machine has a Y-axis, this capability can be added to any Mill/Turn post.

Rotary Feedrates

- Most rotary feedrates are calculated in Degrees Per Minute per rotary segment based on its length. Since the length of each segment is variable, the system outputs a different feedrate for each segment. The resulting rotary feedrate can be a large value based on the Degrees Per Minute calculation.
  - Certain CNCs, such as Haas and Mazak, calculate rotary feedrates using Inverse Time. Any Mill/Turn post can be modified to use Inverse Time for feedrates.

Polar Interpolation posts use inches per minute for rotary feedrate calculations. Any Mill/Turn post can be modified to use Polar Interpolation with inches per minute feedrates.

Appendix

Glossary

This section is intended to provide an overview of terms used throughout the *Multi-Task Machining* guide. Some of these terms may be industry standard but are reproduced here to ensure correct understanding. *Italicized* terms can be found in the glossary.

Axis, Axes	1) Math: X, Y and Z axes of a coordinate system. 2) CNC machine axes: the labels of things that can be programmed to specific linear or polar locations in a G-code program. Labels are occasionally similar to the math definitions. 3) CNC specific: Specific machines may have multiple machine axes moving along the same math axes. Specific machine labels can apply any letter to any machine axis. This is not to be confused with math or CS axes. CNC rotary axes: A rotates around the X-axis; B rotates around the Y-axis; C rotates around the Z-axis.
Balanced Turning	Two tools work on the OD but both use the same depth of cut and no dwell. The second tool effectively cuts nothing but supports the part.
Channels	See *Flow*.
Coordinate System (CS)	1) A right-handed orthogonal XYZ coordinate system orientation. Sometimes applied to a 2D 2-axis orientation and origin definition. 2) A GibbsCAM VNC data definition, numbered as CS1, CS2, etc.
Dual Tool OD Roughing	Two tools work on the OD, usually with a small “Z lag” difference in position. The second finishes before the first and retracts, waiting for the first. Rough the part in half the time. This can be programmed using GibbsCAM MTM Multi-tool processes.
End Sync †	An *Operation sync* that is positioned at the end of an operation.
Flow	An A sequence of machining operations associated with a *toolgroup*; i.e., one toolgroup = one flow, two toolgroups = two flows, etc. Also sometimes referred to as Channels. G-code programs for each flow might be contained in one file or separate files, depending on the machine.
Form Tool	User defined insert shape often used for plunging.
Full Retract	A pre-defined machine axis position (in every axis that has one). This is usually at a travel limit. Usually accessible with a G28 Fanuc-style command.
G28	A Fanuc-style command to move specified axes to their home location.

G30	A Fanuc-style command to move specified axes to a location specified by the user in the control.
Gang Tools	A linear row of tools. Gang tools can be stationary (either fixed or a stationary tool post) or able to move in a single axis.
Guide Bushing	Part of a *Swiss-style machine* that allows the stock to move in and out of the spindle. The diameter of the bushing must match the stock diameter. The length of the bushing limits Z-pull back distance. Also known as Screw Bushing.
Head Stock	Describes the machine body that contains the main spindle on a lathe. It can be bolted to the machine bed ("Fixed Head Stock"), or it can be mounted on a slide ("Sliding Head Stock") with or without a Guide Bushing.
Home	See *Full Retract*.
Load/Unload Spindle †	Used in *Utility processes* to load or unload spindles. "Load" refers to the loading of the initial material into a machine. "Unload" refers to removing the finished part.
Machine Definition Document (MDD) †	In GibbsCAM terminology, the MDD (Machine Definition Document) is where all aspects of a particular machine are organized and stored, including its linear and rotary axes, its toolgroups, spindles (part stations), and utility stations, and how these are associated and organized into Flow Axis Sets, Interop Moves, etc. It also specifies the post processors, coolants, extended cycles, and simulation bodies available to the machine, and it records preferences for work areas, limits, clearances, and many other items. An MDD defines either a generic class of machines (such as a 3-Axis Horizontal Mill) or a specific machine configuration. Multi-Task Machining MDDs are typically machine-specific.
Machine Simulation	An optional product separate from *Multi-Task Machining*. It allows for the precise definition of a machine, its setup, and its tools. It simulates machine and tool motion, with collision detection and movie playback.
Machine Zero	1) CNC: An arbitrary position defined by a machine tool builder for a collection of axes to use as their origin. The main spindle face and centerline is a common machine zero. G53 is sometimes used as a predefined work fixture offset to facilitate G-code programming from this position. 2) MTM: This is the GibbsCAM MTM concept of the main spindle CS. All other CSs are measured from here. MTM's Machine Zero is a math origin, not a machine axis origin.
Motile Spindle †	Part holding spindle that is not fixed in relation to the machine tool (that is, it has one or more axes of motion). This is frequently used with stationary tools.
Index Spindle or Multi-Spindle (3, 5, 6 or 8	An indexing lathe moves a group of spindles between tool positions (toolgroups), typically in a rotary manner. Each tool position is assigned a spindle number. Indexing these spindles has the effect of moving the part

spindles)	from one spindle to the next, loading a new part into the first position and unloading a part from the last position.
Multi-Task Machining (MTM)	A class of machining done on machines that can perform multiple tasks at the same time, typically referring to multi-spindle, multi-turret lathes with live tooling.
Multi-Tool Process, Multi-Tool Operation †	A special rough turning process and corresponding operation where the system allocates alternate machining cuts to multiple tools in a coordinated method. This provides an efficient way to program “dual tool rough” machining.
Off Part Location	Tool change location or a position away from the part. Standard off part location: The off part location defined in the MDD for each TG per spindle, to be used for automatic moves to an off part location. This is triggered by events such as a tool change and occurs between operations. The actual location is set in the MDD. MTG tool change location: A location away from the part that the user has specified with an MTG Utility operation. If an automatic move to an off part location is triggered immediately after an MTG operation, the event will occur at the MTG location instead of a move to the standard off part location.
Operation Synchronization (Op Sync)	Process for controlling multiple machining operations to start or end at a particular time in relation to another process (simultaneous cutting).
Origin	The 0 point of all the axes in a CS.
Part Coordinate System	This is a plane the part is in. It is the same plane as the Spindle CS except the Z origin is shifted based on part placement. Several Part CSs per spindle can be found in the CS List, each representing the appropriate primary plane for specific machining operations (ZX, XY, YZ, and so forth).
Part Shift	Used in *Utility processes* to partially move a part out of the spindle during operations (in other words: the front of the part is machined close to the spindle and then advanced or “shifted” further out to do more machining or to machine between spindles/tailstock etc.)
Pick-Off Spindle	See *Sub Spindle*.
Pinch Turning OD/ID	TG1 roughs the OD. TG2 roughs the ID. Dwell is 0.0. Both tools begin each stroke together. The purpose is to provide support for a long part away from the chuck. The ID or OD cut may finish sooner depending on the length of cuts and feedrates. Not useful in a *Swiss spindle*.
Program	Sometimes used to describe a single flow, especially in machines that require a separate text file for each flow.
Screw Bushing	Part of a screw spindle that holds the part and allows the stock to move in/out of the spindle. The diameter of the bushing must match the stock diameter.

	The length of the bushing limits Z-pull back distance. See also *Guide Bushing*.
Screw Machine	A machine tool with a fixed headstock/spindle, typically cam-operated with several (4-6) “cross slides” using form tools arranged around the spindle. Used to produce large quantities of parts. Also known as Cam Machines and Automatic Machines. See also *Swiss-Style Machine*.
Sliding Headstock Machine	See *Swiss-Style Machine*.
Spindle	A spindle is a place where a part is held, spun and machined.
Spindle Coordinate System	The axis definition for each spindle that, per GibbsCAM standard, places the Z positive axis exiting the spindle face and the origin at the center of the spindle face. GibbsCAM Clockwise/Counter-clockwise is defined by looking at the spindle in the Z- direction.
Standard Spindle	A spindle with no linear axis motion, may have a C rotary axis for milling.
Start Sync †	An *Operation sync* that is positioned at the start of an operation.
Stroke Synching †	A special form of a *Sync Process* where two or more operations start each cutting stroke simultaneously.
Sub Spindle	Any spindle that a part may be transferred to, typically opposite the main spindle.
Swiss Spindle	Part-holding spindle that has a linear motion axis (typically a Z axis) parallel to the spindle axis of rotation and that moves the part in and out of the guide bushing.
Swiss-Style Machine	A machine tool with a “sliding” spindle, typically used to produce small screw-like parts. Also known as a Swiss screw machine, Swiss lathe and Automatic lathe. See also *Swiss Spindle* and *Guide Bushing*.
Sync Marker †	The special symbol shown in the Op Sync dialog to represent an *Operation Sync*.
Sync Process †	Process for controlling multiple machining operations to start or end at a particular time in relation to another process (simultaneous cutting).
System Sync †	Special type of *Operation Sync* where the system creates the sync automatically as a result of a special process or operation. Often used in *multi-tool processes* and *utility processes*.
Tail Stock	Describes the machine body opposing the Head Stock on the same centerline to support long workpieces or shafts on a lathe.
Tool ID # (Identification)	In a simple mill or lathe part, the tool number is the list position. Optionally, a user may override this with a Tool ID #. In MTM, the Tool ID # consists of TG #, and a TG position #. Both are displayed on Tool, Process, and Operation

	tiles.
Toolgroup (TG) †	Any collection of tools, of which only one may be used at a time. The tools in a TG typically move together (there are exceptions). A typical TG is a *turret* or a *gang* slide.
Tool Offsets	An XZ shift for each tool, typically from the TG/turret datum to the tool tip.
Tool Tip	Location used when Offsets are on. G-code position values include the tool offset values because the tool offsets have been programmed "on." See also *Turret Datum*.
Turret	A common lathe tool changer and toolgroup where tools are rotated into position.
Turret Datum	Location used when Offsets are off. G-code position values do not include the tool offset values, because the tool offsets have been programmed "off" and therefore represent the TG datum position. See also *Tool Tip*.
Utility Process Utility Operation †	Special processes that are typically not cutting operations, but ancillary machine-controlled functions such as part mover, part catcher, and tailstock controls.
Utility Tool †	Special tool used for *utility processes*. Allows assignment of tool position to a tailstock or *Sub Spindle* (optional).
VMM †	Virtual Motion Macro. The VMM is a file that provides the link between the *MDD* and the Post Processor to implement custom utility operations. A machine-specific MDD references the matching VMM for MTM machines.
Work Fixture Offsets	Work Fixture Offsets allow the user to enter a shift for every axis in a machine, usually from machine zero or from machine home. The goal is to set up a part CS so the G-code can be programmed from a part origin in each spindle. This shift amount is typically from the TG/turret datum to the part origin, relying on tool offsets to provide the shift for each tool from the TG/turret datum. Typically G54-G59 on a Fanuc.

† Denotes a term used by GibbsCAM.

FAQ (Frequently Asked Questions)

In the following section you will find answers to questions and problems that may arise.

- This operation is not generating any toolpath! What is wrong?

  Typically, an operation will not generate toolpath under several conditions, including the tool is too large or you have specified that an operation is Material Only and there is no material to be cut. With MTM there are some additional possibilities.

a. Open the Tool dialog for the tool you are trying to make an operation with. Check to make sure the tool is oriented correctly (coming in from the correct side of the part) and that the insert is on the correct side of the holder.

b. Check to ensure the operation is set to cut on the correct spindle.

c. Check to ensure the operation is set to cut on the correct side – in other words, X+ or X-.

d. Check the current coordinate system. In some cases being in the wrong CS will prevent toolpath from being generated.

- When I activate rendering, I do not get any stock on a sub-spindle. How do I fix this?

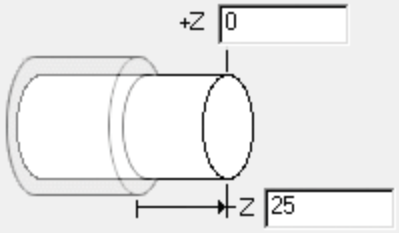
  There are several possible solutions to this problem.

  a. If you have not transferred the part to the sub spindle you must create a set of Utility operations that perform this, e.g. Sub Spindle In, cut off, Sub Spindle Return.

  b. Your Unload Spindle Operation is set to the wrong spindle.

  c. Your Sub Spindle Return that is supposed to move the part does not have With Part selected.

- After a Sub-Spindle Return operation, the transferred part is rendered in the wrong place.



Check the Grip Z value of the Sub-Spindle In Utility operation and the Part Distance From Face setting for the sub spindle in the Document Control dialog. In all likelihood, the Grip Z value plus the part face distance from the spindle do not add up to the part length.

If you have a 50mm part and you grip it in the middle (Z-25, assuming the part face is at the origin) with the sub spindle, it is sticking out of the sub spindle by 25. The setting in the Document Control dialog should reflect that.

- After a Part Shift operation, the part is rendered in the wrong place.

  Check the Sub-Spindle In Grip Z value and the Shift Distance in the Part Shift Utility Operation.

- There are red X's over my utility operations. What does this mean?

  There are two possibilities. You may have changed the MDD used by the part to an MDD that does not support those Utility operations. Switch back to an MTM MDD that does support the Utility operations. If this is not the case, the VMM file that accompanies the MDD has been damaged, removed or not installed. Each MDD has an accompanying VMM. The MDD and VMM files are normally named identically but with different extensions (e.g., `Hard_Conquest65_0001.mdd` and `Hard_Conquest65_0001.v64.vmm`). An MTM MDD will not work without its accompanying VMM and the VMM must be kept in the VMM folder.

- I typed in a rotation angle of 270° and my post output -90°. Why is this?

  GibbsCAM always makes the shortest rotation to minimize cut time.

- I post and I get warnings that certain operations are invalid. What is wrong with the operations?

Chances are, there is nothing wrong with your operations. In all likelihood, you are not using an MTM post processor. Select a different post, make sure it is an MTM post, and try again. MTM posts have the code AML included in their name.

- I have created my part but I need to change the setup. What do I need to look out for?

  The stock size settings in the Document Control dialog may need to be updated to change a part. Especially important are the Spindle 2 settings for part transfers. Additionally, check any Grip Z values in Sub Spindle On Utility operations.

- My rendering sequence is odd. One spindle renders, resets (or reloads) then the second spindle starts to machine.

  The sequence of your operations is set so that the load command is in the middle of your program. Move your load/unload sequence to the start or end of the program.

Conventions

GibbsCAM documentation uses two special fonts to represent `screen text` and `keystrokes or mouse actions`. Other conventions in text and graphics are used to allow quick skimming, to suppress irrelevancy, or to indicate links.

Text

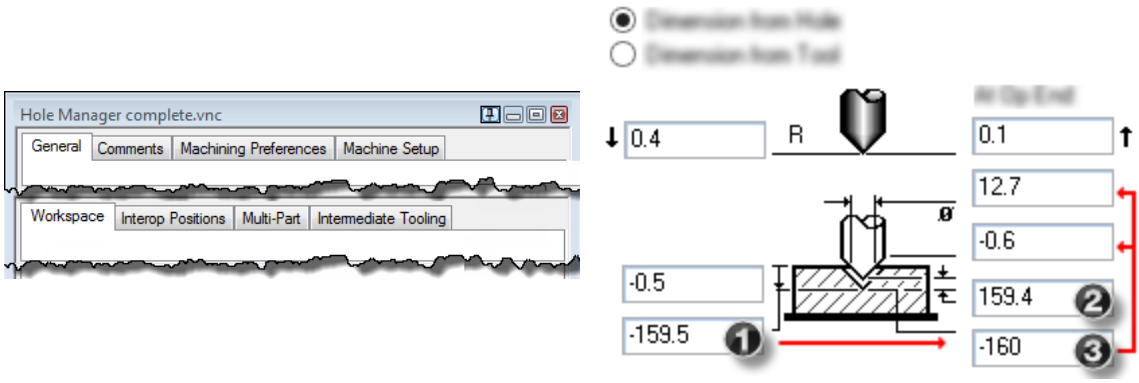
`Screen text`. Text with `this appearance` indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

`Code`. Text with `this appearance` indicates computer code, such as lines in a macro or a block of G-code.

Graphics

Some graphics are altered so as to de-emphasize irrelevant information. A “torn” edge signifies an intentional omission. Portions of a graphic might be blurred or dimmed to highlight the item being discussed. For example:



Annotations on a graphic are usually numbered callouts (as seen above), and sometimes include green circles, arrows, or tie-lines to focus attention on a particular portion of the graphic.

Faint green borders that outline areas within a graphic usually signify an image map. In online help or a PDF viewer, you can click a green-bordered area to follow the link.

Links to Online Resources

Link	URL	Action / Description
Go	http://www.GibbsCAM.com	Opens the main website for GibbsCAM.
Go	https://online.gibbscam.com	Opens a restricted website containing materials available for download. Requires a GibbsCAM Online Services account; to set up an account, contact GibbsCAM Support.
Go	https://store.GibbsCAM.com	Opens the website for the GibbsCAM Student Store.
Go	https://macros.gibbscam.com	Opens a wiki containing documentation and examples of GibbsCAM macros. Requires a GibbsCAM account.
Go	http://kb01.GibbsCAM.com	Opens a Knowledge Base article, **Contour Operations Using Thread Mill Tools**, that explains in detail the correct way to program Contour processes using Thread Mill tools.
Go	mailto:Support@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Technical Support department for GibbsCAM.
Go	mailto:Registration@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Registration department for GibbsCAM.
Go	mailto:Sales@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Sales department for GibbsCAM.
Go	http://www.autodesk.com/inventor	Opens an external website that provides more information on Autodesk Inventor products.
Go	http://www.celeritive.com	Opens an external website that provides more information on VoluMill Ultra High-Performance Toolpath (UHPT) from Celeritive Technologies.
Go	http://www.predator-software.com	Opens an external website that provides more information on a CNC editor and a virtual CNC viewer from Predator Software, Inc.

Index

F

G

H

I

L

M

N

O

P

R

S

T

U

V

W

X

Y

Z