GIBBSCAM® 14

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Geometry Creation

CAMBRIO

Proprietary Notice

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Introduction

Congratulations on purchasing the most productive programming system available! We recommend that you review *Getting Started* and this guide before moving on to *Mill*, *Turning*, *Mill/Turn*, or any other GibbsCAM product guide. The best way to learn the system is to read the reference information and complete the geometry exercises, and then complete the associated tutorials.

This guide is intended for all GibbsCAM users. The information in this guide applies to the basics of creating geometry (and the basics of a part file in general) across all GibbsCAM product modules. We strongly recommend that, at a minimum, you familiarize yourself with the following material:

- About Geometry Creation in GibbsCAM
- Geometry Overview
- Working with Geometry

The tutorials for geometry creation focus on basic milling or turning parts. You should master them before proceeding to the tutorials for Mill or Turning.

Online Help and Balloons

For simple explanations of items and their purpose, use Online Help, and Balloons. Online Help, Balloons, and PDFs are built-in systems of documentation and training information. They are enabled from the Help main menu.

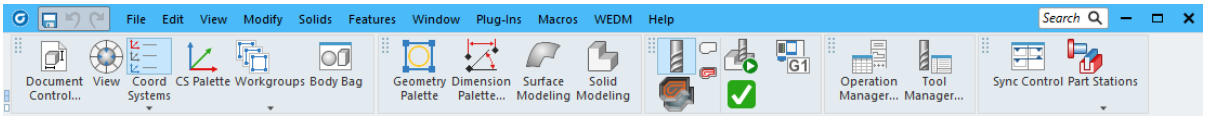
Online Help is a context sensitive Help system. When activated the online help will bring you to a web page that is relevant to the dialog you are working in. Online Help can also be browsed or searched.

Balloons (Ctrl+B) provide reference information about any object that the cursor is placed over.

The PDFs are electronic versions of the GibbsCAM guides. The PDFs may be browsed, searched and print well if you need a hardcopy. The *Common Reference* guide will help you with items contained in the Menu Bar.

System Overview

The workflow is designed to be extremely flexible and to allow the user the freedom to create parts in any way that comes naturally. The “modeless” interface allows the user to have geometry creation, tools, machining capabilities and post processing functions available at all times. However, there are certain basic elements required to create a part. Geometry must be created. At least one tool must be defined and toolpath must be generated before a part can be post processed. The Main palette is organized in a logical manner for building a part. A part does not have to be created in this order, it only serves as a guideline.



About Geometry Creation in GibbsCAM

If you have ever used a design CAD software package to design, draw, or create geometry, the first thing you will notice about GibbsCAM is that the geometry CAD tools are different. This can lead to a number of different possible first impressions, including that different is “bad” especially when it is different from what you have already learned how to use.

In the case of GibbsCAM “different” is not just for difference sake. Throughout our software, we strive for consistency in our interface and usage so that clients can be productive quickly without a steep learning curve. There are reasons why GibbsCAM CAD is different, starting with the fact that it is not intended for the same task or the same user as design CAD software.

There are typically several important steps in modern part manufacturing, including Design CAD, Manufacturing CAD, and CAM.

Design CAD

This is the creation of a new design performed by a design engineer using a design CAD software such as SOLIDWORKS, SolidEdge, AutoDesk, PTC Creo Parametric (Creo Elements/Pro/ENGINEER), Catia, or Unigraphics. The software for this function needs to support the designer's creativity with a variety of freeform CAD capabilities.

Manufacturing CAD

At this point a manufacturing expert receives a design from someone else – another department or another company. The manufacturing expert may need to redraw the file, import CAD files, or repair/correct the design for manufacturing. Additionally, the manufacturing expert may need to create process models, tooling, or fixtures from the part design. A key point is that this is usually not the person doing the original design.

CAM

This is creating the NC program. Once the manufacturing expert has the geometry or part models ready to machine, they can proceed with planning and creating the actual tools, toolpaths and G-code needed to make a good part.

GibbsCAM excels at Manufacturing CAD and CAM, which is everything the manufacturing expert needs to complete the job begun by the design engineer. The users and tasks of GibbsCAM are very different from the users and the tasks of a design CAD software. GibbsCAM CAD is aimed at a different user and a different task and therefore *is* different.

In the infrequent case where the designer and the manufacturer are the same person, they may choose to use GibbsCAM CAD just because they like it, or they may choose to use a design CAD for designing. They would then link files to GibbsCAM for Manufacturing CAD and CAM.

We will look at some of the ways GibbsCAM geometry creation is different and why.

Redrawing Time

When GibbsCAM was first released, in 1993, much CAD work was still sent to the machine shop by blueprint. This required the manufacturing expert to redraw it. GibbsCAM set out to provide tools that allowed a shape to be redrawn faster and easier than design CAD systems could. GibbsCAM CAD tools achieve this, allowing redrawing to be done in a fraction of regular CAD time. Geometry Expert pushes this advantage further by providing parametric capabilities for family-of-parts creation and interactive shape editing.

Just The Finished Part

You do not need to recreate an engineering drawing to make a program. GibbsCAM focuses on creating finished shapes to machine, not recreating an engineering drawing. GibbsCAM does not require that additional geometry be created for entry and exit tool moves; we just need the finished part.

Chaining

Any CAM system using design CAD as the front end has a “chaining” step. Chaining attempts to automatically connect one feature with the next into a machinable shape. This process stops and asks the user “which way?” when more than two features intersect on the screen. In a complex drawing, this can be a cumbersome process. The issue with chaining gets worse when the chaining function finds a small gap which is common with imported geometry. In such a case, you must go back to the CAD, fix the problem, go back to chaining, find the next error, and repeat the steps. GibbsCAM eliminates the chaining step by allowing users to create linked/connected geometry. In effect, when you are finished drawing a shape, you are assured that it is already chained and ready for machining.

Fast and Easy?

There is a big difference between the best way to draw in GibbsCAM and the best way to draw in a design CAD system. GibbsCAM does not have a design CAD style “trim” function. Instead, it uses “connectors” to link one feature with the next, automatically trimming the shape graphically when a feature has two connectors. A machinist can think of drawing a shape like taking a tool around a part in G-code in a sequence of lines and circles.

A design CAD user frequently approaches drawing a part quite differently, creating a large number of untrimmed lines and circles, then going back through them “trimming”. GibbsCAM CAD is always focused on drawing a shape to be machined. You simply need to define the lines and circles in some order around a shape. Going clockwise or counterclockwise doesn't matter; where you start making geometry and the direction you go has no effect on machining, because you are just drawing a finished part shape.

When you start defining your geometry elements in shape order, a wonderful thing happens – there are rarely more than 2 untrimmed elements on the screen at a time. Connecting and trimming are automatic. You can draw a shape in fewer user actions and eliminate chaining all at the same time. This is a win-win situation which makes learning a different approach to drawing (or redrawing) worthwhile.

Geometry Overview

Geometry consists of lines, circles, curves (splines) and points. Geometric features may be connected into complex shapes in 2D or 3D. Connecting points (blue squares) are referred to as a connectors while end points are called terminators and are drawn as yellow squares. Each feature has a reference number label in order of creation. Geometry may be used to create holes, pockets and contours for toolpath.

Geometry may be created using the Geometry Creation features of the system or imported from a CAD program. Additionally, imported geometry may be modified. There are four methods when using the system to create geometry:

- Free-form CAD
- Geometry Expert
- Combining Free-form CAD and Geometry Expert.
- Extracting geometry from a solid or sheet

This chapter details the different functions available for geometry creation and provides detailed explanations of how shapes are created. To gain practical knowledge of the concepts outlined in this chapter, complete the exercises provided throughout this guide.

Free-Form CAD

The free-form CAD tools included in the system are very powerful and easy to use. Creating points and features is as easy as clicking on buttons and entering values. Likewise, connecting features to form shapes that can be machined simply involves selecting the intersecting features and clicking on a button. Several different options are provided for creating points, lines, circles, curves, fillets and chamfers, making it possible to create any shape regardless of how the blueprint is dimensioned. The free-form CAD tools are particularly useful with parts requiring construction geometry.

Geometry Expert

The Geometry Expert is a means of quickly creating geometry of simple parts and the simplified creation of more complex parts. The Geometry Expert is designed to create a single continuous shape. It allows the user to define, create and connect shape features while following along the path of the part. The associative capabilities of the Geometry Expert make editing any existing shape a very easy process.

The Geometry Expert has a tabular format which operates much like a standard spreadsheet. Features are defined by entering data in cells of a row. Each row creates a different feature. The rows define how the features are encountered along the shape path as if drawing a contour.

Combination

Using both the Geometry Expert and the free-form CAD capabilities to create a part shape is a very powerful combination. For example, a simple shaft with chamfers can be created in seconds by using the Geometry Expert to create the horizontal and vertical lines, and the automatic fillet/chamfer option in the Geometry Creation palette to create the chamfers.

Double-click any part of an existing 2D shape to load the geometry into the open Geometry Expert table (except splines). All features of the shape are listed and dimensioned in the rows of the spreadsheet where they can be checked and changed.

Extracting Geometry From Bodies

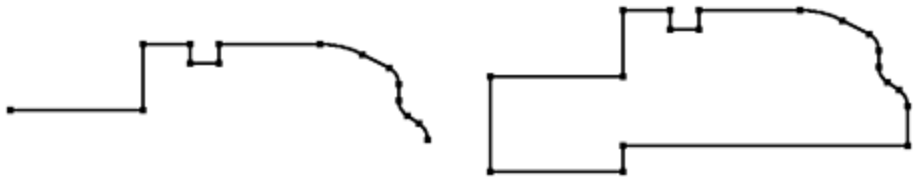
Geometry may also be extracted from solids and sheets. Faces, edges and holes may be extracted from a model. Additionally there is a tool that examines a model and creates geometry for a parting line for that model. A solid option must be available for these options to work.

Working with Geometry

- Shapes and Connectors
- Points
- Features
- Making Connections
- Breaking Connections (Disconnecting)
- Geometry Context Menus

Shapes and Connectors

There are two types of shapes: open shapes and closed shapes. An open shape is a group of connected features. There is a definite beginning and end to the shape. The two ends may or may not be terminated. A closed shape is a group of connected features in which there is no beginning or end. A circle is the most simple example of a closed shape. `Double-click` a feature in a closed shape to select all of the features and points in the shape.



Open Shape Closed Shape

There are two classifications of geometry used by the software – features and points. Lines, circles and curves are considered features.

Points

A point has three states. It can either be a plain point, a connector point, or a terminator point.

- A plain point is used in constructing other geometry or for hole positioning. Plain points are drawn in yellow and are round.
- A connector is used to connect two features together. It is drawn as a blue square.
- A terminator is used to end a shape. It is drawn as a yellow square.

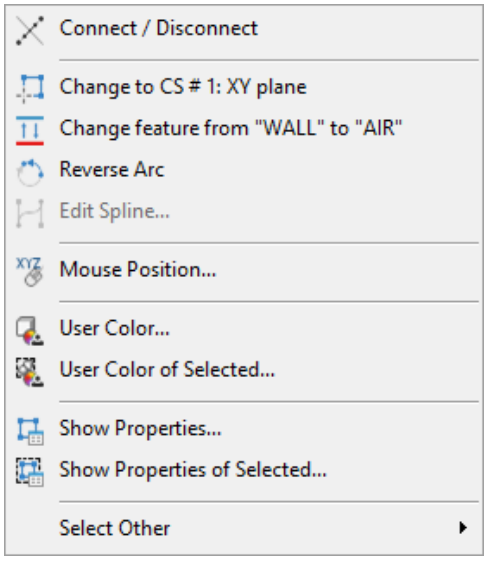
Features

A feature is either a line or a circle. A feature also has three states: unconnected, connected but not terminated, and connected and terminated. A feature can have a maximum of two connectors

and/or terminators attached to it. When the second connection has been added to a feature, it will change colors and become trimmed.

- Unconnected features are yellow. They are used either for constructing additional geometry or can be connected to other geometry to form a shape to be machined. Single features (one line or one circle, for example) that are yellow can be machined by the system without being connected to any other features or having any connection points.
- Connected (but not trimmed) features are features that have only one connection to another feature. The feature will not be trimmed until a second connection is added or when it is terminated. Since this type of geometry has only one connection, it will still be yellow.
- Connected and trimmed features are blue. They have been connected (or terminated) at both ends. Since they already have two connections on them, no additional features can be connected to them.

Making Connections



Most connections are made automatically by the software. If the Point button is chosen on the Geometry Creation palette and two features on the screen are selected, the software will automatically create a connector at the intersection or tangency of the two features. If a connection cannot be made automatically, a plain point will be created at the intersection or tangency of the two features. Once a feature has two connectors on it, it is considered fully connected. A connection must be broken on the original shape before another connection can be added to it.

If a plain point exists at the intersection or tangent point between two yellow features, that point can be turned into a connector. This is accomplished using the Connect-Disconnect button in the Geometry Creation palette, which is described in more detail in Connect/Disconnect , or you may use the right mouse menu to connect selected overlapping/tangent shapes. To change a point into a connector, use the `Ctrl` key to select the point and the two features that intersect at the point. These should be the only items that are selected. Click the Connect-Disconnect button. The point will change to a blue square. If one (or both) of the features is already connected to another feature, it will turn blue and trim with a square yellow endpoint.

The four descriptions listed below explain possible reasons why the Connect-Disconnect button will not perform the desired function.

- One of the features already has two connections on it.
- The point is not tangent to or is not exactly at the intersection of the two features.

- The wrong geometry is selected, either too much or too little in most cases.
- There are multiple features on top of one another.

The best way to tell if there are multiple features on top of one another is to turn on View > Labels. If all of the numbers are clearly visible, there is only one feature at that location. If the numbers are jumbled, there are probably features on top of each other. Try deleting features and re-drawing until there is only one feature left. If the last feature is deleted by accident, use the Undo item from the Edit menu to back up a step. Refer to Cleanup in the Plug-ins Guide for an alternate way to remedy this issue.

Open Shapes

To terminate the last feature of a shape and create an open shape, a point needs to be created at the location where the shape is to end. The feature to be terminated must be yellow to begin with. Select the point and the feature and click the Connect-Disconnect button. The point will become a yellow square. If the feature already has one connector or terminator on it, it will turn blue and trim.

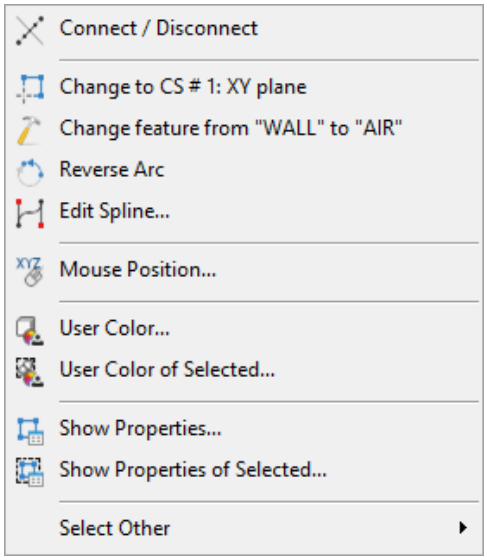
Circles

When connecting circles, the shortest side of the circle will connect and the rest will be trimmed away. To use the larger portion of the circle, select the circle after it has been trimmed and use the Reverse Arc choice from the Modify menu. An alternate solution for circles may be created using the -R function. See -R Creation.

Breaking Connections (Disconnecting)

It is sometimes necessary to break a connection. If the software automatically creates an undesired connection or changes need to be made to the original geometry, then the connection will need to be broken. To break a connection, select the connector or terminator where the connection needs to be broken and click the Connect-Disconnect button or use the right mouse menu Connect/Disconnect option. The connector or terminator will turn into a regular round yellow point and the connected features will turn yellow and extend to their original size. The features can then be changed and reconnected to form a new shape.

Geometry Context Menus



Right-click geometry to open a list of context menu options:

- Connect/Disconnect
- Change to CS #
- Change Feature From "Wall" to "Air"
- Reverse Arc or Edit Spline
- Mouse Position
- Use Color and Use Color of Selected
- Show Properties and Show Properties of Selected

Connect / Disconnect

Available only when two or more items are selected. This option will connect any overlapping or tangential geometric features. It works exactly the same as the **Connect/Disconnect** button. A connecting point will be created if it is required; otherwise, if there is a point where the features are to be connected, then that point will be used. When disconnecting geometry, you must select either the connecting point or select the features to be disconnected; you do not need to select the point and features to disconnect them. For more information on the **Connect/Disconnect** button, see "Connect/Disconnect " on page 39.

Change to CS

This item will change the current coordinate system to the coordinate system of the right-clicked geometry.

Change Feature from "Wall" to "Air"

(Also an item in the Properties dialogs.) Lets you designate the selected geometry as "Wall" or "Air". The menu item will always show which category the geometry can be changed to.

- Wall is the default setting for geometry. If default color settings are in effect, "Wall" geometry displays blue.
- Air changes the behavior of geometry when setting up machining operations. If default color settings are in effect, "Air" geometry displays red.

"Air" geometry acts as a constraint similar to regular geometry, except that the toolpath will overhang this area by the amount specified in the machining dialog. See the *Mill* guide for information regarding machining with "Air" geometry and Overhang.

An additional use for "Air" geometry is in profiles for 2D Form tools. Geometry designated as "Air" will become a non-cutting edge of the 2D Form tool.

Reverse Arc
Available only if the item is an arc. This option will reverse the direction of the selected arc.

Edit Spline


Available only if the item is a spline curve. Opens the **Edit Spline** dialog, which lets you view and optionally modify the spline's knots (points that lie on the curve, shown in yellow) and control points ("shoulder" points, shown in red).

Controls in the **Edit Spline** dialog

- Button 1 prevents or allows modifications to the spline. When the spline is in "allow modifications" mode, you can drag any knot or control point to see the effect on the overall curve.
- Button 2 prevents or allows the addition of knot points.
  To add a new knot point: Select the spline where you want the new knot point.
  System response: The spline tightens: two new control points are added, and the two affected previous control points approach the spline more tightly.
- Button 3 prevents or allows the deletion of knot points.
  To delete a knot point: Select it.
  System response: The spline relaxes into the knots on either side of the deleted point.
- Button 4 prevents or allows the use of tangent features for modifying the spline. For a feature to be used, it and the spline must be connected. A user-specifiable angle value lets you control the sensitivity.
  System response: When you drag the spline's endpoint close enough to a candidate feature, a gray phantom point will appear showing where a tangency would be created. If you release the mouse button during this time, the curve will "snap" to the phantom point and become tangent to the feature.

Mouse Position
Opens the standard Mouse Position dialog, just as if you had chosen main menu View > Mouse Position. For information on Mouse Position, see the *Common Reference* guide.

User Color
Opens a dialog that lets you assign user color to the right-clicked item. For information on user color, see the *Features* guide.

User Color of Selected
Opens a dialog that lets you assign user colors to the selected items. All selected items appear in the dialog, in a list that shows their workgroup, ID, and current color.

Geometry Properties
Opens a dialog that lets you view and modify properties of the right-clicked item. For more information, see "Geometry Properties" on page 16.

Geometry Properties of Selected

Opens a dialog that lets you view and modify properties of the selected items. For more information, see “Geometry Properties” on page 16.

Geometry Properties

The **Geometry Properties** dialog lets you view and modify properties of a single element: a point, line, circle/arc, or spline.

The **Geometry Properties of Selected** dialog lets you view and modify properties of all selected elements (points, lines, circles/arcs, and splines), as follows.

Type

View only. Element type, such as point, line, arc (CW=clockwise, CCW=counterclockwise) or spline.

ID

View only. The unique identifier for the element. Points have IDs of the form P*<n>*. Lines have IDs of the form L*<n>*. Circles and arcs have IDs of the form C*<n>*. B-splines (free-form curves) have IDs of the form B*<n>*.

WG

View only. The number and name of the workgroup that contains the element.

CS

The number and name of the coordinate system that contains the element.

Air/Wall

Elements other than points are designated either “Wall” or “Air”:

- Wall is the default setting for geometry. If default color settings are in effect, "Wall" geometry displays blue.
- Air changes the behavior of geometry when setting up machining operations. If default color settings are in effect, "Air" geometry displays red.

"Air" geometry acts as a constraint similar to regular geometry, except that the toolpath will overhang this area by the amount specified in the machining dialog. See the *Mill* guide for information regarding machining with "Air" geometry and Overhang.

An additional use for "Air" geometry is in profiles for 2D Form tools. Geometry designated as "Air" will become a non-cutting edge of the 2D Form tool.

Rapid / Feed

Lines can be designated either “Rapid” or “Feed”:

- Feed is the default setting, indicating that the tool will move at the speed specified for feedrate. Lines designated as Feed display solid.
- Rapid indicates that the tool will move at the speed specified for Rapiding. Lines designated as Rapid display dashed.

Depth

when the Polar & Cylindrical Milling option is installed, a workgroup can be specified as Rotary, allowing geometry and toolpath to be wrapped about the C axis by rotary interpolation of the C axis in milling operations. The Depth value allows you to specify a radius value for each element in a Rotary workgroup. It takes effect only when the workgroup is in radial mode – that is, when the Wrapped checkbox is selected for its Part Geometry. For full information on Rotary workgroups and radial mode, see the *Mill/Turn* guide.

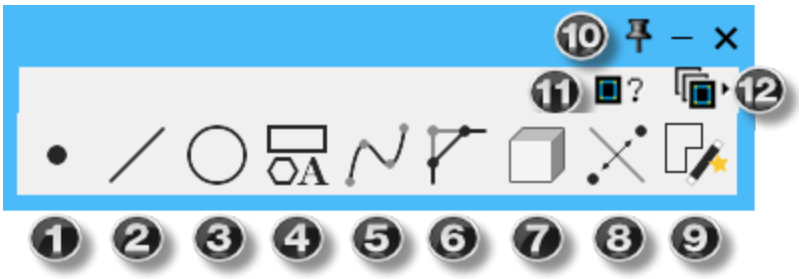
To modify geometry properties

1. Select one or more rows in the table of elements.
2. Select one or more option buttons of:
   - Rapid or Feed
   - Air or Wall
   - Not wrapped
     (Or, to set all selected rows to a wrap depth, select the Wrap Depth option button and enter the radius value.)
3. Click Apply to Selected Items.

Geometry Palette Interface

The Geometry Creation palette contains all of the text creation tools including free-form CAD tools geometry from solids tools and Geometry Expert. The buttons in the palette access sub-palettes and dialogs that allow the user to enter feature specifications and create shapes.

The Geometry palette is where all forms of geometry are created. Sub-palettes create specific types of geometry. The Connect/Disconnect button creates connections, terminators or disconnects features. Geometry Expert allows geometry shapes and features to be defined quickly with a table.



1. Point
2. Line
3. Circle
4. Shape
7. Geo From Solids
8. Connect/Disconnect
9. Geometry Expert
10. Dialog controls

5. Curve
6. Chamfer/Fillet
11. Workgroup Info (Level 1 only)
12. Workgroup list (Level 1 only)

Palette Shortcuts

All nine buttons in the Geometry Creation palette can be accessed from the keyboard by entering the number of their position in the palette, 1 through 9. For example, type a (`1`) instead of clicking on the Point button, a (`2`) for the Line button, a (`3`) for the Circle button etc.

Sub-Palettes

The Geometry palette contains seven sub-palettes that create free-form points, lines, circles, shapes, curves, fillets or chamfers and geometry from solid features. Once a sub-palette is open a selection mode is activated allowing multiple items to be selected without requiring the `Ctrl` key to be pressed.

 **Return**

This button, contained in each sub-palette, will return to the main Geometry palette. Alternately you may press the `Esc` key within any sub-palette.

Descriptor Points

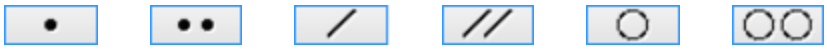
Some geometry dialogs which require point data have a `P` (Point) or `CP` (Center Point) button which allows users to enter the point data directly into the line or circle geometry dialog. When a Descriptor Point button is pressed, the geometry dialog extends so that coordinate for that point can be input. Descriptor Points are not created, or drawn on the screen. In this way they don't clutter the workspace with points that would later become deleted. Descriptor Points are used when a current point is not available to be selected for the particular geometry feature creation.

Dimension Labels

The dimension labels (XYZ) will vary depending on the machine type and/or Coordinate System. For more information, see Coordinate Systems.

Single/Multiple Feature

When creating features some dialogs have two buttons. The single feature button will create the feature and return to the main Geometry palette. The multiple feature button will create the feature and keep the dialog open to create more of the same type of geometry. Press `Enter` to activate the selected button, and pressing `Shift+Enter` will activate the Multiple button.



Point creation | Line creation | Circle creation

Smart Selection and Inferred Features

When you open a sub-palette the system looks at any selected geometry. The system assumes that you want to use the geometry that is selected to define a feature. If only one kind of feature can be defined from the selected geometry the system will automatically take you to the dialog for that feature type. There are variations on this, Smart Selections and Inferred Features.

Smart Selection

A Smart Selection occurs when there is only one possible solution to the combination of selected geometry and the feature type chosen. Think of this as a shortcut.

Example 1
If you have 2 points selected and you click on the Line button the system will take you to the Line Between Two Points dialog with the point data already entered.

Example 2
If the same two points were selected and you clicked on the Circle button the system will take you to the Radius and Two Points dialog.

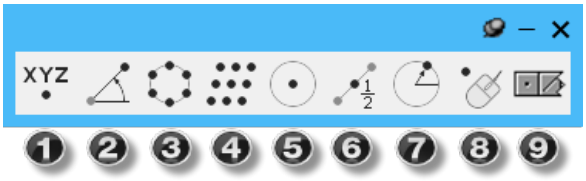
Example 3
If three points are selected and you click the circle button then the system takes you to the Three Features dialog.

Inferred Features

Inferred Features are solutions to creating geometry that do not have a dialog. These are features tangent to two other features. Inferred Features include a point tangent to two features, a line Tangent to a point and a circle, a line tangent to two circles, a circle tangent to two lines, a circle tangent to two circles, a circle tangent to a line and a circle and a circle tangent to a line and a point. Each of these items are fully detailed in Point Sub-Palette, Line Sub-Palette and Circle Sub-Palette.

• Point Sub-Palette

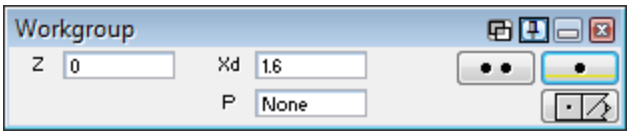
The Point sub-palette which contains a variety of options for creating points in 2D and 3D space. Other options for creating points are available if features are selected prior to opening the Point sub-palette.



1. Explicit
2. Polar Point
3. Bolt Circle
4. Matrix
5. Center Point
6. Mid-Point
7. Point On Arc
8. Mouse Point
9. Return button

XYZ

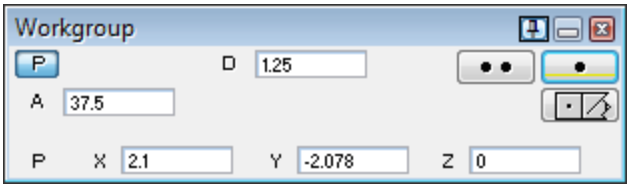
Explicit
An Explicit point can be created by entering coordinate values for that point. Turning dialogs will only contain two axis coordinates as shown. The P value can be used to move existing points by entering coordinates and the point label, for instance P21 and the creation buttons will modify point 21 to the coordinates entered.



Mill Turning

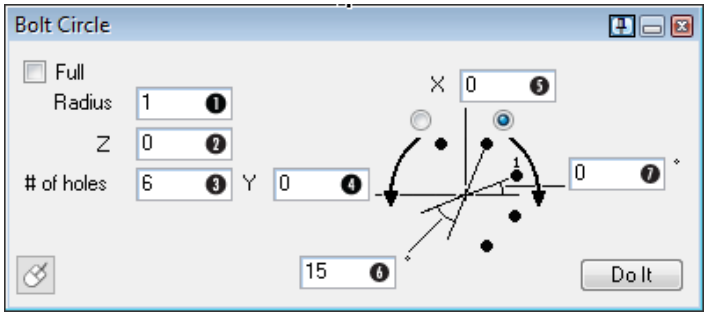
Polar Point

Creates a point at some angle (A) and distance (D) from an existing point (P) or specified coordinates.



Bolt Circle

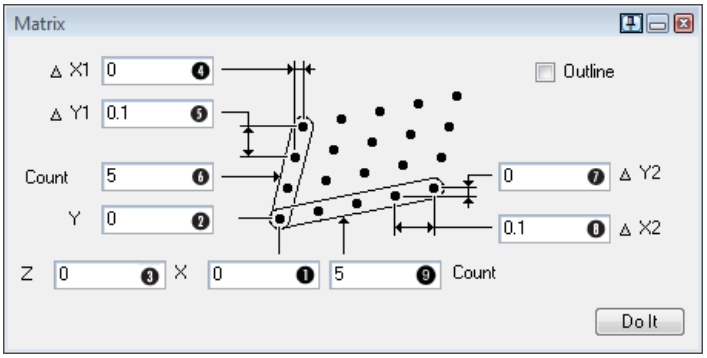
Creates a circular pattern of points. When the Full box is checked, the Degrees Between Points text box is not available. In this case, the system will evenly space the given number of points to complete the circle.



1. Radius of Circle Pattern
2. Z Depth of Points
3. # of Points in the Pattern
4. Y Coordinate of Pattern Center Point
5. X Coordinate of Pattern Center Point
6. Degrees Between Points
7. Angle to 1st Point

Matrix:

Creates a parallelogram pattern of points. When the Outline box is checked, the system will only create the points that define the outline of the parallelogram.



1. X Coordinate of 1st Point
2. Y Coordinate of 1st Point
3. Z Depth of Points
4. Side 1 Change in X
5. Side 1 Change in Y
6. # of Points in Side 1
7. Side 2 Change in X
8. Side 2 Change in Y
9. # of Points in Side 2

Center Point

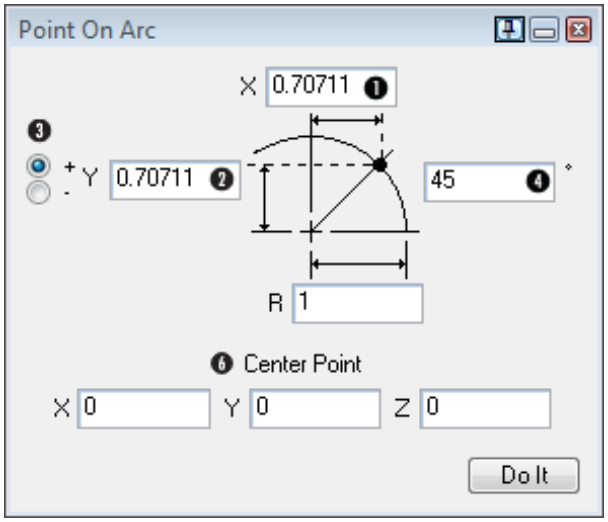
Creates a point at the center of an existing circle. You can also do this function to multiple selected circles. When performing this action be sure you are in the Point Sub-Palette or the Center Point dialog before selecting all the circles you want to make center points for. The Shift-Drag selection works with this feature. Please note that the dialog will still display only the first selected circle's label.

Mid-Point

This will create an unconnected point at the mid-point of a selected line or arc.

Point on Arc

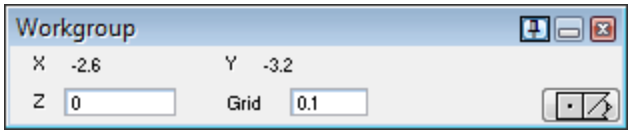
Creates a point along an arc. The radius of the arc and one other value must be entered to calculate the point's position. The variables other than the arc's radius include the angle of the arc, the known `X` coordinate of the point and the known `Y` coordinate of the point. There is a radio button to switch between positive and negative `Y` values.



1. X Coordinate of a known point
2. Y Coordinate of a known point
3. Toggle Between Y+ and Y- value
4. Angle of Point on Arc
5. Radius of Arc
6. Center Point of Arc

Mouse Point

Create point using the mouse. `Click` to create the points. Points will snap to the specified `Grid`. The depth of the points may be changed by modifying the value in the `Z` box.

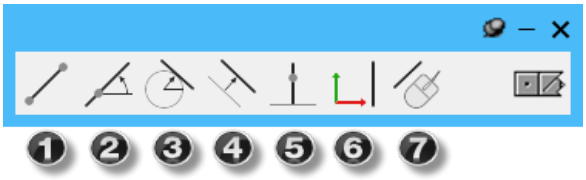


Point Tangent to Two Features

This is an inferred feature. If two features (two lines, two circles or a line and a circle) are selected and you click on the Point button the system will attempt to create a point wherever the features intersect. If the features do not intersect or are not tangent nothing will happen otherwise you will be prompted to choose the proper solution.

Line Sub-Palette

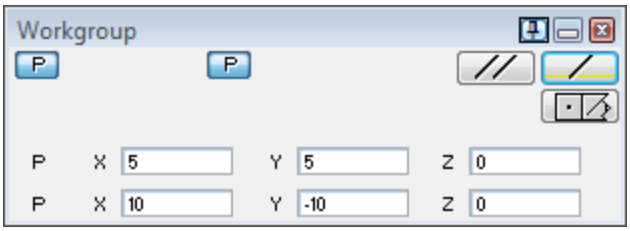
The Line sub-palette creates various types of lines. Lines can be created based on selections. Click to view



1. Two points
2. Point Angle
3. Angle & Tangent Circle
4. Parallel
5. Perpendicular
6. Axis
7. Mouse Line
8. Return button

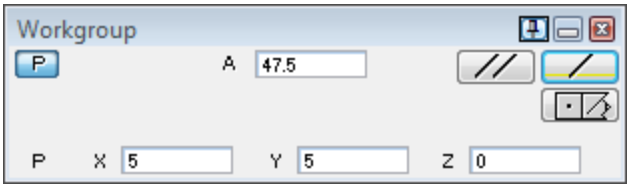
Two Points

Creates a line through two points. The points may already exist, or they can be manually entered by clicking on the P button(s).



Point Angle

Creates a line through an existing point at a specified angle. The point may already exist, or it can be manually entered by clicking on the P button.



Angle & Tangent Circle

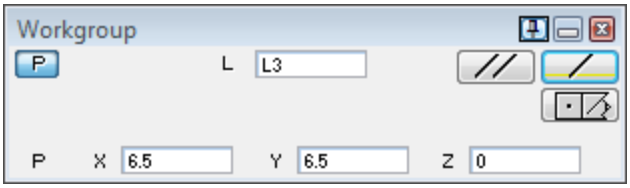
Creates a line tangent to the selected circle at a specified angle.

Parallel

Creates two parallel lines at a specified distance from the selected line.

Perpendicular

Creates a line perpendicular to a selected line through a specified point.

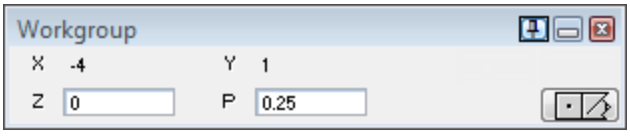


Axis

Creates a horizontal or vertical line parallel to an axis at a specified distance from the origin.

Mouse Line

Creates connected lines using the mouse. `Click` to create endpoints. Points will snap to the specified `Grid`. The `Grid` can be modified while drawing lines as well as the depth value. Lines can be designated as `Rapid` geometry. `Double-click` to create a terminator point, or to exit the dialog. `Click` the first point to close a shape.



Line Tangent to a Point and a Circle

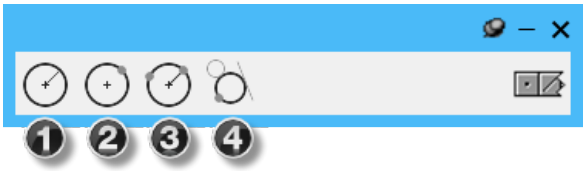
This is an inferred feature. When a point and a circle are selected and you click on the Line button, the system will make a line that is tangent to the point and line. If the point and circle are coincident, then the line will be tangent to the circle at the point; otherwise you will have to choose which the proper solution for a line tangent to the circle through the point.

Line Tangent to Two Circles

This is an inferred feature. When two circles are selected and you click on the Line button the system will make a line that is tangent to the circles. As there is more than one possible solution, you will be prompted to choose the line you need.

Circle Sub-Palette

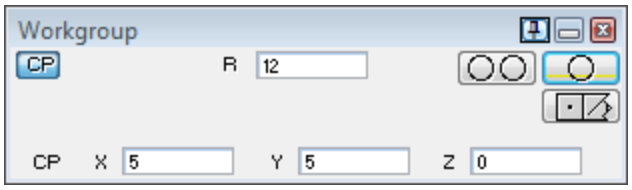
The Circle sub-palette contains options for the creating various circles



1. Radius & Center Point
2. Point & Center Point
3. Radius & Two Points
4. Three Features

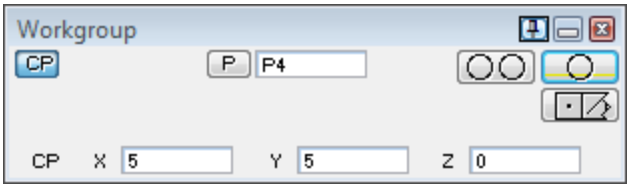
Radius & Center Point

Creates a circle using a selected point or entered coordinate value for the center point and a radius. You may also create circles of the same diameter from multiple selected points. To use this function be sure to access the Circle Sub-Palette or the Radius & Center Point dialog before selecting the points you wish to create circles around. The Shift-Drag selection works with this feature. The dialog will still only show the first selected point's label but clicking the Create Circle button will generate a circle around each point.



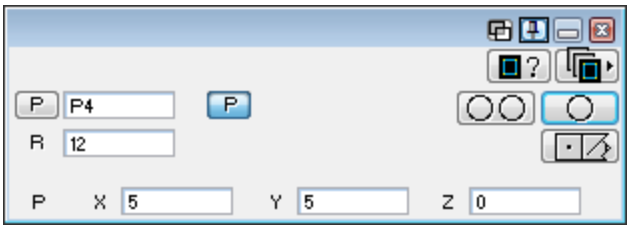
Center Point-Circumference

Creates a circle by selecting a point for the center point and another as a point on the circumference of the circle. The center point may also be entered manually.



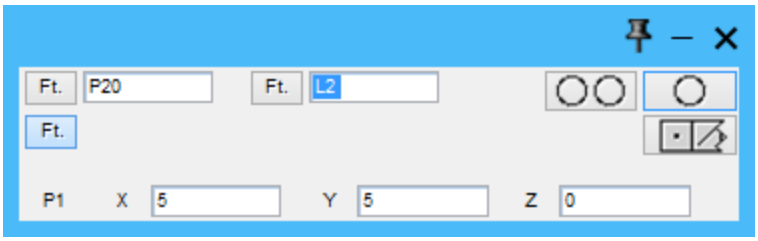
Radius & Two Points

Creates a circle using a radius value and two tangent points to define the center point of the circle. The points may be selected or entered manually.



Three Features

Create a circle by selecting any combination of three points, lines or circles for the circle to intersect or be tangent to.



Circle Tangent to Two Lines

This is an inferred feature. When two lines are selected you can create a circle tangent to the two lines. You will be prompted to specify the circle's radius. When determining the possible points of tangency the length of the line is not considered; internally the system extends the lines out as far as is needed to determine tangency. If the radius is too small to create tangency nothing will happen. As there is more than one possible solution you will be prompted to choose the line you need.

Circle Tangent to Two Circles

This is an inferred feature. When two circles are selected you can create a circle tangent to the two circles. You will be prompted to specify the tangent circle's radius. If the radius is too small to create tangency nothing will happen. As there is more than one possible solution you will be prompted to choose the line you need.

Circle Tangent to a Line and a Circle

This is an inferred feature. When a line and a circle are selected you can create a circle tangent to the two features. You will be prompted to specify the tangent circle's radius. If the radius is too small to be tangent to the features nothing will happen. As there is more than one possible solution you will be prompted to choose the line you need.

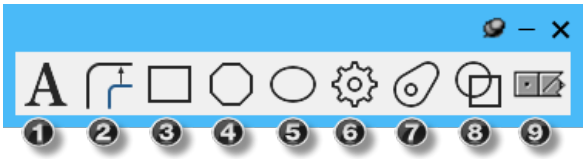
Circle Tangent to a Line and a Point

This is an inferred feature. When a line and a point are selected you can create a circle tangent to the two features. You will be prompted to specify the tangent circle's radius. If the radius is too small to be tangent to the features nothing will happen. If the point and line are coincident the circle will be tangent to the line at the point. There are two possible solutions to this tangency so you will be

prompted to choose the circle you need. If the line and point are not coincident you will have to choose which the proper solution.

Shape Sub-Palette

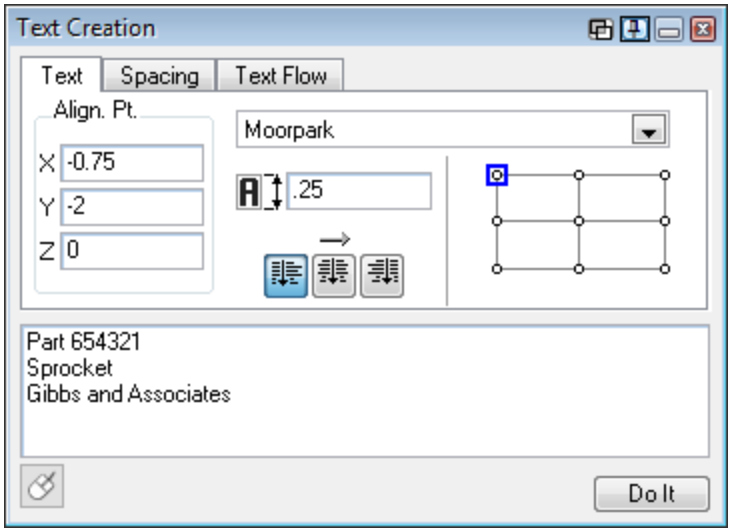
The Shape sub-palette can create various types of shapes including text, offset shapes, rectangles, polygons, ellipses, gears and cams.



1. Text Creation
2. Offset
3. Rectangle
4. Polygon
5. Ellipse
6. Gear
7. Cam
8. Combine Shapes / Trim Shapes
9. Return button

Text Creation

The Text Creation dialog creates spline geometry from any TrueType font and allows you to select the typeface, text size, justification, position, text flow, spacing between characters, words and lines.



Text Tab:
The specifications entered in this window establish what the text will look like and where it will be positioned on the part.

Font List:

This is a list of all typefaces that are available to create the text. Only TrueType fonts (typefaces) can be used to create geometry from text.

Text Size:

This value specifies the height (measured in either inches or millimeters) of a capital 'A' in the selected type face. All other characters will be sized proportionally.

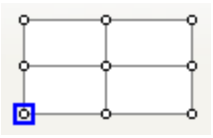
Justification:

These buttons specify whether the text should be left-justified, centered or right-justified. If left justified is selected, all text is aligned so the left edge of each line is in the same X location (or if radial is selected, the same angle). If centered is selected, the center of each line of text is calculated and the text is positioned so the centers all share the same X location or angle. If right-justified is selected, the text is aligned so the right edge of each line is in the same X location (or if radial is selected, the same angle). Justification will only have an effect if there is more than one line of text. The longest line of text is used to specify the left or right edge. All text is then aligned based on one of these locations.

Align. Pt. / Center Pt.:

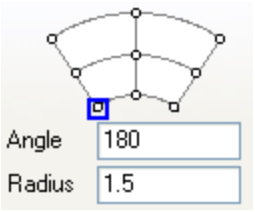
The X, Y and Z values that will be used as the starting point for text creation. The text, Straight or Radial, will be aligned at this point.

Straight Text Alignment:



Text can either be created in a straight line or along an arc. This is designated in the Text Flow window which is described later in this section. If straight text is created, a rectangular area that will contain the text is used for positioning. The diagram specifies how the text will be positioned within the rectangular text area based on the coordinates of the Alignment Point. For example, if the position highlighted in the diagram shown above is selected, the text will be positioned in the lower left corner of the rectangular text area at the Alignment Point. The bottom of the text will be at the Y value of the Alignment Point and the left edge of the text will be at the X value of the Alignment Point. Clicking on the circular position points selects the different possible alignment locations.

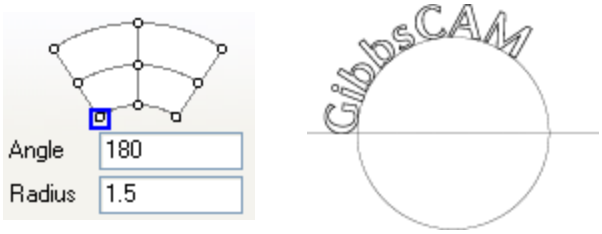
Radial Text Alignment:



If text is created along an arc, a radial area that will contain the text is used for positioning. The radial text area is created by entering an angle value that specifies where along the arc the text will begin and a radius that specifies the size of the arc. The Alignment Point coordinates specify the center point of the arc. The diagram specifies how the text will be positioned within the radial text area. Examples with the Radial Text Alignment information and the resulting text are shown.

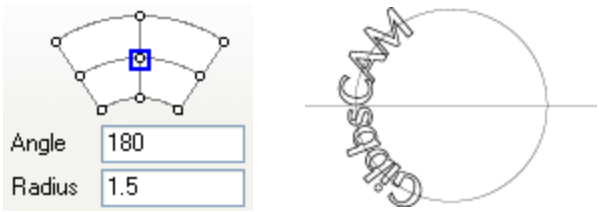
EXAMPLE 1: Inner Radius/Start on Angle:

The picture shows the text as well as the arc and line which designate the radial text area and the starting location of the text. The position selected on the diagram will create the text along the outside of the circle beginning at the line specified by the Angle (the circle and line are drawn only to better visualize the example).



EXAMPLE 2: Center Radius/Center on Angle:

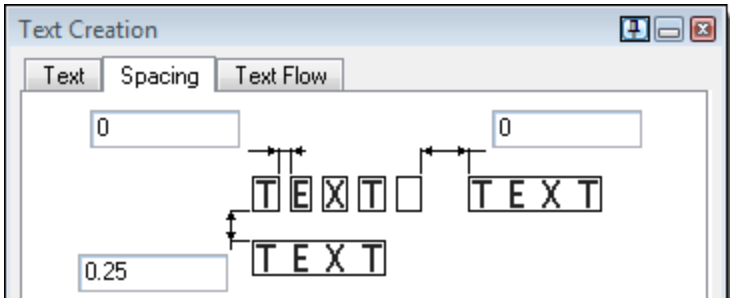
The picture shows the text as well as the arc and line which designate the radial text area and the starting location of the text. The position selected on the diagram will create the text along the centerline of the circle and the center of the line of text will lie along the line specified by the Angle.



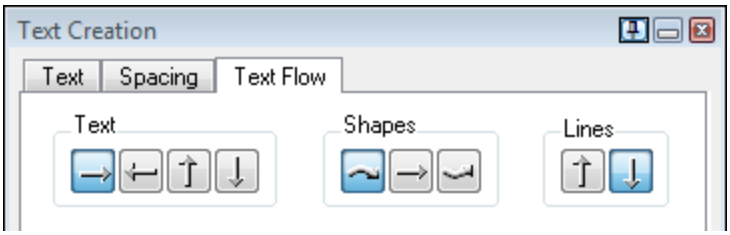
EXAMPLE 3: Outer Radius/End on Angle:

The picture shows the text as well as the arc and line which designate the radial text area and the starting location of the text. The position selected on the diagram will create the text along the inside of the circle ending at the line specified by the Angle.

Spacing Tab:
These options allow you to enter distance between letters, between words, and between lines of the text. Values entered in this window will be added to or subtracted from the standard spacing.



Text Flow Tab:
This section designates the text flow, line flow and shape of the text.



Text:

Text can either be created horizontally or vertically. The first two buttons create text (characters) that will flow along a horizontal line, either from left to right (right arrow) or right to left (left arrow). The last two buttons will make the text flow along a vertical line, either from bottom to top (up arrow) or from top to bottom (down arrow).

Shapes:

These buttons designate whether the text will be created in a straight line or on an arc (these buttons will change in appearance when creating vertical lines of text, but their functionality remains the same). The first button will make the text flow along the arc specified in the Text tab window in a clockwise direction. The middle button creates text in a straight line, and the third button creates text on an arc in a counter-clockwise direction. The selection made for Shapes will affect the Alignment Diagrams in the Text tab window.

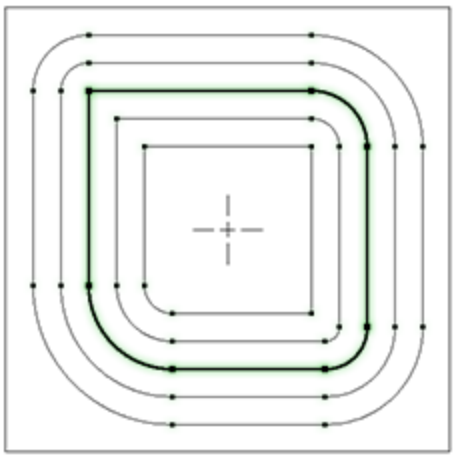
Lines:

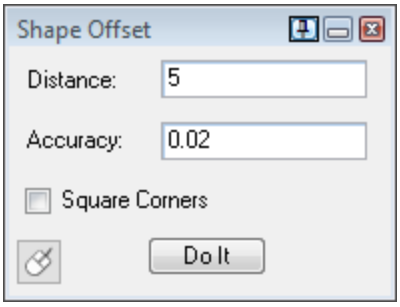
These buttons only have an effect if multiple lines of text are being created (these buttons will change in appearance when creating vertical lines of text, but their functionality remains the same). The first button specifies that the lines flow from bottom to top for horizontal lines of text or from left to right for vertical lines. The second button specifies that horizontal lines of text flow from top to bottom or vertical lines flow from right to left.

As with many other Shape dialogs, the lower left corner provides a button: Mouse Shape lets you create shapes with a quick click-slide-click: `Click` to anchor, slide to get a ghosted preview, and then `click` to create the shape.

Offset

The Shape Offset dialog allows shapes to be offset by a specified distance. Two offset shapes will be created. Offset shapes will adjust to the exact distance entered by a radius, so fillets and corners are added to the offset geometry as illustrated in the image. (To prevent automatic fillets and corners, select Square Corners.) The Accuracy value affects splines only.

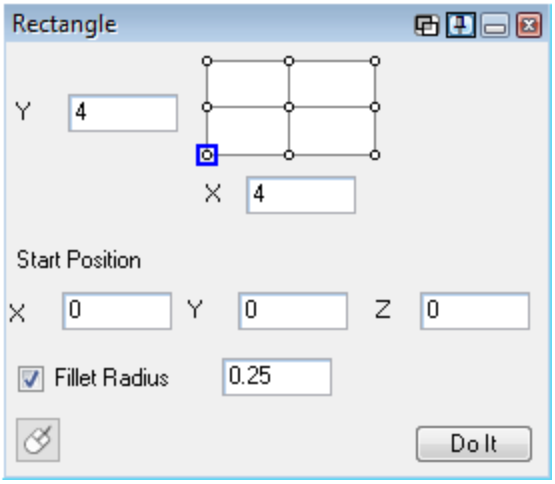




As with many other Shape dialogs, the lower left corner provides a button: Mouse Shape lets you create shapes with a quick click-slide-click: `Click` to anchor, slide to get a ghosted preview, and then `click` to create the shape.

Rectangle

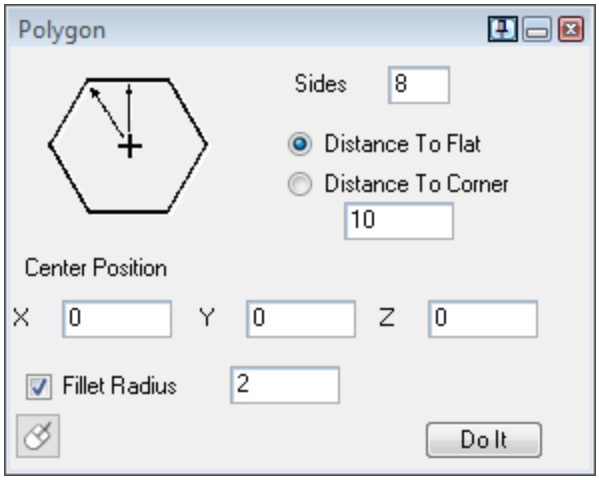
The Rectangle dialog creates rectangles and squares and can automatically add Fillet Radius corners. Define the total length of each side of the rectangle for X and Y, the Start Position of the rectangle by entering the X, Y, and Z coordinates, and click the position on the rectangle to use for the Start Position as the center point, an edge midpoint, or a corner point. 



As with many other Shape dialogs, the lower left corner provides a button: Mouse Shape lets you create shapes with a quick click-slide-click: `Click` to anchor, slide to get a ghosted preview, and then `click` to create the shape.

Polygon

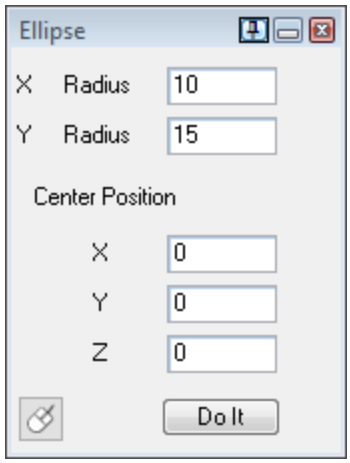
The Polygon dialog creates multi-sided shapes (polygons). Specify the number of Sides, the Center Position, the Distance To Flat or Distance To Corner. There is also a checkbox which allows the user to specify a Fillet Radius on each corner of the shape.



As with many other Shape dialogs, the lower left corner provides a button: Mouse Shape lets you create shapes with a quick click-slide-click: `Click` to anchor, slide to get a ghosted preview, and then `click` to create the shape.

Ellipse

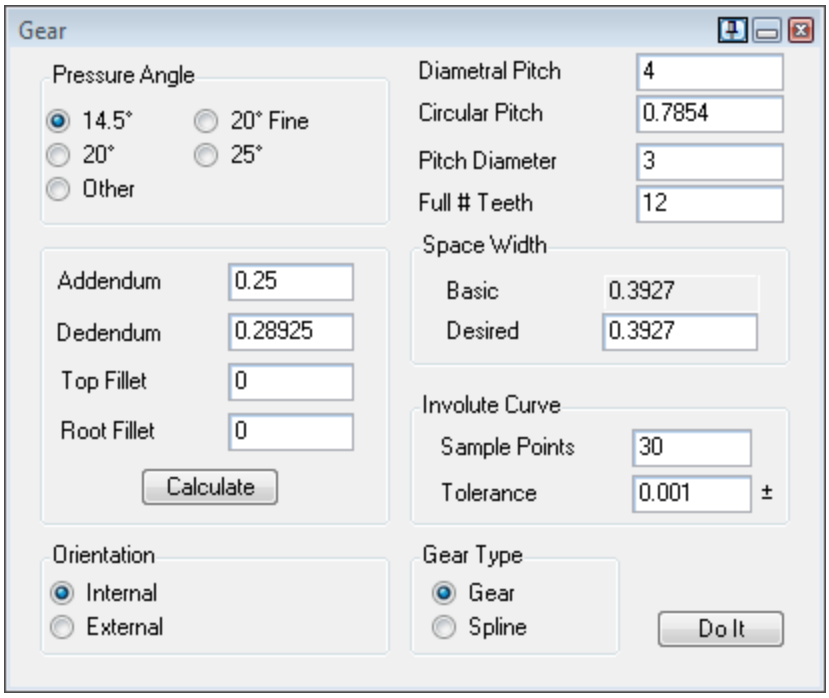
The Ellipse dialog creates vertical and horizontal spline ellipses. Enter the X and YRadius, and the Center Position. A larger X value creates an horizontal ellipse while a larger Y value creates a vertical ellipse.



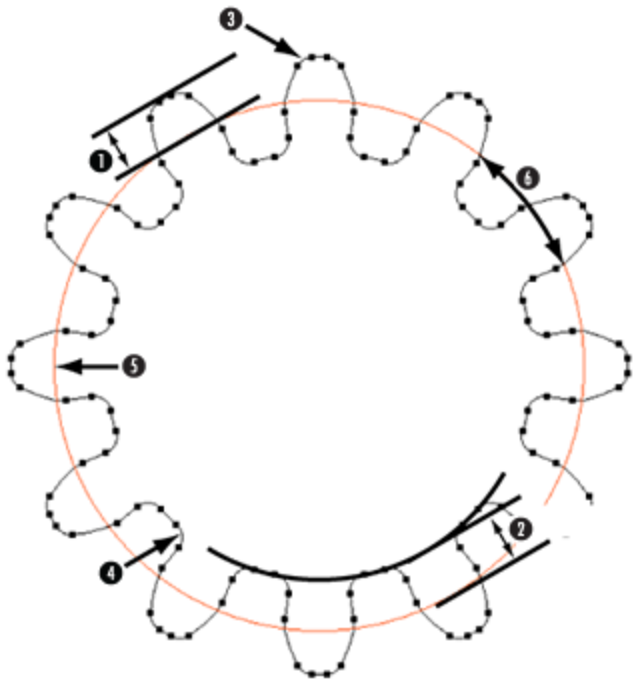
As with many other Shape dialogs, the lower left corner provides a button: Mouse Shape lets you create shapes with a quick click-slide-click: `Click` to anchor, slide to get a ghosted preview, and then `click` to create the shape.

Gear

The Gear dialog creates complex gears by defining a single tooth of the gear. Using Modify > Duplicate And... > 2D Rotate to create the rest of the gear. For more information about creating gears, see the Geometry Creation tutorial.



1. Addendum / Major Diameter
2. Dedendum / Minor Diameter
3. Top Fillet
4. Root Fillet
5. Diametral Pitch
6. Circular Pitch



Gear Definition

The Machinery's Handbook is an excellent source for information on Gearing. All of the calculations and formulas are based on this information (ANSI standard B6.1-1968).

Pressure Angle
The angle between the top of the tooth and the point of intersection between the tooth and its Pitch Diameter. Select Other to specify a different value.

Addendum / Major Diameter
The radial distance between the pitch circle and the top of the tooth. Changing the gear type from Gear to Spline changes Addendum to Major Diameter.

Dedendum / Minor Diameter
The radial distance between the pitch circle and the bottom of the tooth. Changing the gear type from Gear to Spline changes Dedendum to Minor Diameter.

Top Fillet
The convex portion of the tooth where it joins the top of the tooth.

Root Fillet
The concave portion of the tooth where it joins the bottom of the tooth.

Calculate
The values for Addendum and Dedendum (or, when Gear Type is Spline, for Major Diameter and Minor Diameter) and values for Top Fillet and Root Fillet can be calculated from the Pressure Angle, Pitch diameter, Diametral Pitch, Circular Pitch and Full # Teeth.

Orientation
Specifies whether the gear is internal or external.

Diametral Pitch
This is the ratio between the number of teeth and the number of inches/mm of pitch. This value can be used to Calculate the other dependent values for the gear.

Circular Pitch
The length of the arc of the pitch circle between the center and adjacent teeth. This value can be used to Calculate the other dependent values for the gear.

Pitch Diameter
The diameter of the pitch circle. This value can be used to Calculate the other dependent values for the gear.

Full # Teeth
The number of teeth that a gear will contain. This value will be automatically created from the Diametral Pitch and Pitch Diameter values entered. Remember, although that the full gear is not created until the user creates the teeth with the Duplicate And... feature. The Full # Teeth value minus one will be entered into the Duplicate And... text box for full gear creation.

Space Width / Tooth Width
Circular thickness: Space Width for Internal orientation, or Tooth Width for External orientation. It describes the space between the teeth of the gear. The Basic value is calculated for users based upon other defined specifications of the gear; however, you can bypass this value by defining your own value in the Desired text box. The value is calculated to a Basic or zero slope value. A tooth can be narrowed or a space can be enlarged depending on the gear to be defined.

Involute Curve
A profile of one side of a gear tooth. The system will translate this data into a cubic spline which is more readily machined, because an involute curve cannot be directly interpolated by a CNC.

Sample Points
The number of points along the gear's Involute Curve to take into consideration when calculating the spline.

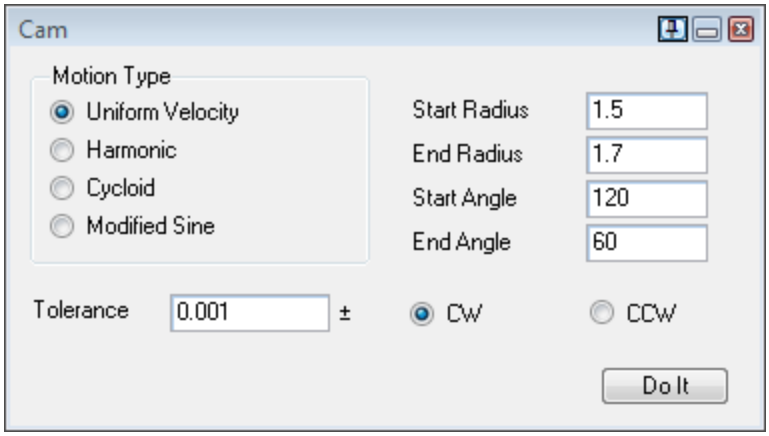
Tolerance
How close to the Sample Points the spline must pass.

The system will very closely approximate the Involute Curve with the default values, but tighter tolerances may be used if needed. Notice that this is independent of Machining Preference: To hold a 0.001" Tolerance to the Involute Curve, use a Tolerance of 0.0005" in the gear creation dialog and a Tolerance of 0.0005" in the Machining Preference.

Gear Type
Select the type of gear to be created.

Cam

This dialog helps you create the geometry for cams.



Motion Type:
This selection will determine the shape of the cam. Depending on the need of the user., the motion type may be selected from Uniform Velocity, Harmonic, Cycloid, or Modified Sine.

Tolerance:
This is the greatest distance allowed between the points on the cam and the arcs generated to approximate the best possible cam. In general, a large value here creates fewer arcs, but the deviation between the desired shape and the created shape is large. A small value should result in a more accurate shape with significantly more arcs.

Start Radius:
This is the smallest circle drawn to the cam shape. It is also commonly referred to as the Base Circle.

End Radius:
This is the largest circle that the cam shape will be drawn to.

Start Angle:
This is the angle from which the motion of the cam will begin.

End Angle:
The angle at which the motion will end.

CW / CCW:
This selection will dictate the direction of the motion from the Start Angle to the End Angle, either clockwise (CW) or counter clockwise (CCW).

Combine Shapes / Trim Shapes

This geometry palette lets you manipulate shapes as regions that can be combined, trimmed, or broken into segments. Simple examples are shown in sample parts `Geometry_Shape*.vnc`, and

use cases are illustrated in the workgroups of sample part `Geometry Region Function Palette.vnc`.

Shape Union

Creates the smallest shape that includes all selected regions, selected in any order.

Shape Subtraction

Starting from the first-selected region, subtracts the next-selected region or regions.

Shape Intersection

Creates the largest shape that is shared in common by all selected regions, selected in any order.

Trim Shapes In

Snips off outer geometry and/or regions from elements containing portions that lie outside the region. If two or more regions are in the selection set, elements/regions are trimmed against the largest region.

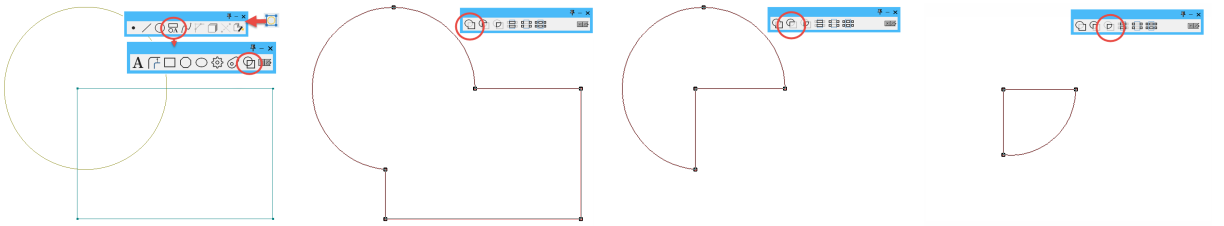
Trim Shapes Out

Snips away inner geometry and/or regions from elements containing portions that lie inside the selected region. If two or more regions are in the selection set, elements/regions are trimmed against the first-selected region.

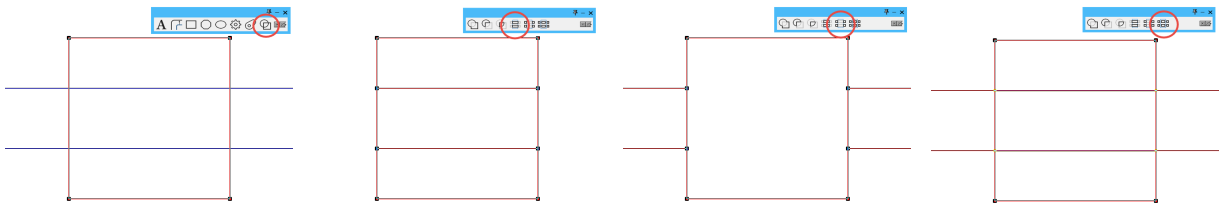
Slice Shapes

Breaks overlapping elements in the selection set, creating separate elements so that none overlap.

Each of the Combine / Trim functions adds terminator points wherever needed.



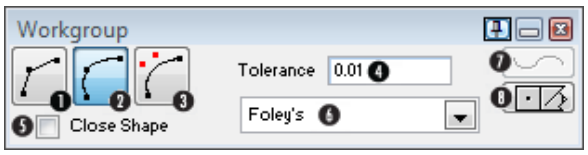
Sample part: "Geometry_Shape_Combine.vnc"	Union	Subtraction	Intersection



Sample part: "Geometry_Shape_Trim.vnc"	Trim Shapes In	Trim Shapes Out	Slice Shapes

Curve Sub-Palette

The Curve sub-palette which provides three methods for creating curves through a series of pre-points. The curve will travel through the points in the order they were selected. Modify > Sort can be used to sort the points in the correct order to create the desired curve. After the curve is created, you can right-click it and use **Edit Spline** to modify it.



1. Line Fit button
2. Curve Fit button
3. Control Point button
4. Tolerance setting
5. Close Shape checkbox
6. Blending Method menu
7. Do It button
8. Return button

Line Fit

This method creates straight lines between the selected points. If a Tolerance is specified, any points that are co-linear within the specified tolerance will be approximated by a single line, rather than several small lines between the points. If Close Shape is turned on, the system will create a closed shape, meaning that the first and last lines will be connected. The Blending Method pop-up menu is not applicable when using the Line Fit method.

Curve Fit

This method creates a B-Spline curve through the selected points. The Tolerance value is designed to reduce the mathematical complexity of the finished curve and should only be larger than 0.00 when generating curves from a large number of points. When a 0.00 tolerance is used, the system will create a smooth curve that passes through every selected point. When a value larger than 0 is used, the system uses a different method to produce the curve, and any points that lie within the Tolerance specified will be skipped. Even if no points lie inside the Tolerance, a completely different curve will be generated by the system than if a 0.00 Tolerance was used. The curve generated by zero Tolerance will be smoother than a curve generated by a non-zero Tolerance.

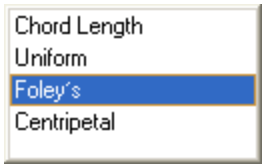
Control Point

This method produces a B-spline curve using the selected points as control points. When using control points, only the first and last point must actually lie on the curve. The other points are used to specify the shape and direction of the curve. The Close Shape, Tolerance and Blending Method specifications are not applicable when using the Control Point method.

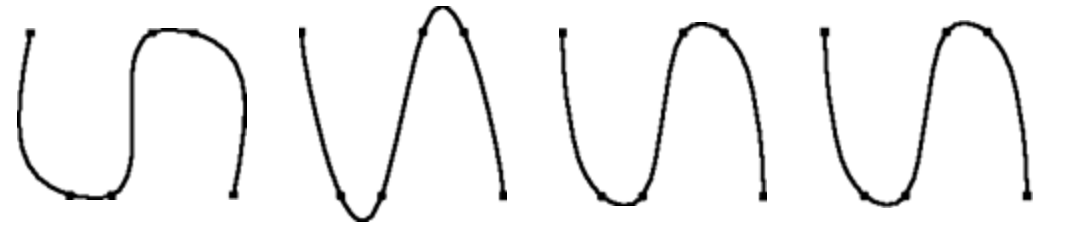
Close Shape

This will create a closed, continuous shape.

Blending Methods



The blending methods control how the curve will be approximated through the selected group of points. There are an infinite number of possible curves that can be drawn between a series of points. The way the curve is calculated between the points is based on the blending method. There are four options for the Blending Method. They are Chord Length, Uniform, Foley's and Centripetal. One way to explain the differences between these blending methods is to imagine a point traveling along the curve being created. The difference between methods is how much time the point spends traveling along the curve between the points that define the curve. The Chord Length and Uniform methods are general curve calculation methods. The Foley's and Centripetal methods are based on the Chord Length and Uniform methods and attempt to create a closer approximation of the desired curve. Each blending method is described below.



Chord Length | Uniform | Foley's | Centripetal

Chord Length

This method creates a curve between points proportional to the distance between points. That is, the farther apart two points are, the "longer time" it spends traveling between them, which is why it creates wider curves. The closer two points are together, the more the curve flattens out, because "less time" is spent traveling between the points.

Uniform

This method attempts to create curves of equal length between points of equal distance from each other.

Foley's

This method takes into account the angle between adjacent points. The larger the angle, the "more time" is spent between the points, causing the curve to be less pointed than the Uniform method.

Centripetal

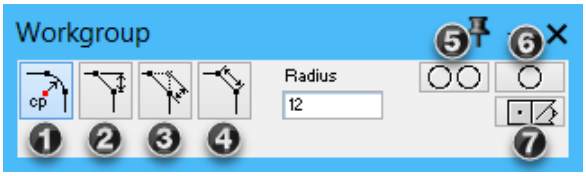
This method is based on the Chord Length method. It uses the calculated square root of the Chord Length method to blend the curve between the selected points. This makes for a slimmer curve.

For information on **Edit Spline** functionality, an option on the right mouse menu, see "Geometry Context Menus" on page 14.

Chamfer and Fillet Sub-Palette

The Chamfer-Fillet sub-palette provides options for creating fillets and chamfers. This sub-palette is only accessible when one or more points are selected. The options available in this palette will only

work on fully-connected points. Multiple fillets and chamfers can be created at one time by using this option when multiple points are selected.



1. Fillet button
2. Chamfer-Side
3. Chamfer-Depth
4. Chamfer-Length
5. Create multiple chamfer/fillets
6. Create single chamfer/fillet
7. Return button

Fillet button
Creates a fillet for selected connector points.

Chamfer-Side
Creates a chamfer based on a side value for selected connector points. Geometry Expert uses this type of Chamfer.

Chamfer-Depth
Creates a chamfer based on a depth value for selected connector points.

Chamfer-Length
Creates a chamfer based on a length value for selected connector points.

Multiple chamfers
Select one or multiple points to apply a chamfer/fillet, click this button to apply, dialog remains open.

Single chamfer/fillet
Select one or multiple points to apply a chamfer/fillet, click this button to apply. Dialog closes.

Return button
Return to previous level.

Connect/Disconnect

The Connect/Disconnect button is used to connect, disconnect and terminate geometry. It becomes available when either a point is selected or two intersecting/tangent features are selected, (this feature can also be accessed by pressing the 8 key on the keyboard). To connect geometry simply click this button.

Clicking on the Connect/Disconnect button will:

- Connect or disconnect tangent or intersecting geometry. The system will attempt to use an existing point at the intersection or tangency to connect features otherwise it will create a new point to connect the features. Refer to the Shapes and Connectors section in this chapter for details.
- Terminate or unterminate open shapes.

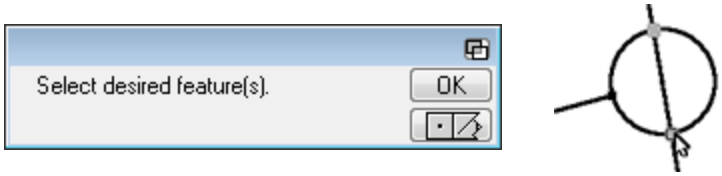
A point may have three states:

- Unconnected (the point is yellow and round).

- Being used as a connector between geometry features (the point is a blue square).
- Connected to geometry and being used as a terminator (the point is a yellow square).

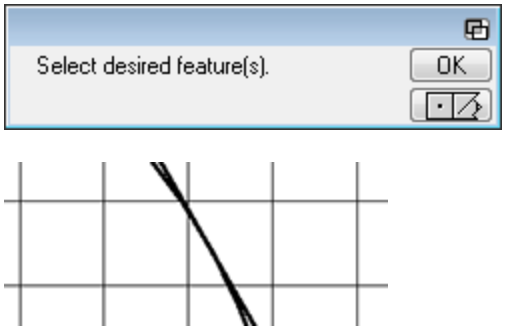
Multiple Connections

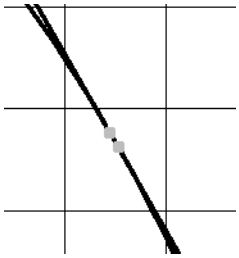
When connecting geometry that has multiple solutions, as with connecting a line to a circle, the Point Selection dialog will appear. The Point Selection dialog asks for the correct connection point (s) to be selected. You may select one or more intersection points. If only one intersection is available, the system will disable the other connection possibilities. Press the OK button after selecting the intersection to create the connection. Press the Esc key to cancel any connections.



Overlapping Connections

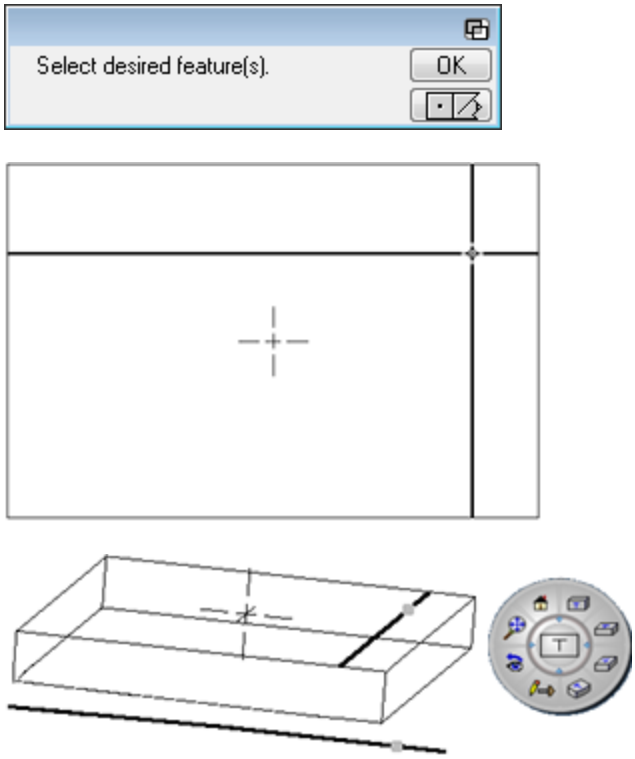
If the Point Selection dialog comes up when creating a connection but no points are visible, the points may be too close together to be displayed. You need to zoom in to reveal the points. This will generally happen with connections tangent to a circle. The following image illustrates two tangent points available in connecting two features. The line actually has two tangent points so close together that the points overlap and become invisible.





An example of a possible connection that cannot be seen (left), and the points after zooming in (right)

Sometimes connecting points cannot be displayed, such as when they are in the same horizontal and vertical position but at a different depth and the view is looking down the depth axis. When the possible connections are overlapping the system displays the cross hair as shown. Simply change the view orientation until the points become visible.



After rotating the geometry, the hidden connection points are revealed

Geometry From Solids Sub-palette

The Geometry from Solids sub-palette contains options for extracting edges and holes from solid models.

1. Geometry Extraction
2. Hole Extraction
3. Parting Line
4. Outline
5. Return button

Geometry Extraction

The geometry extraction function creates geometry from both edges and faces of solids and sheets. Your selection can include both edges and faces, i.e. a combination of both edges and faces are selected on a body. In order to view edges of a solid or sheet, the system must be in Edge Selection mode. Connected shapes will be created if the selected edges create a closed loop. Clicking on the Do It button in the Geometry Extraction dialog will create geometry from any selected edges or faces (in Face Selection mode).

Typically, this function will extract the selected edges as splines or curves. However, if the resulting spline edge can be converted to lines or circles within the specified tolerance, the extracted geometry will consist of lines and circles. The Geometry Extraction dialog allows users to enter a tolerance value which will be used to determine whether or not the selected edges will approximate lines and circles within the given tolerance. A large tolerance value will convert more of the edge splines to lines and circles, while a tight tolerance will keep the entities as they are defined in the part model. A tolerance of zero is recommended when extracting geometry that is definitely a circle or a line.

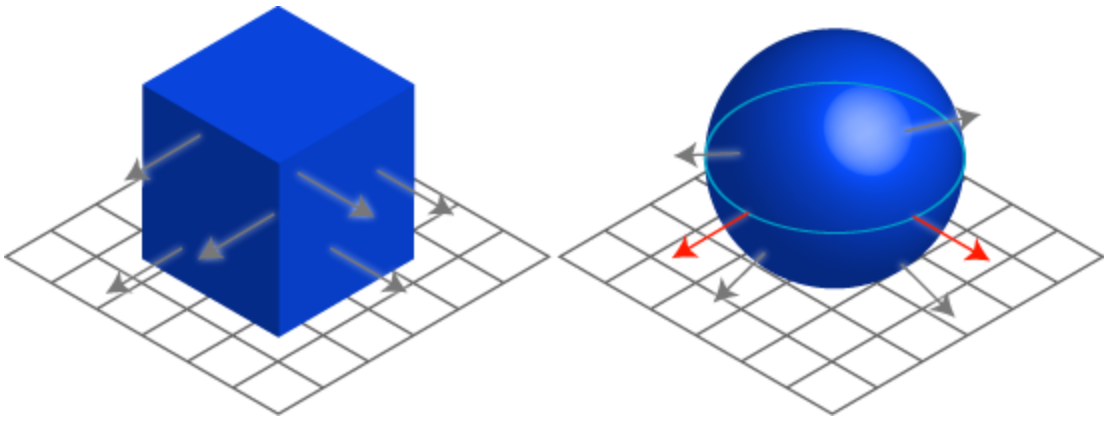
Hole Extraction

This function is used to create circles from holes in solids or sheets. This is very useful when a model contains drill holes. In order to create a drilling operation, points or circles must be selected. This function allows the user to extract circles from the existing holes on a model in order to have geometry to select for drilling operations.

When using this function, either a solid or sheet can be selected and the system will scan the faces of the selection and find all holes. A hole must have walls which are perpendicular to the current CS. Additionally, the edge loop of the hole must be either a circle or a spline – it cannot be a polyline (line segments approximating a circle). When hole extraction is performed, the resulting geometry will all be circles. Any hole edge loops which are splines but fall within the Circularity Tolerance specified will be extracted as circles. Edge loops which are splines that do not approximate a circle within the given tolerance will not be extracted. The depth of the extracted geometry will be based on the bottom of the hole(s), making it easy for the user to determine the depth for the drilling operation.

Parting Line

This function creates a parting line curve from selected faces or bodies. The parting line curve is generated where a face's surface normals are parallel to the current coordinate system but not on an edge. Think of this as a unique point on a face where the normals transition to and across parallel to the CS. This parting line curve can be used to help create a parting line surface. This function will not work on many shapes, such as cubes. A cube has an infinite number of points where surface normals are parallel to the CS. There are other tools in the system that may be used to get the boundary of shapes such as a cube.

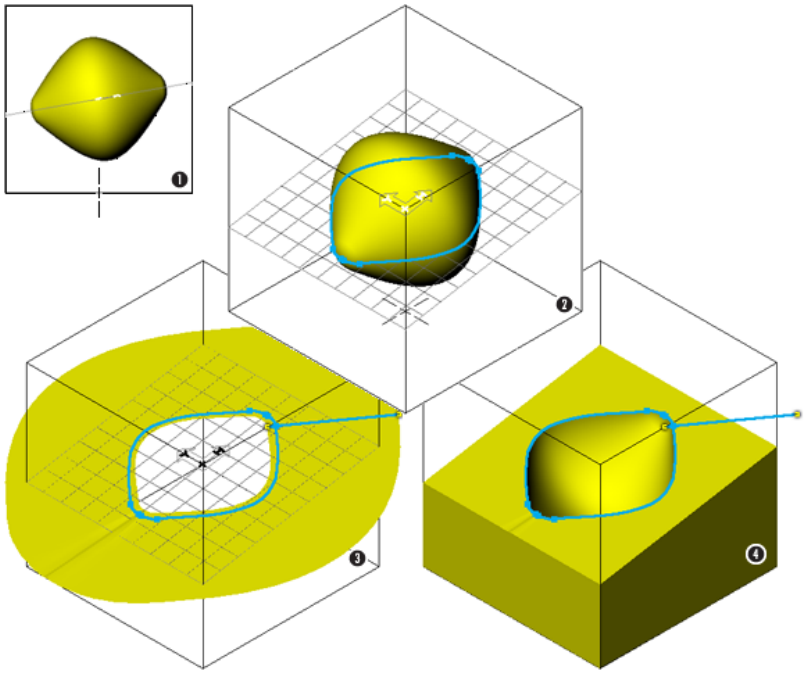


A cube showing many parallel normals and a sphere showing normals (in red) that provide a unique solution

As previously stated, this function can be used to help you create a parting line surface – which is the surface along which a mold pulls apart. Typically, a mold is pulled apart into two pieces, either two cavities or a cavity and a core. These two pieces must match and seal perfectly along the parting line surface in order for the mold to produce the desired part. With simple molds, the parting line is often a simple plane through the middle of the part. However, there are more complex situations where a planar parting line will not work.

Before using this function your model should be fully prepared for making a mold model, including adding any draft angles. To use this function, select all the faces that the parting line will be on or select the entire solid. The parting line function uses the depth axis of the current coordinate system as the draw axis – the axis on which the mold will be pulled apart. The parting line curve is the curve on which the surface normal vector is parallel to the coordinate system (or normal to the draw axis) at every point. The parting line function creates geometry which can then be used to create a parting line surface.

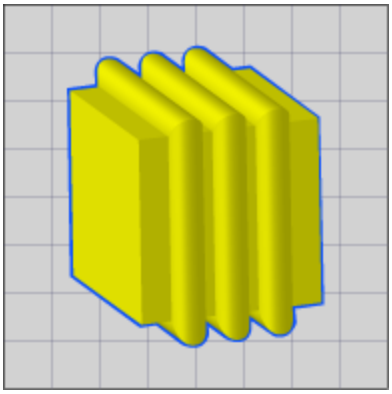
A good way to create a parting line surface from parting line geometry is to sweep a straight line along the parting line geometry, creating a sheet which will intersect the solid. The straight line is the drive curve and should intersect and slightly overlap the parting line geometry which is the base curve. Once the parting line surface is created, the part model can be subtracted from a cube to create the mold and then sliced with the parting line surface to create the two halves of the mold.



1. Original model
2. Parting Line geometry is created
3. A sheet is swept about the parting line geometry
4. The mold model is made using the swept shape

Outline

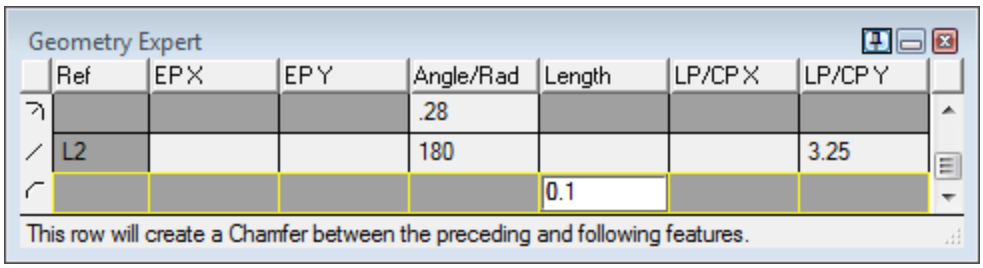
This function will create geometry that is an outline of the selected faces on single or multiple solids and/or sheets. The geometry is created at a depth of 0 in the current CS. This is useful for getting a profile of a shape where edges are not easily selected or available, where you may want the profile of a shape for an extrusion or to get the profile of a selected area for solid machining.



Geometry Expert

The Geometry Expert button opens the Geometry Expert dialog which looks and acts very much like a spreadsheet. Geometry Expert allows the user to create connected shapes by entering

feature dimensions in the rows of the spreadsheet. This feature is fully detailed in Geometry Expert Interface.



Geometry Expert Interface

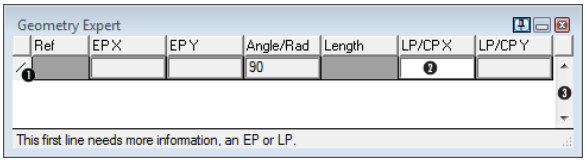
Creating a shape using the Geometry Expert is analogous to contouring a shape, indicating such items as location, direction and the distance being traveled. As the shape is created, geometric contour logic principles are followed to allow automatically connect and assume logical geometric shapes to require the least amount of input from the user. The Geometry Expert, as its name implies, provides the user with a built-in consultant on the rules and principles of geometry.

The Geometry Expert is a very powerful editing tool. Feature dimensions can be adjusted by simply changing the values in the cells. Geometry Expert handles all of the calculations and adjustments to the other features that are affected by the changes.

- How Geometry Expert Works
- Creating Shapes Using Geometry Expert
- Geometry Expert Table
- Additional Information
- -R Creation

How Geometry Expert Works

Geometry Expert is set up much like a standard spreadsheet. Feature specifications are entered into cells which are contained in rows. Each row of the spreadsheet defines a feature.



1. Feature Row
2. Cell
3. Scroll Bar

Row Border Color

The current row is outlined in either yellow or black. Yellow indicates the row contains adequate information. Black indicates the row does not contain enough information to define geometry. The prompt at the bottom will indicate the status of the row and define what is needed or what will be created from the current data.

Creating Shapes Using Geometry Expert

When creating a part using Geometry Expert, the first thing that must be decided is the starting feature and the direction to travel around the part—either clockwise or counter-clockwise. Features are defined in the order that they appear along the path of the shape. This process continues until the last feature of the shape intersect the first feature of the shape or end points are defined. If the shape is a closed shape the last feature row will be the Close Shape feature type.

Creating Features

When a row contains enough information to define the feature press the `Enter` key. The next row will be highlighted and the feature type may change or the `Angle/Rad` will switch based on the last row.

Feature Types

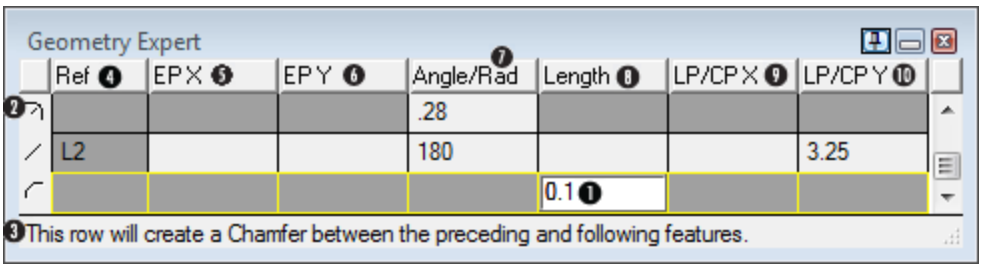
There are several feature types including lines, circles, fillets, chamfers, and post. Each feature is selected by changing the feature type button in the first cell of the row. For arcs, the selected feature type indicates the direction—either clockwise or counter-clockwise. For lines, the angle value also indicates the direction. For example, a horizontal line can either be defined with an angle value of 0° or 180°; both will draw the same line, but in opposite directions.

Fillets, Chamfers and tangent arcs will be draw when the features surrounding them are completely defined. These are called “floating features”. When the feature is created a reference number (`Ref`) such as `L1` (line 1) or `C4` (circle 4) is entered in the row. This is the actual geometry’s label (`View > Labels`).

In some cases, no information is required for the feature to be defined and the row will be yellow. This is because it is still possible to add information in subsequent rows that will fully define the feature. For example, a circle with a start point defined using a previous features end point and tangent to the following feature.

Geometry Expert Table

The Geometry Expert allows the user to create connected shapes by entering feature dimensions in the rows of the spreadsheet.



1. Current Row/Cell
2. Feature Type
3. Prompt
4. Reference #
5. Horizontal Endpoint
6. Vertical Endpoint
7. Line Angle/Arc Radius
8. Chamfer Side or Line Length
9. Horizontal Line point or Center Point
10. Vertical Line point or center point

Feature Type

There are six options available for the feature type. They are: Line (1), Chamfer (2), Fillet (3), Clockwise Arc (4), Counter-clockwise Arc (5), and Close Shape (6). Each row must have a feature type selected. To select the feature type, click the feature type button which will access the possible selections. Drag the mouse to the desired feature so that the button appears depressed, and let go of the mouse button. That feature will now appear as the feature type for that row. In some cases, depending on the previous feature, some of the feature type options may be disabled to indicate that they are not valid selections. Also, depending on the feature type selected, some cells may be disabled, indicating that the selected feature type does not require that dimension. The feature type can also be selected using `Alt+` the corresponding number.

Ref

This is the feature's reference number. The letter indicates what type of feature it is (L for line, C for circle) and the number indicates the creation order. Also displayed with View > Labels. These reference numbers may change during the course of geometry creation but will not affect the shape.

EP (H)

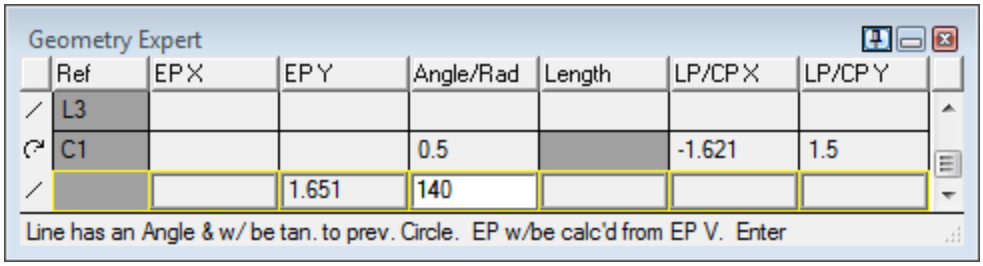
The horizontal endpoint of the feature. When a feature is defined with an endpoint, the system will draw the feature and trim it at the specified endpoint. Endpoint specifications are required if the next feature needs a start point in order to be correctly defined.

EP (V)

The vertical endpoint of the feature.

Example

The row shown will create a 140° line with an endpoint drawn at Xr 1.651. The next feature that is defined in the spreadsheet will start at the endpoint of this feature.



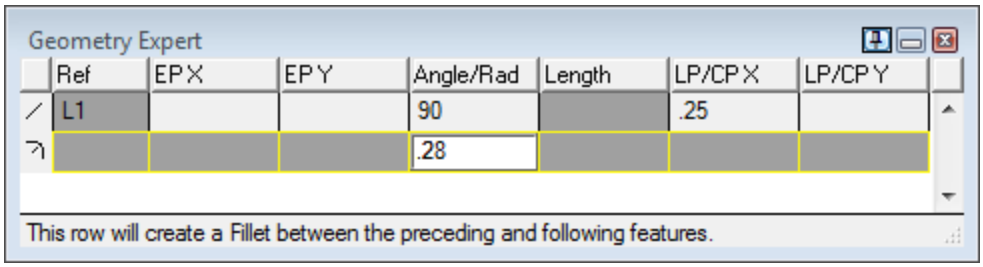
Angle/Rad

The number entered in this cell is dependent on the feature type selected. If the feature type is a line, this number specifies the angle of the line. The angle value for a line defaults to either 90°/270° or 0°/180°, allowing for the creation of vertical or horizontal lines. The system toggles between these angle values, which makes the creation of intersecting horizontal and vertical lines a very

quick and easy process. The user can change the default values by simply entering the new numbers in the cells. If the feature type is a arc or fillet, this number specifies the radius.

Example

This row will create a fillet between the previous and next feature with a radius of 0.28.



Length

The line length or chamfer side height.

LP/CP (H)

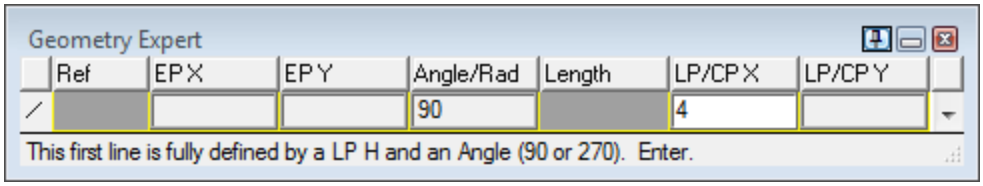
The horizontal line point or center point. A line point is any point tangent to the line.

LP/CP (V)

The vertical line point or center point.

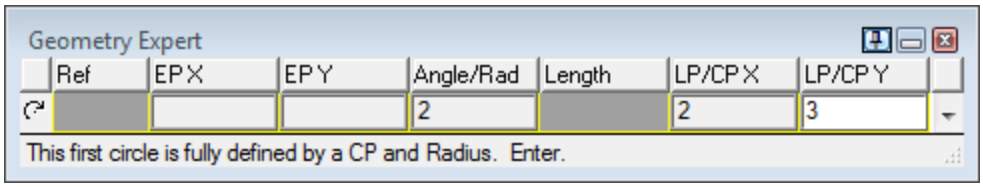
Example

This row will create a 90° line at X4. When defining either horizontal (0°/180°) or vertical (90°/270°) lines, only one LP coordinate is required. Refer to the section on Half Points in this chapter for more information.



Example

This row will create a circle with a radius of 2 and a center point at X2, Y3.



Prompt:

The information contained in the prompt indicates the status of the current row. The prompt also indicates if the system has automatically deleted a cell value because the feature row contained too much information that overdefined the feature. This aspect of Geometry Expert is referred to as the Auto Delete function, which is explained in the Expert Aids section of this chapter.

Close

When Geometry Expert is closed, all information is cleared.

Expert Aids

There are several items built into Geometry Expert that are designed to guide the user through geometry creation. They include Prompting, Auto Delete, and Error Balloons.

Prompting

The Prompting information appears across the bottom of Geometry Expert. The prompt tells the user what actions are being taken by the system based on the information provided by the user. The user should be able to follow the prompts through the creation process to get a good idea of what is happening.

Auto Delete

The Auto Delete function of Geometry Expert is intended to reduce errors resulting from features being overdefined because too much information has been entered into the feature row. Geometry Expert is designed so that the user needs only to enter the minimum amount of information to define a feature. The system will automatically delete the first entry made in the feature row when the feature is overdefined. Auto Delete is necessary in order to facilitate the associative capabilities of Geometry Expert. Note that the default values, such as the line angle, are considered the first entry rather than any information entered by the user. The prompt will indicate what information is being deleted.

Error Balloons

Geometry Expert also contains Error Balloons which appear on the screen whenever a problem occurs. The most common error messages appear when the system requires more information for a feature being entered. For example, if a line needs another value for an endpoint, an Error Balloon will come up indicating that a V or H value is needed to calculate the endpoint. The Error Balloons use the letters V and H (Vertical and Horizontal) rather than X and Y (for Mill) or Xd/Xr and Z (for Turning.) This way the error messages remain the same regardless of what module is being used.

Another common error message indicates that the feature being defined cannot intersect with the previous feature. The non-intersecting feature can still be created, but the message indicates to the user that the continuity of the shape has been broken and subsequent features created will not connect to the existing shape.

Additional Information

Defaults

Geometry Expert has defaults for the feature type and line angle. The standard feature type default is a line. When Geometry Expert defaults to a line, it also enters an angle value, either 90°/270° or 0°/180°, depending on the angle of the last line entered. Sometimes a line is not a possible feature

type option, in which case the system defaults to an arc. This only occurs when the previous feature is a floating line. Geometry Expert dimensions the line according the preceding and subsequent feature specifications. Floating features are not drawn on the screen until the system contains the necessary information.

Point Selection

Despite all of its expertise, Geometry Expert can't always know the correct intersection point to use for a connector. When there are two or more equally valid points of intersection, the system will draw both points and the point selection dialog will ask to select the appropriate point. Once the user has selected the desired intersection point, Geometry Expert will make the appropriate connection and continue along in the spreadsheet.

Half Points

In some cases, only one coordinate (either the horizontal or the vertical) is required to define the feature. This is referred to as a half point case. Half points are valid when Geometry Expert is able to calculate the other half of the coordinate value from information contained in the spreadsheet.

When a valid half point is entered, the row will be highlighted in yellow, allowing the user to enter the feature row. If the half point entered is not valid, the row will be highlighted in black and the prompt will indicate what additional information is necessary to enter the row. If an incomplete row is entered, an Error Balloon will come up indicating what additional information is required. There are three cases where half points are valid. They are listed and explained below.

Half Line Point

A half line point is valid only when creating either a horizontal line (angle value = 0° or 180°) or vertical line (angle value = 90° or 270°). If creating a horizontal line, a Y coordinate must be given for a valid half line point. If creating a vertical line, an H coordinate must be given for a valid half line point. Line points are not part of the shape, but are only used to calculate the position of the line.

Half End Point

A half end point is only valid if the line is otherwise completely defined. Given either the vertical or horizontal coordinate of the end point along with the other information that defines the line, the system can calculate the other half of the end point. If a half end point is used in the case of 0°/180° or 90°/270° lines, the half end point will function like a half line point, in that an end point will not be drawn, although the correct line will be created.

Half Center Point

A half center point is valid when a circle has a radius value and is tangent to the preceding feature. (There must be a preceding feature.) Given the radius and the vertical or horizontal component of the center point, the system can calculate the other half of the center point by the assumed tangencies.

Floating Features

Floating features are features whose defining row does not contain all of the information necessary to draw the feature. Floating feature rows are different from incomplete feature rows. With floating

feature rows, the information contained in the current feature row and the preceding rows is inadequate to completely define the feature and draw it. However, subsequent features should provide the necessary information to define the floating feature. Floating feature rows are outlined in yellow, indicating that the row has enough information to proceed.

Incomplete feature rows do not contain enough information to create the feature, and no amount of information entered in the following rows will make the feature definable. Incomplete feature rows are outlined in black and indicate that more information is needed for a valid feature.

Inserting and Deleting Rows

Choose Edit> Insert row to insert a row into the Geometry Expert. New rows are created above the selected row. When an inserted row is entered, the system will recalculate the shape and attempt to incorporate the new feature into the existing shape, if possible.

Choose Edit > Delete Row to delete a row. Only rows with created geometry can be deleted. Deleting rows can also be done with the keystroke Alt+K.

Arcs vs. Fillets

Arcs with only a radius value and fillets appear to be very similar at first glance, but actually use two completely different methods for calculating the circles. A fillet takes a sharp point produced by an intersection between two other features and changes it into a radius. It is created after the intersection between the two features is completed. Because of this, it is dependent on the intersection of the other two features to exist. This means that the system cannot use the fillet to calculate features that follow it in the spreadsheet. Fillets are not used to calculate the features that surround it.

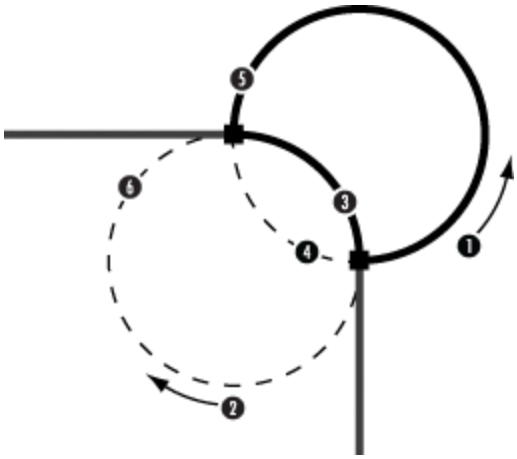
An arc with only a radius value is created tangent to two other features. These two features do not have to intersect. The arc is treated as an actual feature and can be used when calculating other features of the shape.

This is particularly important when dealing with floating lines (lines with limited information) that have specific tangency requirements. Geometry Expert assumes that floating lines are going to be made tangent to the preceding feature. In most cases this is adequate, but sometimes a floating line is supposed to intersect the previous circle and be tangent to the next circle. This is called a “forward” tangency.

In the case of forward tangencies, if there is a radius between the previous circle and the floating line, then an arc (rather than a fillet) must be used to allow for the necessary tangency calculation. The line will be made tangent to both the arc used as a fillet and the next arc. A fillet cannot be used in this case because the fillet will be ignored by the system until the intersection is completed, and the correct intersection cannot be created without the arc. If there is no fillet, an arc with a radius of zero will need to be created. This will allow the system to create the line tangent to the following circle, while creating a sharp point at the intersection of the previous feature. For a practical example of forward tangencies, refer to the Geometry Creation tutorial.

-R Creation

Geometry Expert has -R (large arc) functionality that provides the ability to toggle between two possible solutions for an arc between two features. This is completely different from the Reverse Arc (Ctrl+T) feature, which changes the direction of the arc. -R provides access to opposite tangent points on the circle. Normally the smaller radius will be used when creating geometry, -R offers a way to override that default. These features may be created in Geometry Expert or via free-form CAD. Figure 5 below illustrates the difference between Reverse Arc and -R.



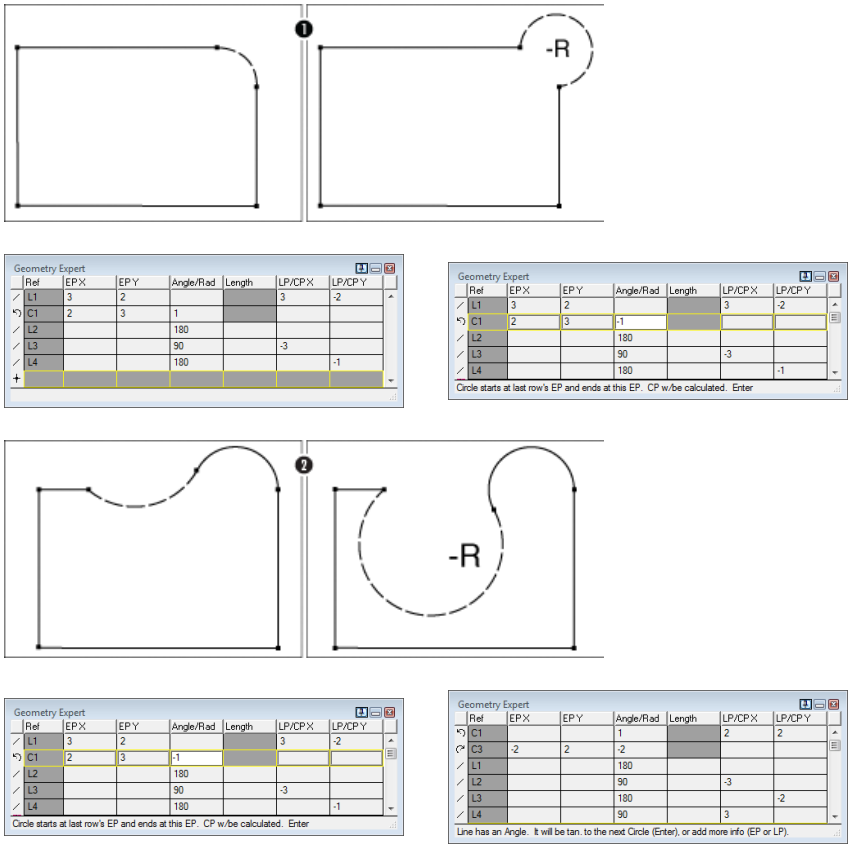
1. Clockwise
2. Counter Clockwise
3. Default Arc with R
4. Reversed Arc
5. Default Arc with -R
6. Reversed Arc with -R

Via Geometry Expert

When using Geometry Expert the -R function may be applied to an arc defined in one of two ways. First, the arc may be defined by being tangent to the previous feature (a line or arc) and a specific endpoint. Second, the arc may be defined by a radius and an endpoint, and the feature previous to the arc must be defined by an endpoint. The previous feature may be a line or an arc. To apply the -R functionality simply place a "-" (minus sign) before the arc's radius. Figure 6 provides an example of this.

Via Free-Form CAD

Geometry created by standard CAD functions (free-form CAD) may be loaded into Geometry Expert and modified to a use the larger arc solution. For this to work, the free-form arc must be an arc that is tangent to a previous feature (line or arc) and a given end point. To apply the -R functionality, simply place a "-" (minus sign) before the arc's radius once the geometry has been loaded into Geometry Expert.

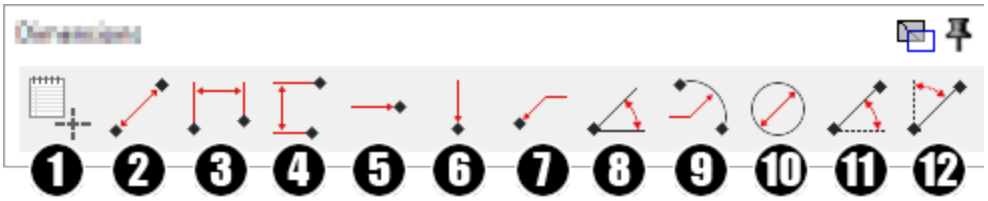


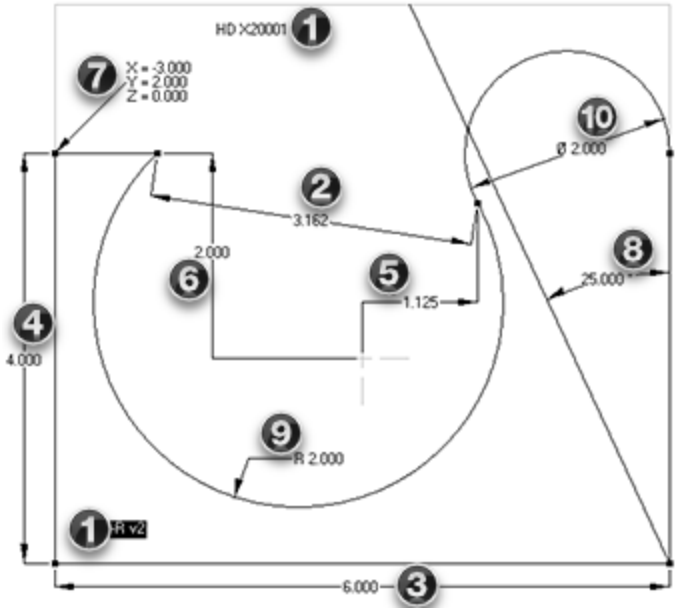
1. Example of an arc defined by a radius and endpoint with a previous feature defined by an end point.
2. Example of an arc defined by a radius and endpoint that is tangent to the previous feature.

Examples of -R use in Geometry Expert

Dimensioning

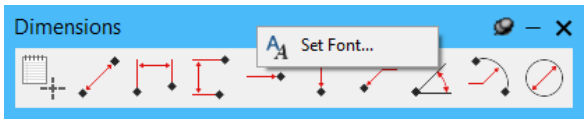
The Dimensions palette is accessed from the View > Dimensions Palette or from the Main palette in the Level 2 interface. The Dimensioning tool gives the ability to place text and show part dimensions. Dimension values are calculated from the current CS and visible based on the current workgroup. Any part of the dimension line, (line, arrowheads or text) may be dragged to reposition the dimension. Selection mode is active while the Dimensions palette is active.

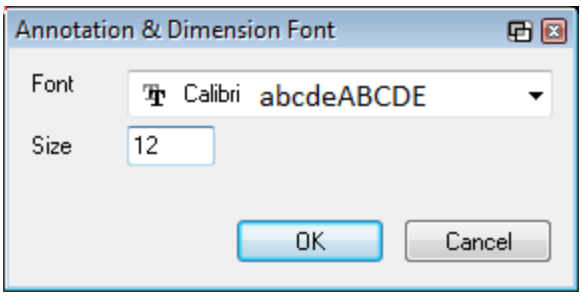




1. Text
2. Align
3. Horizontal
4. Vertical
5. Horizontal From Origin
6. Vertical From Origin
7. Point
8. Angle
9. Radius
10. Diameter
11. Angle to H Axis
12. Angle to V Axis
13. Multiple Dimensioning mode

The font used and the size of the the annotation text is controlled by right-clicking on the Dimensions title bar and selecting Set Font. This opens a dialog that displays the fonts that are installed on your system.





Text
Enter a note on the part. Place the cursor where you want the note to appear and enter the text.

Align
Gives the distance between two selected points.

Horizontal
The horizontal distance between two selected points.

Vertical
The vertical distance between two selected points.

Horizontal From Origin
The horizontal distance of a point from the origin of the current CS.

Vertical From Origin
The vertical distance of a point from the origin of the current CS.

Point
The explicit coordinates of a point in the current CS.

Angle
The interior angle of two lines. The exterior angle may be determined by selecting the dimension and choosing `Modify > Reverse Arc` (`Ctrl-T`). This button will also determine the angle between three points.

Radius
The radius of a selected arc or circle.

Diameter
The diameter of a selected arc or circle.

Angle to H Axis
The angle between a selected line and the horizontal axis.

Angle to V Axis
The angle between a selected line and the vertical axis.

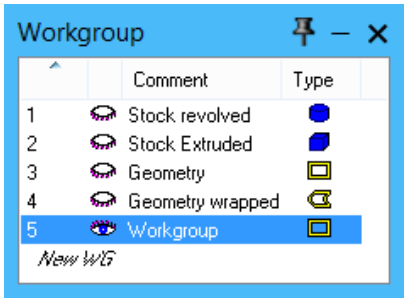
Multiple Dimension mode

Multiple Dimension mode lets you create multiple dimensions with maximum power and efficiency. In this mode, you select a dimensioning type and then, after you select a base feature: Hovering over another feature provides a preview of the dimension; `Ctrl`+click sets the dimension; and `Ctrl`+drag adjusts the dimension's position.

Workgroups

All created geometry is placed in the active workgroup. Workgroups are separate layers used to separate different groups of geometry, including custom stock. One or more workgroups may be viewed at a time, however there may only be one active workgroup. The viewing, selection and creation of workgroups is accomplished with the Workgroup list. Workgroup information and selection can also be performed through the Workgroup dialog, accessed from the Main palette.

Workgroup List



The Workgroup list controls the workgroups. Click `New WG` to create additional workgroups. The active workgroup is highlighted. To switch workgroups, `click` the name of the desired workgroup. The eye icon show and hide the workgroups. `Click` the column header to sort the list. `Right-`

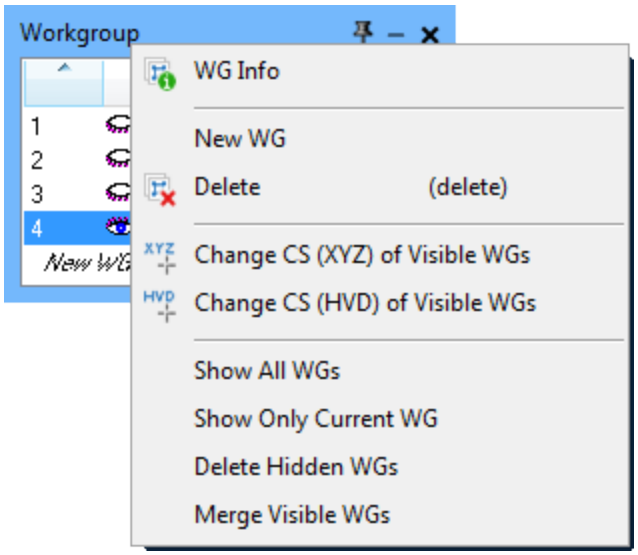
click or a workgroup entry in the list to open a context menu which accesses basic Workgroup functions and some Modify menu items. The right mouse menu is described in “Workgroup Right Mouse Menu” on page 56.

Background Workgroups

To see inactive geometry contained in other workgroups double-click the eye to make it visible. To select multiple workgroups to be viewed as background workgroups, use the Shift key to select a contiguous group of workgroups and the Ctrl key to add or remove a single workgroup from the selection. Double-click to change the status of an eye from visible to invisible.

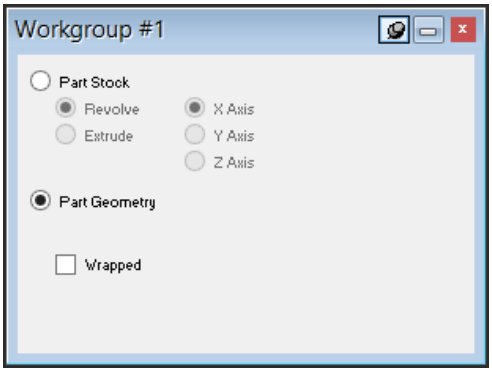
Workgroup Right Mouse Menu

You can right-click the Workgroup title bar or a Workgroup list entry to open a context menu. This contains actions that are commonly used with workgroups:

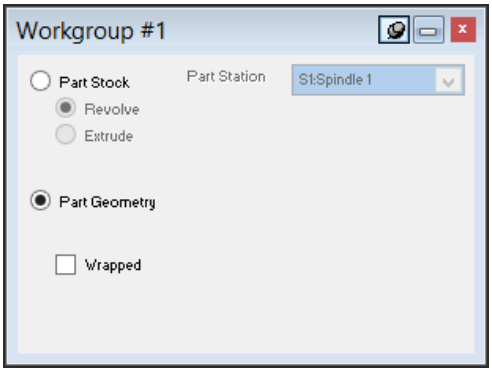


WG Info

The WG Info choice opens a dialog that allows you to specify the behavior of geometry in this workgroup:



Workgroup Info dialog for Mill part



Workgroup Info dialog for Turning part

Choose Part Stock to use the geometry as a stock shape, or choose Part Geometry to specify that geometry in the workgroup is used to define the part.

Part Stock

If Part Stock is selected, the geometry in the workgroup will be used for any calculations that need to look at the stock dimensions, such as Auto Clearance and Material Only. The geometry can be used as the starting stock condition in rendering and simulation, so long as it is a closed shape. When used in this way, the stock shape can also be used in machining operations to adjust the toolpath according to the amount of material to be removed based on the stock shape.

Extruded geometry in Turning workgroups is extruded along the depth.

New WG

This option creates a new, empty workgroup in the next available number slot. That is, if you have WGs numbered 1, 3 and 4, creating a new WG will make WG #2.

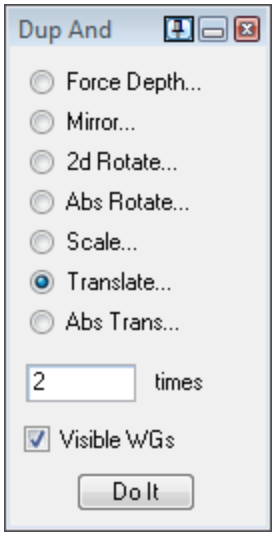
Delete

This option deletes the current workgroup. Any geometry contained within the workgroup will also be deleted. You may also press the Delete key to delete WGs.

Duplicate Visible WGs

This option will duplicate all the geometry within currently visible workgroups. New workgroups are not made, only geometry within each workgroup is created. This is functionally the same as pressing Ctrl D to duplicate geometry but the function is applied on a larger scale.

Duplicate And...



This option will duplicate geometry and apply a transformation to the geometry in the current workgroup. This option is functionally identical to the Duplicate And... command found in the Modify menu, except that it applies the modification to all geometry in the workgroup. For more information on Duplicate And..., see the *Common Reference* guide.

Force Depth...

This option will move all geometry in the current workgroup to a specified depth. For more information on Force Depth, see the *Common Reference* guide.

Mirror...

This option will reflect or flip all geometry in the current workgroup about a specified point. For more information on Mirror, see the *Common Reference* guide.

2d Rotate...

This option will rotate all geometry in the current workgroup about a specified point. For more information on 2D Rotate, see the *Common Reference* guide.

Scale...

This option will magnify or shrink all geometry in the current workgroup by a specified amount. For more information on the Scale function, see the *Common Reference* guide.

Translate...

This option will move all geometry in the current workgroup by a specified amount. For more information on Translate, see the *Common Reference* guide.

Segment Spline...

This option will break all splines in the current workgroup into line segments. For more information on Segment Spline, see the *Common Reference* guide.

Sort...

This option allows you to sort all points in the current workgroup. For more information on Sort, see the *Common Reference* guide.

Reverse Arcs in Visible WGs

This option will toggle the direction of all arcs in the current workgroup, i.e. clockwise arcs become counter clockwise and vice versa. For more information on Reverse Arcs, see the *Common Reference* guide.

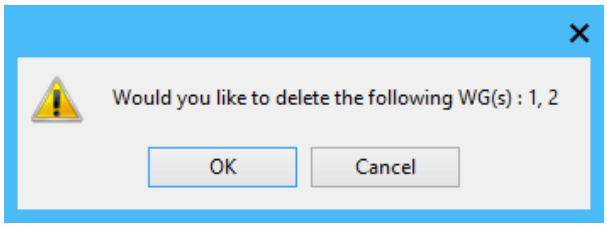
Change CS (XYZ) of Visible WGs
This option will change all geometry in the current workgroup from whichever coordinate system it is aligned to the current CS. The geometry stays in its current location. For more information on Change CS (XYZ), see the *Common Reference* guide.

Change CS (HVD) of Visible WGs
This option will change all geometry in the current workgroup from whichever coordinate system it is aligned to the current CS. The geometry will move to be in the same relative position within the CS. For more information on Change CS (HVD), see the *Common Reference* guide.

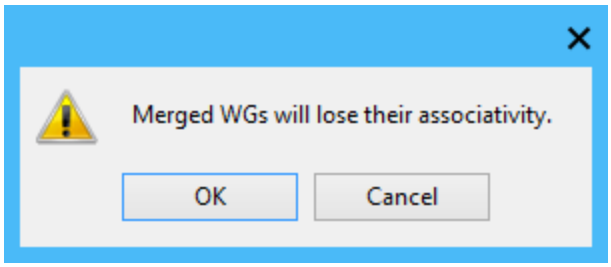
Show All Workgroups
Switches all the eye icons to open () to display all geometry. The non-active geometry is displayed in grey.

Show Only Current WG
Changes all the eye icons except currently active WG to shut (). Only geometry in the currently selected Workgroup will be displayed.

Delete Hidden WGs
This command will delete all WGs where the “eye” icon is shut (). A one-time warning appears.

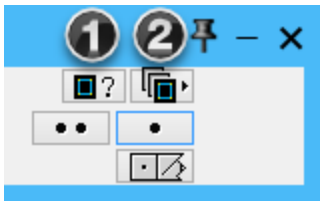


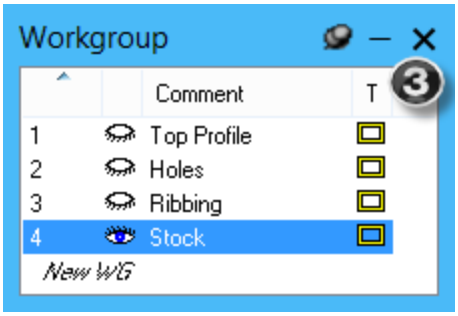
Merge Visible WG’s
Open the eye on the Workgroups that are to be merged. (). A confirmation message will be displayed.



Level 1 Interface and Workgroups

Accessing workgroups is slightly different in the Level 1 interface. Workgroups and the Workgroup Info are accessed through the Geometry Creation palette. Clicking the Workgroup List button opens the Workgroup List, which functions identically as in Level 2, while clicking and holding the Workgroup List button accesses a menu which allows quick switching of the active workgroup. A new workgroup may also be created from this menu.





1. Workgroup Info button
2. Workgroup List button
3. Workgroup List menu

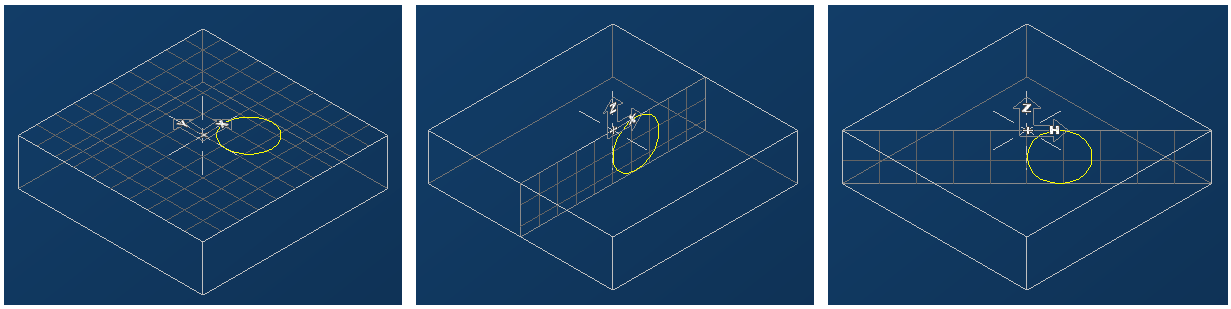
Workgroup Manager

A Workgroup Manager is available from the Window menu which provides information in a tabular format about the points, lines and circles contained in the current workgroup. Print and Save to file functionality is available from the Manager dialog. The list is customizable and can be sorted, saved and printed.

For more details on the manager capabilities see the *Common Reference* Guide.

Coordinate Systems

Geometry may be created in, or moved to, different coordinate systems. The *Advanced CS* guide is the best place to get more information about this type of geometry. Geometry can be defined using these other planes with the Advanced CS module. The image is an example of three different coordinate systems. The circle in each coordinate system is at the same horizontal, vertical and depth position relative to each coordinate system.



Geometry is not contained in a coordinate system the way it is with workgroups. The coordinate system used to define geometry can be thought of as an attribute of the geometry and its orientation to the rest of the part. Coordinate systems are used for 3D geometry creation, rotary part orientation for machining, multiple work fixture offsets and as a basis for solid modeling. Coordinate systems are only available in the Level 2 interface.

3D Geometry

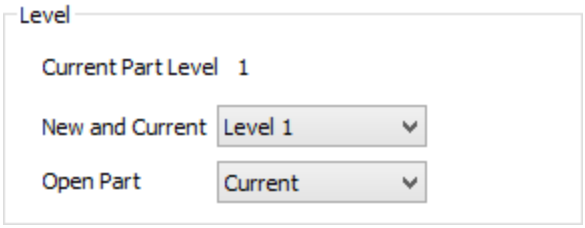
Creating non-planar geometry is the same as planar except you would use connectors that lie in different coordinate systems.

Printing the Part Geometry

After the part geometry has been created, it can be printed. Geometry can either be printed in black and white or in color. When the desired geometry is on the screen, choose Drawing (Ctrl+P) from the Print sub-menu under the File menu. To adjust the way the image will print, choose Printing from the Preferences sub-menu in the File menu. The Printing Preferences dialog shown allows the user to specify how the software will handle the background color. If the printer being used is black and white, choose the Black on White option to ensure that all portions of geometry, including those that are of a light color, can be seen in the printout.

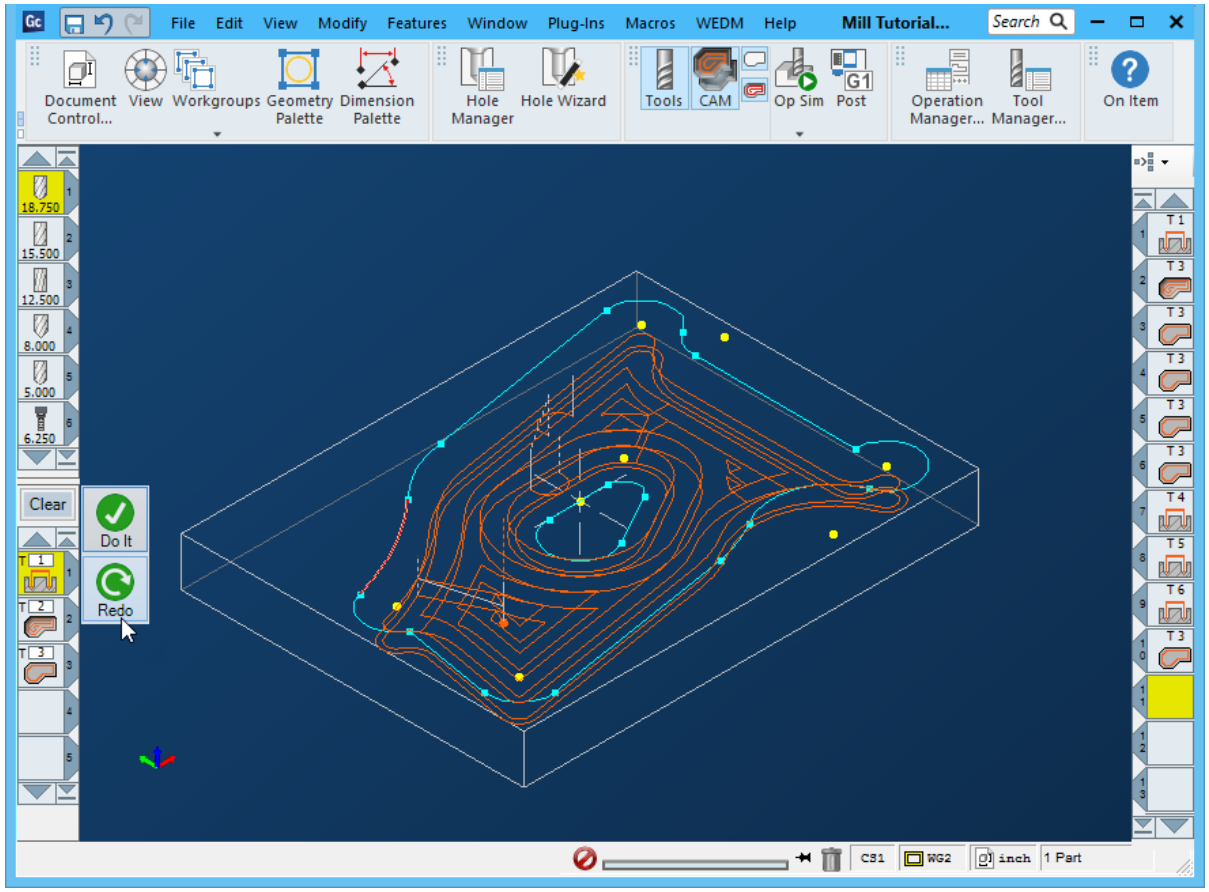
Appendix

GibbsCAM has two interface levels: Level 1 and Level 2. Level 2 is the default and provides a more complete, feature-rich environment. Level 1 is a simpler interface that some users may prefer if they do not need all the options or flexibility that Level 2 offers. You can think of Level 1 as a training interface that hides the more complicated features. This section details the different interface options found in Level 1.



Interface

The interface is different in Level 1. The Floating Toolbar is not present and the Commands Palette is simpler.



Workgroups

To access different workgroups in Level 1, the Workgroup list and information dialogs are also located on the geometry palette.

Not Included In Level 1

Level 1 can be displayed with any MDD but is truly only useful with generic lathe or 3-axis mill MDDs. All other MDDs require access to items not present in Level 1. Additionally, there are several things that cannot be done in the Level 1 interface.

- Any surfaces or solids manipulation as described in any of the solids manuals which includes:
  - Global Tolerance settings
  - Surface Machining
  - Solids will not be visible or selectable until switching to Level 2.
- Rotated Coordinates as used in Advanced CS, Mill/Turn or Multi-Task Machining
  - All Coordinate system options are hidden in Level 1, including the grid, lists and palette.
- Advanced Contour and Roughing options
  - Stay In Stock
  - Material Only
  - Advanced Entry And Exit
  - Hit Flats
  - Open Sides - limited to fixed parameters based on tool size
- Access to some workspace context menus is disabled.

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

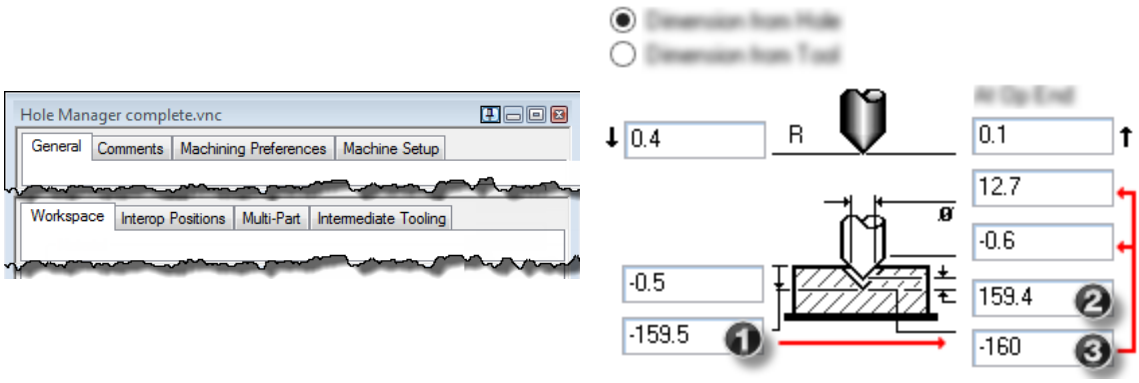
`Screen text`. Text with `this appearance` indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`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.

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