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Geometry Creation Tutorials

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GEOMETRY CREATION TUTORIALS

These tutorials are designed to introduce you to the concepts associated with Geometry Creation. As none of these tutorials involve machining, these are excellent places to start if you need to understand the basic ideas surrounding Geometry before moving on to machining concepts. We will cover Free-Form Exercises, Geometry Expert Exercises, and Combination Exercises.

FREE-FORM EXERCISES

The exercises contained in this chapter are designed to introduce the basic concepts of geometry creation using the Free-Form CAD tools contained in the Geometry Creation palette. For the part prints for the exercises, see "Part Prints" on page 158.

The first few exercises give very detailed step-by-step instructions for creating part geometry, while the later exercises become less detailed. Exercises 1, 2, and 5 are done using the Mill module, while Exercises 3 and 4 are done using the Lathe module. Because these exercises deal with only geometry creation (and not machining), they can be created using either module. The only differences will be in the setup of the document. It is recommended that you read through the Getting Started manual before attempting these tutorials.

Exercise 1: Shapes and Connectors

This exercise creates a very simple shape to demonstrate how the Free-Form CAD tools create and connect geometry. There is no example part print at the end of the manual for this exercise.

Creating Shapes

Part Setup

Create the Part

1. `Click` the Documents button in the Toolbar command group.

   This dialog contains file management options as well as general information about the part, such as the type of machine being used, stock measurements and material information. Typically, when creating a part, you would modify these settings at this point to reflect your desired options. For this tutorial, we will configure these settings in the next section.

Configure Machine Information

2. `Click` the New button.

New
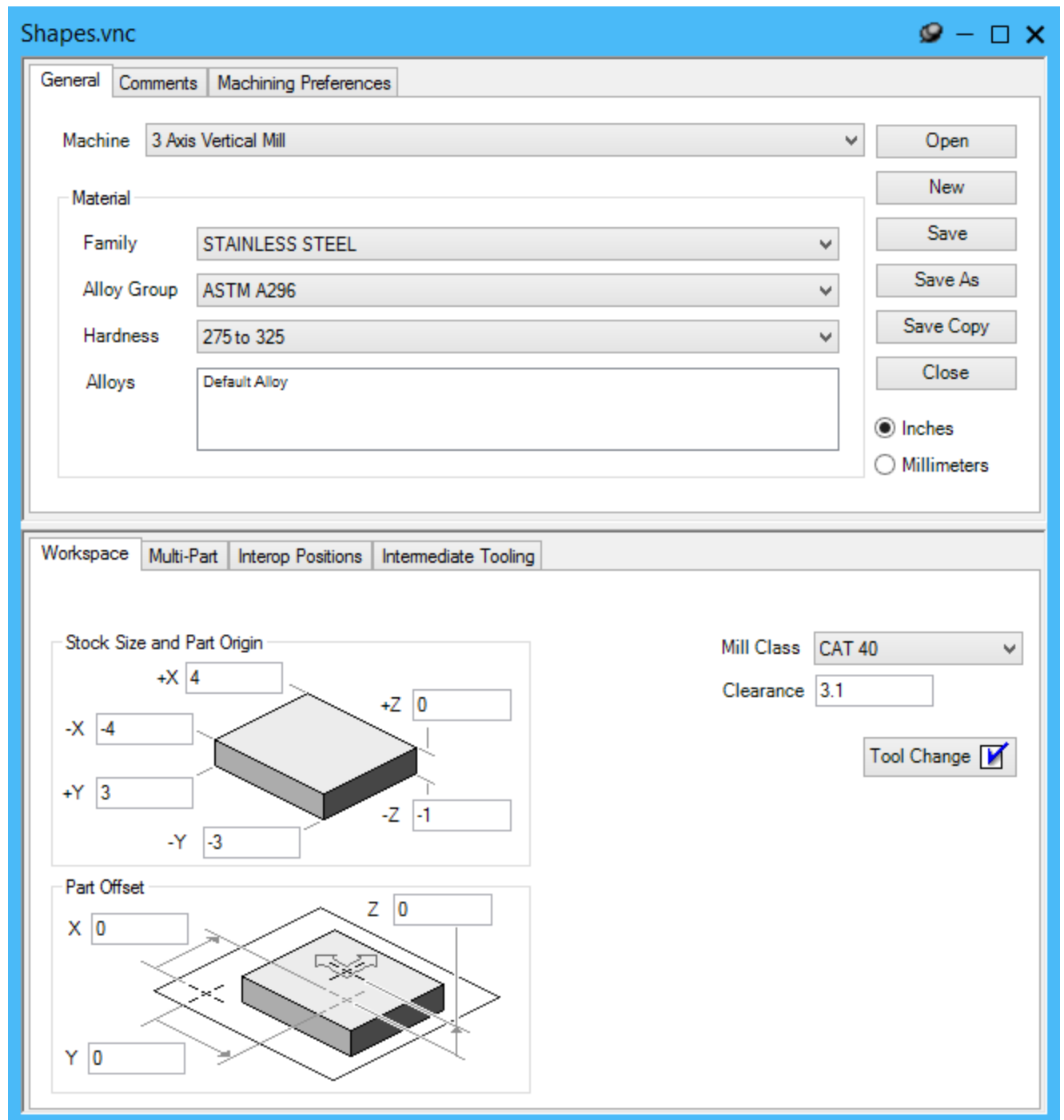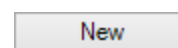
Type Shapes in the File name text box.

3. Save the tutorial part in a convenient location. The parts created in this manual will be used in

the other manuals.

4. `Click` the Save button.

Save

5. In the Machine list, select 3 Axis Vertical Mill.

The Material information is fully covered in the Getting Started manual. For this exercise, match the Material settings shown.

6. Select inch for the measurement type.

7. Enter the values as displayed above.

8. Clearance Plane values are explained in the Machining chapter and tutorials of the Mill and Lathe manuals.

Because machining processes will not be created in this exercise, the stock and clearance values input into this dialog are merely to provide a stock outline to contain the shape we will create.

9. Type "`This is the first tutorial part`" in the Comment text tab.

If a comment is entered in the text tab, it will appear in the Open dialog when a part is selected. This comment will also appear in the G-code. A part comment is not necessary but is often helpful in identifying parts.

10. Close ⊠ the Documents dialog.

You can press Ctrl + W if you wish to use the keyboard shortcut.

To finish this section, we must ensure that your view accurately depicts the steps in the remainder of the tutorial. This can be accomplished through either keyboard shortcuts, the View Menu (which lists the keyboard shortcut for each selection) or the View Palette.

To use keyboard shortcuts, press Ctrl + H then Ctrl + U.

or

To use the View Menu, select View 🏠 home

then View 🔍 Unzoom.
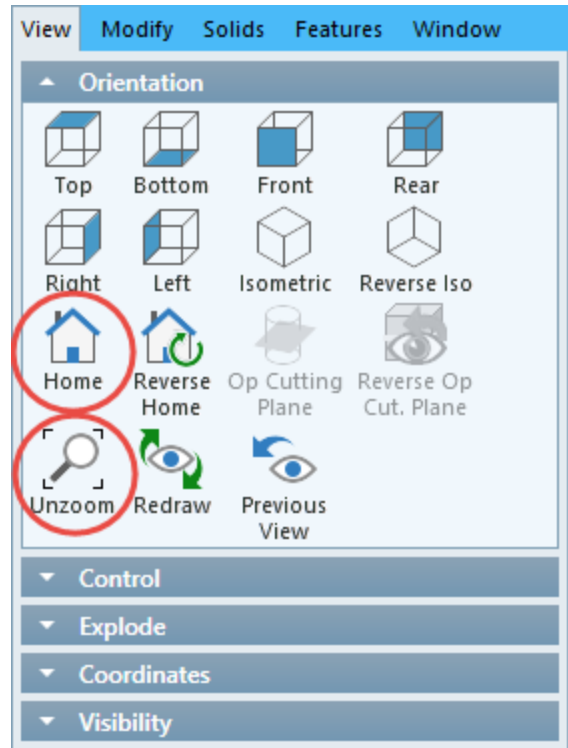
or

Click the view button in the Toolbar command group to open the view palette. Press the Home, then the No Zoom icons on the palette.

ⓘ The options available in the Machine list are dependent on the modules installed. The selection made for the Machine Type sets the MDD (Machine Definition Document) for the current part. Selecting the correct MDD is important because it affects how the part will be drawn and machined.
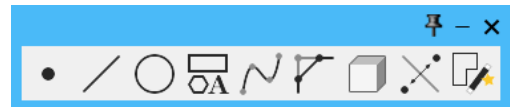
Geometry Basics

Axis Lines
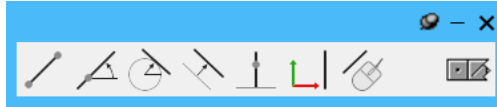
Open the Geometry palette.

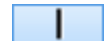🔲 or F2

1. Open the Line sub-palette.

    Please note that all buttons in the palettes can be selected by pressing the number on your keyboard that corresponds to the button's position in the palette. For example, the Line sub-palette can be accessed by pressing [2] as the Line sub-palette is the second button.

2. Select Axis line.

3. Select the vertical line mode.

    We will draw the first line at X -1 and Z 0.

4. Click the Multiple Lines button.

    The Multiple lines button tells the system that you wish to create more of this type of line. The first line will be drawn as shown.

    The colors associated with lines/backgrounds can be different depending on your settings. Here the default color of any unconnected geometry line is yellow.

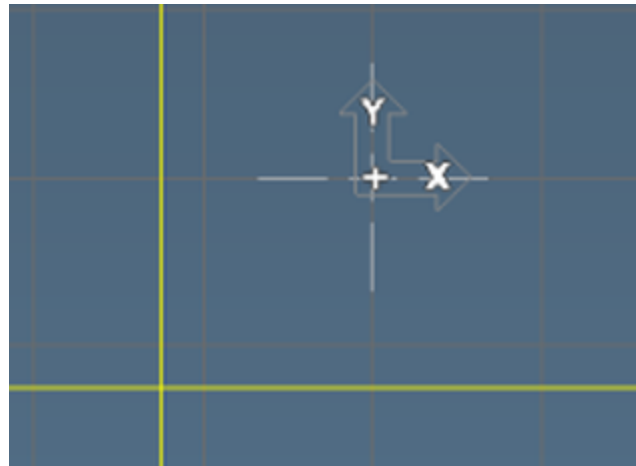5. Switch to the horizontal line mode.

    This will change the X value to a Y. Since the value for the horizontal line is the same, nothing needs to change.

6. `Click` the Single Line button.

If you accidentally press `Enter` or the Multiple feature when you are done, you can press the `Esc` key or Return button.

Tangent Circle to Line

1. Return to the Main Geometry Palette.

(press the `Esc` key or Return button)

2. Open the Circle sub-palette.
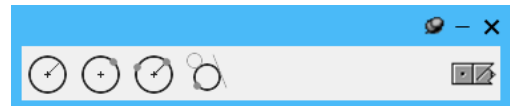
3. Select the two lines you have just created.

A modifier key such as `Shift` or `Ctrl` is not required when sub-palettes are open. The selection mode is designated with the white arrow pointer.

They will change to a dark blue color.

4. Input the value shown.

Workgroup

L L1   L L2

R 0.5

The order in which the lines appear in the selection boxes (L) in the dialog is determined by the order in which you selected them. In this exercise, the order is not critical.
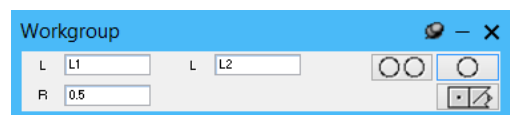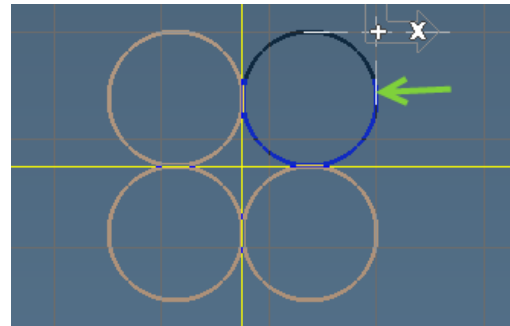
5. `Click` the Single Circle button.

All possible circle tangencies will be drawn, allowing you to select whichever circle is necessary to create the required part geometry. If more than one feature is selected, no automatic connection will occur.

6. Select the upper right circle shown.

    The four possible circle tangencies to the lines are drawn.

Once a feature is selected, the OK button becomes active.

7. `Click` the OK button.

When any button is active and waiting for a response you can use the Enter key or the space bar rather than using the mouse.

The circle trims to an arc creating 2 blue square points called connector points.
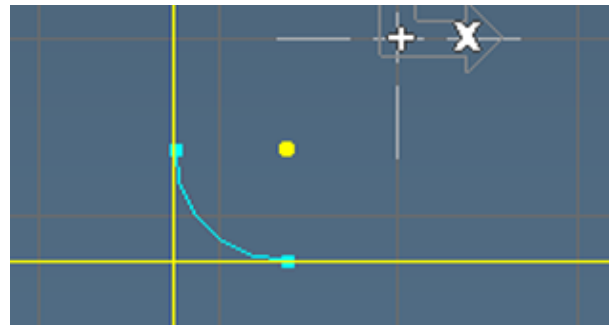
Connector points designate connections between elements. The system will create trim points or connector points (blue squares) rather than points (yellow round dots) whenever possible. Each type is interchangeable as we will demonstrate later on in this tutorial.

Explicit Point

Return to the Main Geometry Palette.

1. Open the Point sub-palette.

2. Select Explicit point.

XYZ

An Explicit point requires you to define the point's location.

3. Input the value shown.

| X | 3 | Y | 2 |
| Z | 0 | P | None |

4. `Click` the Single Point button.

You will notice that the button is highlighted and therefore active (light blue band around it). This means you can also select this by pressing the space bar or Enter key.
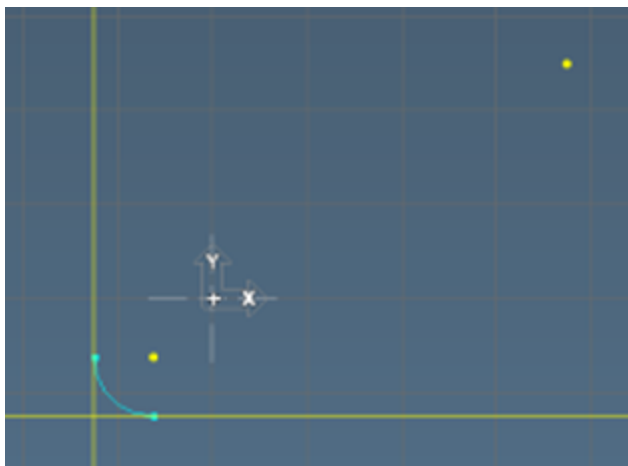
Pressing Shift + Enter or Shift +`space` will change the mode to "multiple" and will create the feature, remaining on the same dialog to allow you to create another point.

Angled Line Through a Point

1. From the Main Geometry Palette, open the Line sub-palette.

2. Select Point Angle line.

3. `Click` (select) the point we just created.

Input the value shown.

Workgroup

| P | P4 | A | 15 |

4. **Click** the Multiple Lines button.　　　　　　Alternatively you can press [Shift] + [Enter].
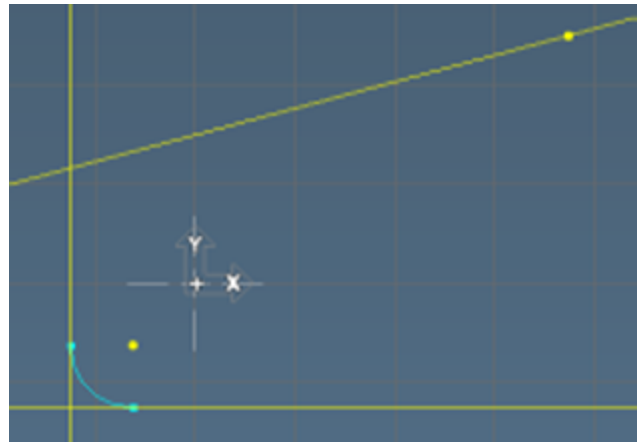
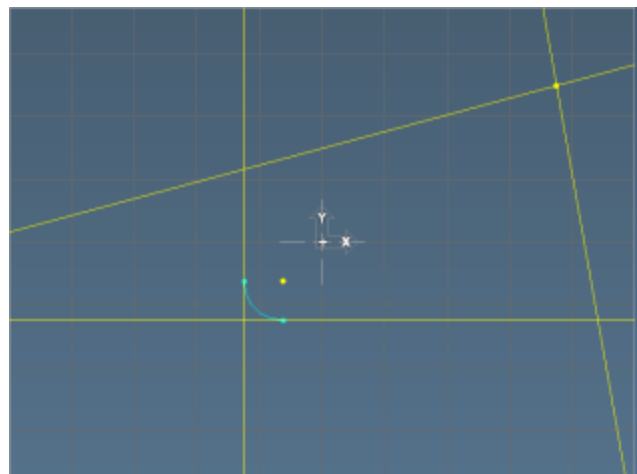Your screen will look like this

5. Change the value as shown.
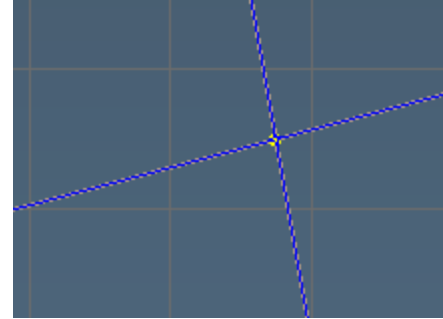
6. **Click** the Single Line button.

In this case a connector point is not created. We will need to manually connect the intersection of the angled lines.

Connecting Lines - 1

1. `Ctrl+click` anywhere along each of the angled lines in the upper right of the shape.

   The lines will change to a dashed blue.

   The `Ctrl` key activates the selection mode (<u>white</u> arrow).

2. `Click` the Connect-Disconnect button on the Main Geometry palette.

   (Or you can click the right mouse button which brings up a pulldown menu - choose Connect/Disconnect.)

   The plain point (circle) is converted to a connector (square.) Neither of the lines that intersect at that point trim because each has only one connector point.
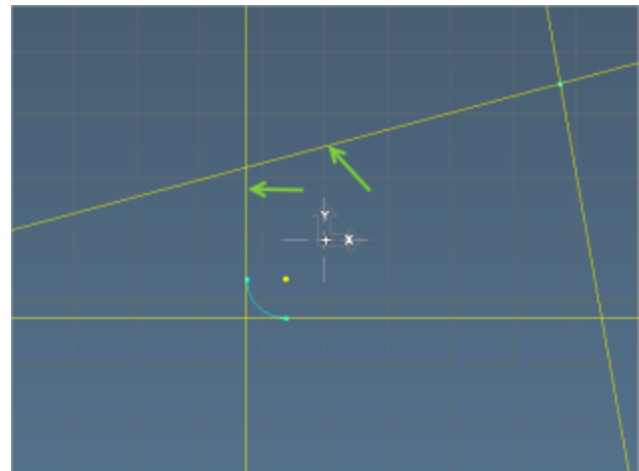
Connecting Lines - 2

1. From the Main Geometry Palette, open the Point sub-palette.

2. Select the two lines in the upper left of the shape.

   The lines will change to a dashed blue.

   We are again in selection mode. Selecting two lines while the Point sub-palette is open will open the line-line intersection point dialog that will create a point and connect the lines.

3. `Click` the Multiple Points button.

The lines become light blue (the default color for connected geometry) and trim because they now have two connector points.
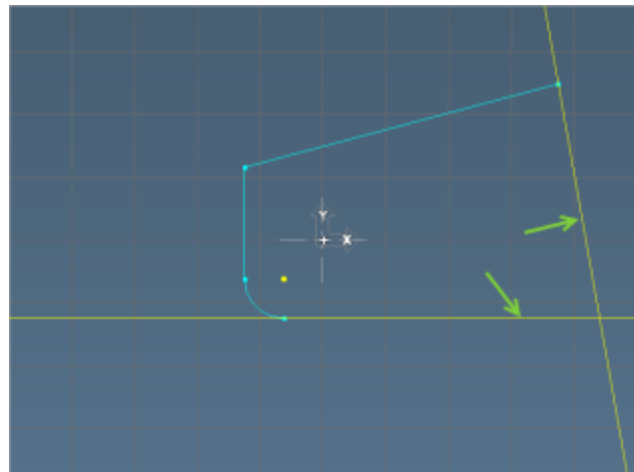
4. Select the last two unconnected lines (bottom right).

5. `Click` the Single Point button.

The shape is now closed.

Circle with Radius and Center Point

1. Open the Circle sub-palette.

2. Select Radius & Center Point.

3. Select the point shown (top right).

4. Input the value shown.

5. `Click` the Single Circle button.

The circle is untrimmed, indicating there are no connections - the lines that it intersects already have two connector points each.

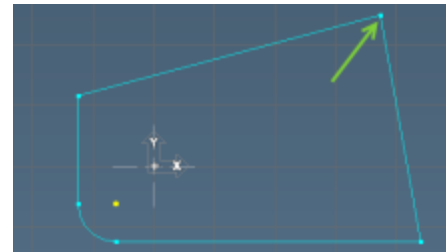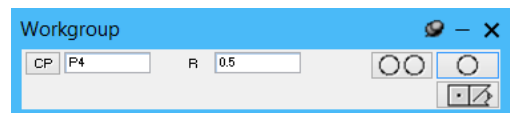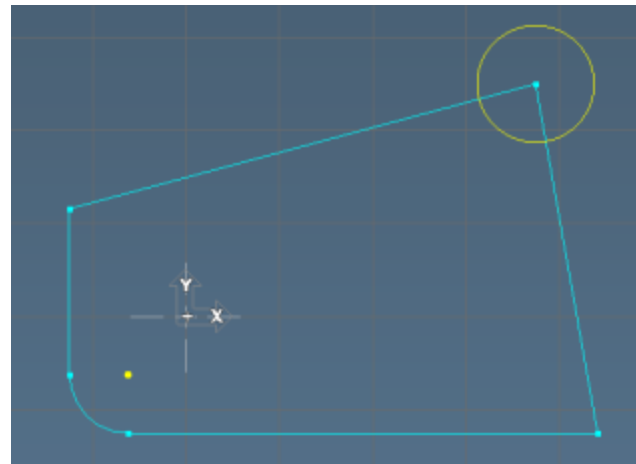We want this new circle to be the corner of our shape. There are two methods of achieving this, one uses the Combine function to add the shapes together, the other uses the Connect-Disconnect function. First we will use Connect-Disconnect.

Connecting a Circle and Line - 1

Firstly, the point at the intersection of the lines (the center point of the circle) must be disconnected.

> ⓘ A quick way to enter data into a text box is by using the interrogation cursor. If there is a reference value, `Alt+click` the reference geometry to load the data into the selected text box. Additionally, all the usable data for the dialog can be loaded with `Shift+Alt+click`.

Breaking Connections - 1

Select the center-point of the circle we just created.

1. `Click` the Connect-Disconnect button.

The point changes from a blue square to yellow round point. The lines also become untrimmed.
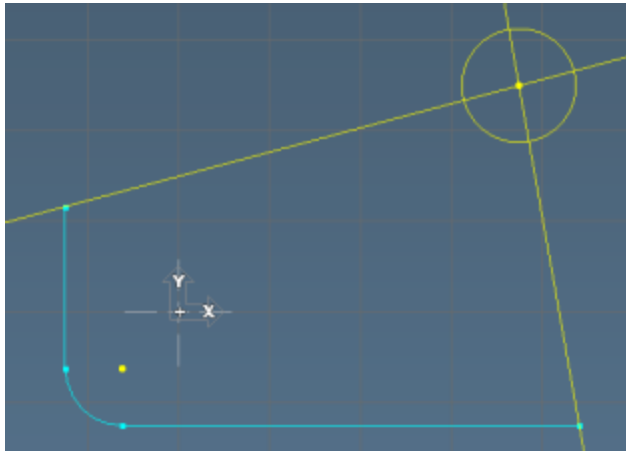
While the Connect/Disconnect button is a perfectly valid way to disconnect the lines, in this exercise we won't need the center point. As such, we will use a slightly different method.

2. Choose Edit ↰ Undo.   Ctrl + Z   This will reconnect the lines.

Breaking Connections - 2

Delete the center point of the circle.

Select the center point, press Delete

The same disconnection happens but there is no longer a center point. We could easily recover this point using the Line-Line intersection point.

Connecting

1. Open the Point sub-palette.

●

2. Select the line and circle as shown.
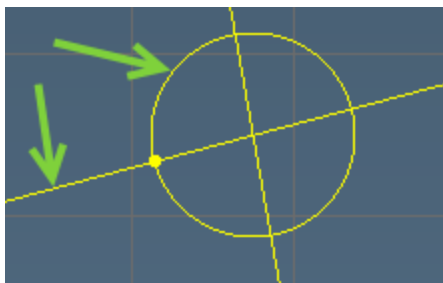
When a circle is intersected and not tangent to a line, the 2 intersecting points will highlight for you to select the features necessary for the part. One or both of the points can be selected for incorporation into the part geometry.

3. Click the Single Point button.   [●]   Two points are highlighted where the line intersects the circle.

4. Select the left point as shown.
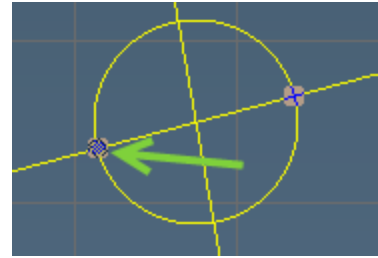
5. `Click` the OK button.

   The point becomes a connector, and the top line trims.
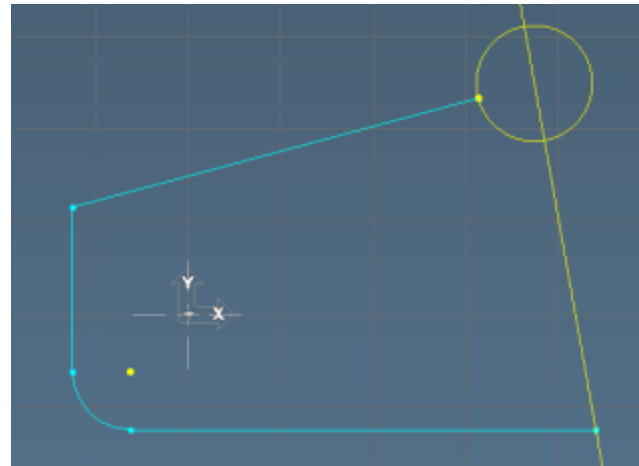
6. Select the unconnected circle and vertical line and `click` the Connect-Disconnect button.

   The two points of intersection are displayed.

7. Select the bottom point.
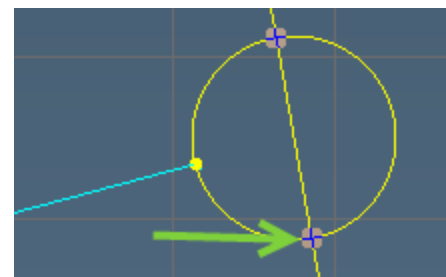
8. **Click** the OK button.

   The resulting closed shape will look like this:
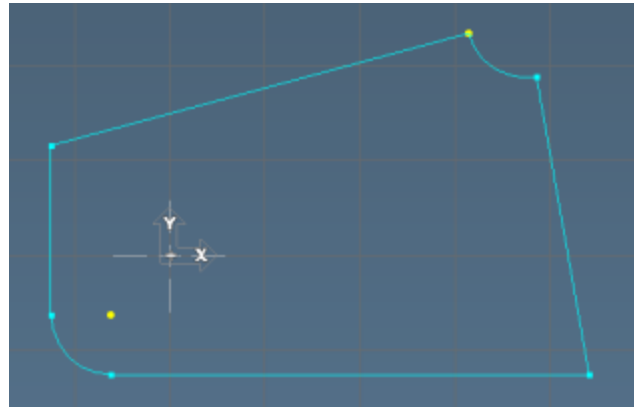
Reversing Arcs

By default, all circles trim to an arc of less than 180° when they have two connector points. In this case, we want the arc that is greater than 180°.

1. Select the arc.

2. Choose Modify > Reverse Arc

   Ctrl + T
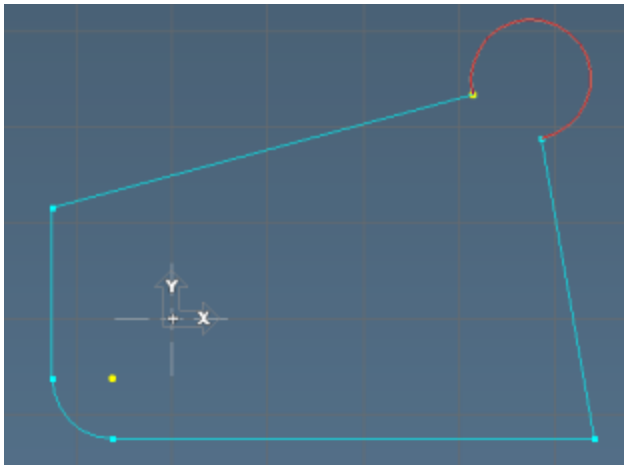
The arc is now reversed.

Connecting a Circle and Line - 2

This method of adding shapes together uses the Combine Shapes function.

1. Step back to this stage of the construction using Edit>Undo.
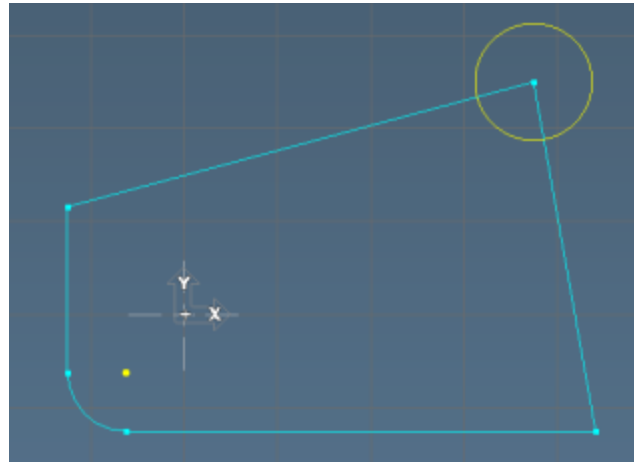
2. `Double-click` one of the blue lines to select the main shape, then `Ctrl-click` the circle.

   In the Geometry Palette select the Shape sub-menu OA and then Combine Shapes.

3. Click the Union button.

   The shapes combine.
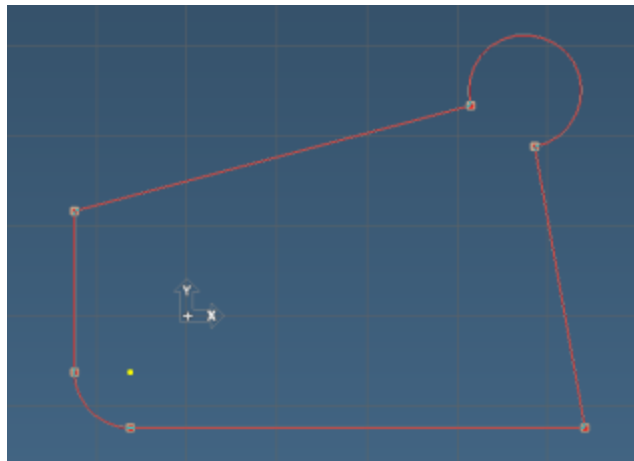
Terminating Points

There is a third type of point called a terminator. A terminator is graphically shown as a yellow square. A terminator is similar to a connector point in that it connects a feature to itself. Terminators are useful in finishing an open shape.

1. Delete the arc.

   The two lines un-trim.
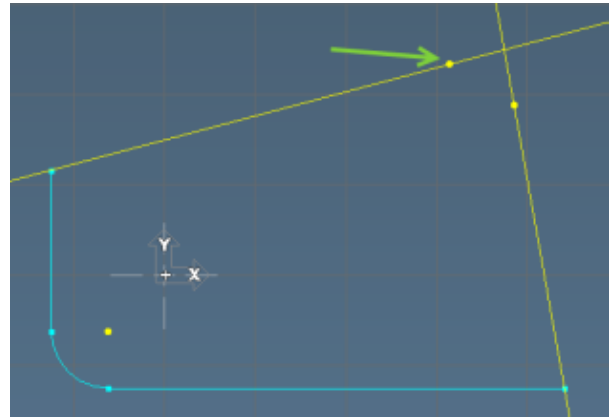
2. Select the point shown and `Ctrl+click` the line that intersects it.

   In order to create terminators, a point and the feature that goes through that point must be selected.

3. `Click` the Connect-Disconnect button.

   The point becomes a square, indicating a terminator point. The line turns blue and trims to signify that it is fully connected.
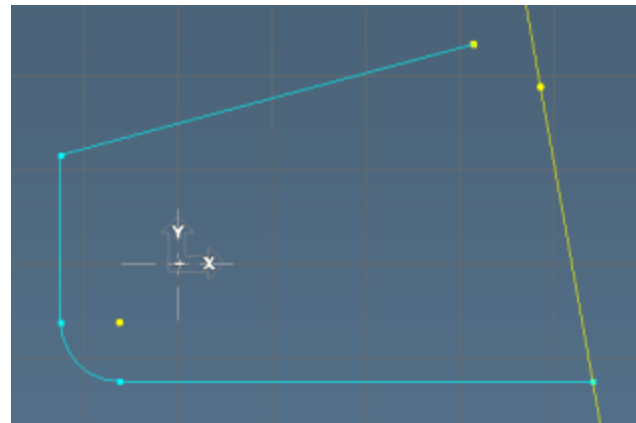
4. Select the center-point of the circle we just created.

5. Select the other point and untrimmed line.

6. `Click` the Connect-Disconnect button.

The geometry is now a fully-trimmed open shape.
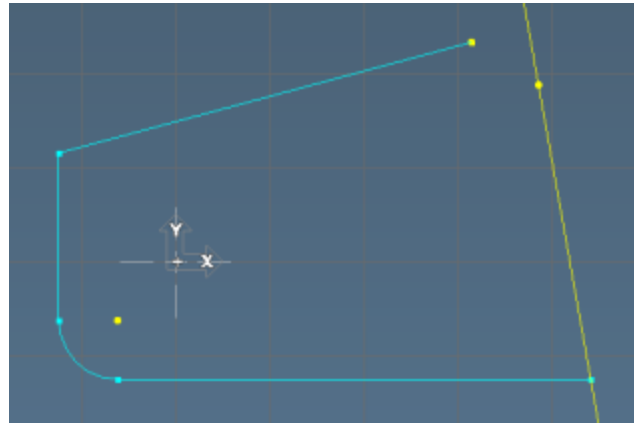
7. `Click` File> 💾 Save to save your work.

Dimension Palette

We can now add dimensions to the shape we just created.

1. Click the Dimension Palette command.

   To begin with we will set the font and size of the dimensions for this part.

2. Right-click anywhere within the title bar of the Dimensions Palette, and click Set Font.
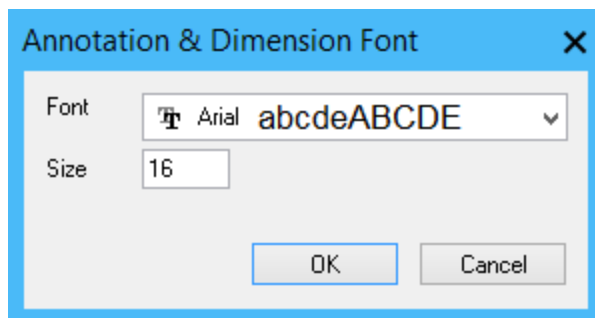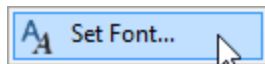
Change the font size to 16. This changes all existing and subsequent annotations, and controls the size of both the dimensions and any text within this file.

Click OK to accept.

3. Click the Point to point dimension button. This places a linear dimension between two selected points.

    The command becomes highlighted in blue.

4. Click the point shown in the screenshot below to highlight it. Now press and hold the Ctrl key. Hover your cursor over any of the other points in the part. You will notice the dimension between two points will display in a light grey colour.
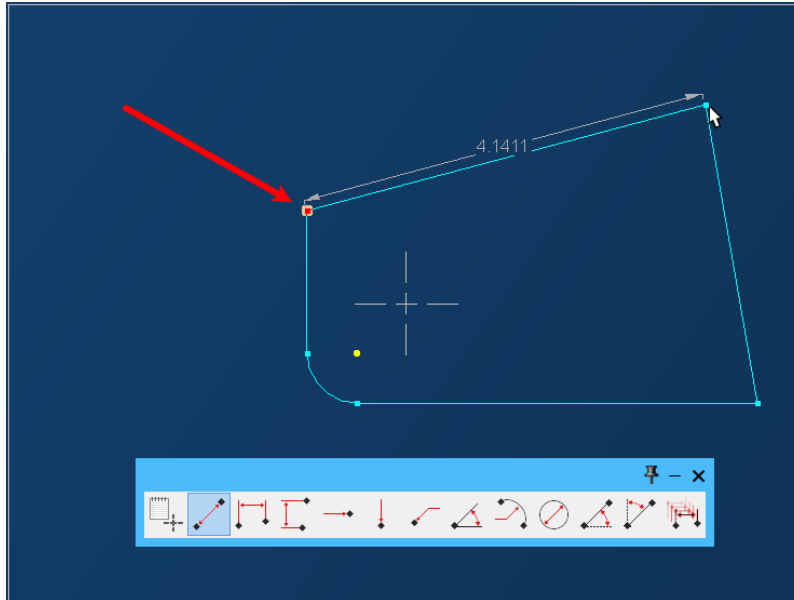
5. Hover the cursor over other points in the workspace until the dimension you require is displayed, at which time, click the second point. The color will change to white, to indicate it is now added to the part display.

6. Click the Angle Dimension command. This places a 2D angular dimension between two selected lines.

    You will note that the Point to point dimension command we used previously becomes deselected, and the angle command is now highlighted in blue.

7. Click the line indicated by the red arrow below to highlight it and once again press and hold the Ctrl key. Hover over any of the other lines to display the angle between them. (The closer to the ends of the line, the smaller/larger the leader line will become.) Click the desired distance along the second line to add the angle dimension.
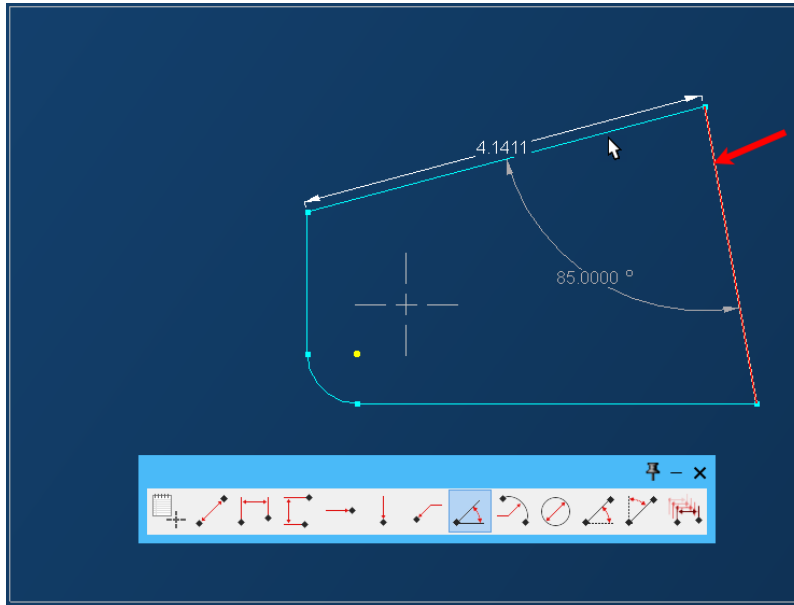
8.   Finally, click and highlight the Text command.

    You will notice that the cursor changes to a brown "I" insert cursor.

9.   With the text command highlighted, click in the workspace where you wish text to appear, and simply start typing. When you have finished typing, click away and the text command then becomes deselected.

To delete an item

    To delete a dimension or text, click and highlight it (It will change to a brown color.) If required, press and hold the Ctrl key to click and select multiple items. Pressing the Del key on your keyboard will then delete the highlighted item(s).

Moving existing text or dimensions

You can move existing text or dimensions within the workspace.

To move text, Click, hold and drag the white text (a brown box appears) and drop the brown box where you wish to reposition the text.

To move a dimension, click either the (white) leader line or the value and drag it to the required position.

Exercise 2: Mill Tutorial

This exercise creates the geometry for the part we will machine in the Mill manual.

Setting up the Part
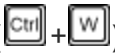
1. Open the Documents dialog (`F1`) ![icon].

2. `Click` the New button.

    New

3. Type "`Mill Tutorial`" in the File name text box.

4. Save this file where it can be easily accessed in the future. We will machine this part in the Mill manual.

5. `Click` the Save button.

    Ensure the Machine list has a 3 Axis Vertical Mill.

6. Select the inch measurement type.

    Input the values for the stock size shown.

    +X 3.7
    -X -3.7
    +Y 2.7
    -Y -2.7
    +Z 0
    -Z -0.75
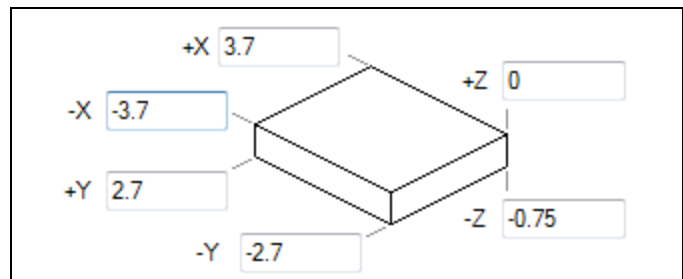
    The information contained in the dialog can be changed at any time.

7. Close the Documents dialog.(`Ctrl` + `W`)

    The preferred process to create the geometry for a particular part is to choose a starting location and work in either a clockwise or counter-clockwise direction around the part. By using this method, features will become connected at the first opportunity which simplifies geometry creation.

Creating the Geometry

Multiple Explicit Points

1. Open the Geometry palette.
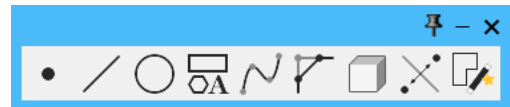
    We will now create five Explicit points.

2. Open the Point sub-palette.

3. Select Explicit point.

XYZ

4. Input the coordinate values shown.

   (This will create a point at 0,0 - you
   may not be able to see it as it is hidden
   by the axis lines)

Workgroup		
X 0	Y 0	•• •
Z 0	P None	

5. Click the Multiple Points button..

   ••

6. Change the values and press Enter.

Workgroup		
X -3	Y 0	•• •
Z 0	P None	

7. Change the values and press Enter.

Workgroup		
X -3	Y -2	•• •
Z 0	P None	

8. Change the values and press Enter.

Workgroup		
X 3	Y -2	•• •
Z 0	P None	

9. Change the coordinate values
   shown..

Workgroup		
X 3	Y 2	•• •
Z 0	P None	

10. Click the Single Point button.

    •

    If you accidently press Enter or the Multiple feature when you are done, you can press the Esc
    key or Return button.

You should now have the five points shown. (The one under the axis is hidden)
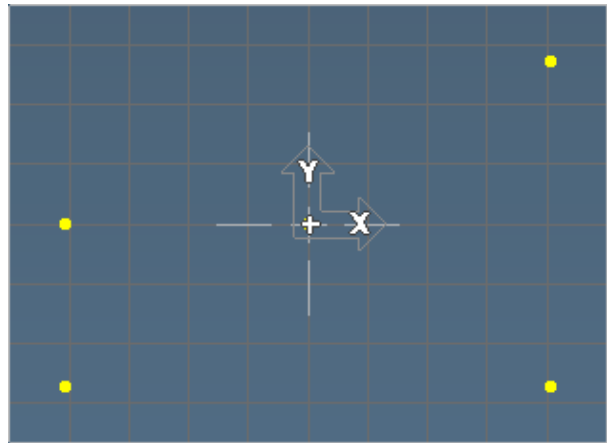
Radius and Center Point Circles

We will now create circles around each of the points.

1. Open the Circle sub-palette.

2. Select Radius & Center Point.

3. Select the point at the origin. (This is the first point created at X0, Y0)

The CP text box displays a label identifying the point selected as the center point of the circle.

4. Input the value shown for the radius (R2)
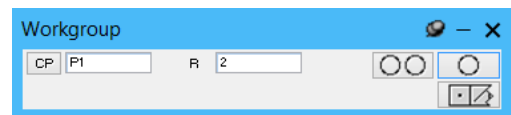
The system assigns a label to all geometry created. Geometry is defined by a letter (P for point, C for circle, and L for line) and a number indicating the order of creation. These labels are displayed with View > Labels or Ctrl+L. Many geometry dialogs display the label(s) for selected geometry.

Labels

1. Choose `Ctrl` + `L`)   or
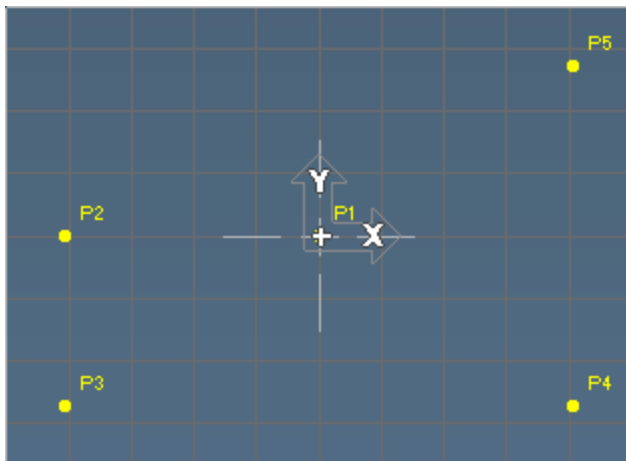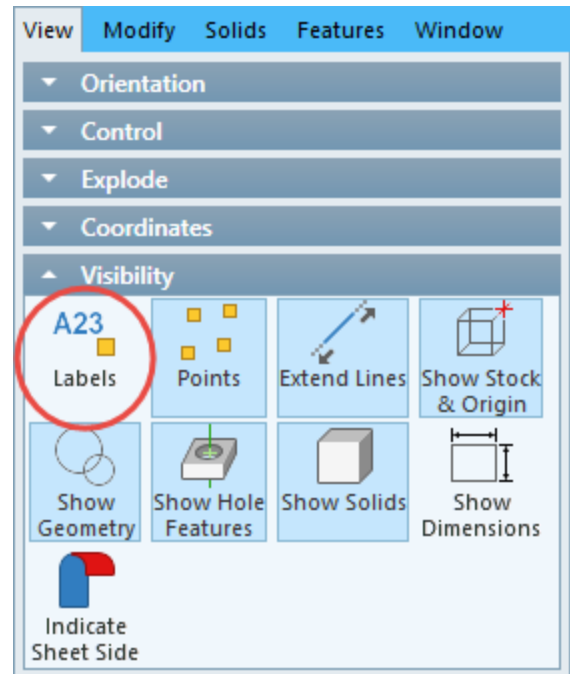
All the points created have been labeled. Labels can be turned on at any time without affecting geometry creation, and are especially helpful if there are any connection problems.

2. Choose View A23 labels again, to turn them off.

3. Click the Multiple Circles button.

   This button functions in the same way as the Multiple Points button. It allows for the creation of more than one circle using the same method (in this case, selecting an existing point to act as the center point, and then entering a radius value).

4. Select the point shown.
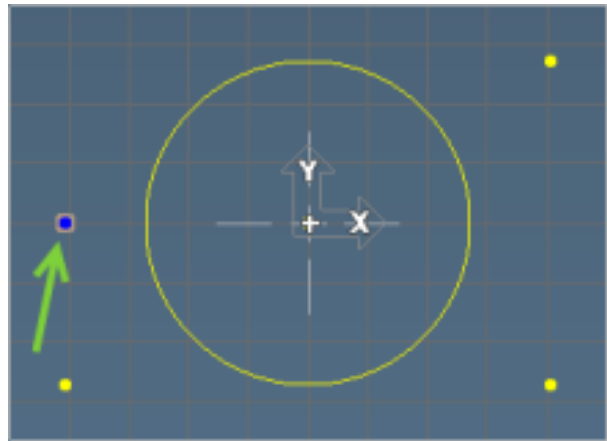
5. Input the radius value shown.
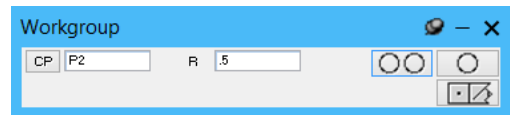
6. Press the space bar.

This is the same as pressing Enter or the Multiple Circles button.

7. Select the point shown and press Enter.
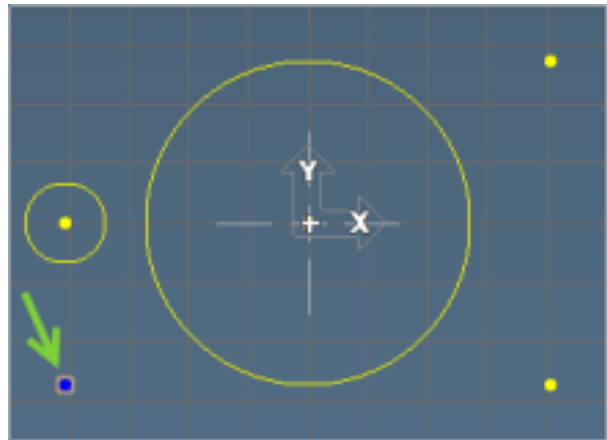
The radius is the same for the next three circles, so there is no need to change the value.

8. Select the point shown and press `Enter`.
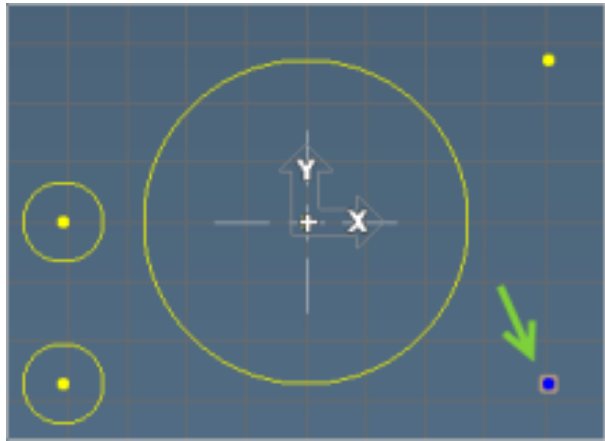
9. Select the point shown and `click` the Single Circle button.
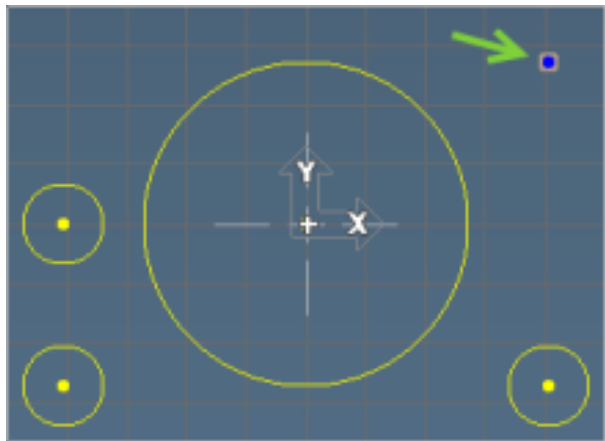
We could have easily have defined the circles with descriptor center points; however, we will be using some of the center points.

Line Tangent to Two Circles - 1

1. Open the Line sub-palette.

2. Select the two circles shown.

   (Remember the Ctrl key)
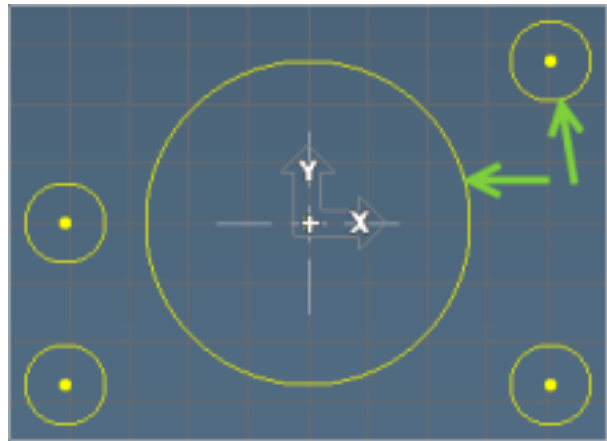
3. `Click` the Single Line button.

All four lines that are tangent to the two circles will appear.

4. Select the line shown and press the space bar.

   The line is added in blue and connected between the two circles.
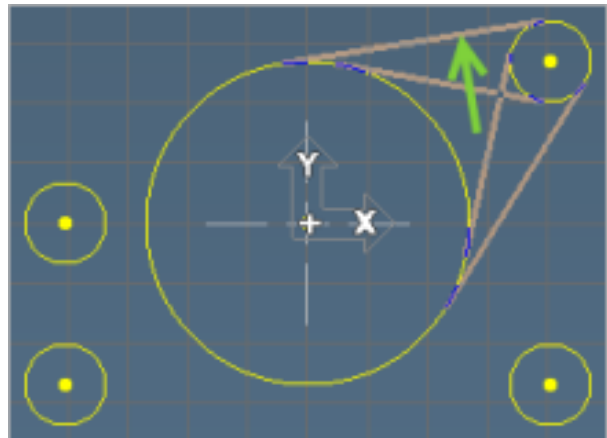
Arc - Tangent to Two Circles - 1

1. Open the Circle sub-palette.

2. Select the two circles shown.

   (Remember the Ctrl key)
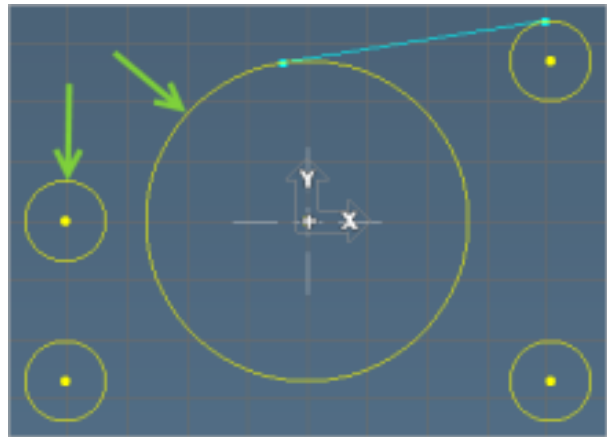
3. Input the radius value shown.

4. **Click** the Single Circle
   button.

   The eight circle tangencies
   between the two circles are
   displayed.
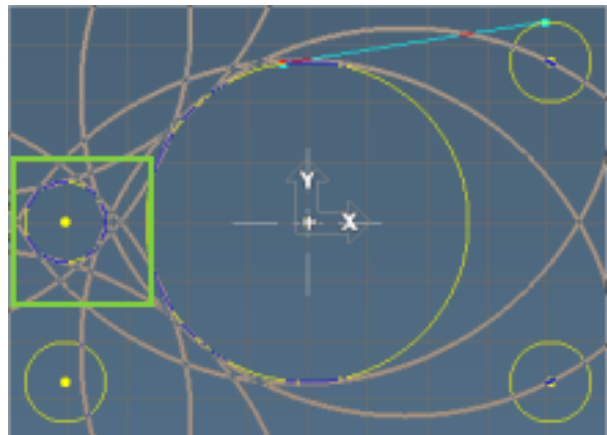
When the arc desired is larger than both the selected arcs there are eight possibilities. This can look confusing so we will zoom into the area we want.

Drag a marquee around the tangencies to zoom in.

5. Select the circle shown.

6. **Click** the OK button.

7. `Click` the View Command button.

   The View palette is also referred to as the Trackball because of the trackball centerpiece. The surrounding buttons change the view to standard views and other View menu options. It also contains an Unzoom button, which will redraw the screen to its original scale.

8. `Click` the Unzoom button or (Ctrl + U)
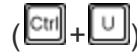
9. `Click` the View Command button again to close it.

10. Open the Circle sub-palette.

11. Select Radius & Center Point circle.

12. Select the point shown.
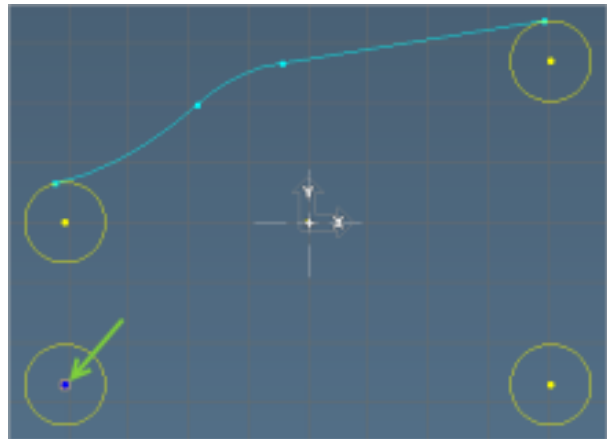
13. Input the radius and press `Enter`.
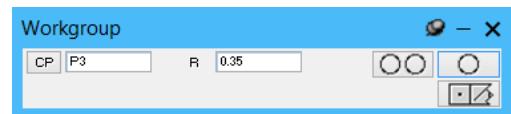
Tangent Line from Point - Circle

There are two possible lines when choosing the tangent line from a circle to a point.

1. Open the Line sub-palette.

2. Select the point and inner circle shown and press Enter.
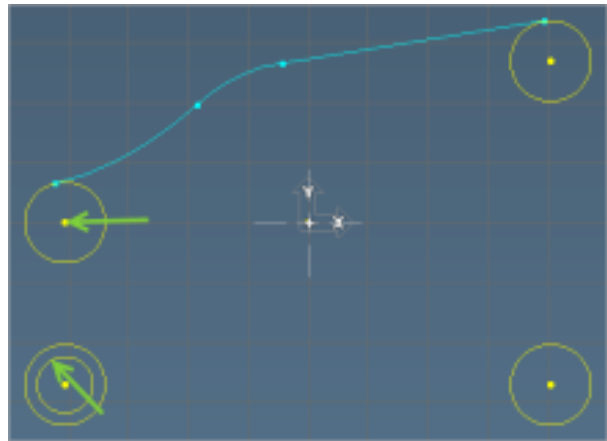
3. Select the left line and click the OK button.
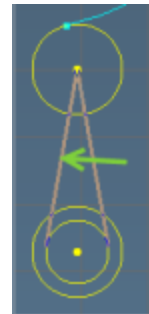
Parallel Line

1. Open the Line sub-palette.

2. Select Parallel line.

3. Select the line we just created.

4. Input the value shown.

5. Click the Single Line button.
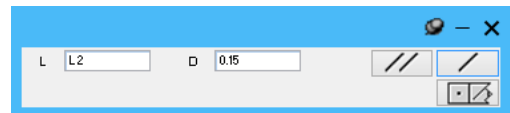
6. Select the left line and press OK.
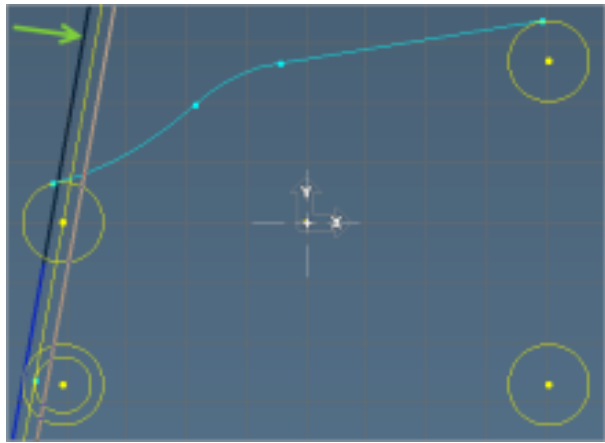
Connecting a Circle and Line

1. Select the Line and the Circle
    shown, click the
    Connect/Disconnect button.
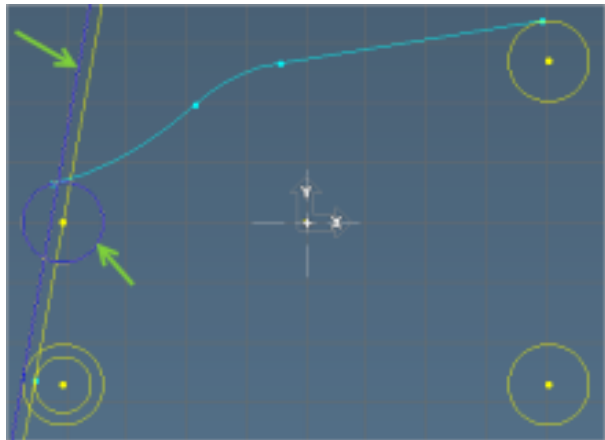
2. Select the bottom intersection
    point.
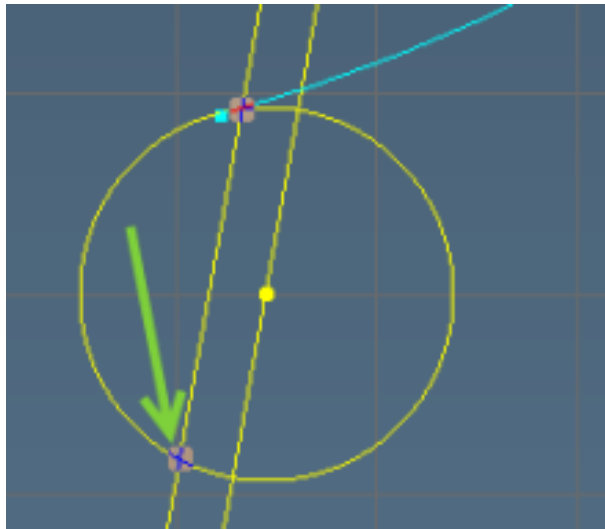
3. Click the OK button.   The circle will trim to an arc and become blue because it

now has two connector points.

4. Connect the circle and line shown.
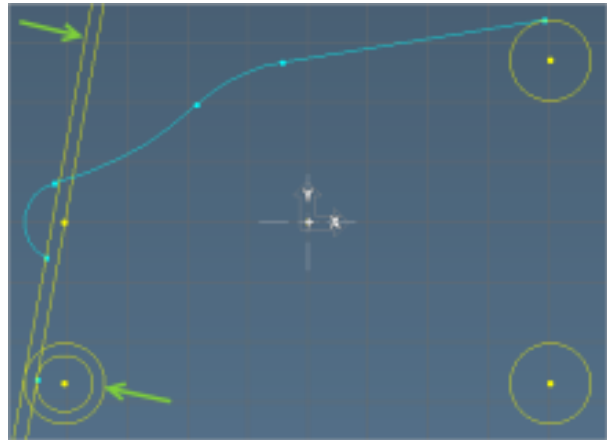
Duplicating Geometry

1. Select the arc shown.
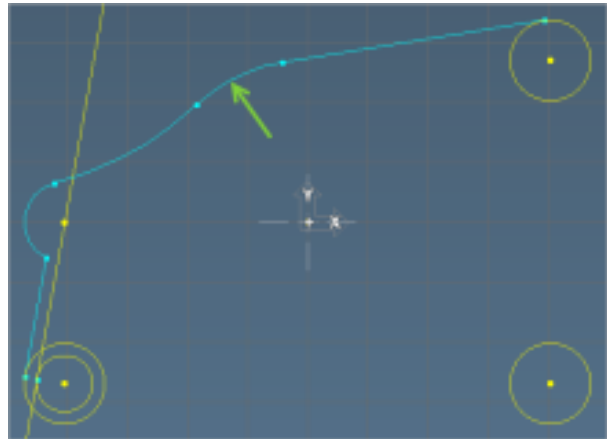
2. Choose Modify Duplicate. (Ctrl + D)
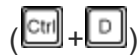
Duplicate will create a duplicate of the selected geometry.

3. Click somewhere off of the part to deselect the geometry.

   Geometry sub-palettes must not be open for deselecting in this manner. If you have a white arrow you are in selection mode.

Line Tangent to Two Circles - 2

1. Open the Line sub-palette.

2. Select the two circles shown.
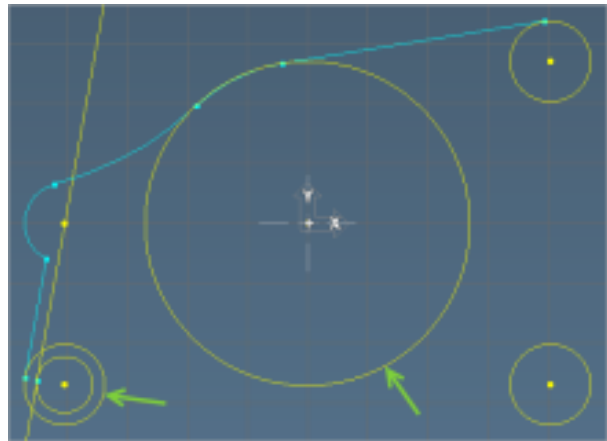
3. `Click` the Single Line button.
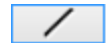
4. Select the line shown and `click` the OK button.
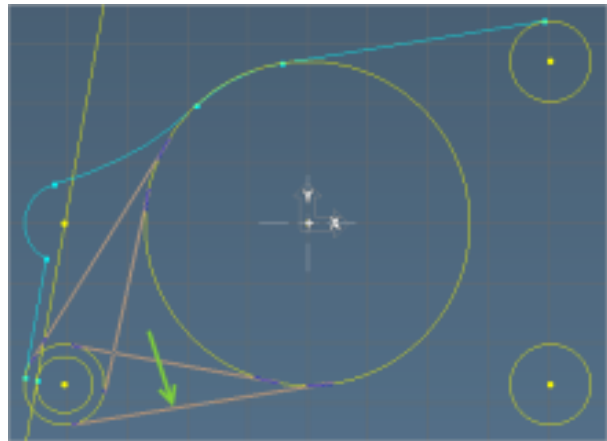
5. Open the Circle sub-palette.

Arc - Tangent to Two Circles - 2

1. **Click** the two circles shown.
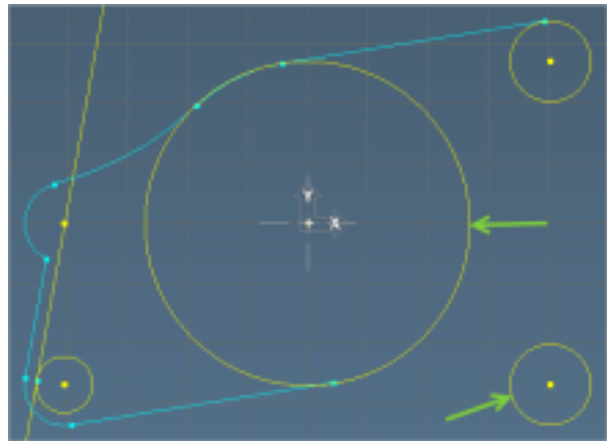
2. Input the values shown.

3. **Click** the Single Circle button.

In this case because the tangent radius is smaller than the largest circle and larger than the distance + diameter of the smaller we only have two circles to choose from.

4. Select the bottom circle as shown.
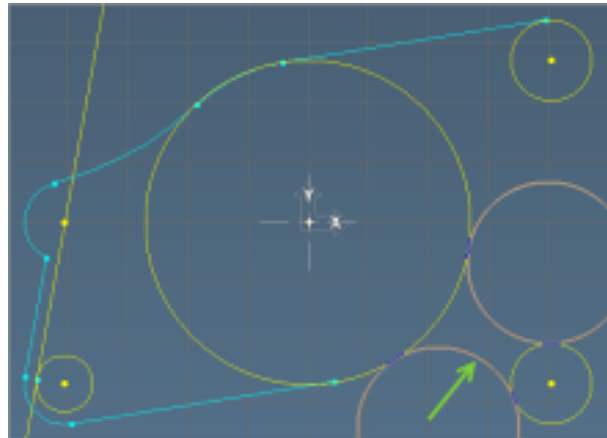
5. **Click** the OK button.

Line Between Two Points

1. Open the Line sub-palette.

2. Select the two points shown.
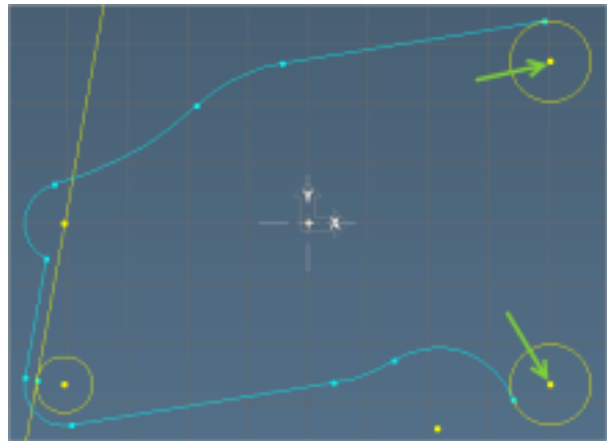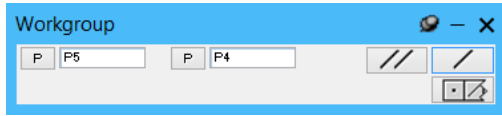
3. Click the Single Line button.
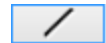
A vertical line is drawn through the two points.

4. Connect the circle and line shown.
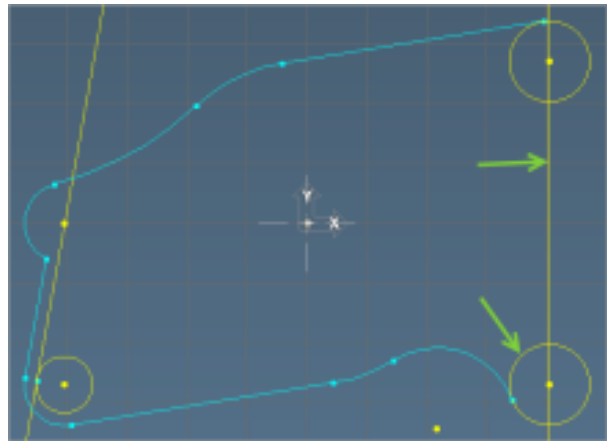
5. Select the top intersection point. Click OK.
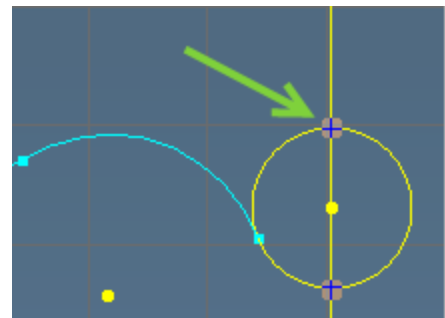
Reversing Arc Direction

1. Select the arc shown..

   The smallest arc is the default.
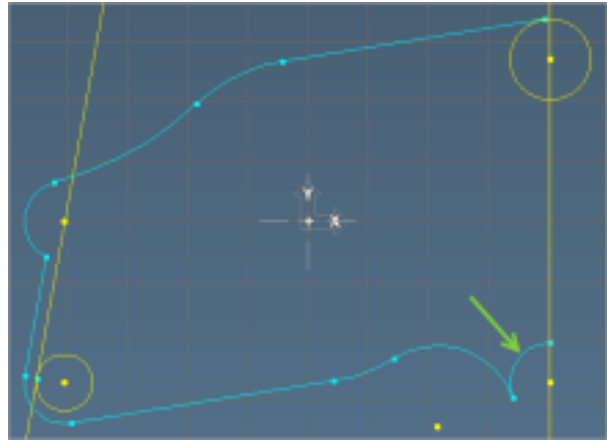
2. Choose Modify ↻ Reverse Arc.

   We have chosen to use the larger arc.

   `Click` somewhere off of the part to deselect the geometry.

Arc - Tangent to a Line and Circle

1. Open the Circle sub-palette.

2. Select the line and circle shown.
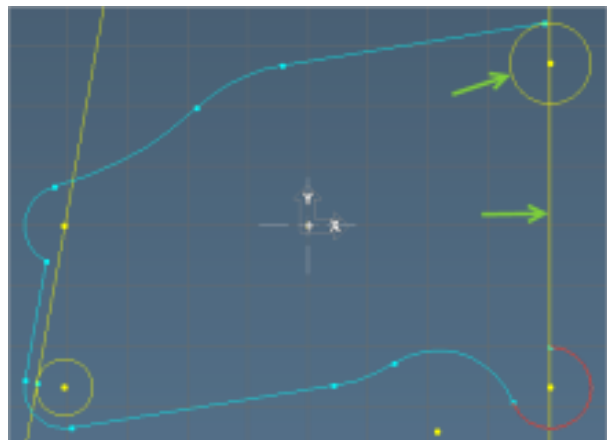
3. Input the radius values shown.

   Workgroup

   L L5   C C5
   R 0.5
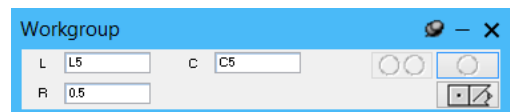
4. `Click` the Single Circle button.

5. Select the bottom right circle shown.

6. `Click` the OK button.

Selecting a Shape

1. `Double-click` the line shown.
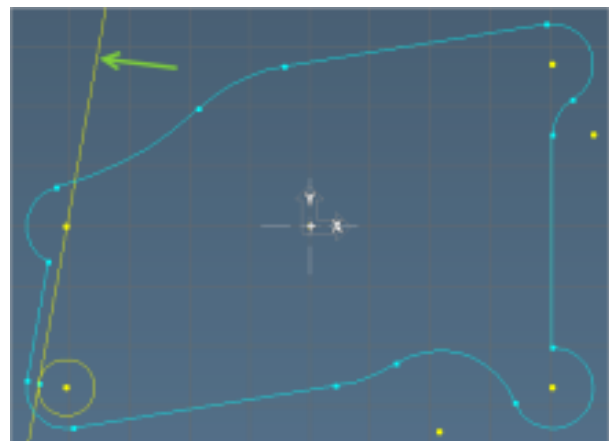
This will select all connected geometry to the line.

2. `Click` the Trash button.

Boss Geometry

1. Open the Point sub-palette.

2. Select Polar point.

3. Select the point at the origin as shown.

4. Input the values shown.

> Workgroup   − X
> P | P1   D | 1   ••   •
> A | 15
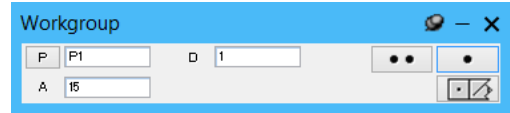
5. **Click** the Single Point button.

> This creates a point one inch from the selected point at an angle of 15°.

6. Open the Circle sub-palette.

7. Select Radius & Center Point circle.

8. Select the point at the origin as shown.
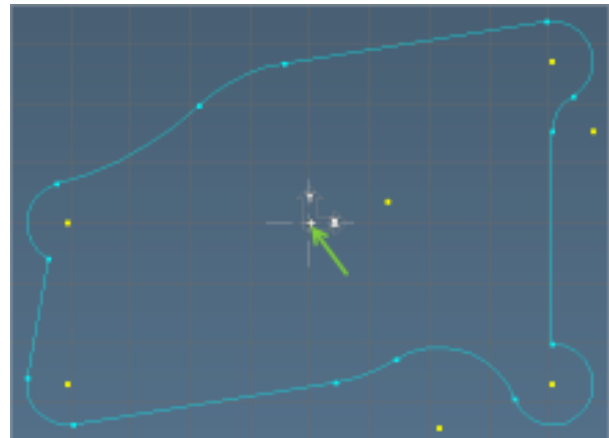
9. Input the radius value shown.

> Workgroup   − X
> CP | P1   R | 0.5   OO | O
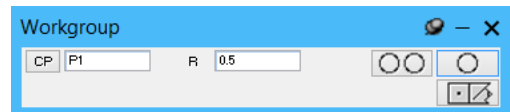
10. **Click** the Multiple Circles button.

11. Select the point shown.
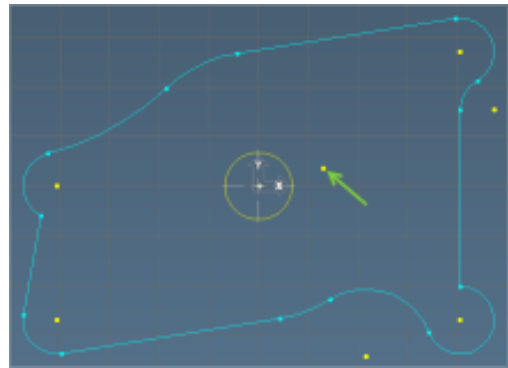
12. Input the radius value shown.
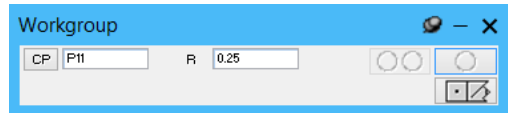
13. Click the Single Circle button.

14. Open the Line sub-palette.

15. Select the two circles shown.
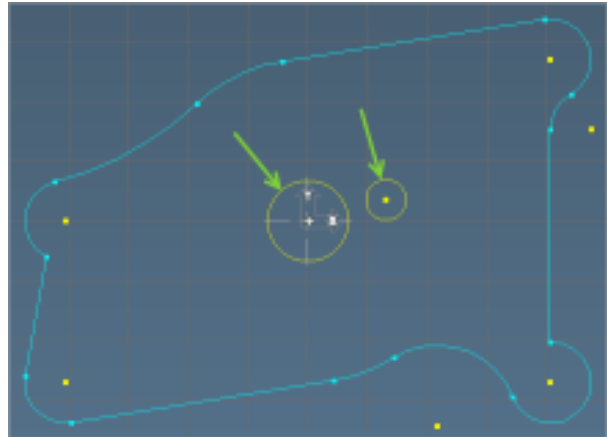
16. Click the Single Line button.
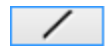
17. Select the two outside lines shown.
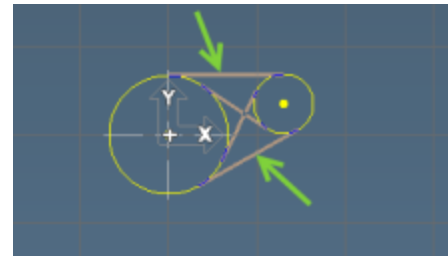
18. `Click` the OK button.

19. Select the arc shown.
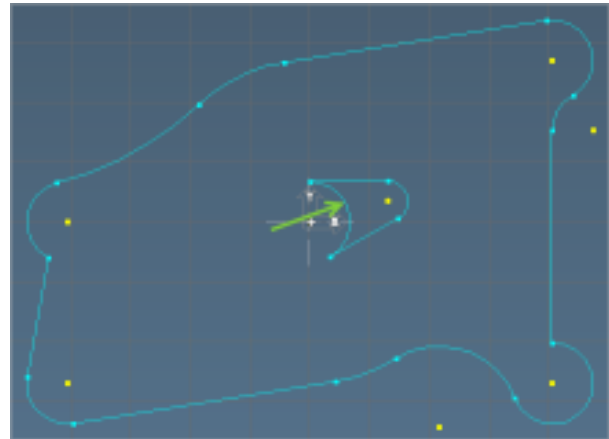
20. Choose Modify ↺ Reverse Arc.

The geometry for this part is now complete. Make sure that you save this on your computer where it can be conveniently accessed. We will machine this part in the Mill manual.

Exercise 3: Lathe Tutorial

This exercise creates the geometry for the part we will machine in the tutorial of the Lathe module.

Geometry for Lathe Parts

Setting Up the Part

1. Open the Documents dialog..   🖳   or (Ctrl+1) or   F1

2. Create a New file called `Lathe Tutorial`.

3. Select a Horizontal Lathe from the Machine list.

4. Select the inch measurement type.

5. Input the values shown.
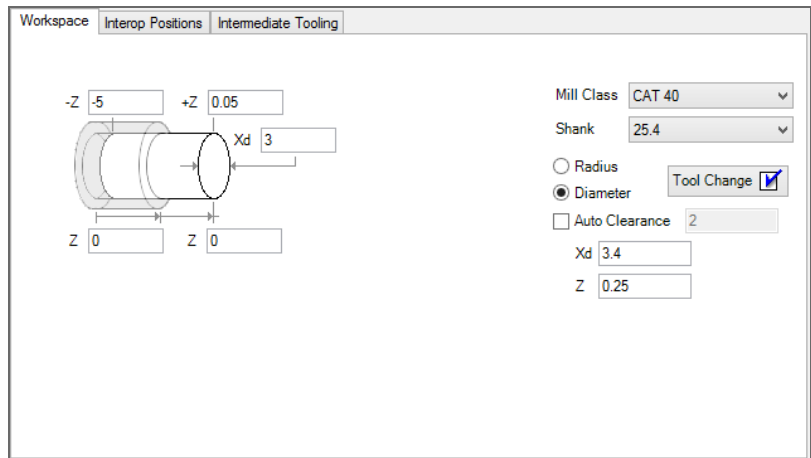
The stock size will be used to draw the workspace outline and origin marker. The workspace is drawn round. The Clearance options and Tool Change positioning are not important now and are described in the Lathe manual.

6. Select the Diameter option for the X Dimension style.

   Enter a Comment. (Remember this should help describe the part.)

7. Close the Documents dialog.

In this example, we will begin on the face of the thread, and work counter-clockwise. Turning parts are symmetrical about the X axis. Therefore, we will only need to draw the top cutaway profile of the part.

Creating Geometry

This tutorial will take you through creating geometry for a Lathe Part using the GibbsCAM Geometry Palette.
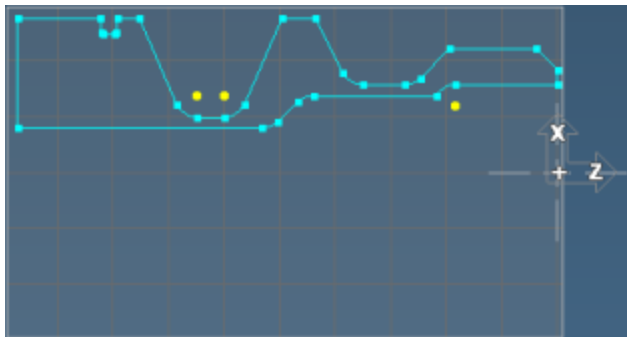
1. Open the Geometry Creation palette.
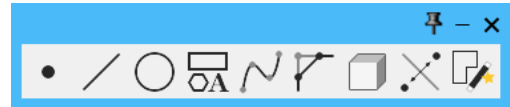
2. Open the Point sub-palette.

3. Select Explicit point **XYZ** .

Input the values shown.

4. Click the Single Point button.
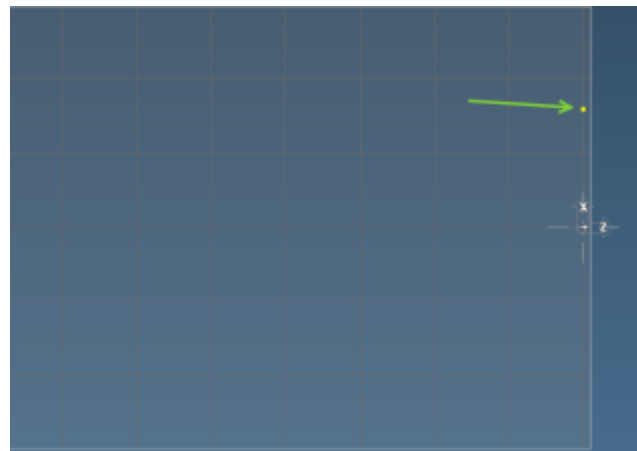
This will be our start and end point. It is created for reference only. (If your view is different, try clicking View > Home.)

Face

1. Open the Line sub-palette.
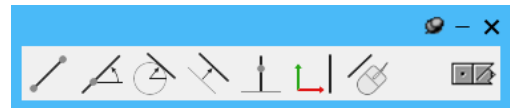
2. Select Axis line.

3. Select the vertical line mode.   Input the value shown. (Z = 0).
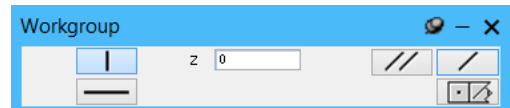
4. Click the Multiple Lines button.

OD

1. Switch to the horizontal line mode.

   Input the value shown. (Xd = 2.25).
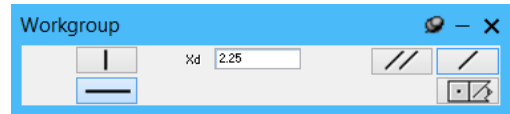
2. Click the Single Line button.

   The horizontal line is drawn.

3. Select the two lines shown.
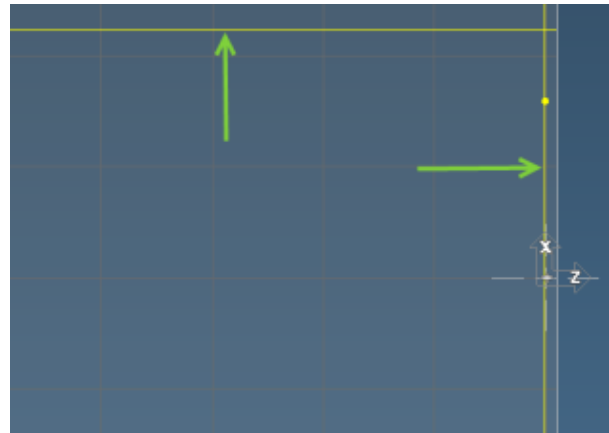
4. Choose View A23 Labels.

   All geometry will now be defined by a letter indicating the type of geometry (P for point, C for circle, L for line) and a number indicating the order of creation.

5. Shift+Drag a marquee to select the two lines and click the Connect-Disconnect button.
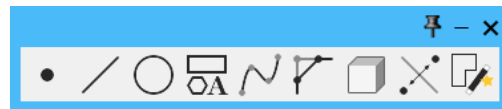
   (If your view zooms to your marquee rather than selecting the lines, ensure that you are not in a sub-menu of the geometry palette.)

The point (P2) is created as a connector point.
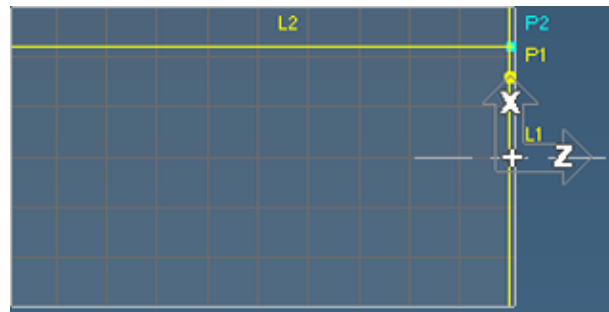
6. Turn off A23 ☐ Labels.

7. Open the Line sub-palette.

8. Select Point Angle line.

9. Click the P button

This area allows you to create a descriptor point which the angled line intersects.

10. Input the values shown. (A= 45, Xd = -0.975, Xd = 2.25)

Workgroup

| P | A | 45 | // | / |
| P | Z | -0.975 | Xd | 2.25 |
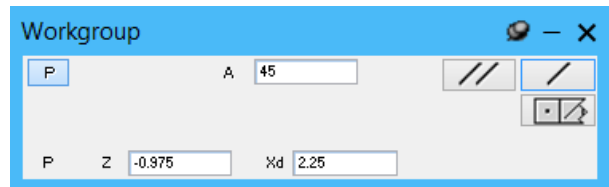
11. Click the Single Line button.

Using the Connect-Disconnect button, we will connect the two lines at the point where they intersect.

12. Connect the two lines shown.
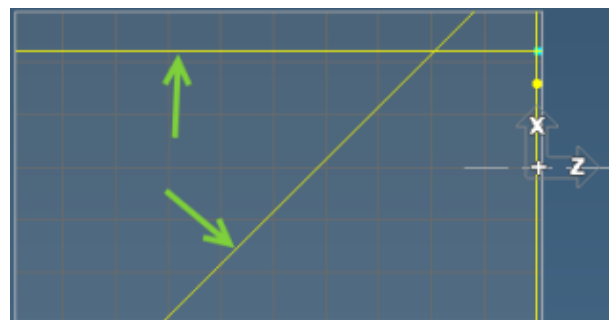
The line will trim because it has two connector points.
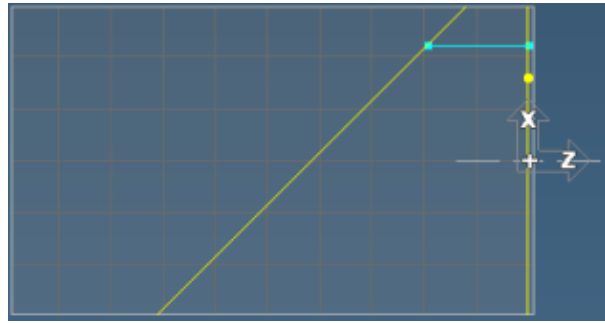
13. Open the Line sub-palette.

14. Select Axis line.

15. Select the horizontal line mode.
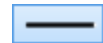
16. Input the value shown. (Xd = 1.6).

17. Click the Single Line button.
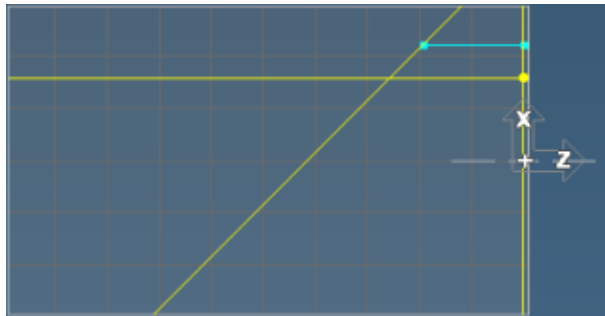
18. Open the Line sub-palette.

19. Select Point Angle line.

20. Click the P button

Input the values shown. (A = -63, (Z = -2.2, Xd = 2.8).

Workgroup

| P | A | -63 |
| P | Z | -2.2 | Xd | 2.8 |
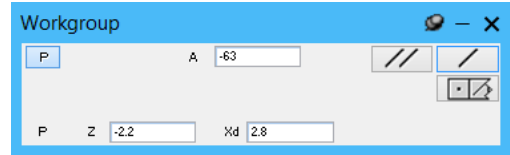
ⓘ Angle values follow the standard Cartesian coordinate system, as shown. Negative values are acceptable as input.

```
              (-270°)
               90°
                ▲
                |
                |
                |
180°  ◄─────────┼─────────► 0°
(-180°)         |
                |
                |
                ▼
              270°
              (-90°)
```
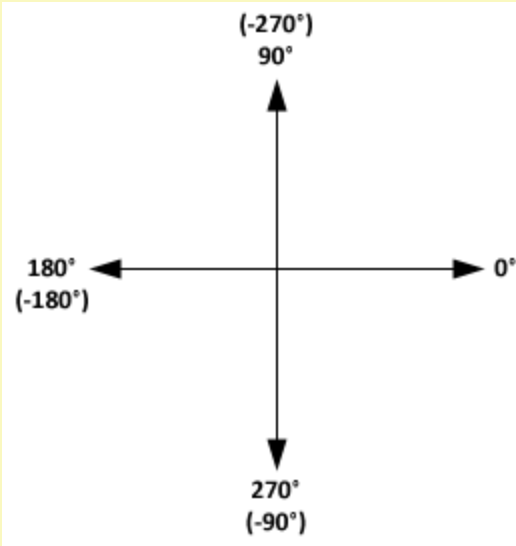
21. Click the Single Line button.

22. Connect the two lines shown.
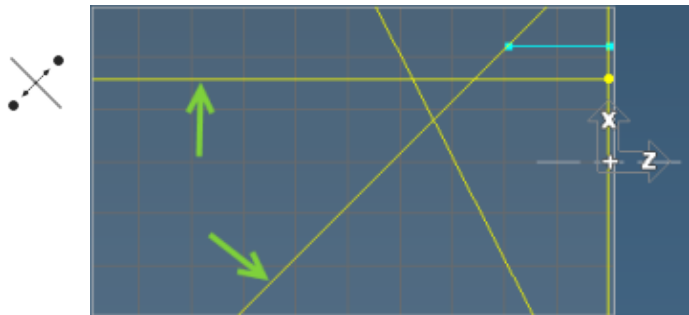
23. Connect the next two lines shown.
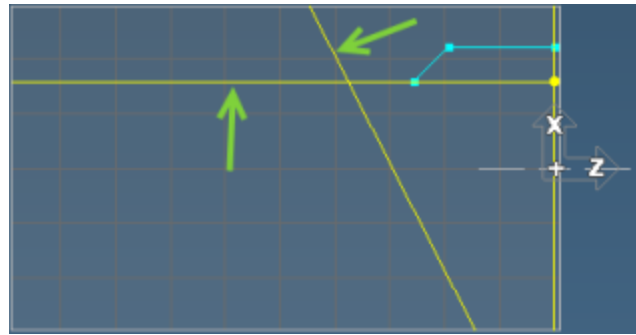
24. Your screen should now look like this
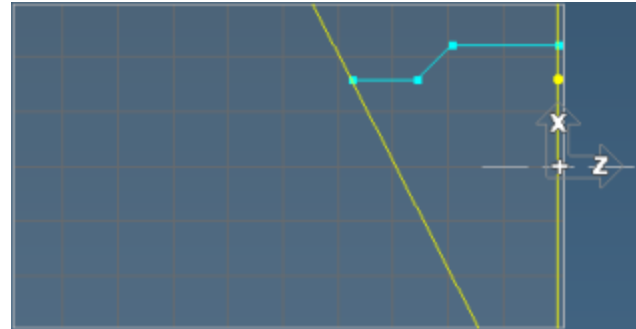
25. Open the Line sub-palette.

26. Select Axis line.

27. Select the horizontal line mode.

28. Input the value shown. (Xd = 2.8).

Workgroup

Xd 2.8

29. Click the Single Line button.

30. Connect the two lines shown.
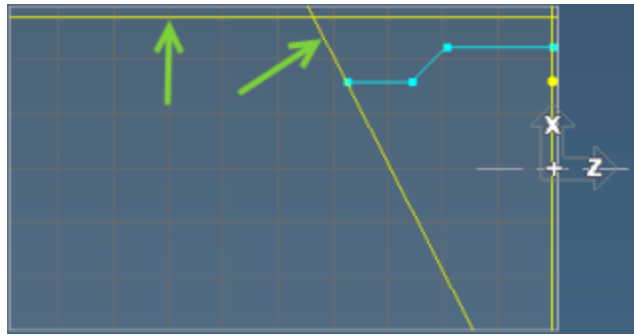
Your screen should now look like this
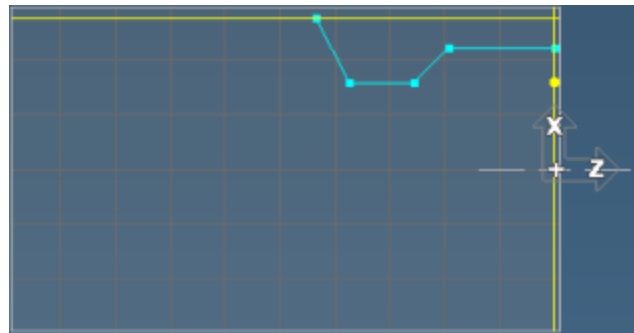
31. Open the Line sub-palette.

32. Select Point Angle line.

33. Click the P button [P] and input the values shown. (A = -114, Z = -2.5, Xd = 2.8).

Workgroup

P   A -114

P   Z -2.5   Xd 2.8
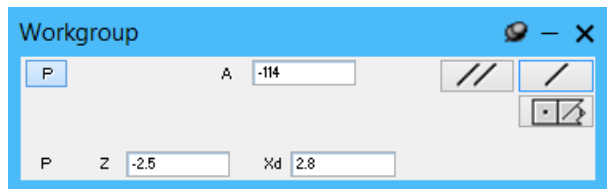
34. Click the Single Line button

35. Connect the two lines shown.
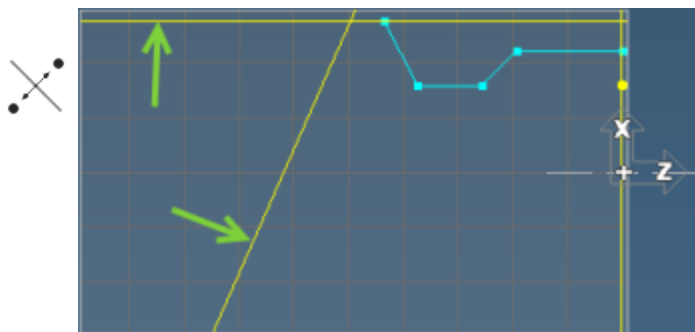
Your screen should now look like this
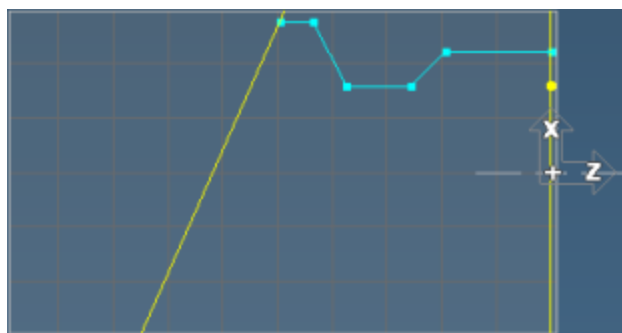
36. Open the Line sub-palette.

37. Select Axis line.

38. Select the horizontal line mode.

39. Input the value shown. (Xd = 1).

Workgroup

Xd 1

40. **Click** the Single Line button.
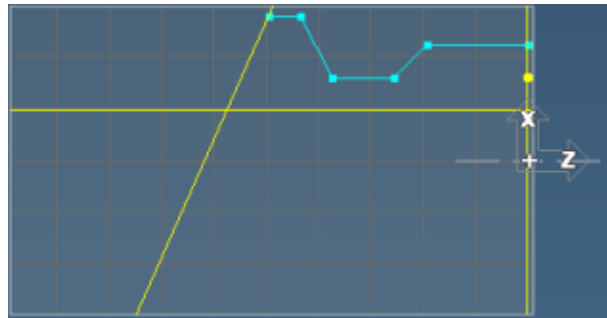
41. Open the Line sub-palette.

42. Select Point Angle line.

43. **Click** the P button and input the values shown. (A = **-66**, Z = **-3.8**, Xd = **2.8**).

44. **Click** the Single Line button.
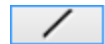
45. Open the Circle sub-palette.

46. Select the lines shown.

    Remember the **Ctrl** key is not needed because a sub-palette is open.
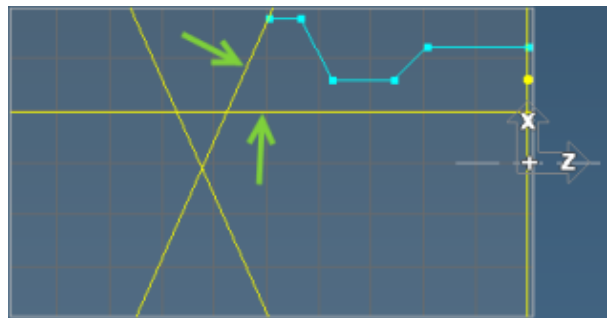
47. Input the radius value shown. (R = .2).

Workgroup			⦿ − ✕
L L7	L L8	⭕⭕	⭕
R .2			⬚
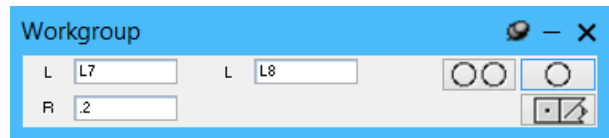
48. Click the Multiple Circles button.

⭕⭕

The four circle tangencies between the two lines selected will be displayed.

49. Select the upper left circle. Click the OK button.
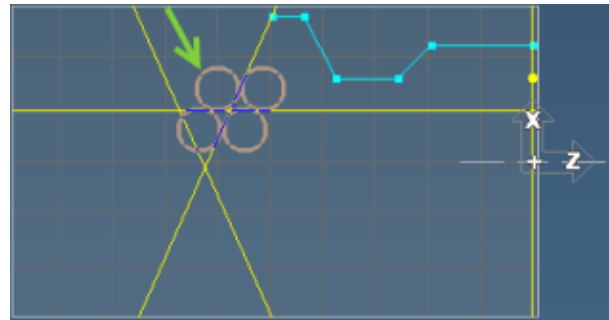
50. Select the lines shown. The radius should still be defined as 0.2; therefore, no editing is necessary.
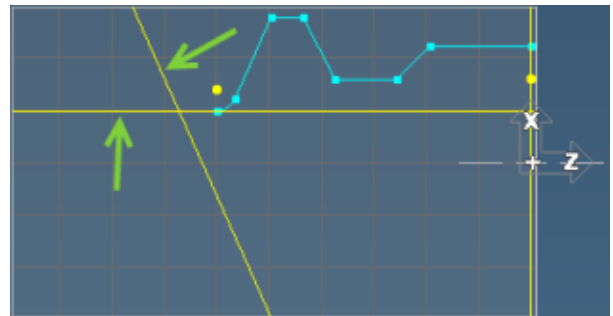
51. Click the Single Circle button.

⭕

52. Select the upper right circle as shown.

Click the OK button.
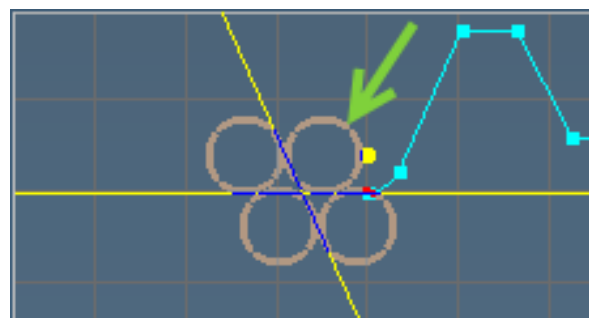
53. Select the line shown below and choose

    Modify ⊞ Duplicate.
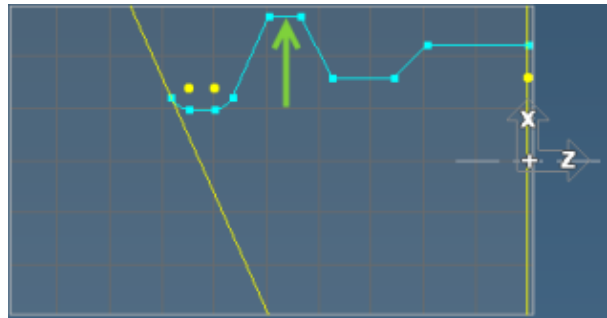
54. Connect the lines shown.
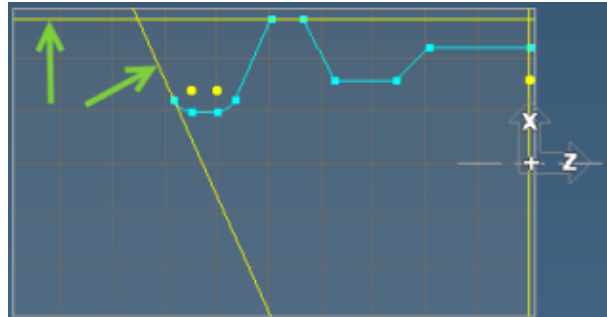
Your screen should now look like
this
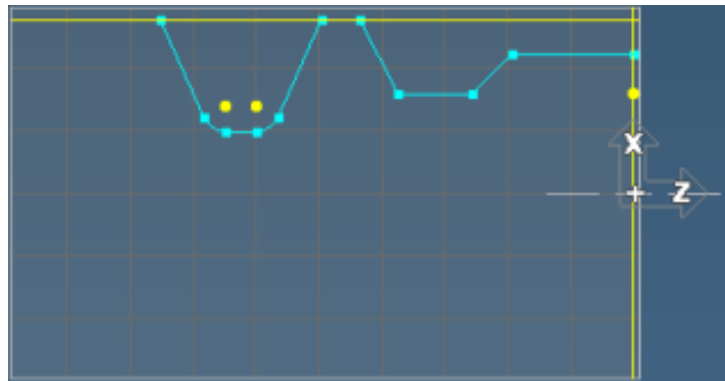
55. Open the Line sub-palette.

56. Select Point Angle line.

57. Click the P button P and input the
    values shown. (A = 85, Z = -4 Xd = 2.8).

Workgroup

P   A   85

P   Z -4   Xd 2.8
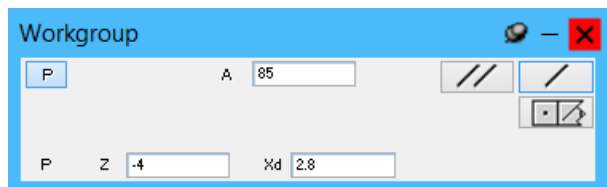
58. Click the Multiple Lines button.
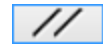
59. Change to the values shown. (A = 95
Z = -4.15, Xd = 2.8).

Workgroup

| P | | A | 95 | | |

| P | Z | -4.15 | | Xd | 2.8 |
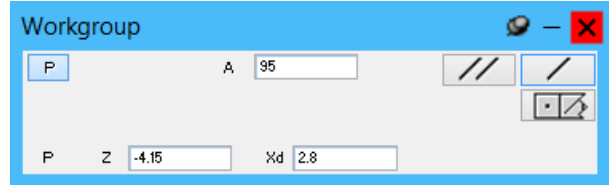
60. Click the Single Line button.

Your screen should now look
like this
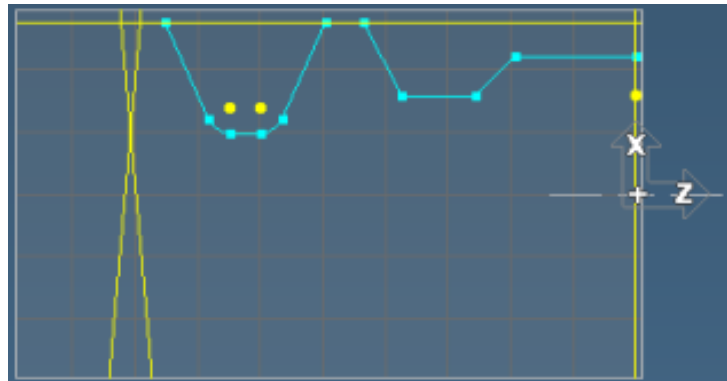
61. Open the Line sub-palette.

62. Select Axis line, then select horizontal line mode.

63. Input the value shown. (Xd = 2.52).

Workgroup

| | Xd | 2.52 |
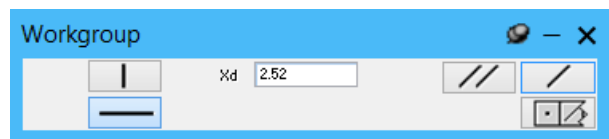
64. Click the Single Line button.

65. Connect the lines shown.
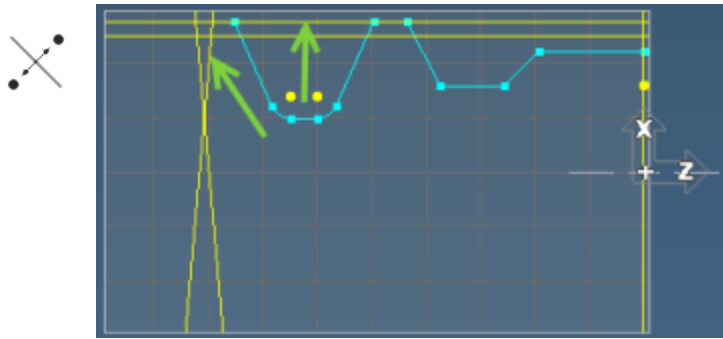
66. Next connect these lines.
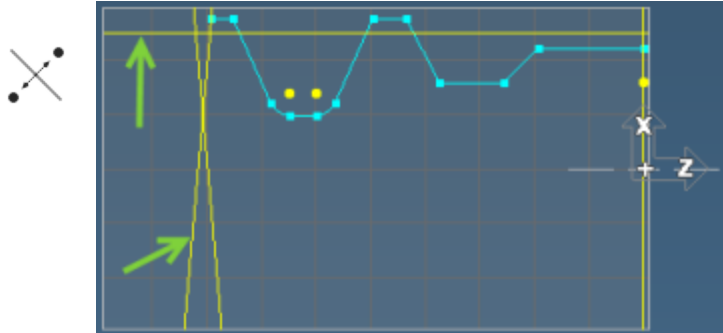
67. Select the lines as shown, then connect them.
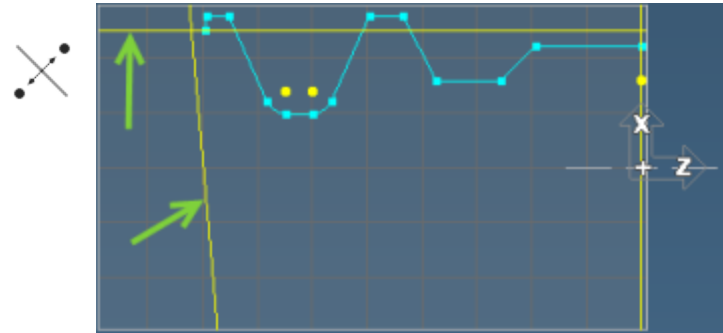
68. Select the line shown below

    and choose Modify
    Duplicate. ($\boxed{\text{Ctrl}} + \boxed{\text{D}}$)
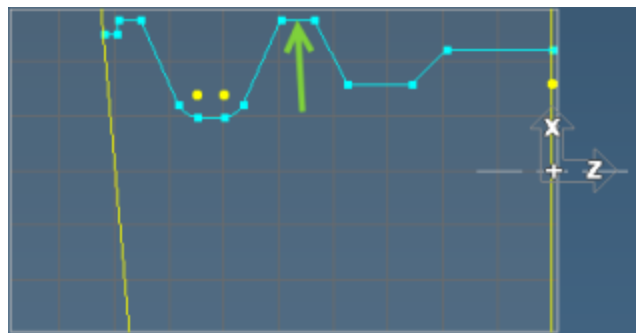
69. Connect the two lines as shown.
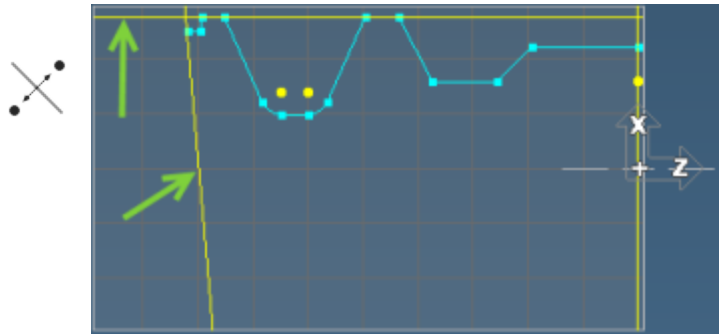
Backface

1. Open the Line sub-palette.

2. Select Axis line.

3. Select the vertical line mode.

4. Input the value shown. (Z = -4.9).

Workgroup

Z   -4.9

5. Click the Single Line button.
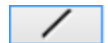
6. Connect the lines shown.
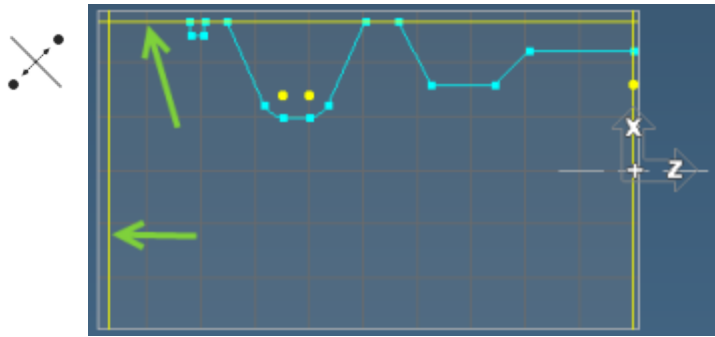
Your screen should now look like this
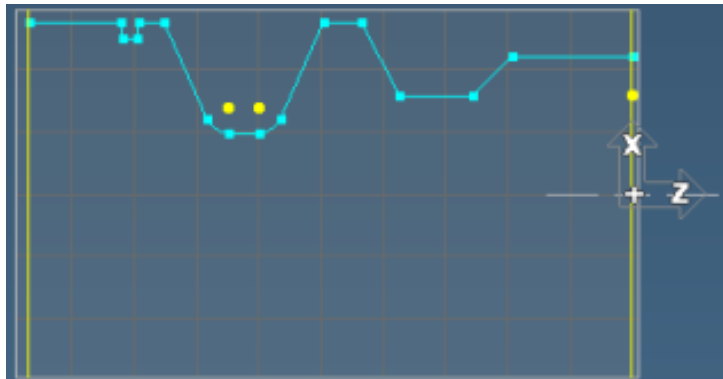
7. Open the Line sub-palette.

8. Select Axis line.

9. Select the horizontal line mode.
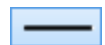
10. Input the value shown. (Xd = .8).
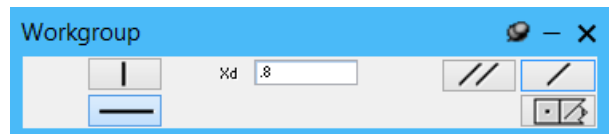
11. Click the Single Line button.

12. Connect the lines shown.
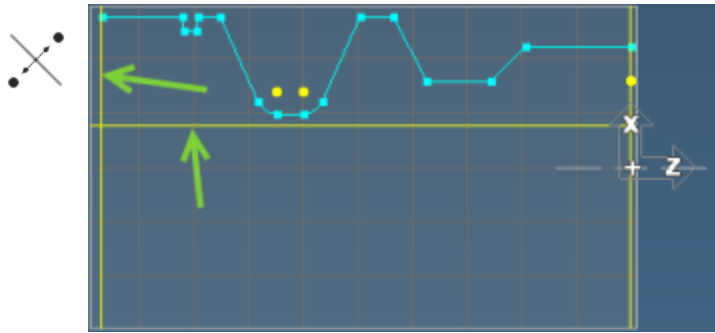
Your screen should now look like this
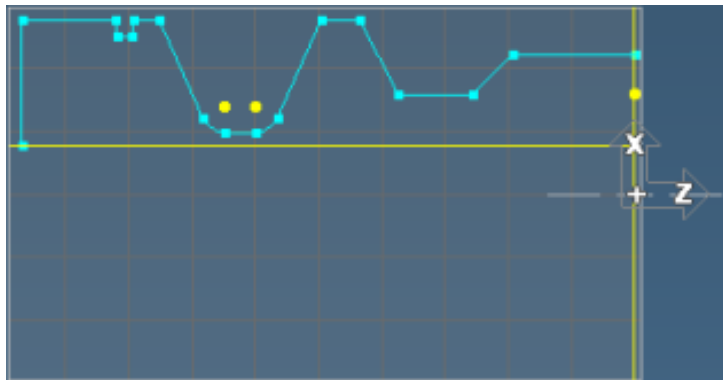
ID

1. Open the Line sub-palette.

2. Select Point Angle line.

3. Click the P button ⬚P⬚ and input the values shown. (A = 45, Z = -2.6, Xd = 0.8).
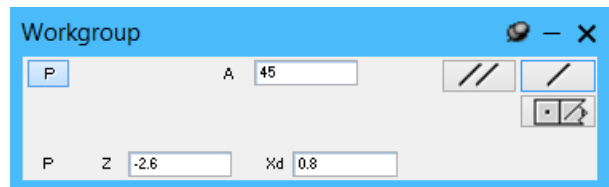
4. Click the Single Line button

5. Connect the lines shown.
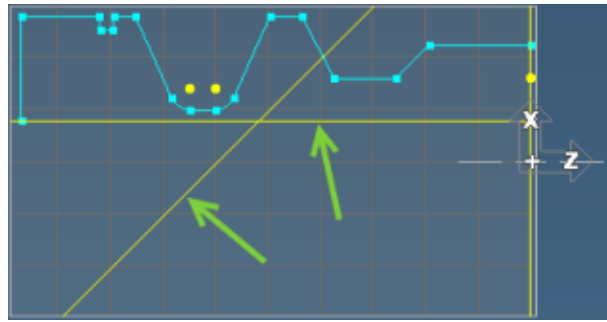
6. Open the Line sub-palette.

7. Select Axis line.

8. Input the value shown. (Xd = `1.4`).

Workgroup

Xd 1.4

9. `Click` the Single Line button.

10. Connect the lines shown.
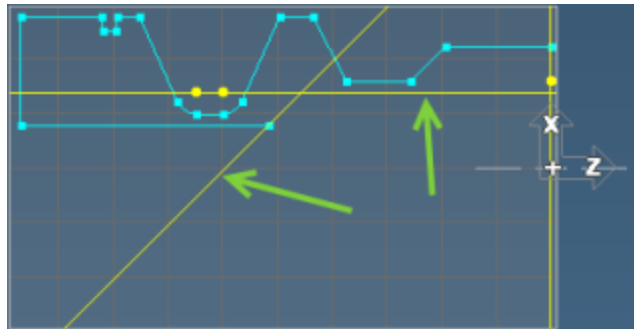
11. Select the line shown below
    and choose Modify ⊞
    Duplicate.

    Ctrl + D
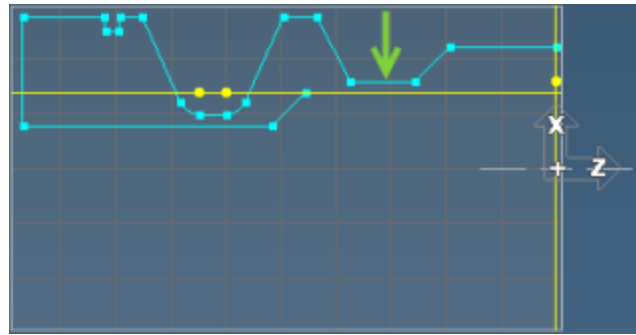
12. Open the Point sub-palette.

    ●

13. Select Explicit point.

    XYZ
    ●

14. Input the values shown (Z = -1.1, Xd =
    1.4).

15. Click the Single Point button.

    ●

16. Select the point and line shown.
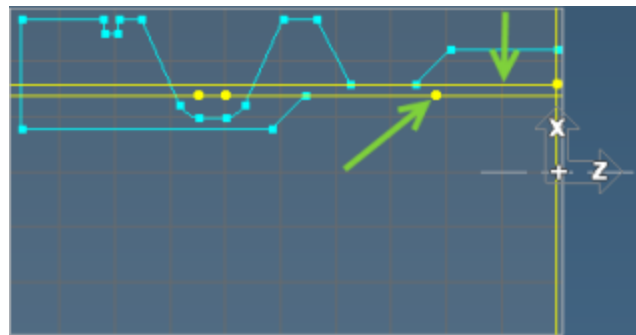
17. Open the Circle sub-palette.

    ○

18. Input the value shown. (R = `0.2`).

Workgroup				
P	P23	L	L19	
R	0.2			

19. `Click` the Single Circle button.

20. `Click` the OK button.
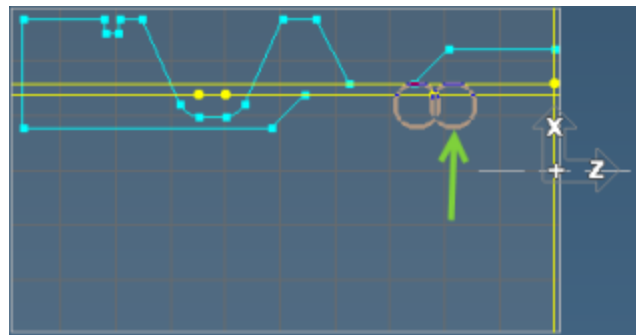
The circle selected will be drawn and a connector point will be created between the circle and the line. When tangent features are created they will create connector points when possible.

Explicit Connect

Connect the circle, point and line shown.
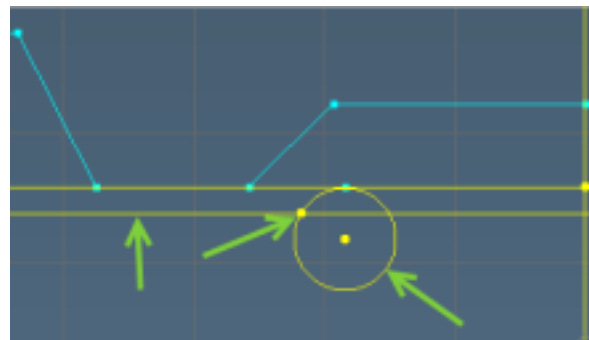
When using a point and two features to make connections the multiple features dialog is bypassed because the point is specifying the connection position. This is usually the case when connecting non-tangent circles with other features.

Closing the Shape

Connect the lines shown.
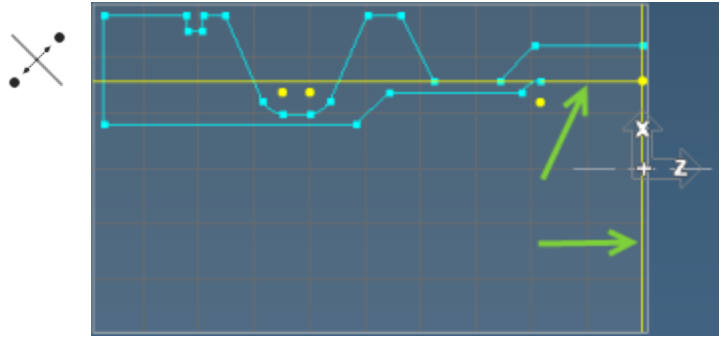
Notice the point has now been replaced by a connector even though we didn't select it.

Adding Fillets

Now we will create the fillets and chamfer specified on the print.

1. Ctrl+click the 4 points shown.
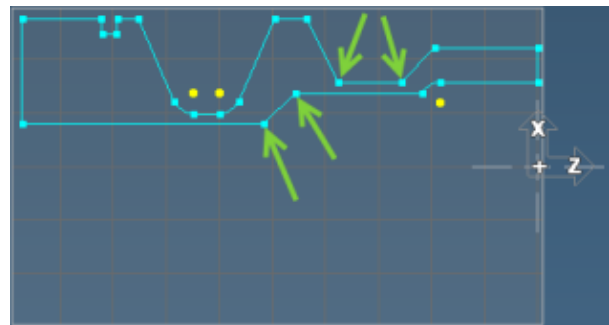
2. Open the Fillet-Chamfer sub-palette.

3. Select the Fillet mode.

4. Input the value shown. (R = 0.2).
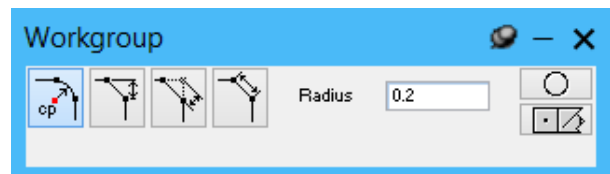
5. `Click` the Single Circle button.

6. `Ctrl+click` the 2 points shown.
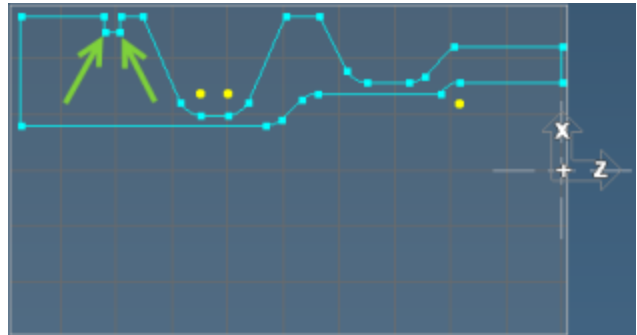
7. Open the Fillet-Chamfer sub-palette. (The Fillet mode should still be active.)

8. Input the value shown. (Radius = `0.015`).
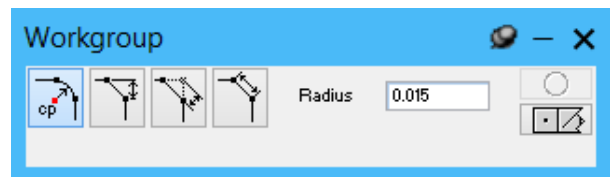
9. `Click` the Single Circle button.

Adding a Chamfer

1. Select the point shown.
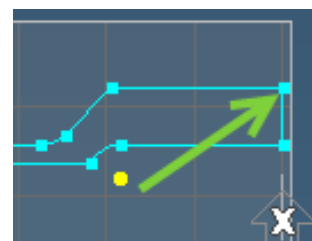
2. `Click` the Chamfer-Side button.

Workgroup

Side   0.2

3. Input the side value shown (Side = .2).

4. Click the Single Line button.

The geometry for this part is now complete.

Save this part.

We will machine this part in the Lathe manual.

Exercise 4: Shuttle

OD

1. Create the horizontal lathe part shown.

-Z -4   +Z 0.1

Xr 2.5

Z 0   Z 0

2. Select the Radius and Inch options.

3. Open the Geometry palette.

4. Open the Circle sub-palette.

5. Select Radius & Center Point circle.

6. `Click` the CP button `CP` and input the values shown. (R = `1`, Z = `-1`, Xr = `0`).

Workgroup

| CP | | R | 1 |
| CP | Z | -1 | Xr | 0 |
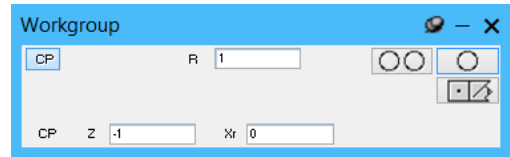
7. `Click` the Single Circle button.

8. Open the Line sub-palette.

9. Select Axis line.

10. Select the horizontal line mode.
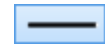
11. Input the value shown (Xr = `0.875`) and press `Enter`.

Workgroup

| | Xr | 0.875 |

12. Connect the circle and line shown.
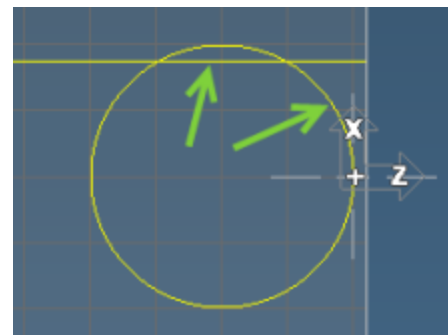
The two possible points of intersection between the selected line and circle will be drawn, and the dialog shown will come up on the screen asking you to select the desired features.

13. Select the right point as shown and
    `click` the OK button.
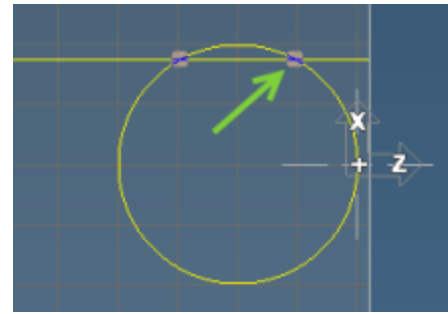
14. Open the Line sub-palette.

15. Select Axis line.

16. Select the vertical line mode.

17. Input the value shown. (Z = `-1`).

18. `Click` the Single Line button.

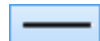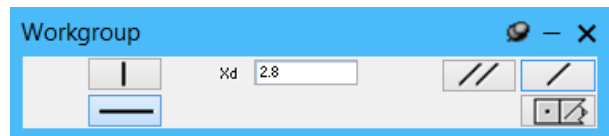19. Connect the lines shown.
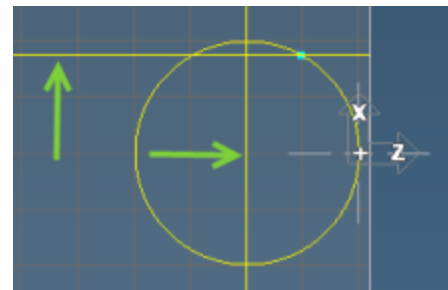
20. Open the Point sub-palette.

21. Select Explicit point.

XYZ

22. Input the coordinate values shown. (Z = -1, Xr = 0.938).
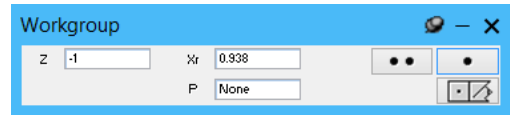
23. Click the Single Point button.

24. Zoom in on the area shown.

    Because some of the points are very close together, we will zoom in on the area to make the necessary geometry selections much easier.
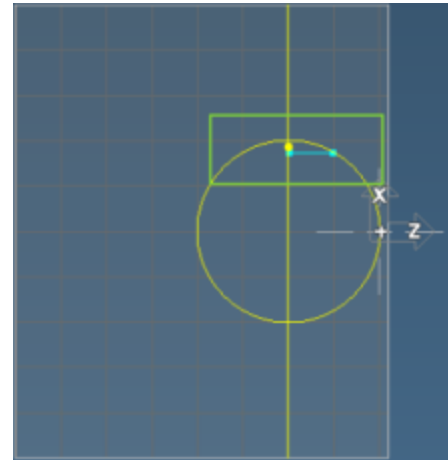
25. Open the Circle sub-palette.

26. Select Radius & Center Point circle.

27. Click the CP button CP and input the values shown. (R = 0.5, Z = -1.621 Xr = 1.5).
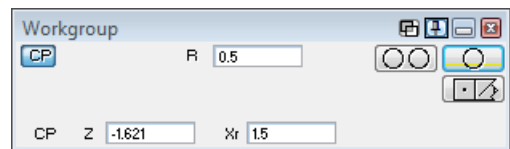
28. Click the Single Circle button.

29. Open the Line sub-palette.

30. Select the circle and point shown.
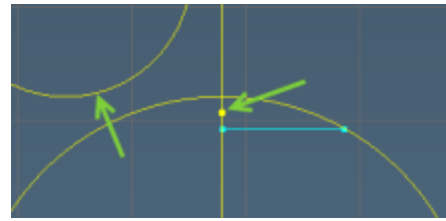
31. Click the Single Line button.

32. Select the bottom line shown.
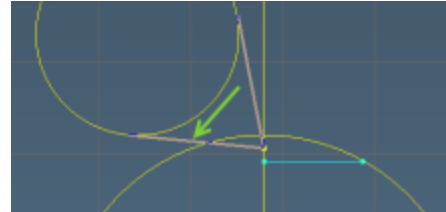
33. Click the OK button or press

The point created between the line and the circle is a connector point. However, the point that was used to create the line remains a plain yellow point. We will need to manually connect that point.

34. Connect the lines shown.
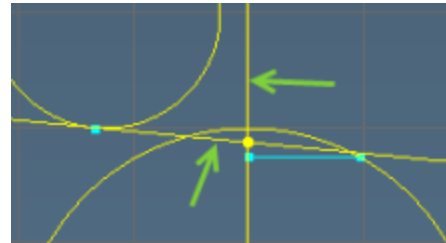
35. Open the View palette and click the No Zoom button.

This will fit the workspace stock in the window.

36. Open the Line sub-palette.

37. Select Angle & Tangent Circle line.

38. Select the circle shown.
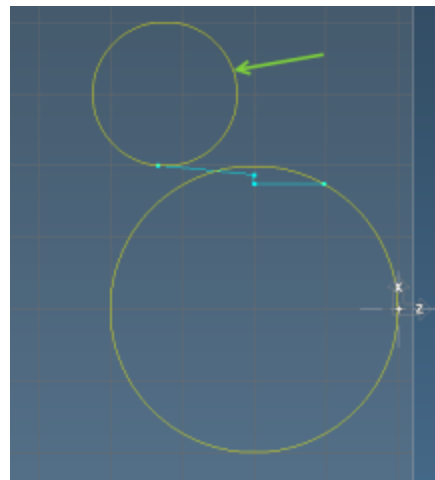
39. Input the angle value shown. (A = 140).

40. Click the Single Line button.

41. Select the line shown and press Enter
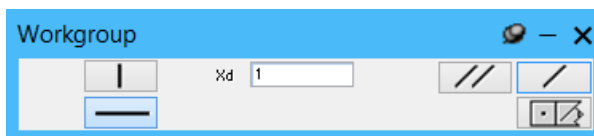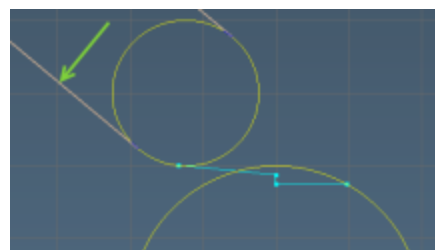

42. Open the Line sub-palette.


43. Select Axis line.

44. Select the horizontal line mode and enter the value shown. (Xr = 1.651).
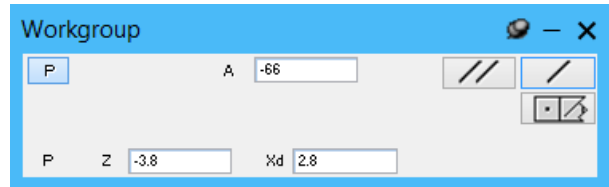
45. Click the Single Line button.

46. Open the Point sub-palette.

47. Select the two lines as shown.
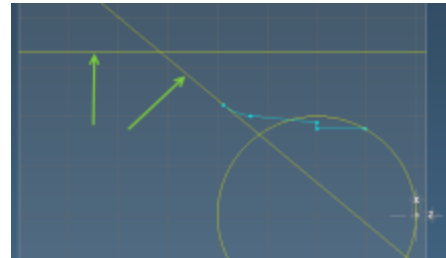
48. Click the Single Point button.

   The point created is a connector point and the angled line is trimmed.

49. Select the horizontal line and click the Trash button.

   The point will change to a plain point and the angled line will change to yellow and become untrimmed.
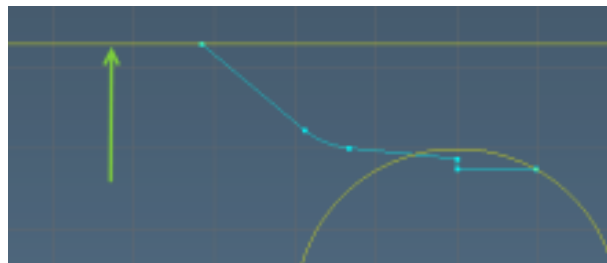
50. Open the Line sub-palette.

51. Select Point Angle line.

52. Click the P button [P] and input the values shown. (A = 135, Z = -3 Xr = 2).

Workgroup

P	A 135	// /
		·◻
P	Z -3	Xr 2
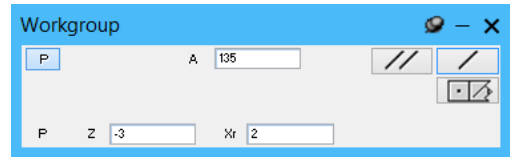
53. Click the Single Line button.

54. Zoom in to the area in the upper left corner as shown.
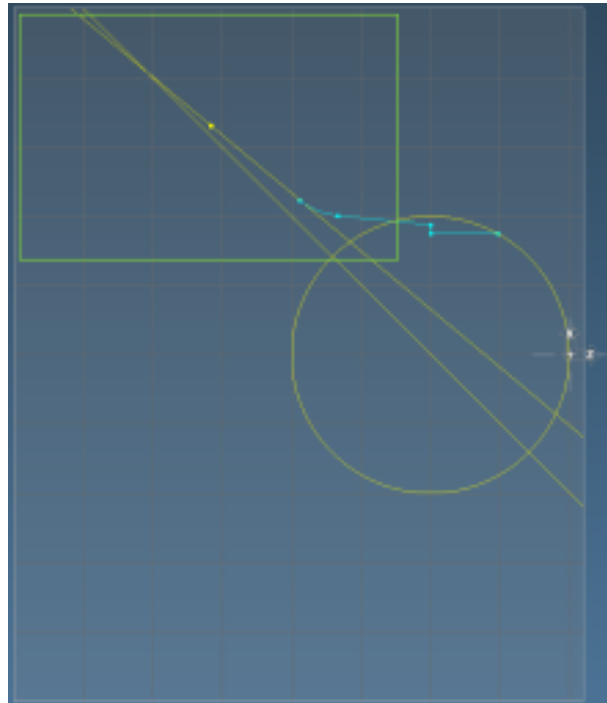
55. Open the Circle sub-palette.

56. Select the point and line shown.

57. Input the radius value shown.

Workgroup				⊘ − ✕
P	P6	L	L5	OO O
R	0.187			·▱
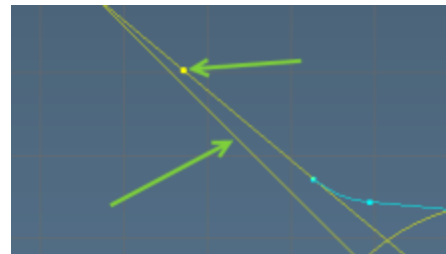
58. `Click` the Single Circle button.

59. Select the upper left circle as shown.

60. `Click` the OK button.　　　or　　　Press the [Enter] key.

One of the points created is a connector point. We will need to manually connect the other plain point.

61. Connect the point, line, and circle shown.
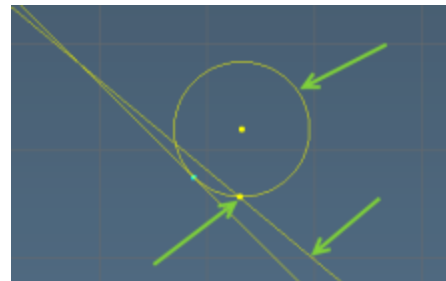
The point will become a connector point, and the line and circle should trim, and become blue.

62. Choose View > Unzoom
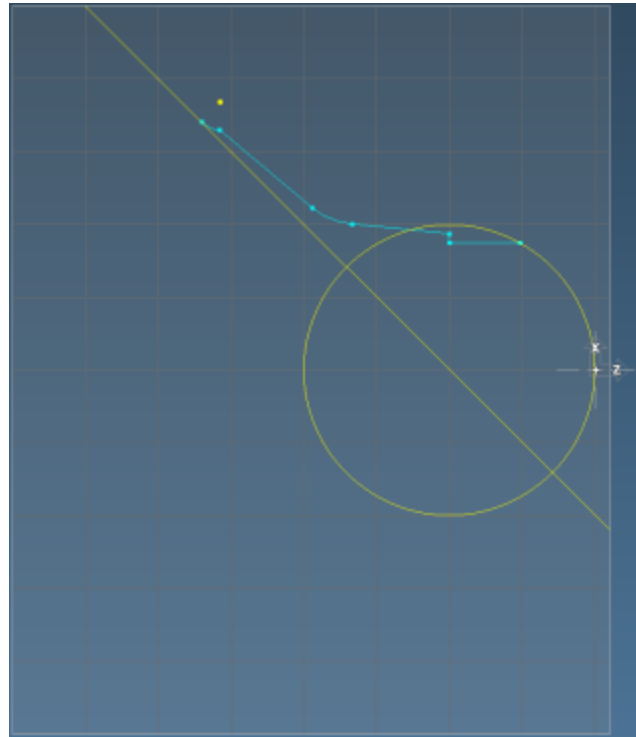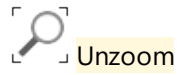
63. Open the Line sub-palette.

64. Select Axis line.

65. Select the vertical line mode and enter the value shown. (Z = -3).

66. Click the Single Line button.
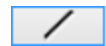
67. Connect the lines shown.
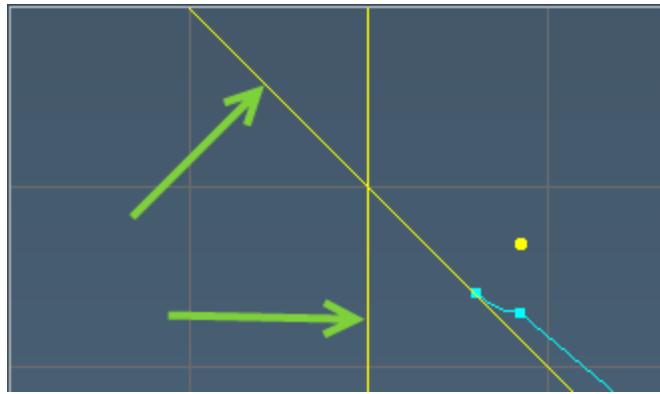
68. Open the Line sub-palette.

69. Select Axis line.

70. Select the horizontal line mode ___ and enter the value shown. (Xr = 0).

Workgroup			
I	Xr 0	//	/
___			⊡⊿
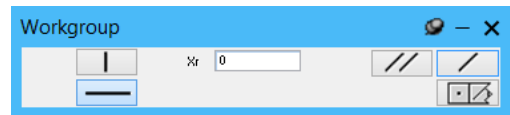
71. Click the Single Line button.
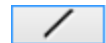
72. Connect the lines shown.
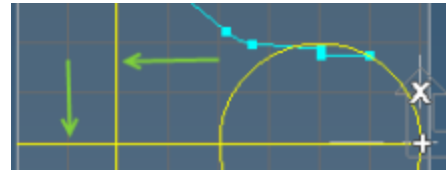
73. Connect the lines shown.
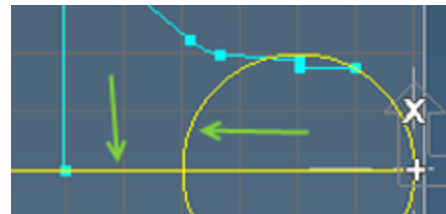
74. Select the right point and press

Enter

The geometry for the part is now complete and fully connected.
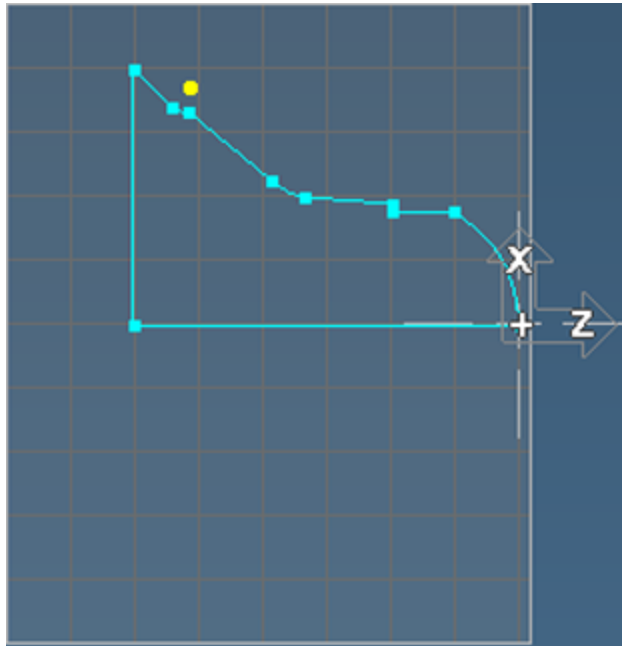
Exercise 5: Text Creation

In this example we will create text geometry. We will use the same text to show the different methods of alignment, spacing. and text flow.

Horizontal Text

Left Justified

1. Before starting the tutorial, you may need to adjust your view area to see the whole part (Home>Unzoom.)

2. Create a new 3 Axis Vertical Mill part with the following stock specifications:+X = 5, -X = -5, +Y = 4, -Y = -4, Z = 0, -Z = -1. Leave all other items at their default settings.

3. Open the Geometry palette.

4. Open the Shape sub-palette.

5. Open the Text Creation dialog.

6. Input the values shown into the Text Creation dialog.
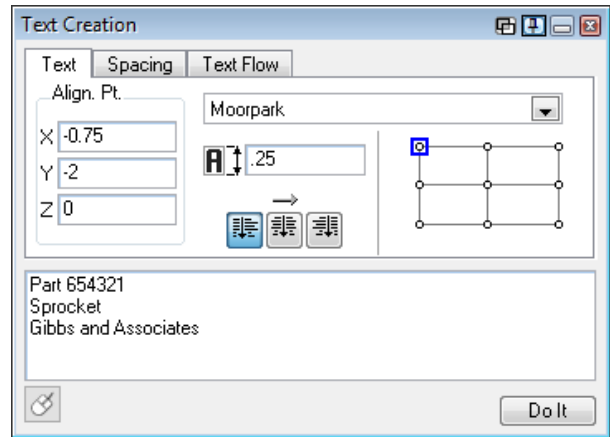
The font list contains all the available TrueType fonts. Moorpark is an installed TrueType font specifically designed for centerline engraving. If you have many fonts in the list, type the first letter of the name of the font to navigate to it faster.

7. Click the Do It button.

We have added lines at X-0.75 and Y-2.0 to show how the text aligns.
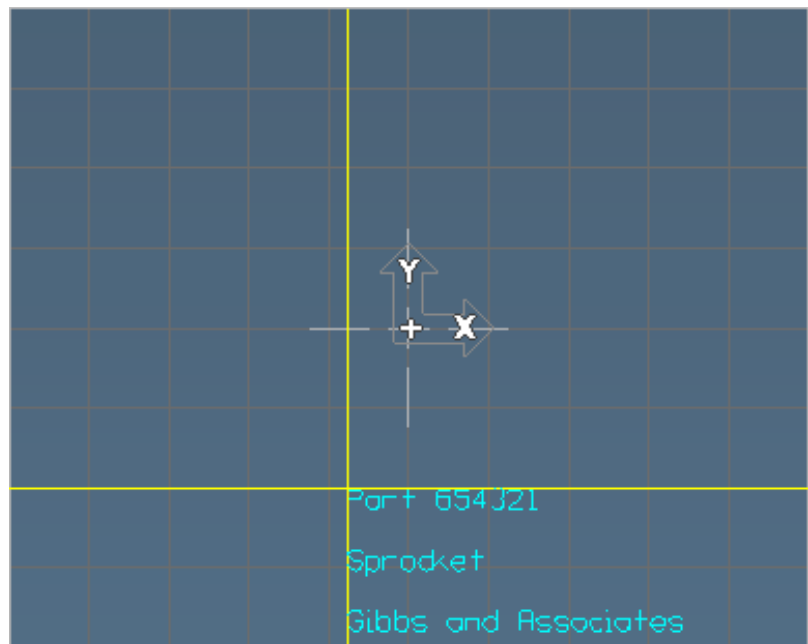
8. Choose View and ensure that under the visibility submenu, the points icon is unchecked (white background). This will make the text easier to read.

The lines on the screen illustrate the text alignment according to the align point and the position selected on the Alignment Diagram. The text starts in the upper left corner of the rectangular text area.

Centered

1. Create a new Workgroup

2. Click the Workgroup button, click
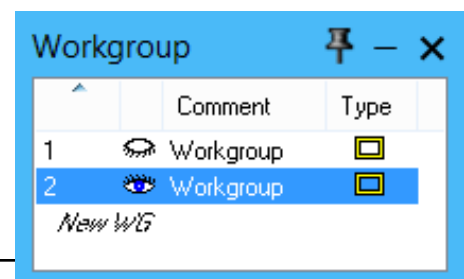
New WG. The eye icon 🕷 in the Workgroup menu indicates which Workgroup is visible.

After creating the new Workgroup, ensure that only Workgroup 2 is visible.

3. Open the Text Creation dialog.

A

4. Enter this information in the Spacing tab.

   (The text you typed previously still remains.)
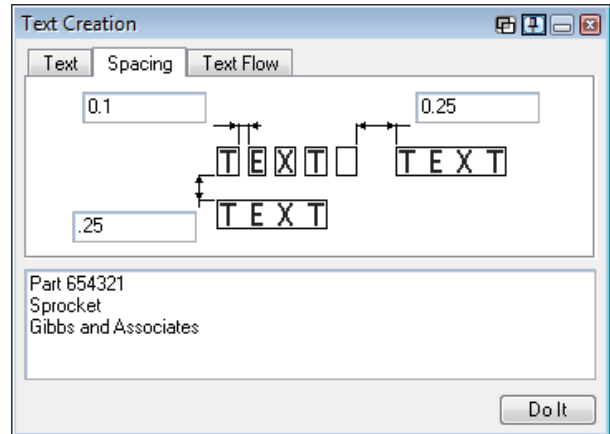
The values entered for spacing are added to the default spacing.

5. Enter this information in the Text tab.
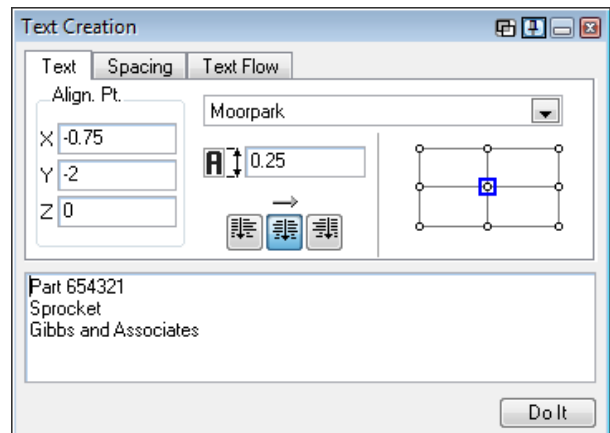
6. Click the Do It button.

Again, we have added lines at X = -0.75 and Y = -2 to visualize the text alignment.
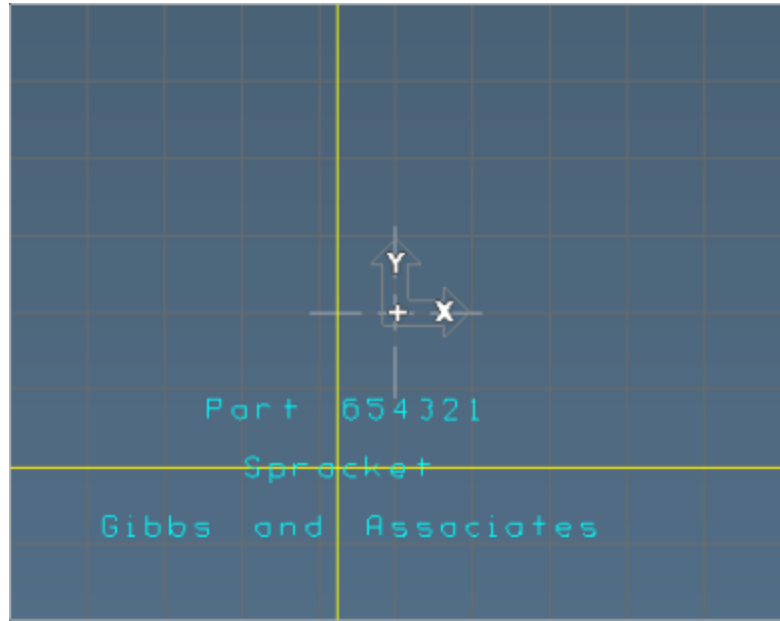
Vertical Text

1. Create a new Workgroup 3.

2. In WG:3 open the Text Creation dialog.

3. Enter this information in the Text Flow tab.
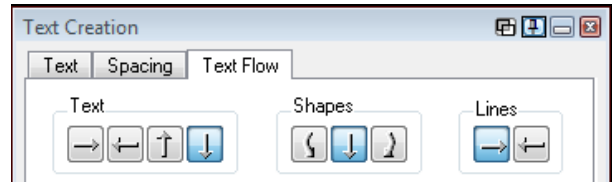
4. Enter this information in the Spacing tab.

    (The text input previously remains.)
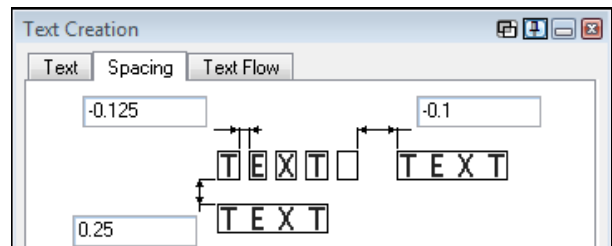
5. Enter this information in the Texttab.
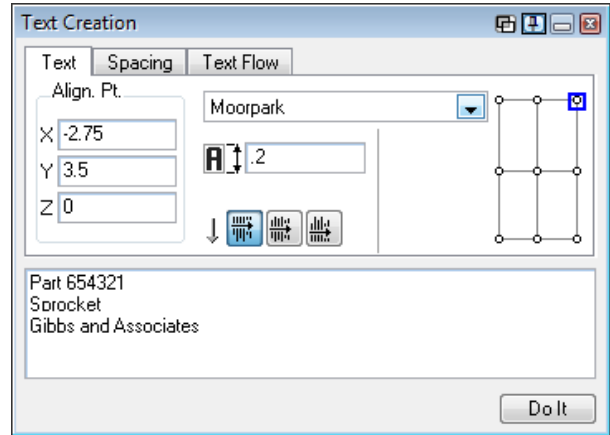
6. Click Do It.

The text is aligned in the upper right corner of the text area as specified by the alignment dialog. We have added a line at X=-2.75 and Y=3.5 to show the alignment.
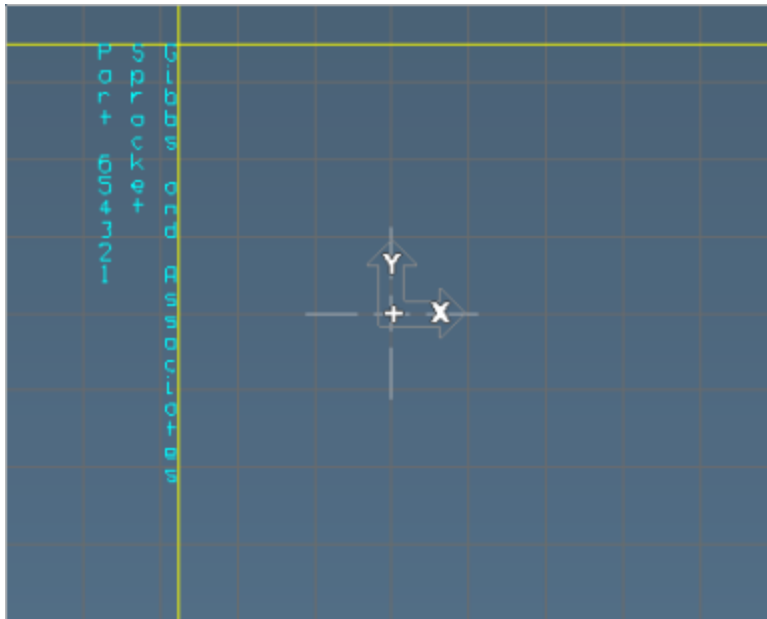
Text On An Arc

1. Create a WG4 and switch to it.

2. Enter this information in the Text Flow tab.

3. Enter this information in the Spacing tab.
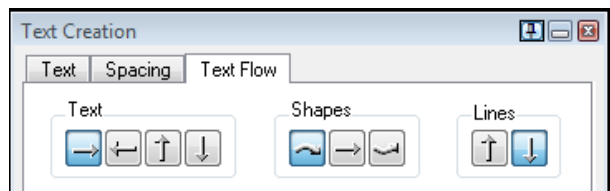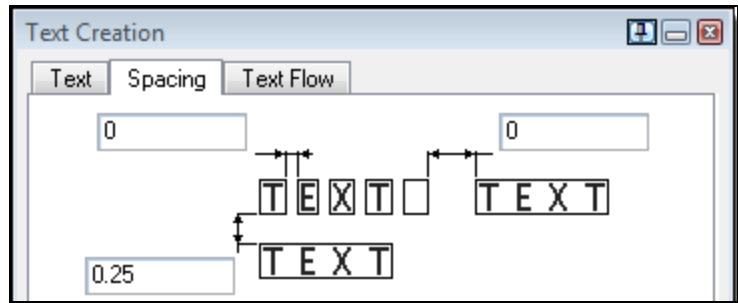
4. Enter this information in the Text tab.

(The text input previously remains.)
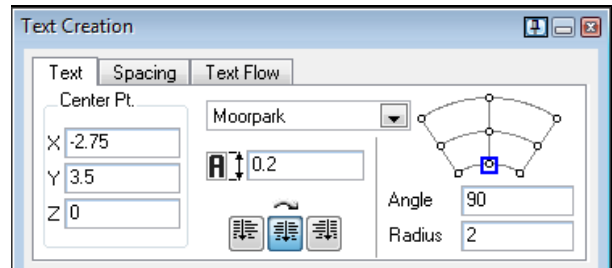
Note that the alignment diagram is different. The Radius specifies the size of the arc and the Angle indicates the position on the arc that the text will begin. Also, a center point for the arc must be entered (rather than an alignment point).

5. Create the text.
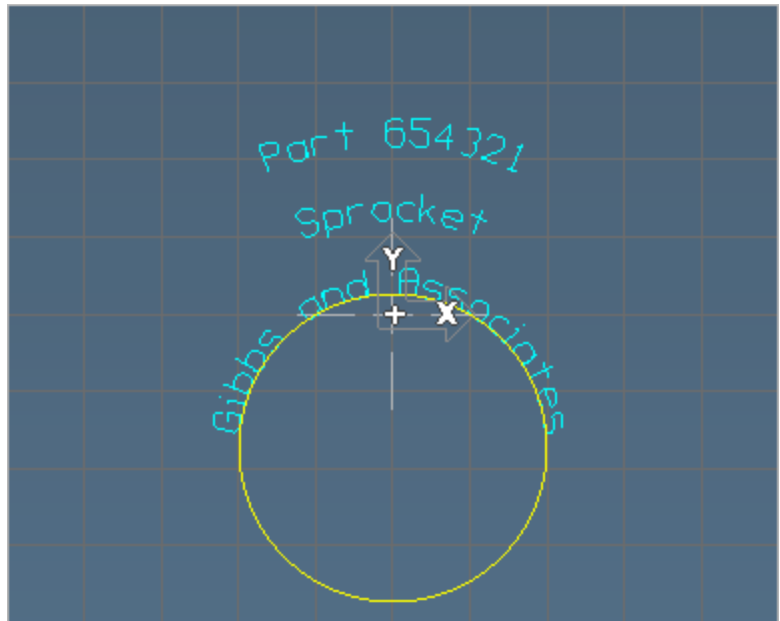
In order to show the alignment we have added a circle with centre point and radius as specified with the Angle, Radius and Center Pt. values entered in the dialog. The text is positioned along the outside of the arc and is positioned along the center of the y axis line as specified by the angle.

Exercise 6: Gear Creation

Creating a Gear

In this exercise we will create an involute spur gear. We will walk through the steps. The gear has the following known metric dimensions.

Outside Diameter: 11.666mm

Pitch diameter: 10mm

Inside Diameter: 7.916mm

Pressure Angle: 25°

Number of Teeth: 12

Typically, the Pitch Diameter and the Number of Teeth are known values. We will use these values to obtain the value for the Diametral Pitch.

Diametral Pitch = `# Teeth ÷ Pitch Diameter`

(Diametral Pitch: 12 ÷ 10 = 1.2)

1. Create a new 3 Axis Vertical Mill part with the following stock specifications:+X = 6, -X = -6, +Y = 6, -Y = -6, Z = 0, -Z = -1.

2. Open the Shape sub-palette.

3. Open the Gear dialog.

4. Set the Pressure Angle to 25°.

   Set the Orientation to External and the Gear Type to Gear.

5. Input the Diametral Pitch value, Full # Teeth amount.

Diametral Pitch	1.2
Circular Pitch	2.618
Pitch Diameter	10
Full # Teeth	12

The other Pitch settings are automatically calculated.

6. Input the Involute Curve settings as shown.

Involute Curve		
Sample Points	60	
Tolerance	0.001	±

7. **Click** the Calculate
    button.

    [Calculate]

Addendum	0.833
Dedendum	1.042
Top Fillet	0
Root Fillet	0.313

[Calculate]

The formulas for calculating the Addendum and Dedendum are as follows:

Addendum = (Major Diameter - Pitch Diameter) ÷ 2

Dedendum = (Pitch Diameter - Minor Diameter) ÷ 2

8. **Click** Do It to generate the shape of a single tooth.

    Do not deselect the tooth geometry - it is drawn in red as shown below.
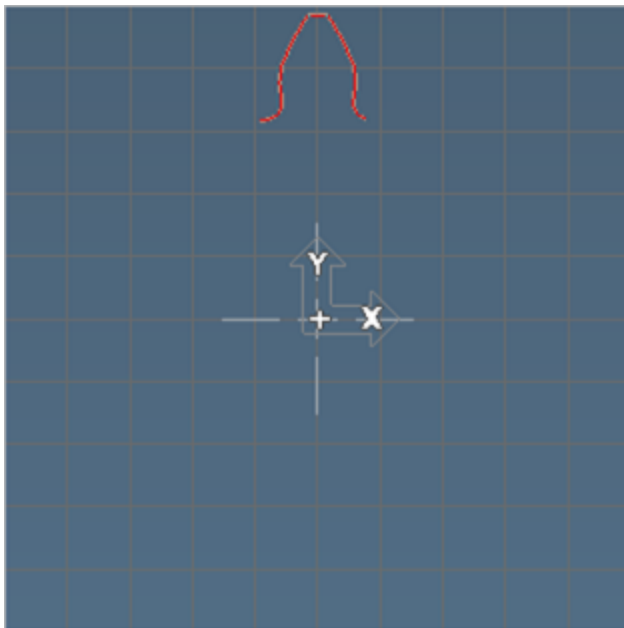
9. From the Modify menu select Dup + 2D Rotate.

The 2D rotate calculates the number of times to repeat the geometry to create the complete gear. Additionally, when you create the duplicated geometry it is automatically connected.
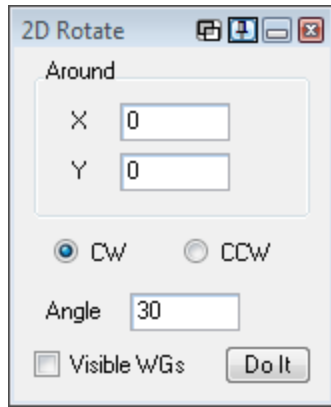
10. **Click** the Do It button.

Your screen will now look like this:
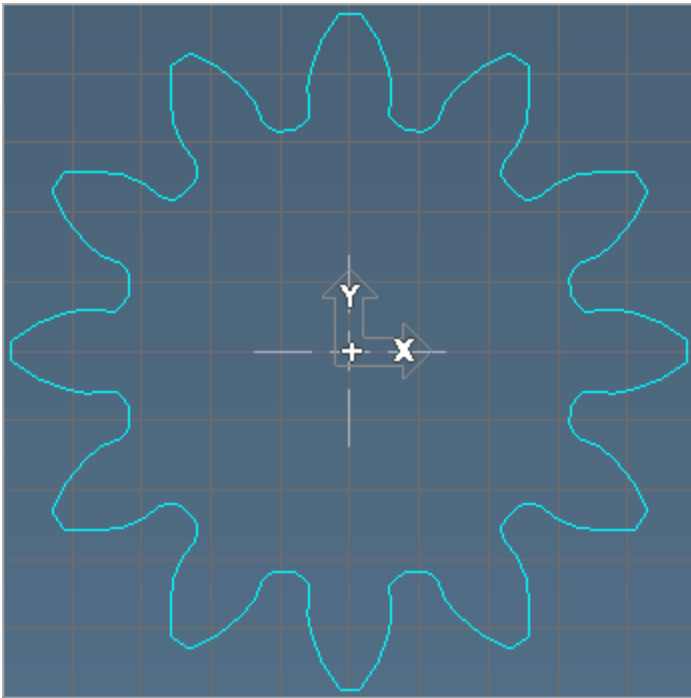
Exercise 7: Overlapping Features

A good understanding of connections and connector points is necessary to become proficient. While connecting a shape is a simple process, this exercise details how to create a connected shape when individual features overlap one another.

Overlapping Connections

Circles

This exercise provides minimal direction on how to build the part. Be sure to check your progress carefully against the illustrations at each step. The part is a simple shape. It consists of six circles, and no lines.
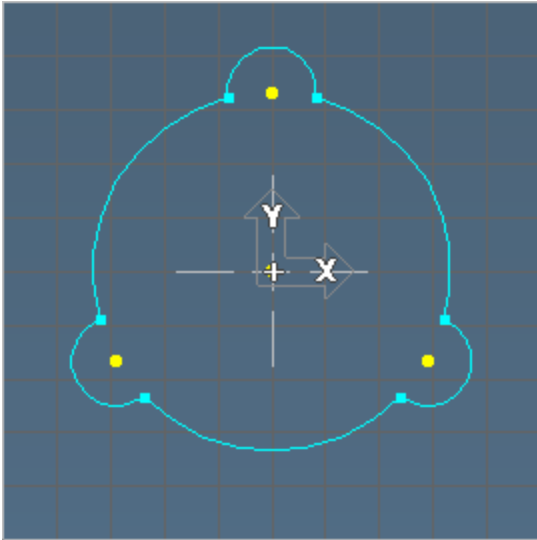
1. Create a 3 Axis Vertical Mill Vertical Mill part with the following stock specifications: +X = 3, -X = -3, +Y = 3, -Y = -3, Z = 0, -Z = -1. Dimensions should be in inch.

2. Create P1 (point 1) at X0Y0.

   Use the Bolt Circle button (in the point sub-menu) to create the other three points on a two-inch radius.
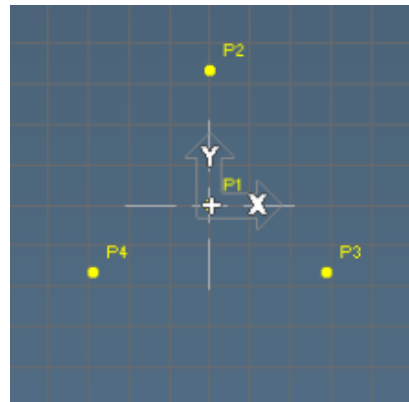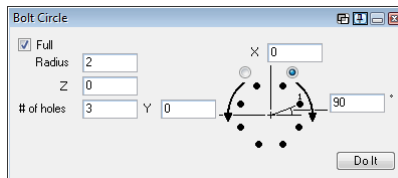
3. Create three 0.5" circles around P2, P3, and P4.
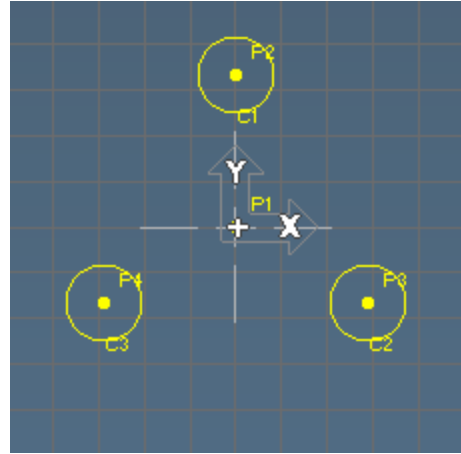
4. Create one 2.0" circle around P1.
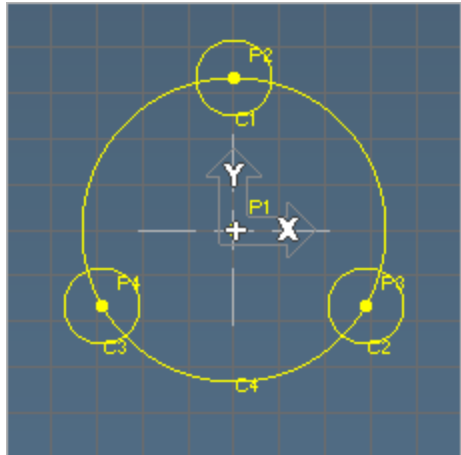
There are now two ways to proceed:

For the sake of completeness, we will first show you the long method, where you add features and connections one by one.

Then, we will show the short method where you can combine multiple shapes in one step.

Sequentially Add Features and Connections

C4 is one of three 2.0" circles needed. Putting two more circles on top of C4 makes it difficult to tell which one you are clicking on. The best approach for all shapes is to work around the shape in a specific direction (clockwise or counter-clockwise), adding features and connections as you go. In this example, you can use C4 as any one of the three circle segments (or arcs). The key idea is to fully connect C4 before adding another 2.0 circle to the workgroup. Starting with C1, we'll proceed clockwise around the shape.

Let's connect C4 between C1 and C2. Select C1 and C4 circles and use the Point button or the Connect/Disconnect button (or press the number 8) to find the intersections between C4 and C1.

1. Select the point shown.

Select desired feature(s).   OK
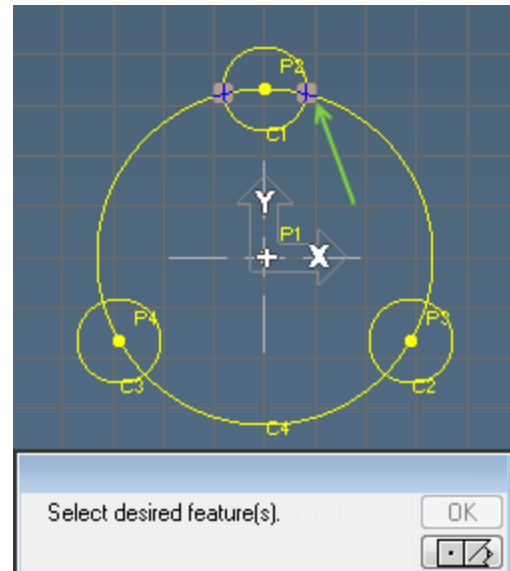
P5 is drawn at the selected intersection of C1 and C4.

2. Now connect C4 to C2. Use the Point button again.
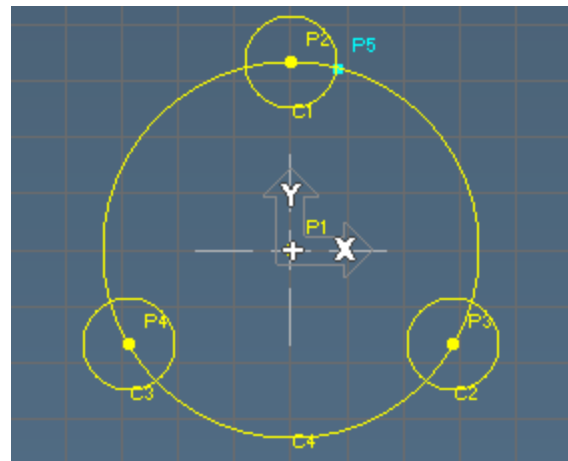
3. Connect C4 to C2 by selecting the point shown.
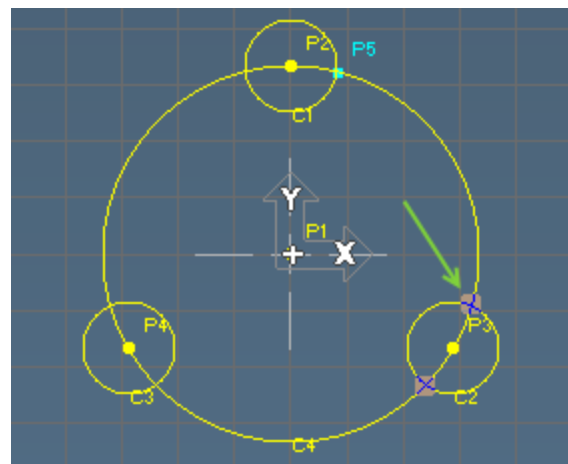
P6 is drawn at the selected intersection of C2 and C4, completing the connection for this feature.
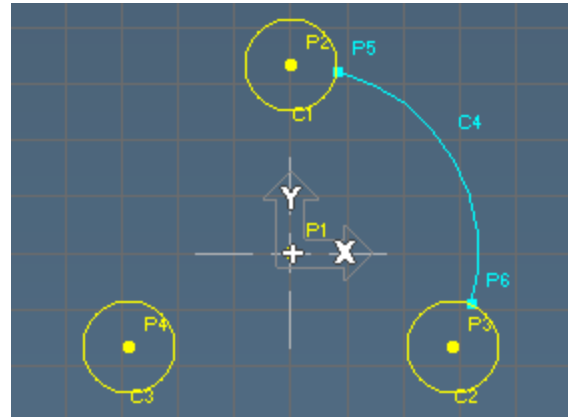
4. Add the next 2.0" circle around P1 or duplicate C4.
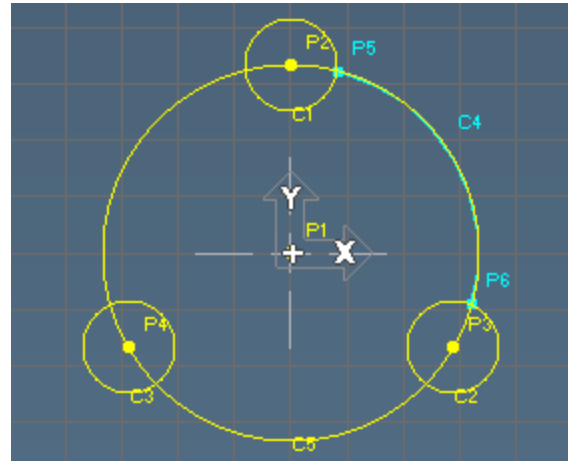
C5 now overlaps the arc C4. Selecting C5 where it overlaps C4 will only select C4. The system will always select the lowest-numbered circle in the overlapping area. The selection order of geometry that overlap are as follows: points, lines, then circles. If you click C5 where it does not overlap C4, C5 will be selected and highlighted as a full circle.

5. Connect C5 to C2 using the point shown.

P7 is drawn at the selected intersection of C2 and C5. It is the second connection for C2, therefore completing this feature. Use ⟳ Reverse Arc (`Ctrl+T`) if necessary.

6. Now connect C5 to C3 using the point shown.
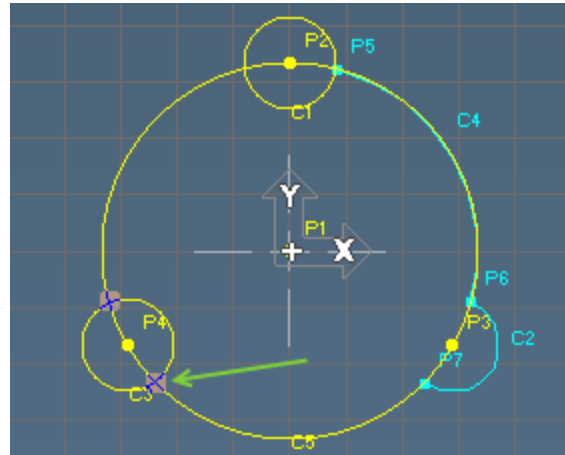
P8 is drawn at the selected intersection of C3 and C5.
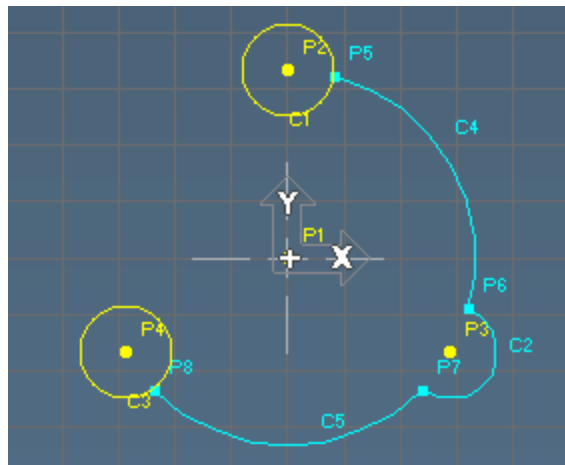
7. Create the last 2" circle around P1.
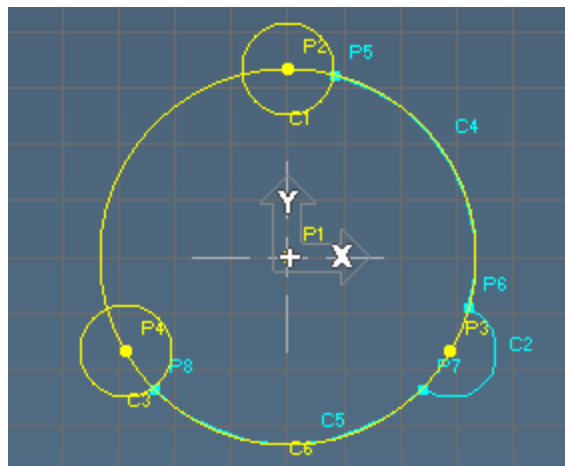
8. Connect C6 to C3 using the point shown.
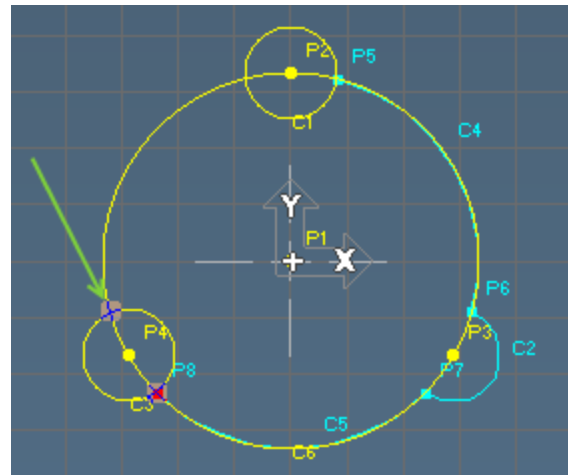
P9 is drawn at the selected intersection of C6 and C3.

9. Connect C6 to C1.
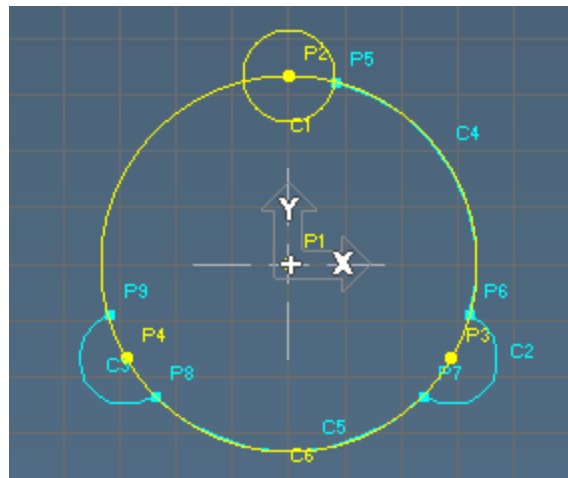
10. Connect C6 to C1 using the point shown.
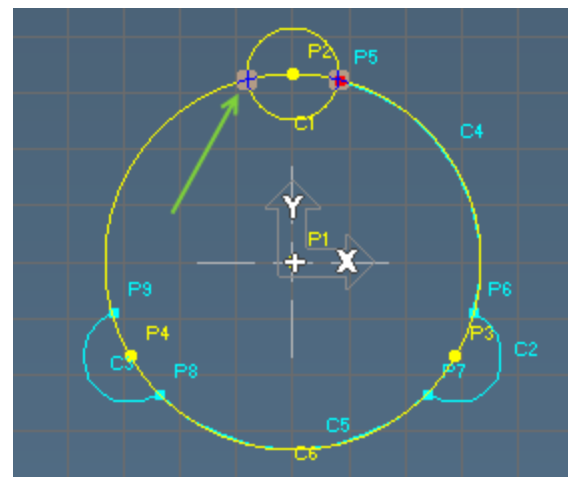
P10 is drawn at the selected intersection of C6 and C1, creating fully-connected geometry. The shape

A23

is now complete. Choose View Labels for a clearer view.
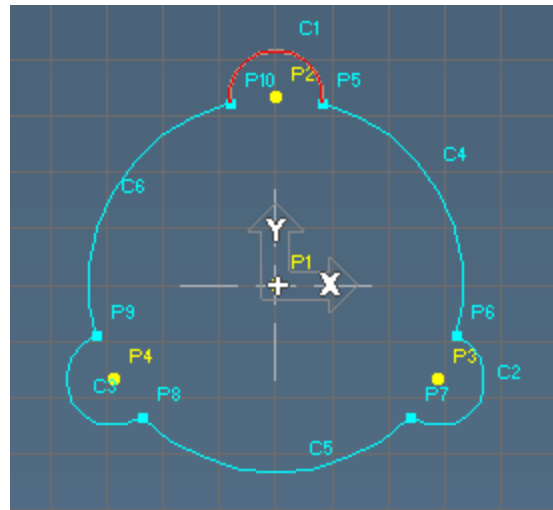
Short method (optional): Combine Multiple Shapes.

If you are comfortable with the concepts of connecting and disconnecting, you can optionally use a different method that can be much quicker.

1. Undo steps 5 through 14 (for example, press `Ctrl+Z` about 12 times), until you have restored the four unmodified circles.

2. Select a circle, and then `Ctrl+select` each of the other three circles.
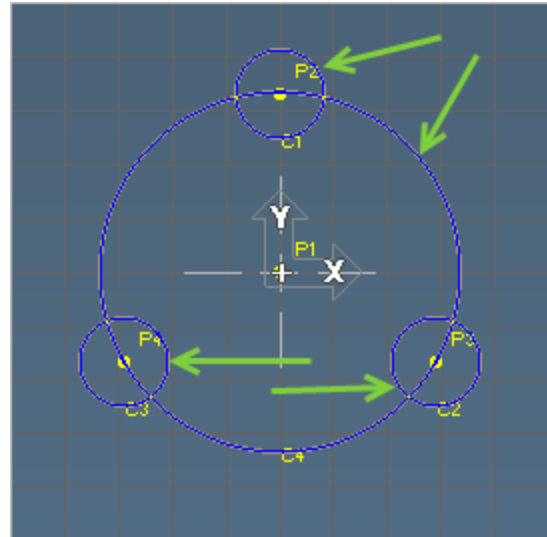
3. In the Shape $\overline{OA}$ sub-palette, select

   Combine Shapes and then

   select the Shape Union button
   .
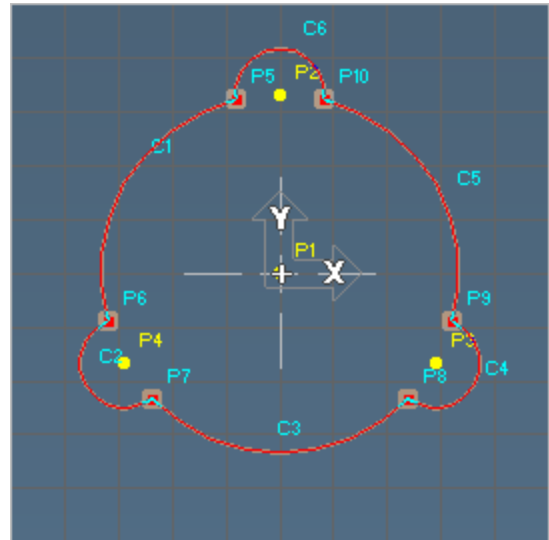
Remember that the directions and methods used in the system to create a shape have no effect on how the shape is machined. The cut can begin anywhere, stop anywhere, and cut in any direction. The tool can cut outside, inside, or on the shape. This will be covered in more detail in the Lathe and Mill manuals.

GEOMETRY EXPERT EXERCISES

The exercises contained in this chapter are designed to introduce you to the basic concepts of geometry creation using Geometry Expert. For part dimensions for each of these exercises, see "Part Prints" on page 158 . Lathe examples can be completed with Mill, and vice-versa.

Exercise 1: Shaft

Horizontal and Vertical Lines

In the first exercise, we will create a simple shaft containing only horizontal and vertical lines. Because of the feature type defaults and automatic angle toggling, the creation of a simple shaft can be done very quickly with Geometry Expert. For the part print for this exercise, see "Shaft" on page 163.

1. Open the Documents dialog.

2. Create a Horizontal Lathe - Generic shank MDD part with the following dimensions.
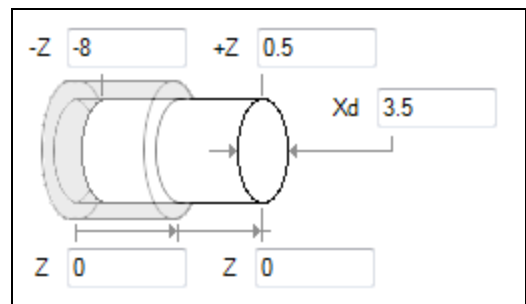
3. Select inch and Diameter for the dimension style.

4. Rotate your view to display the work area (Home - Un-Zoom)

5. Open the Geometry palette.

6. Open the Geometry Expert.

   The first row (current row) is highlighted in black because the row needs more information in order to define the feature.

   Notice that when Geometry Expert opens, the feature type selected is a line, and the angle value is 90°.

   The feature we will start with is the vertical line at Z0.

7. Input the values shown.

Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
✓			90		0	

This first line is fully defined by a LP H and an Angle (90 or 270). Enter.
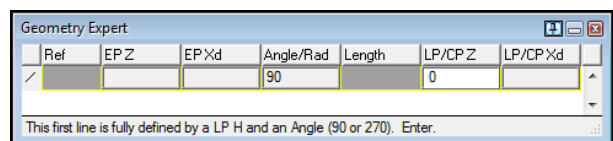
When defining either horizontal or vertical lines, it is only necessary to enter a half point (a value for one axis) for the line point.

> When only one coordinate value is entered, it is referred to as a half point. In certain cases only a horizontal (H) or vertical (V) coordinate value is needed in order to define the feature.

Once enough data to create a feature is entered the row is highlighted in yellow.

8. Press Enter.

We have switched labels on (View>Labels checked) so you can see what happens.
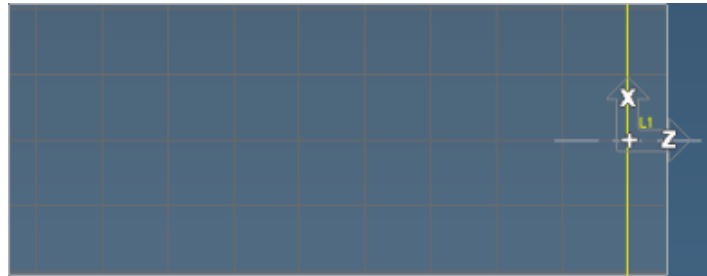
This will create the line and a new row, highlighting the most logical cell to enter data for the feature type selected.

The second row is a line feature type and the angle is switched automatically to a horizontal line. (We have switched labels on (View>Labels checked) so you can see what happens.)

9. Input the values shown.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
/	L1			90		0	
/				180			1.5

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

Since the purpose of Geometry Expert is to create a single shape, connections are assumed. Therefore, when features do not have a point of connection an error message will be displayed.
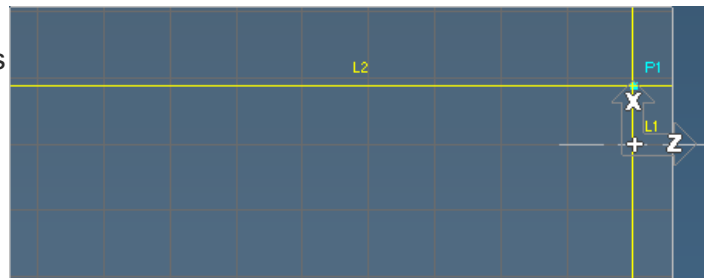
10. Input the values shown.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
/	L1			90		0	
/	L2			180			1.5
/				90		-3	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

Notice that the line angle has again switched to a vertical angle.
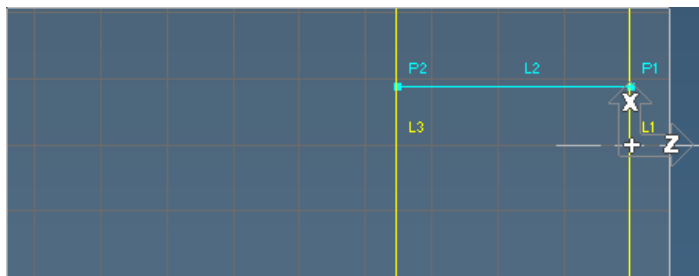
11. Input the values shown and press Enter.

At this stage the display becomes a little crowded so Labels have been switched off.
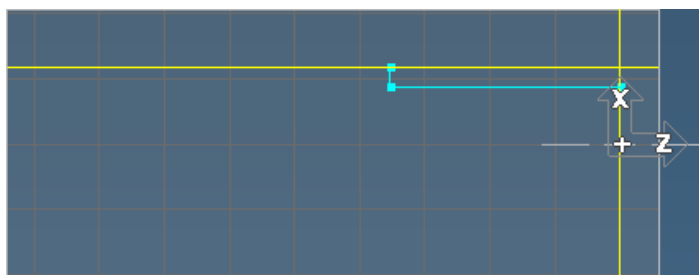
12. Input the values shown and press Enter.

Once a feature has been drawn on the screen, a reference number is assigned and entered in the Ref column.
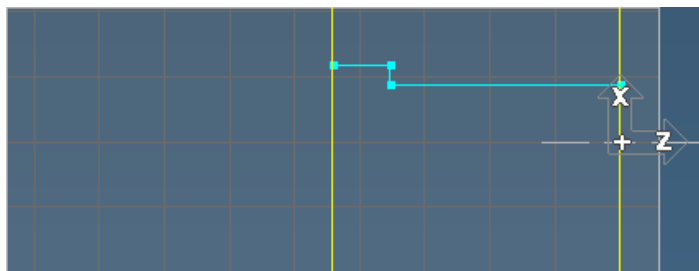
13. Input the values shown and press Enter.
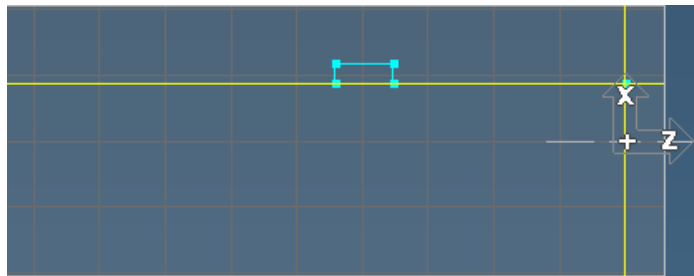
14. Input the values shown and press `Enter`.

Geometry Expert						
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L5			90		-3.75	
L6			180			1.5
			90		-5	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

15. Input the values shown and press `Enter`.

Geometry Expert						
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L6			180			1.5
L7			90		-5	
			180			2

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

16. Input the values shown and press `Enter`.

Geometry Expert						
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L7			90		-5	
L8			180			2
			90		-6	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

17. Input the values shown and press Enter.
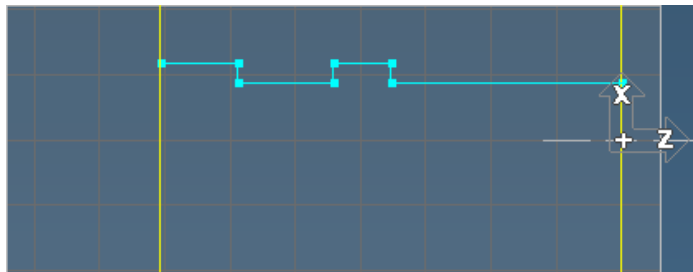
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L8			180			2
L9			90		-6	
			180			3.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

18. Input the values shown and press Enter.
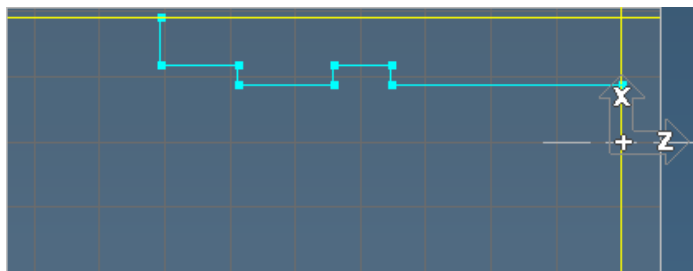
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L9			90		-6	
L10			180			3.25
			90		-7.5	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

19. Input the values shown and press Enter.
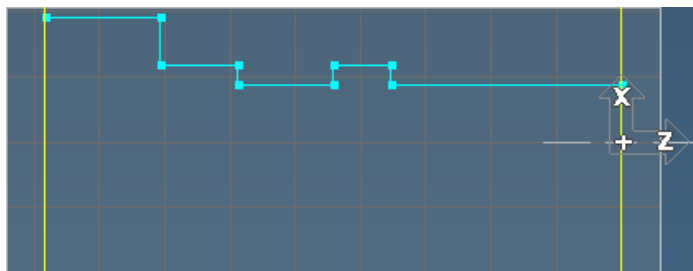
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L10			180			3.25
L11			90		-7.5	
			180			3

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

20. Input the values shown and press
    Enter.

Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
L11			90		-7.5	
L12			180			3
			90		-7	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

21. Input the values shown and press
    Enter.

Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
L12			180			3
L13			90		-7	
			180			2.5

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

22. Input the values shown and press
    Enter.

Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
L13			90		-7	
L14			180			2.5
			90		-7.25	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

23. Input the values shown and press Enter.
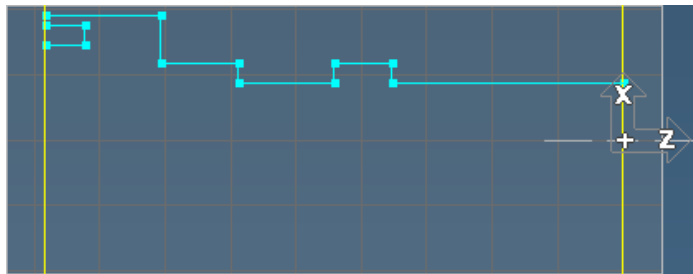
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L14			180			2.5
L15			90		-7.25	
			180			1.5

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

24. Input the values shown and press Enter.
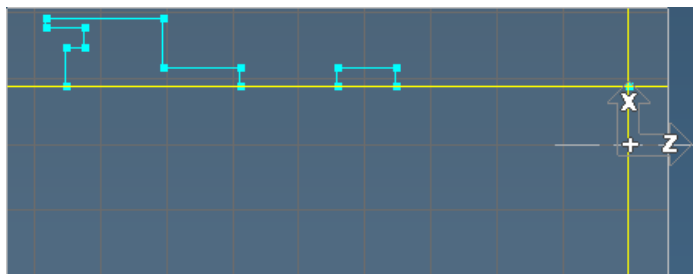
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L15			90		-7.25	
L16			180			1.5
			90		-6	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

25. Input the values shown and press Enter.
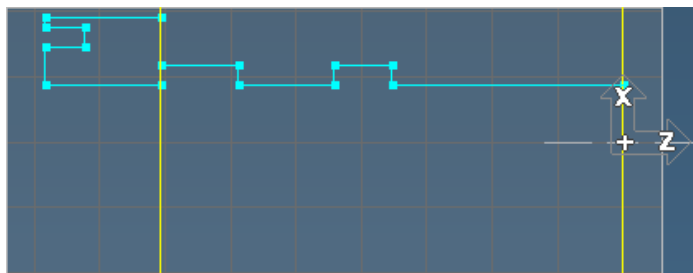
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L16			180			1.5
L17			90		-6	
			180			0

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

26. Input the values shown and press
    `Enter`.

Geometry Expert							
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
L17			90		-6		
L18			180		0		
			90		-4		

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

27. Input the values shown and press
    `Enter`.

Geometry Expert							
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
L18			180			0	
L19			90		-4		
			180			1	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter
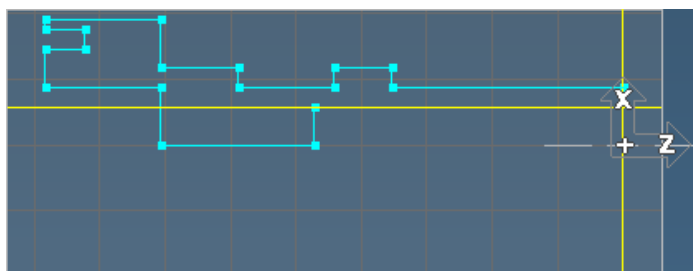
28. Select the Close Shape feature
    type (`Alt+6` or click the line
    icon in the far left column of the
    geometry expert dialog to
    display the menu below.)

The Close feature type connects the first and last feature at their point of intersection.

Geometry Expert							
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
L19			90		-4		
L20			180			1	
+							

The shaft is complete and fully connected
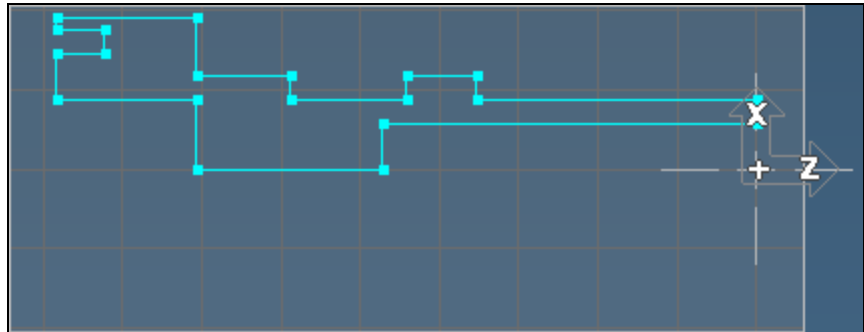
Save this part.

Exercise 2: Chassis

In this exercise, we will introduce adding different feature types such as fillets and chamfers. Refer to Part Print #2: Chassis for this exercise.

Chamfers and Fillets

1. Open the Documents dialog.

2. Create a 3 Axis Vertical Mill part with the following workspace stock dimensions.

+X 7.5	+Z 0
-X -0.5	
+Y 4.5	-Z -1
-Y -0.5	
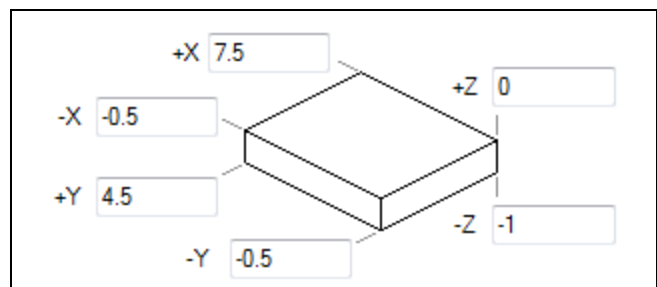
3. Select inch for the measurement type.

4. Open the Geometry palette.

5. Open the Geometry Expert.

6. Input the value shown and press Enter.
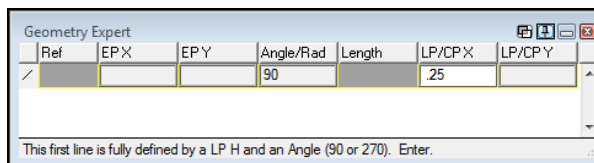
7. Change the view to Home, no zoom.
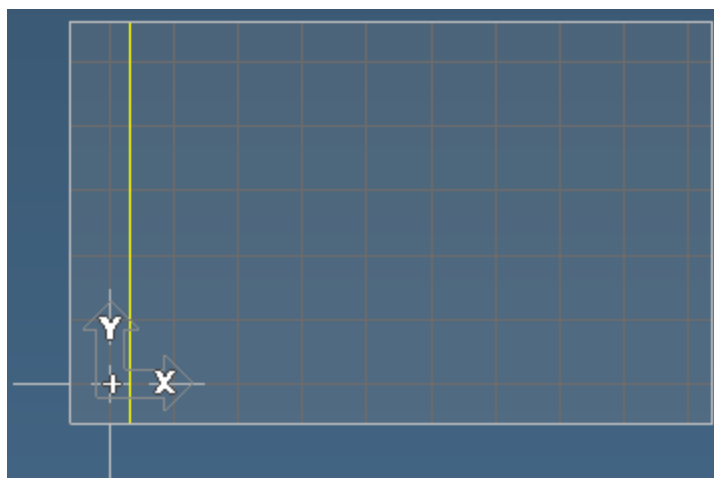
8. Change the feature type to a Fillet (3rd button) as shown.
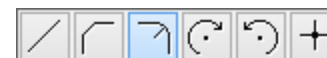
The feature type can be selected using keyboard shortcuts. For example pressing Alt+3 is the same as selecting Fillet feature type.

9. Input the value shown.
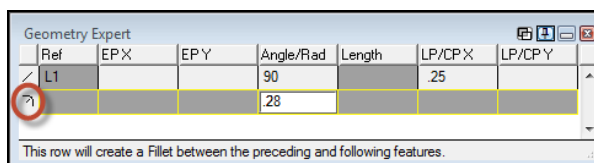
Only a radius value is required when creating fillets. Notice the prompt. "This row will create a Fillet between the preceding and following features." Fillets and chamfers are entered between the two features they connect with. Fillets and chamfers can only be created between two intersecting features at their point of intersection.

10. Press the Enter key.

Since there isn't enough information to define the complete Fillet nothing is drawn. When the next feature is completely connected and trimmed the Fillet will become complete and be

created. Fillets, Chamfers and tangent features are completely dependent on the features that they exist between and can only be seen once the features they are created from are completely defined.

11. Input the 2 values shown and press Enter.

Geometry Expert						
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L1			90			.25
			.28			
			180			3.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

12. The two lines drawn are connected.
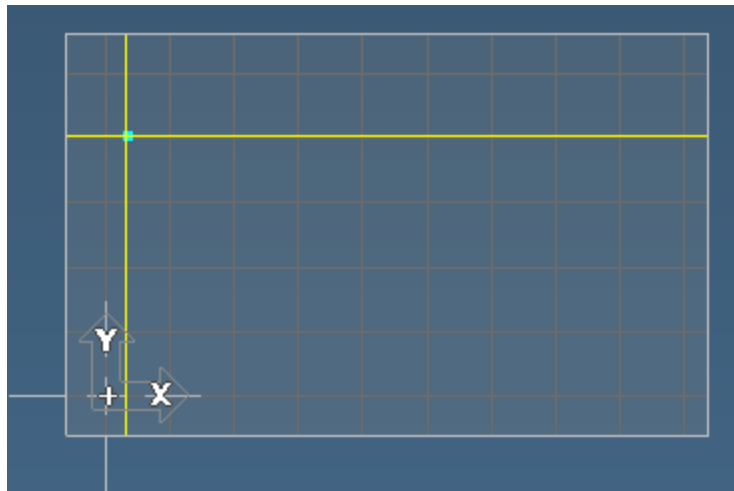
13. Change the feature type to a Chamfer (Alt+2).

Input the value shown.

Geometry Expert						
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
			.28			
L2			180			3.25
				0.1		

This row will create a Chamfer between the preceding and following features.

Chamfers created in Geometry Expert are dimensioned according to the Side length. To create chamfers that are dimensioned differently, use the Chamfer palette in the free-form CAD tools.

14. Press the Enter key.

Again, the chamfer will be drawn when the features it exists between are fully connected.

15. Input the value shown and press Enter.

Geometry Expert						
Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L2			180			3.25
				0.1		
			90		0.75	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

Your screen will look like this
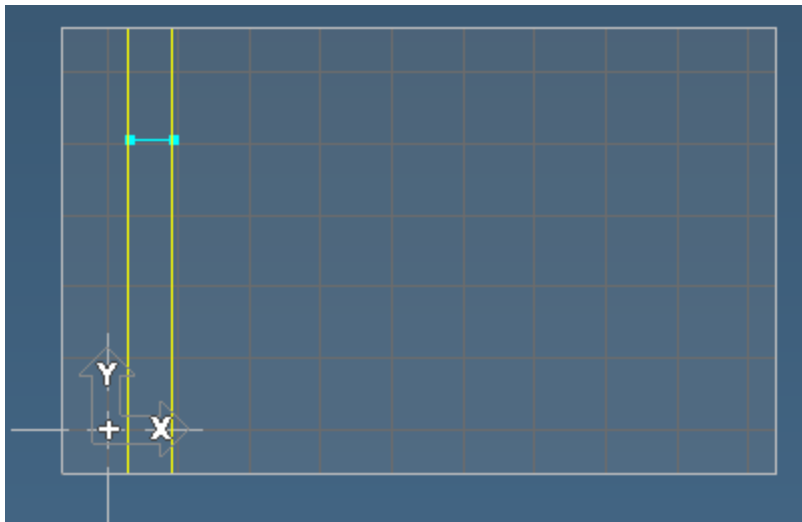
16. Select the Fillet feature type (`Alt+3`).

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
⌒					0.1		
╱	L3			90		0.75	
⌐				0.28			

This row will create a Fillet between the preceding and following features.

The radius value for the fillet will default to the previous radius entered for a fillet. Therefore, you don't need to enter a radius value.

17. Press the `Enter` key.

18. Input the value shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
╱	L3			90		0.75	
⌐				0.28			
╱				180			3.75

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

The chamfer is now drawn because the preceding and following lines are fully connected.
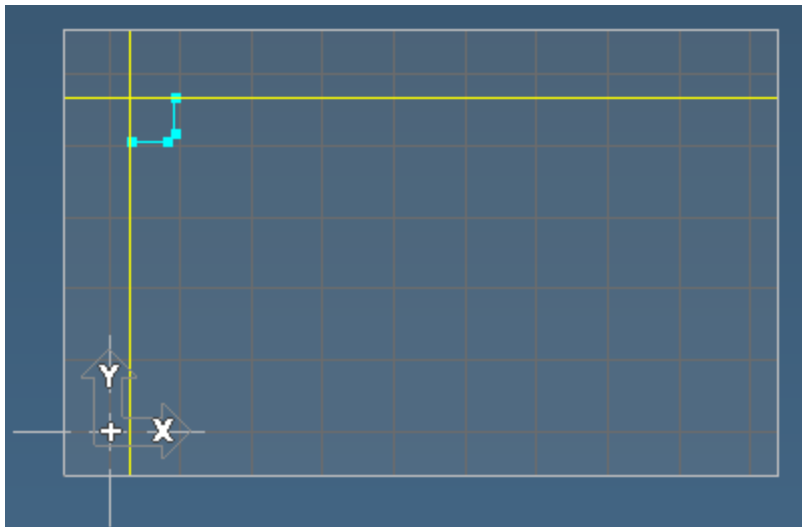
19. Select the Fillet feature type (Alt+2) and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
			0.28			
L4			180			3.75
			0.28			

This row will create a Fillet between the preceding and following features.

20. Input the value shown and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L4			180			3.75
			0.28			
			90		2.25	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

The fillet is now drawn because the preceding and following features now have two connector points.
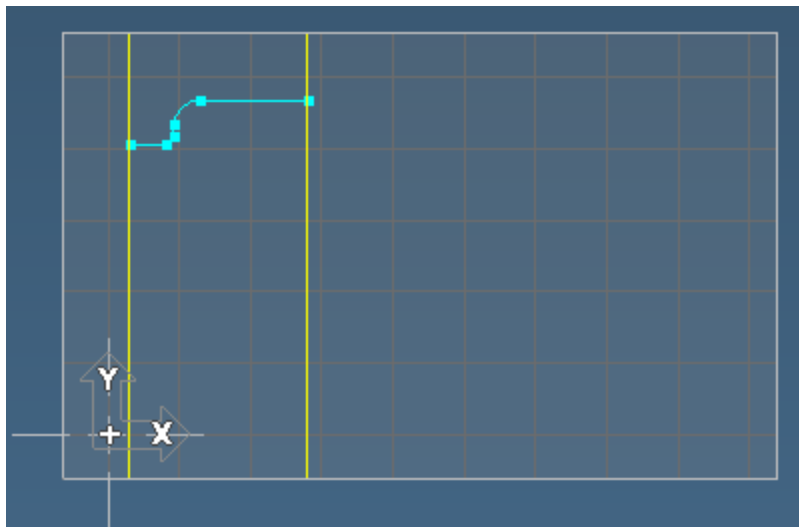
21. Select the Chamfer feature type (`Alt+2`) and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
↗			0.28			
L6			90		2.25	
⌒				0.1		

This row will create a Chamfer between the preceding and following features.

The length value for the chamfer also defaults to the previous value entered.

22. Input the value shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L6			90		2.25	
⌒				0.1		
			180			3.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

While we could continue making fillets and chamfers in this manner, from this point on, we will only create the horizontal and vertical lines. Then, we will use the free-form CAD tools to create the remaining fillets and chamfers.

23. Input the value shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
⌒				0.1		
L7			180			3.25
			90		2.75	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter
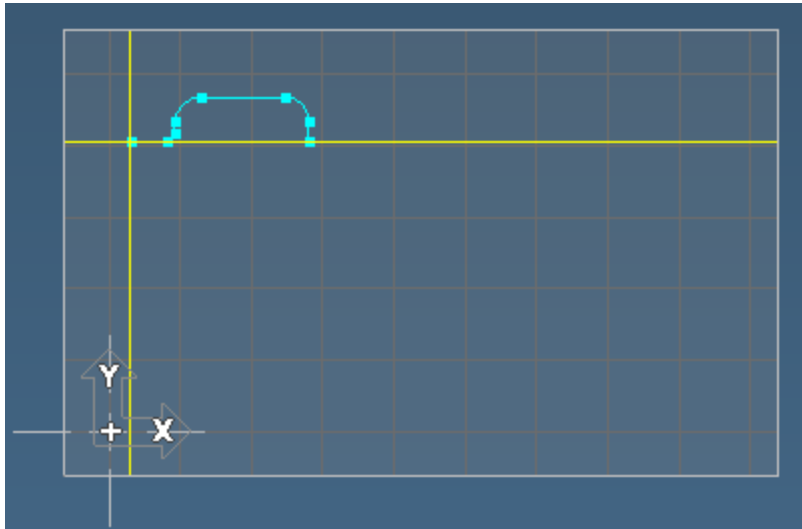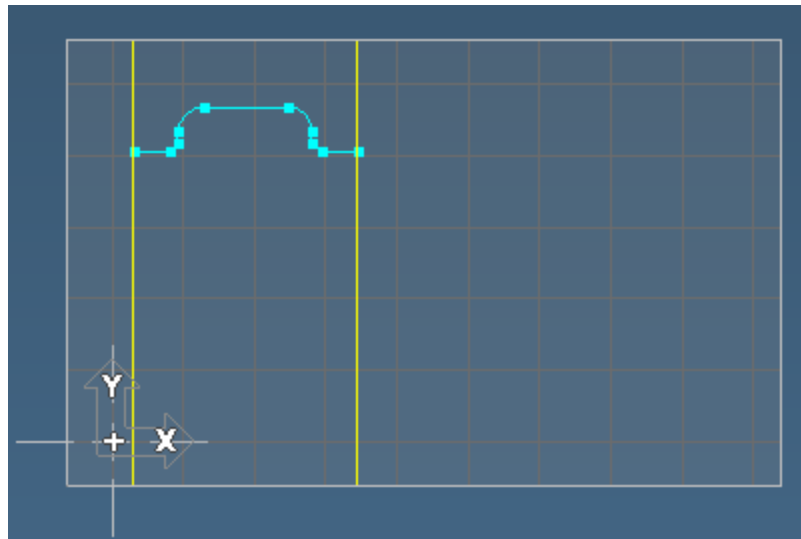
24. Input the value shown and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L7			180			3.25
L8			90		2.75	
			180			3.75

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

25. Input the value shown and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L8			90		2.75	
L10			180			3.75
			90		4.25	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

26. Input the value shown and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L10			180			3.75
L11			90		4.25	
			180			3.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

27. Input the value shown and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L11			90		4.25	
L12			180			3.25
			90		4.75	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter
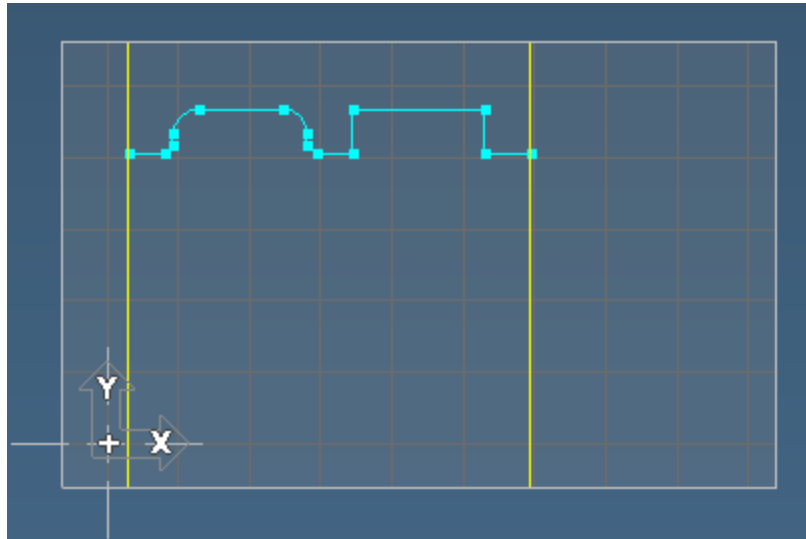
28. Input the value shown and press
    Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L12			180			3.25
L13			90		4.75	
			180			3.75

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

29. Input the value shown and press
    Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L13			90		4.75	
L14			180			3.75
			90		6.25	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter
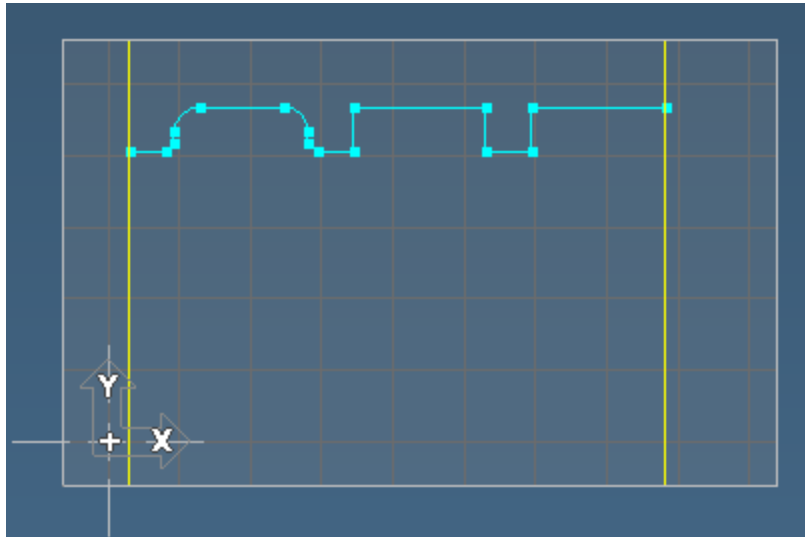
30. Input the value shown and press
    Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
✓ L14			180			3.75
✓ L15			90		6.25	
✓			180			3.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

31. Input the value shown and press
    Enter.

Ref	EP Y	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
✓ L15			90		6.25	
✓ L16			180			3.25
✓			90		6.75	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

32. Input the value shown and press
    Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L16			180			3.25
L17			90		6.75	
			180			0.75

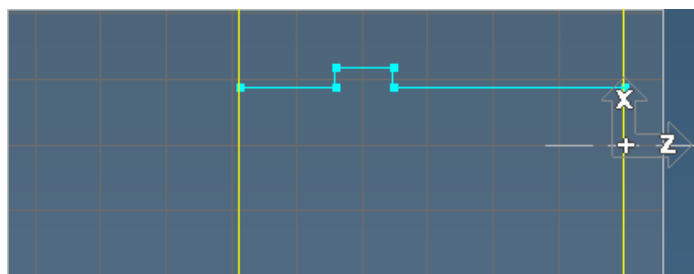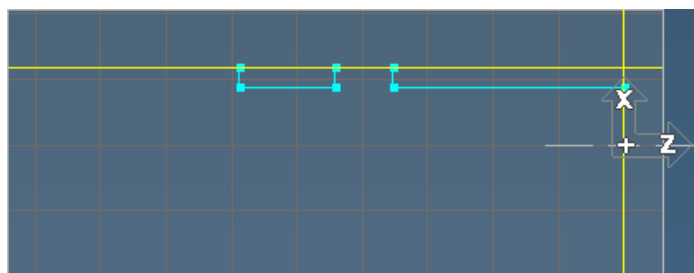Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

33. Input the value shown and press
    Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L17			90		6.75	
L18			180			0.75
			90		6.25	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

34. Input the value shown and press
Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L18			180			0.75
L19			90		6.25	
			180			0.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

35. Input the value shown and press
Enter.

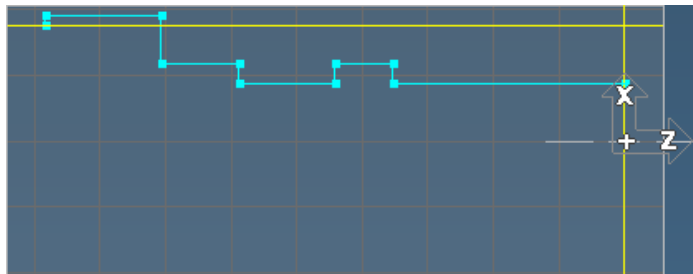| Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L19			90		6.25	
L20			180			0.25
L21			90		4.75	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

36. Input the value shown and press
    Enter.

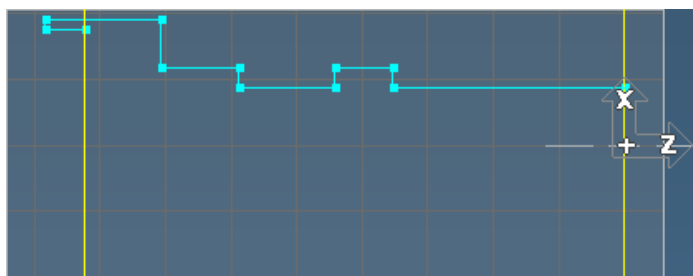| Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L20			180			0.25
L23			90		4.75	
			180			0.75

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

37. Input the value shown and press
    Enter.

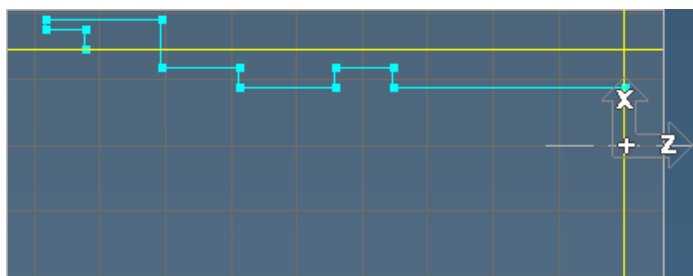| Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L23			90		4.75	
L22			180			0.75
			90		4.25	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter
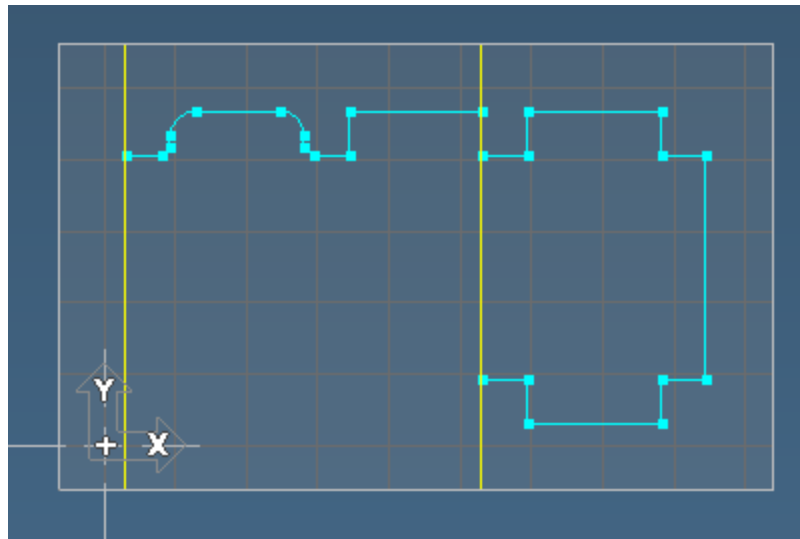
38. Input the value shown and press
    Enter.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
✎	L22			180			0.75
✎	L24			90		4.25	
✎				180			0.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter
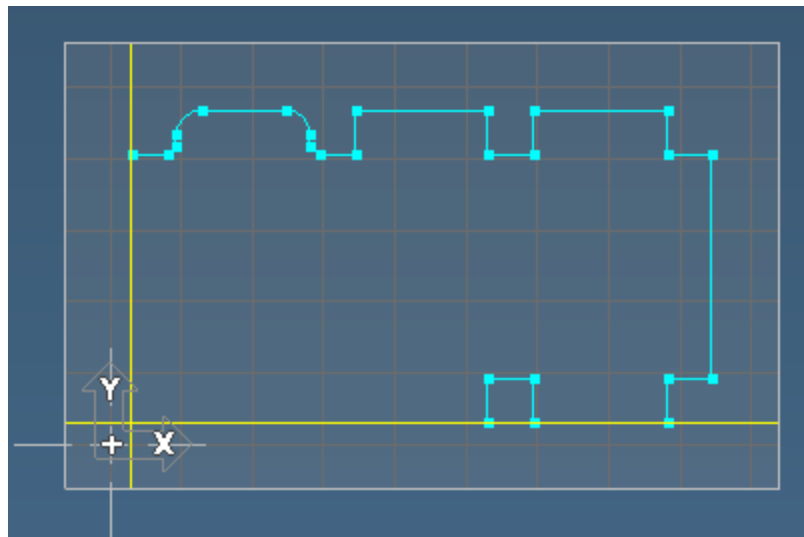
39. Input the value shown and press
    Enter.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
✎	L24			90		4.25	
✎	L25			180			0.25
✎				90		2.75	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

40. Input the value shown and press Enter.

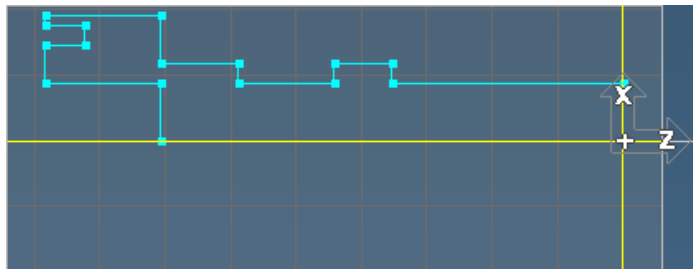|	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✓	L25			180			0.25	▲
✓	L26			90		2.75		▤
✓				180			0.75	▼

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

41. Input the value shown and press Enter.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✓	L26			90		2.75		▲
✓	L27			180			0.75	▤
✓				90		2.25		▼

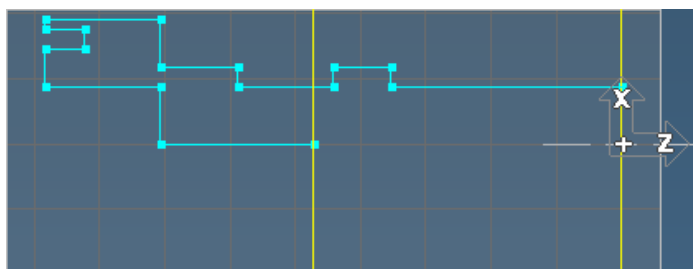Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

42. Input the value shown and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L27			180			0.75
L28			90		2.25	
			180			0.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

43. Input the value shown and press Enter.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L28			90		2.25	
L29			180			0.25
			90		0.75	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

44. Input the value shown and press Enter.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✓	L29			180			0.25	
✓	L30			90		0.75		
✓				180			0.75	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter
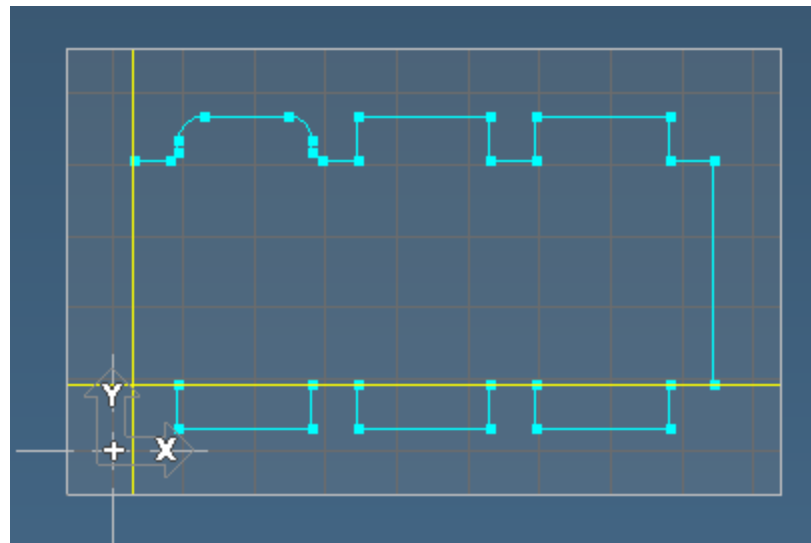
45. Select the Close Shape feature type for the last row.

46. Close Geometry Expert.

47. **Ctrl+click** the outside corners as shown.
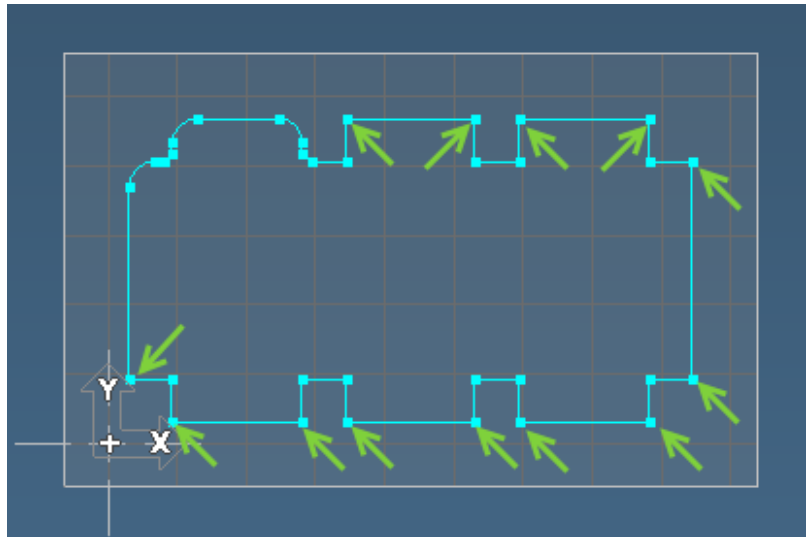
If you accidentally click something that you didn't want to and it becomes selected, click it again to deselect it.

48. Open the Fillet/Chamfer sub-palette.

49. Select the Fillet mode.

50. Input the value shown and press **Enter**.
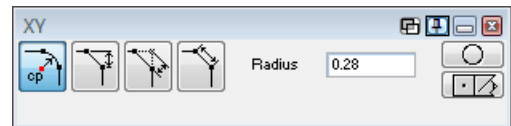
Fillets are created at all of the selected corners.

51. Ctrl+click the inside corners as shown.
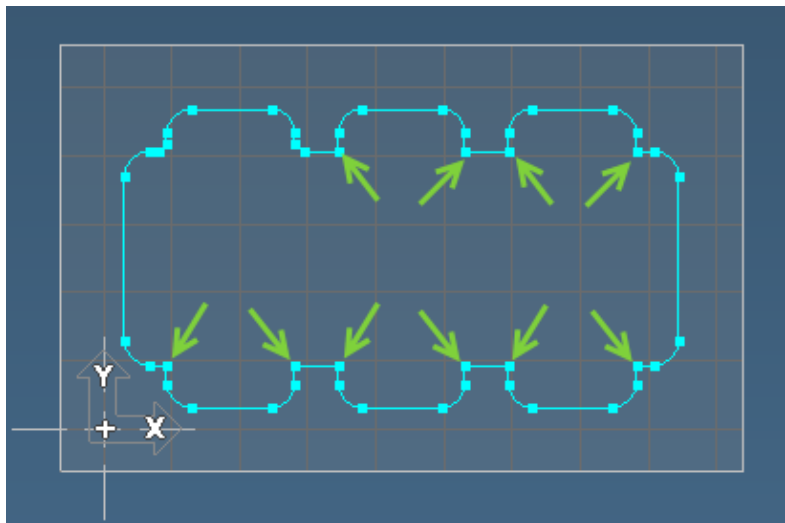
52. Open the Fillet/Chamfer sub-palette.

53. Select the Chamfer - Side mode.

This is the mode that Geometry Expert uses when creating chamfers.

54. Input the value shown and press Enter.
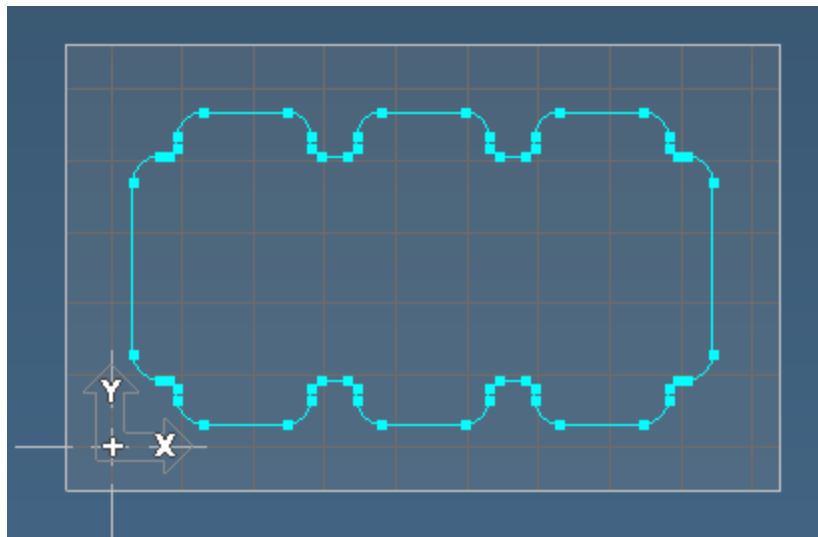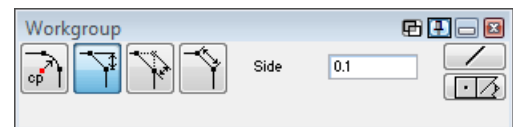
55. `Click` the Single Line button

The Chassis is now complete.

Save the file.

Exercise 3: Shuttle

In this exercise, we will create a slightly more complex shape containing arcs and angled lines. This exercise will also introduce how the Select Feature dialog works. For the part print for this exercise, see "Shuttle" on page 161.

Arcs and Angled Lines

1. Create a horizontal lathe part with the following workspace stock dimensions.
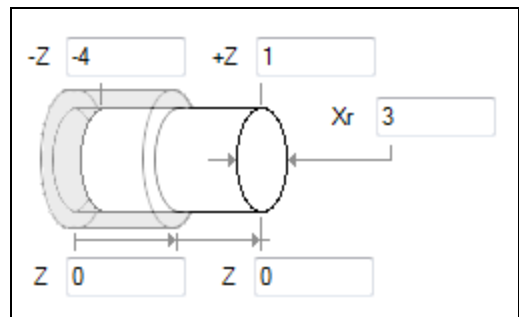
2. Select the Radius option for the X dimension style.1

Mill Users
If you want to create this as a Mill part, input the stock values shown. The images shown in Geometry Expert will be slightly different. In Lathe, the horizontal values shown are entered as Z coordinates, and the vertical values are entered as Xr coordinates. If you are working in Mill, the horizontal values are entered as X coordinates, and the vertical values are entered as Y coordinates.
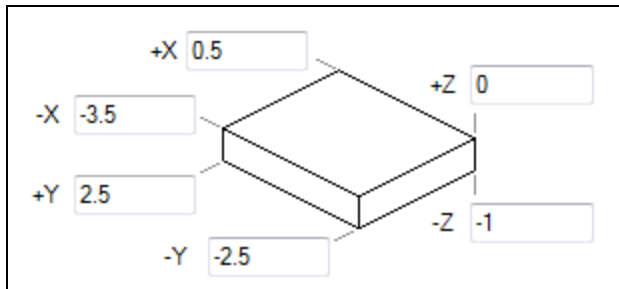
1. Open the Geometry Expert.

2. Select Counter-Clockwise Arc or press `Alt+5` for the first feature.

Remember, the feature type can also be selected using keyboard shortcuts [`Alt+(1-7)`].

3. Input the values shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
↻			1		-1	0

This first circle is fully defined by a CP and Radius. Enter.

4. Input the values shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
↻ C1			1		-1	0
/			180			.875
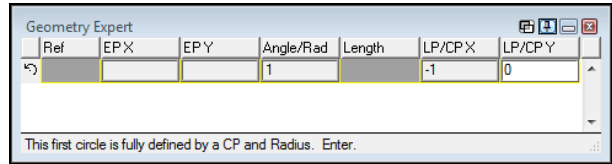
Line has an Angle (0, 180) & LP V. It will intersect the prev. Circle. Enter
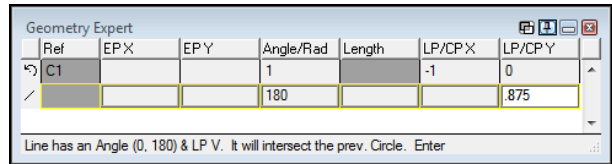
There are two possible points of intersection between the line and the circle. When this is the case, the Select Feature dialog will appear.

Select desired feature(s). OK

5. Select the right point shown and `click` the OK button.
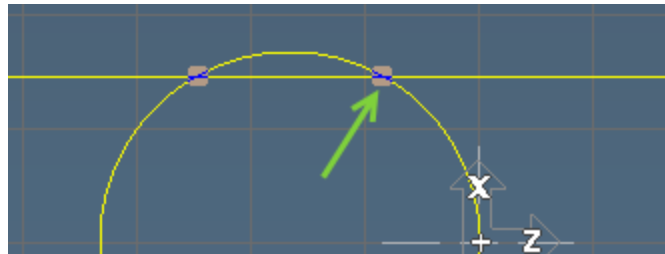
You can now continue creating connected features in Geometry Expert.

6. Input the values shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
↻ C1			1		-1	0
/ L21			180			.875
/	-1	0.938	90			

Line has an Angle & EP. Will intersect w/prev. Line. Enter
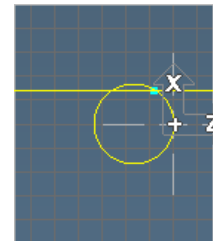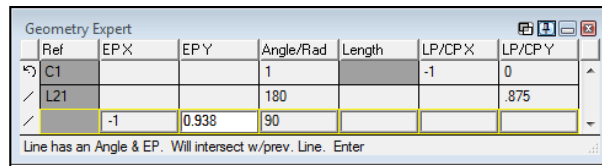
7. Delete the angle value and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
/ L1			180			.875
/ L2	-1	0.938	90			
/						

This line has no data. It will start at the last row's EP & be tan. to the next circle. Enter or add.
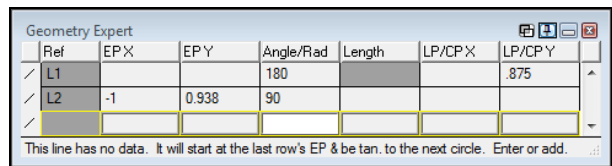
Notice the prompt. Although this row contains no data other than the feature type, following features will provide the system with the required information to define and draw this line. It will not be created until all necessary information has been entered. This is an example of a floating feature.

The next row's feature type will default to a clockwise arc.

8. Input the values shown and

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
/ L2	-1	0.938	90			
/						
↻			0.5		-1.621	1.5

Circle is defined by it's R & CP. The prev. Line will be tan to this and it's preceding EP. Enter
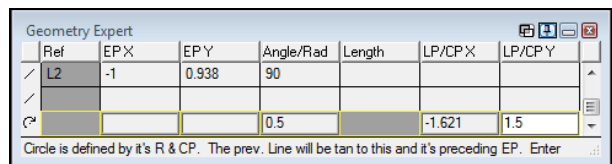
press `Enter`.

The previous line and the arc are drawn.

9. Input the values shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
/	L3						
↻	C2			0.5		-1.621	1.5
/	L4		1.651	140			

Line has an Angle & w/ be tan. to prev. Circle. EP w/be calc'd from EP V. Enter
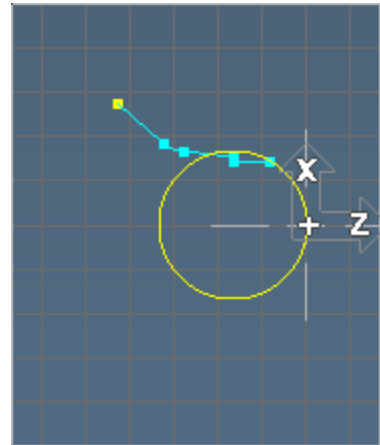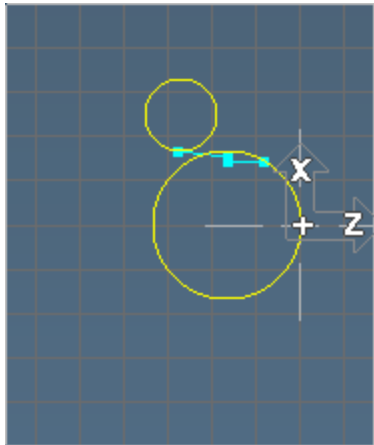
A half end point is valid in this case. As the prompt indicates, the system will calculate the other endpoint coordinate with the information it has, and draw the endpoint on the screen. An endpoint is necessary on this line because the following arc is not tangent to this line. The end point of the preceding feature will be used as the start point of the circle. Note that it is important the angle value entered indicate the proper vector direction of the line because the next feature is an arc.

10. Select Clockwise Arc feature type (`Alt+4`).

11. Input the values shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
↻	C3			0.5		-1.621	1.5
/	L4		1.651	140			
↻				0..187			

This circle will start at the last row's EP and be tangent to the next line or circle. Enter

This is another example of a floating feature, this time a circle. The circle will be drawn once we define the next feature. The system uses the information from the previous row (in this case the endpoint) and the information from the following row (which will provide a tangency point) to draw the arc.

12. Input the values shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
/	L4		1.651	140			
↻				0..187			
/				135		-3	2

Line has an Angle & LP. Prev. Circle w/be tan. Enter

(If the line is truncated make sure
`View>Extend lines` option is checked.)
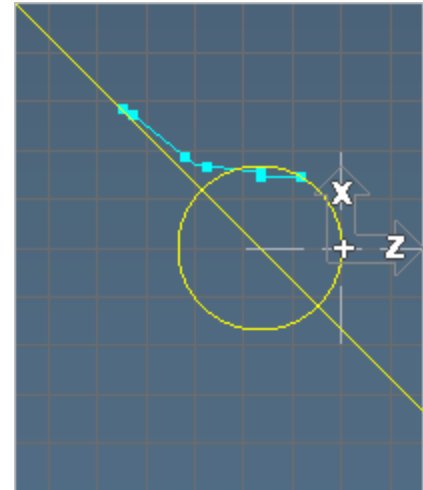
13. Input the values shown and press
    `Enter`.

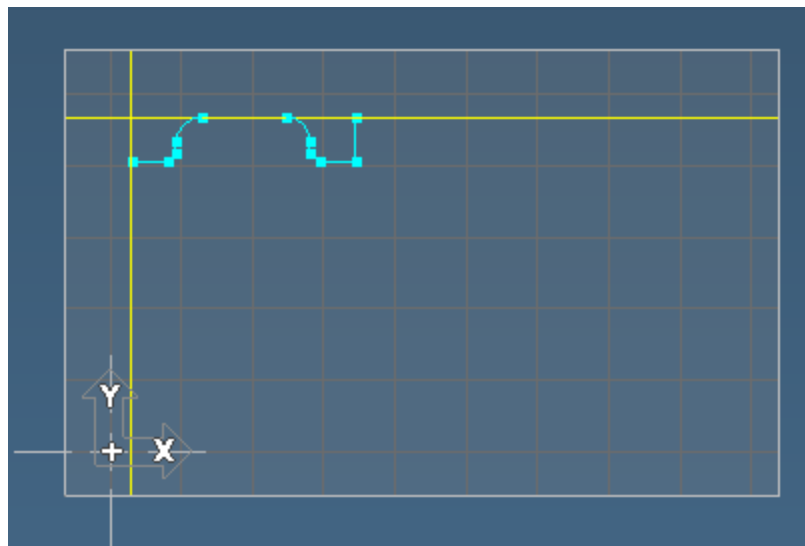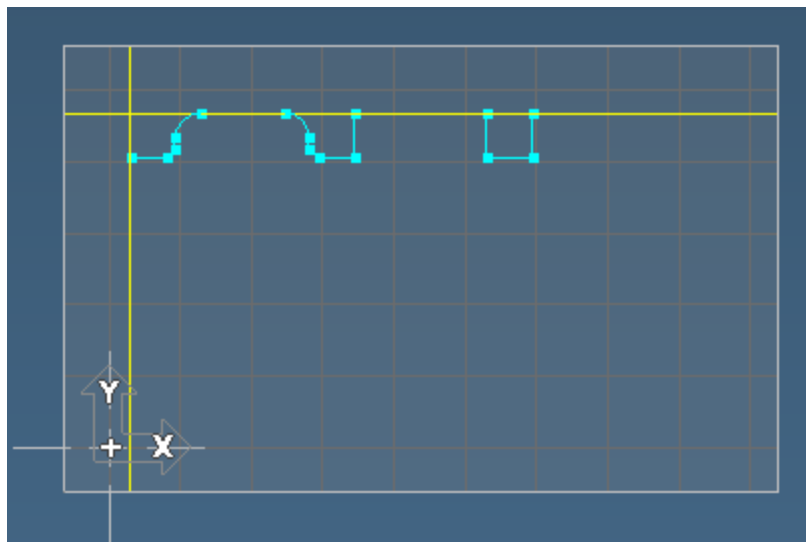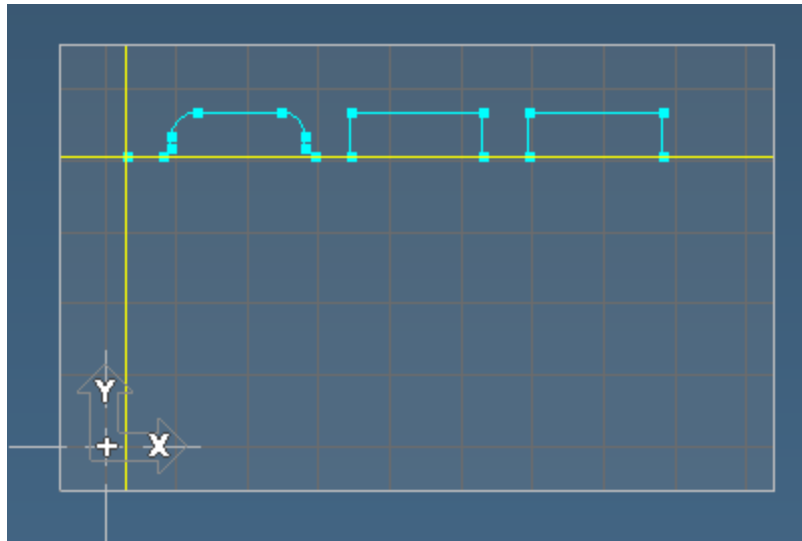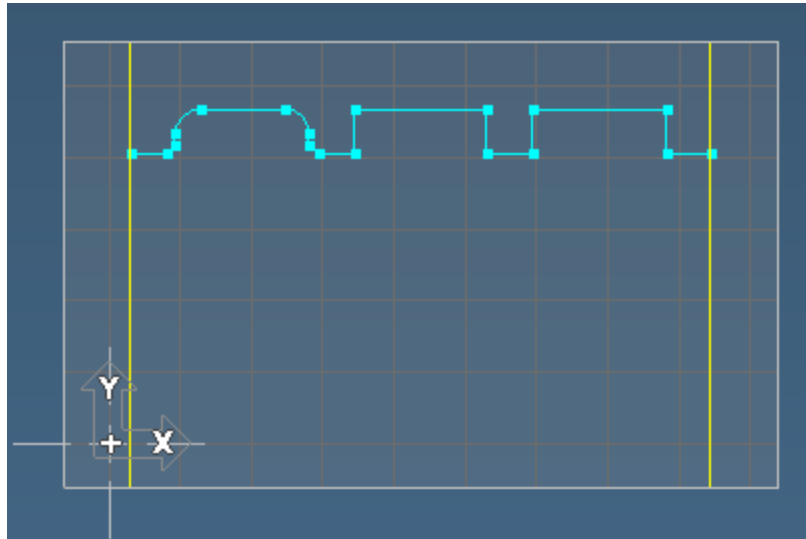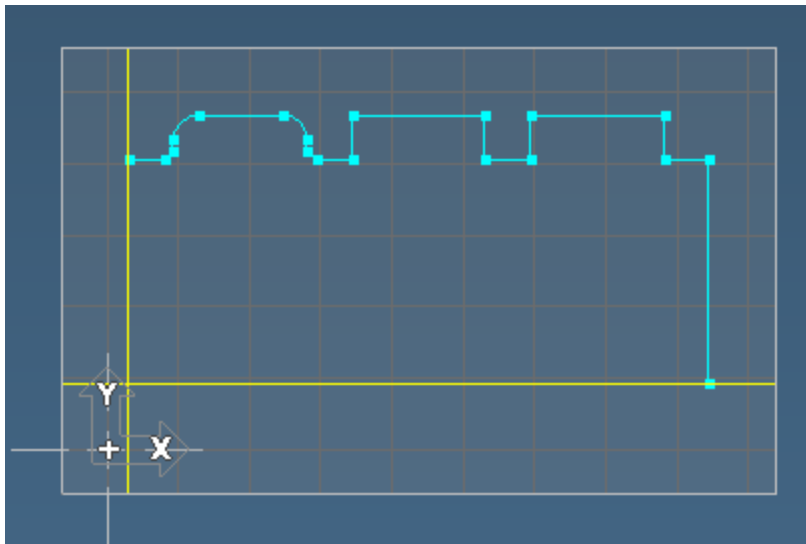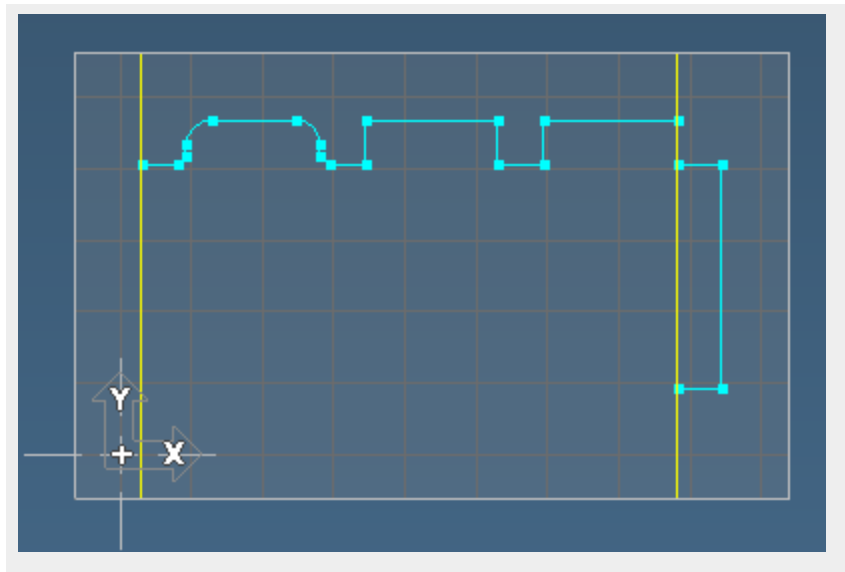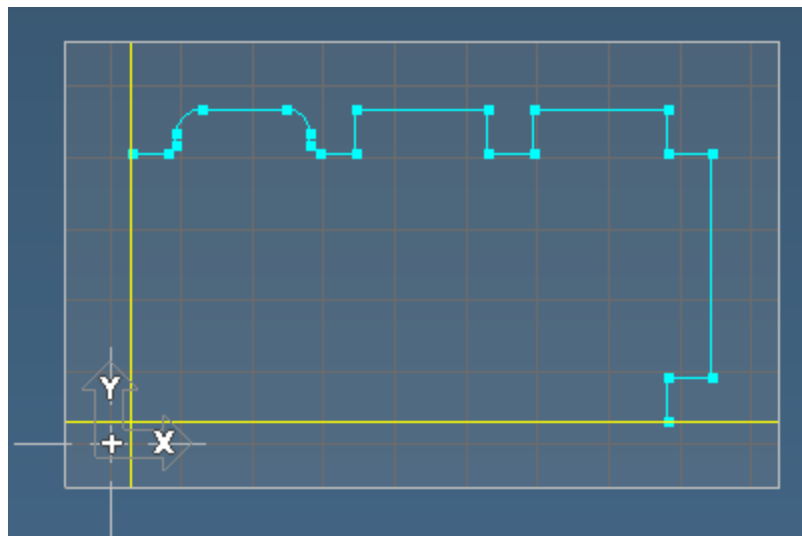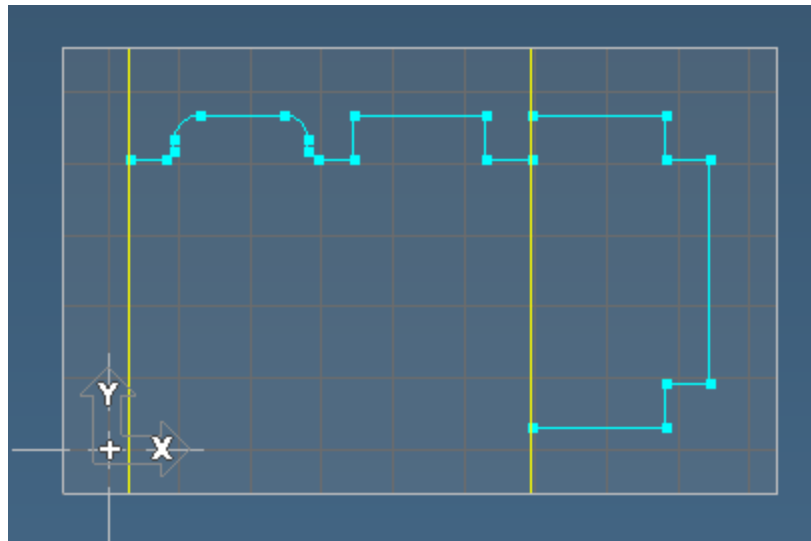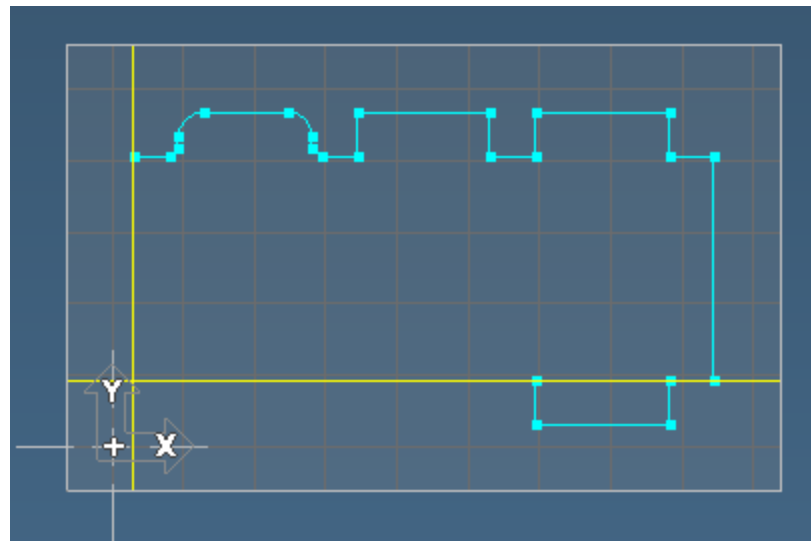| Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
C3			0.187			
L5			135		-3	2
L6			90		-3	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

14. Input the values shown and press
    `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L5			135		-3	2
L6			90		-3	
L7			0			0

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

15. Select the Close Shape feature type (`Alt+6`).

There are two possible intersection points, so the system will ask you to select which intersection point should be used. There is an option in the Edit menu which allows you to change the intersection point chosen in the select features dialog once the connection has been made. This Edit menu item toggles between Use Intersection #1 and Use Intersection #2.

16. Select the right point shown and
    press `Enter`.
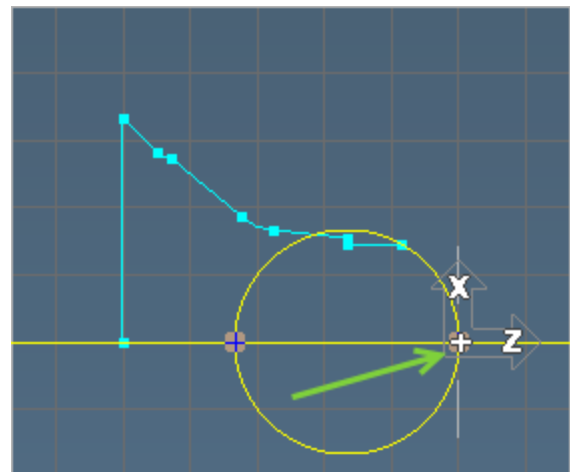
17. Resize Geometry Expert so all rows are visible.

18. Select the cell shown.

Notice that the row becomes highlighted and is now the current row. Also, the arc that is defined by this row is now selected.

19. Change the radius value to `0.4` and press `Enter`.

The radius is adjusted and maintains the connections.

Exercise 4: Lathe Tutorial

Make sure that you save this file in a place where you can easily access it, because it will be used in the machining tutorial found in the Lathe manual. This part is also created in the Free-Form CAD Exercises.

Lathe Tutorial

1. Create a lathe part with dimensions set as inch with the following workspace stock dimensions.
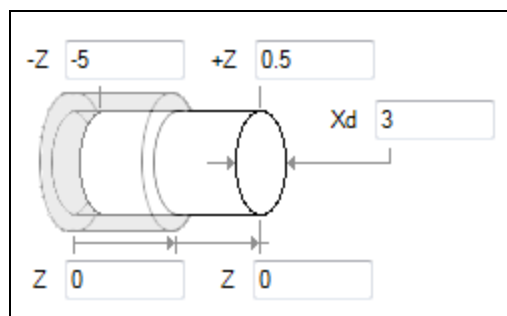
2. Choose Diameter for the X Dimension Style.

3. Open the Geometry Expert.

4. Input the value shown and press `Enter`.

Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
⁄			90		0	

This first line is fully defined by a LP H and an Angle (90 or 270). Enter.
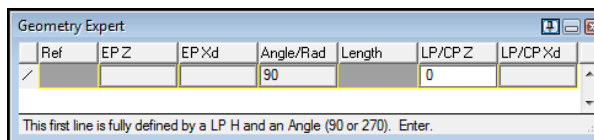
5. Select the Chamfer feature type (`Alt+2`).

6. Input the value shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
⁄ L1			90		0	
⌒				0.2		

This row will create a Chamfer between the preceding and following features.
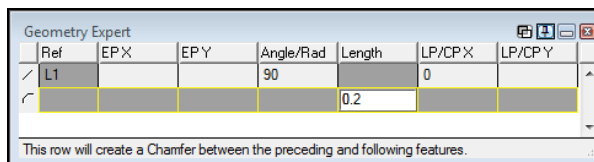
This chamfer will be drawn when we close the shape.

7. Input the value shown and press `Enter`.

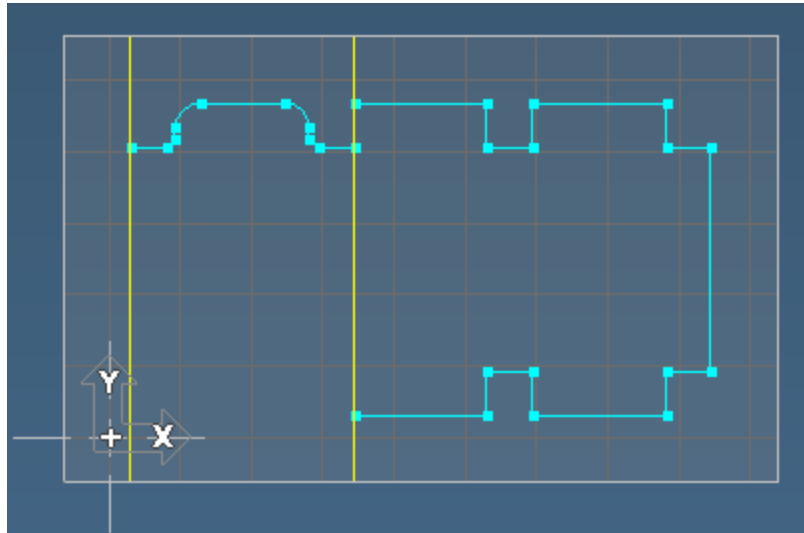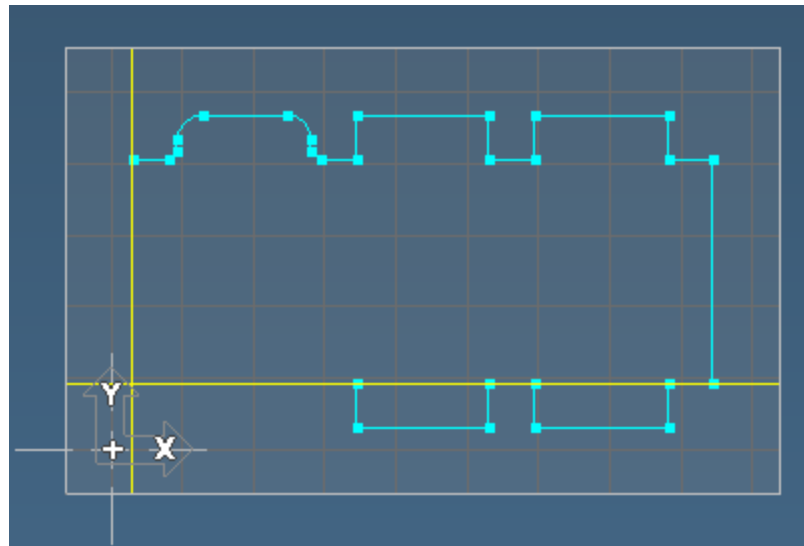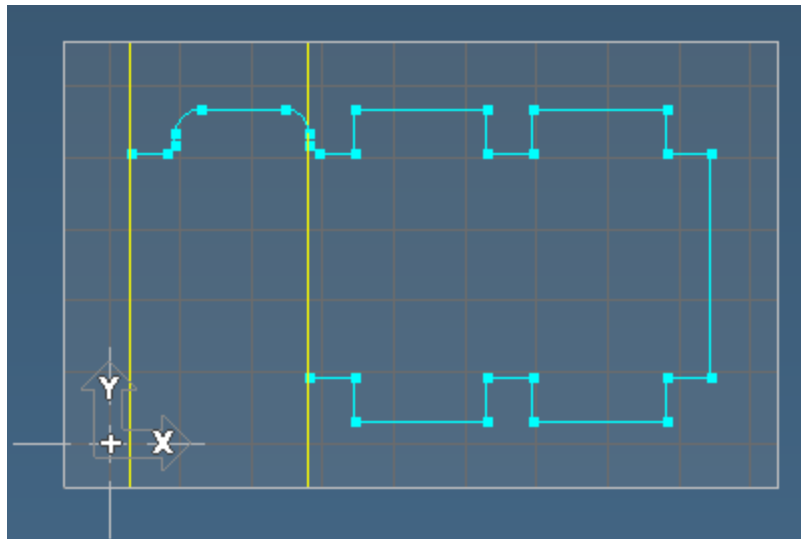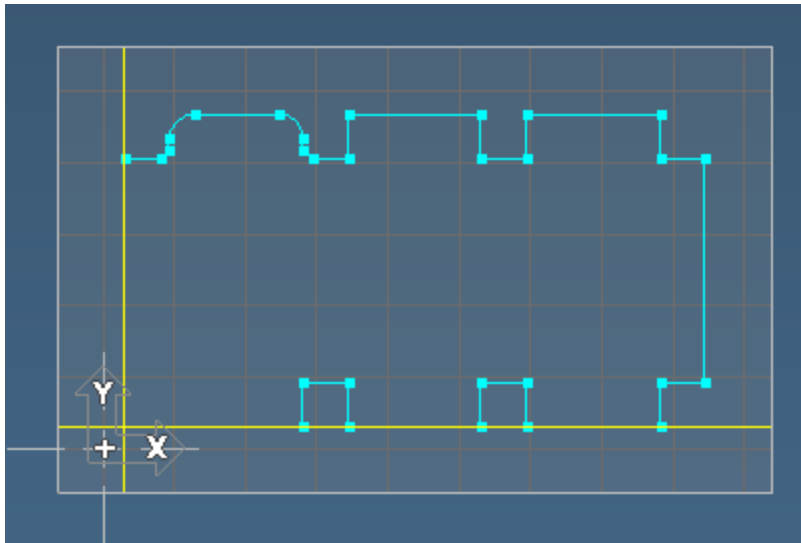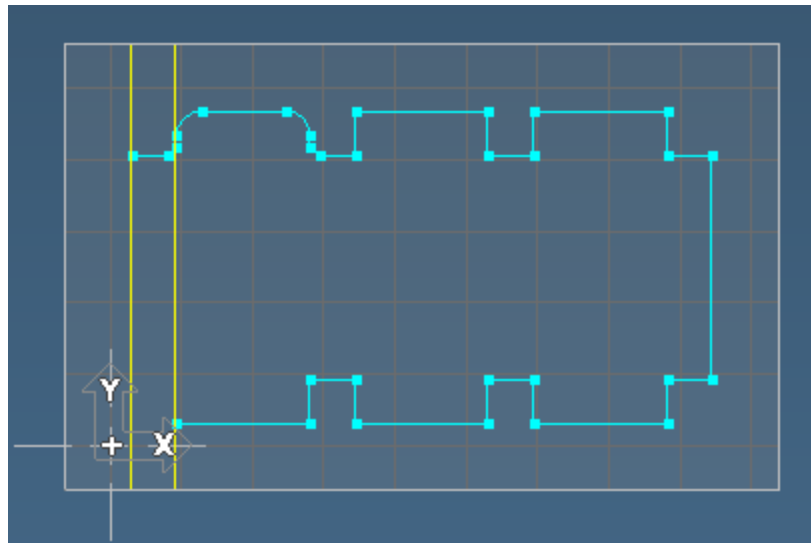| Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
⁄ L1			90		0	
⌒				0.2		
⁄			180			2.25

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

8. Input the values shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
⌒				0.2		
⁄ L2			180			2.25
⁄			45		-0.975	2.25
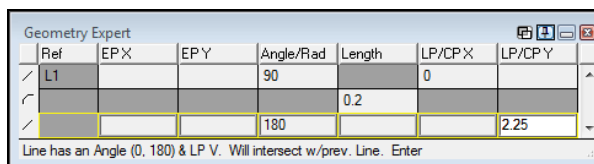
Line has an Angle & LP. Will intersect w/prev. Line. Enter
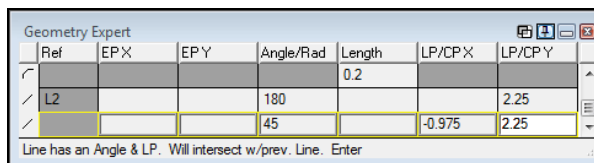
If the line does not draw properly make sure `View>Extend lines` is checked)
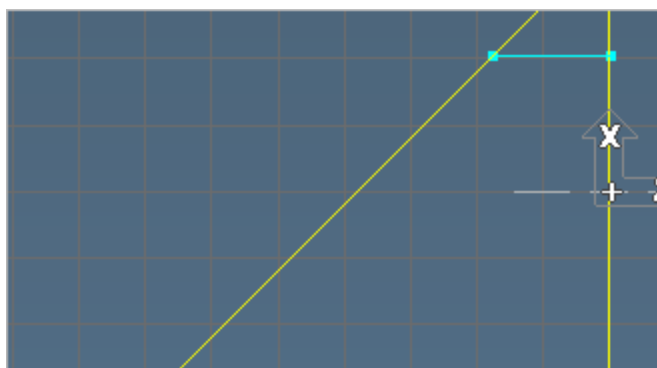
9. Select the Fillet feature type (`Alt+3`).

10. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
╱	L2			180			2.25
╱	L3			45		-0.975	2.25
⤸				0.2			

This row will create a Fillet between the preceding and following features.

This Fillet will be drawn when the next feature is defined.

11. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
╱	L3			45		-0.975	2.25
⤸				0.2			
╱				180			1.6

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

12. Select the Fillet feature type (`Alt+3`) and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
⤸				0.2			
╱	L4			180			1.6
⤸				0.2			

This row will create a Fillet between the preceding and following features.

The radius defaults to the last entered value.

13. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
╱	L4			180			1.6
⤸				0.2			
╱				117		-2.2	2.8

Line has an Angle & LP. Will intersect w/prev. Line. Enter

14. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
⤸				0.2			
╱	L5			117		-2.2	2.8
╱				180			2.8

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

Both fillets are created because the preceding and following features are fully connected.
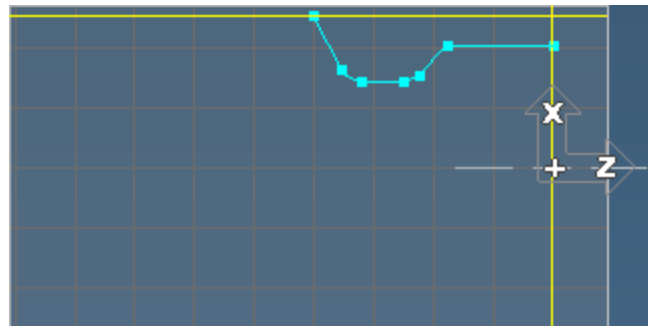
15. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
╱	L5			117		-2.2	2.8
╱	L6			180			2.8
╱				66		-2.5	2.8

Line has an Angle & LP. Will intersect w/prev. Line. Enter

16. Select the Fillet feature type (`Alt+3`) and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✎	L6			180			2.8	
✎	L7			66		-2.5	2.8	
↰				0.2				

This row will create a Fillet between the preceding and following features.

17. Enter the angle and line point value shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✎	L7			66		-2.5	2.8	
↰				0.2				
✎				180			1	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

18. Select the Fillet feature type (`Alt+3`) and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
↰				0.2				
✎	L8			180			1	
↰				0.2				

This row will create a Fillet between the preceding and following features.

19. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✎	L8			180			1	
↰				0.2				
✎				114		-3.8	2.8	

Line has an Angle & LP. Will intersect w/prev. Line. Enter

20. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
↰				0.2				
✎	L9			114		-3.8	2.8	
✎				180			2.8	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter
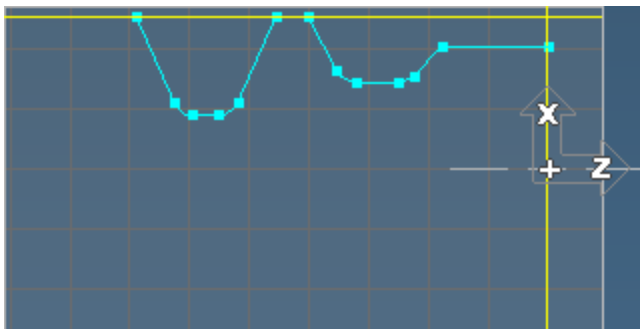
21. Input the values shown and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✎	L9			114		-3.8	2.8	
✎	L10			180			2.8	
✎				85		-4	2.8	

Line has an Angle & LP. Will intersect w/prev. Line. Enter

22. Select the Fillet feature type (`Alt+3`) enter the radius value shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
✎	L10			180			2.8	
✎	L11			85		-4	2.8	
⤵				0.015				

This row will create a Fillet between the preceding and following features.

23. Input the values shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
✎	L11			85		-4	2.8	
⤵				0.015				
✎				180			2.52	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

24. Select the Fillet feature type (`Alt+3`) press `Enter` to accept the default value shown.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
⤵				0.015				
✎	L12			180			2.52	
⤵				0.015				

This row will create a Fillet between the preceding and following features.

25. Input the values shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
✎	L12			180			2.52	
⤵				0.015				
✎				95		-4.15	2.8	

Line has an Angle & LP. Will intersect w/prev. Line. Enter

26. Input the values shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
⤵				0.015				
✎	L13			95		-4.15	2.8	
✎				180			2.8	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

27. Input the values shown and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
✎	L13			95		-4.15	2.8	
✎	L14			180			2.8	
✎				90		-4.9		

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter
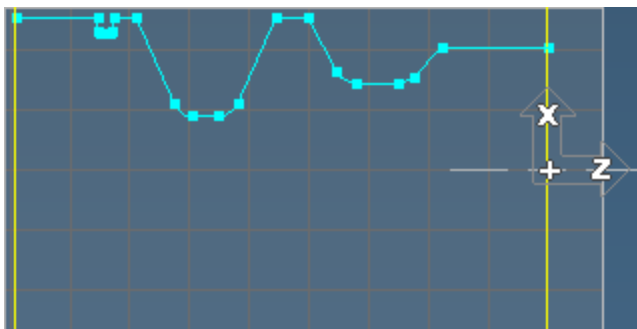
28. Input the values shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
✎ L14			180			2.8	
✎ L15			90		-4.9		
✎			180			0.8	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

29. Select the Fillet feature type (`Alt+3`)

Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd	
✎ L15			90		-4.9		
✎ L16			180			0.8	
⤻			0.2				

This row will create a Fillet between the preceding and following features.

30. Change the radius and press `Enter`.

31. Input the values shown and press `Enter`.

Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd	
✎ L16			180			0.8	
⤻			0.2				
✎			45		-2.6	0.8	

Line has an Angle & LP. Will intersect w/prev. Line. Enter
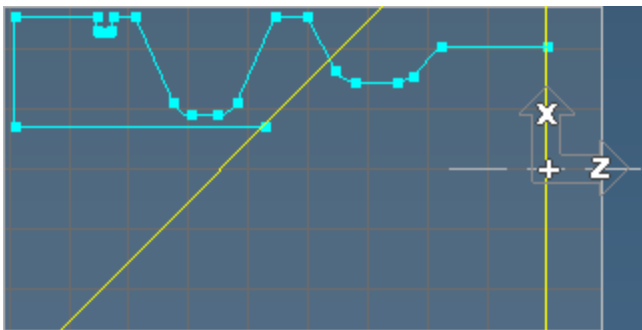
32. Select the Fillet feature type (`Alt+3`) and press `Enter`.

Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd	
⤻			0.2				
✎ L17			45		-2.6	0.8	
⤻			0.2				

This row will create a Fillet between the preceding and following features.

33. The radius defaults to the last entered value.

34. Input the values shown and press `Enter`.

Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd	
✎ L17			45		-2.6	0.8	
⤻			0.2				
✎	-1.1	1.4	180				

Line has an Angle & EP. Will intersect w/prev. Line. Enter

This line must be defined with an endpoint because the next feature is an arc that is not tangent. Whenever an arc is not tangent to the previous line, the arc needs to have a start point.
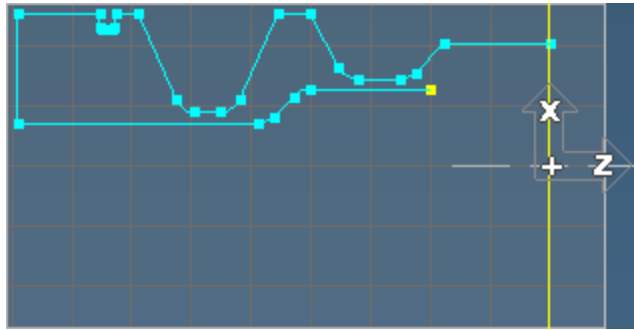
35. Select the Clockwise Arc feature type (`Alt+4`).

36. Input the values shown and press `Enter`.

	Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
↰	C8			0.2			
╱	L18	-1.1	1.4	180			
↱				0.2			

This circle will start at the last row's EP and be tangent to the next line or circle. Enter
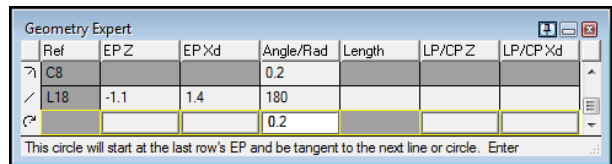
Notice the prompt.

Implicit Angle Tangent Line

37. Input the values shown and press `Enter`.

	Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
╱	L18	-1.1	1.4	180			
↱				0.2			
╱				0			1.6

Line has an Angle (0 or 180) & LP V. Prev. Circle w/be tan. Enter
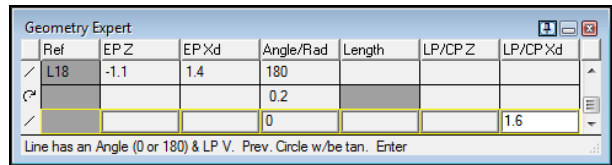
An angle value of 0° must be entered for this line because it is connected to an arc, and we must indicate the exact direction of the line. Line direction is determined by the angle value. Since we defined this part in the CCW direction of the shape, this line is `0`. We could have defined the part in the clockwise direction, and the angle value would have been 180°.

38. Selecting the Close Shape feature type (`Alt+6`).

	Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
↱	C9			0.2			
╱	L19			0			1.6
+							
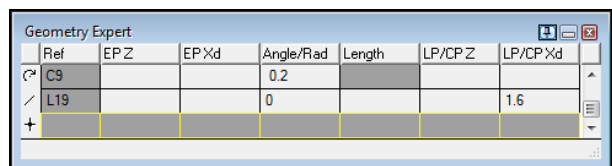
Notice that the chamfer we created at the beginning has been drawn because the lines it is between are now fully connected.
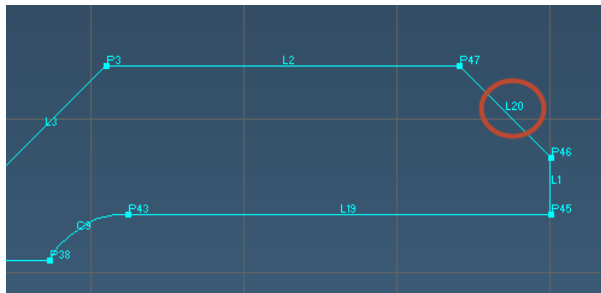
Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
L1			90		0	
L20				0.2		
L2			180			2.25
L3			45		-0.975	2.25
C1			0.2			
L4			180			1.6
C2			0.2			
L5			117		-2.2	2.8
L6			180			2.8
L7			66		-2.5	2.8

Save this part.

Exercise 5: Doodle

In this exercise, we will show how the Geometry Expert can aid in fixing a rough sketch of the part geometry using the mouse line tool. For the part print for this exercise, see .

Editing Free-Form Geometry

Again this part can be done with either the Mill or Lathe workspace stock definition.

1. Create a lathe part with inch dimension and with the following workspace stock dimensions.
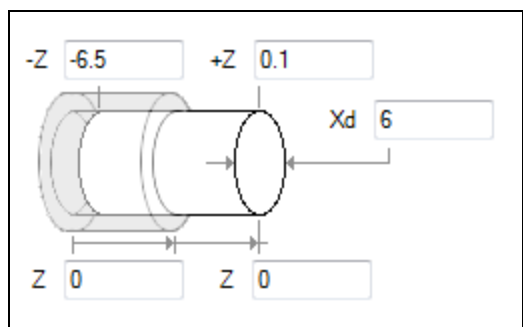
2. Select Diameter for the X dimension style.

3. Open the Geometry palette.

4. Open the Line sub-palette. (2)

5. Select Mouse line.

We will start sketching the part from the vertical line at Z -6.25. Don't worry too much about the coordinates shown in the Mouse Line dialog, the dimensions will be adjusted using Geometry Expert.

DO NOT draw the chamfer on the part. We will insert that in the spreadsheet.

6. **Click** near the location of point 1.
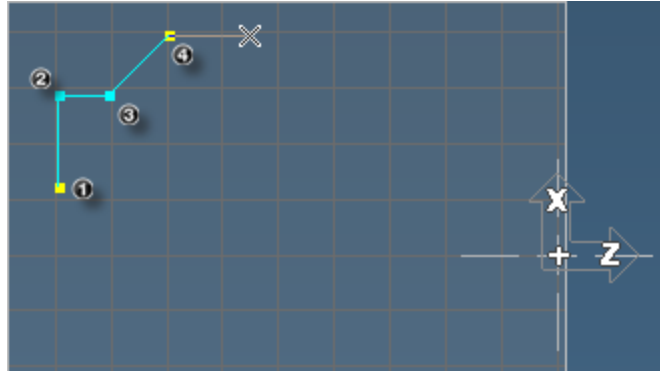
7. Continue creating the part following the part print until the shape is closed (when you click on the first point again).

   Your finished part should look something like this:
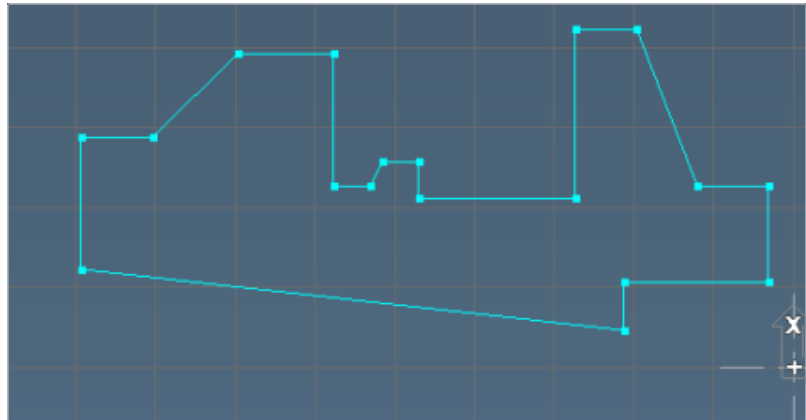
This is a very quick process because we aren't concerned with proper dimensions. We only need an approximate shape or have the same number of features (excluding the chamfer).

8. Open the Geometry Expert.

Loading Free-Form Geometry

1. **Double-click** the first line we created as shown.
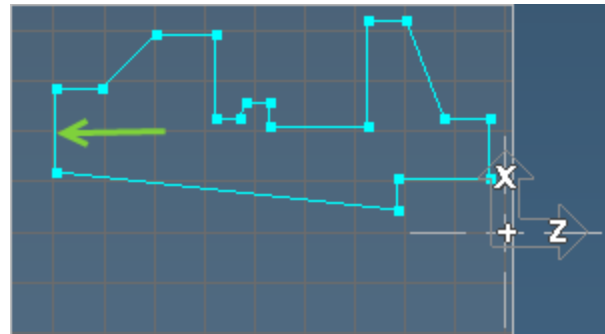
All connected features will be loaded into the Geometry Expert spreadsheet with the dimension values. The first feature listed in the spreadsheet will be the feature that you clicked on to load the shape. All the other features will be listed in creation order from that first feature.

2. Scroll to the top of the Geometry Expert using the up arrow on the slider bar as shown below.

	Ref	EP Z	EP X	Angle/Rad	Length	LP/CP Z	LP/CP X	
/	L7			90		-6		
/	L8			0				
/	L9			45		-5.5	4.2	

Line has an Angle. It will be tan. to the next Circle (Enter), or add more info (EP or LP).

If you selected the line as shown above, the first feature should be a line with an angle close to 90° around Z -6.25. If your reference numbers don't exactly match those shown in the pictures, don't worry about it.

Click in one of the cells in the first row.

3. Change the values as shown and press Enter.

	Ref	EP Z	EP X	Angle/Rad	Length	LP/CP Z	LP/CP X	
/	L7			90		-6.25		
/	L8			0			4.2	
/	L9			45		-5.5	4.2	

This first line is fully defined by a LP H and an Angle (90 or 270). Enter.

The row can also be reached using the arrow keys or the mouse. When you have changed the values you will find that the Enter key won't scroll to the next row.

The feature is highlighted in red and will change according to the new dimensions. To move down to the next row in the spreadsheet you must use the arrow keys or click the cursor within a cell of another row. Be careful to use Backspace to delete any values, as the Del key will delete the entire row.

4. Change the second feature as shown and press Enter.

	Ref	EP Z	EP X	Angle/Rad	Length	LP/CP Z	LP/CP X	
/	L7			90		-6.25		
/	L8			0			4.25	
/	L9			45		-5.5	4.2	

Line has an Angle (0, 180) & LP V. Will intersect w/prev. Line. Enter

Continue to adjust the part dimensions in this manner. You can select the geometry or the row and edit the values to match the part print. Be careful to use Backspace to delete, as the Del key will delete the entire row.
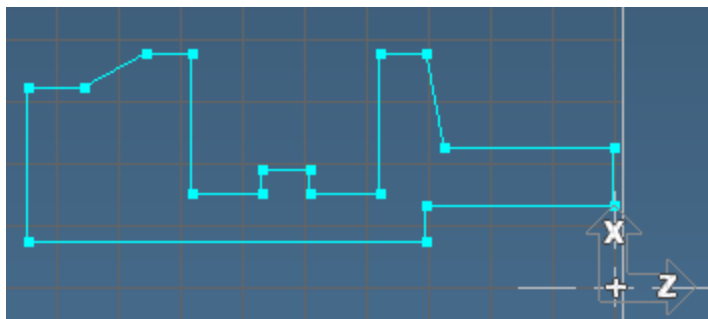
Because of the existing connections and locations of points, when you make some of the adjustments it may look like something is wrong; the features and overall shape may not look the way it should. Don't worry, keep going through and changing the dimensions, and Geometry

Expert will adjust the shape correctly. You may find that editing down each vertical line is faster. If somehow the Geometry Expert fails in some way, you may have to fix the geometry with the mouse line and connect button. Geometry Expert will not be able to regenerate some broken connections.

When finished, the Geometry Expert should appear as shown. Notice that the close shape row is already in the spreadsheet. It was entered in the spreadsheet when we double clicked and loaded the shape because it was a closed shape.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
L1			90		-6.25	
L2			0			4.25
L3			30		-5	5
L4			0			5
L5			270		-4.5	
L6			0			2
L7			90		-3.75	
L8			0			2.5
L9			270		-3.25	
L10			0			2
L11			90		-2.5	
L12			0			5
L13			100		-2	5
L14			0			3
L15			270		0	
L16			180			1.75
L17			270		-2	
L18			180			1

Your part should now have accurate dimensions.
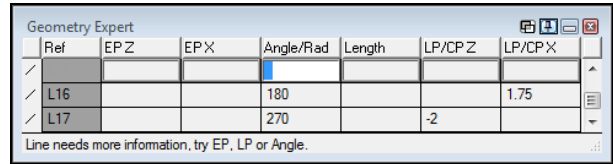
Inserting a Row

1. Select the line at Z = 0.

Ref	EP Z	EP X	Angle/Rad	Length	LP/CP Z	LP/CP X
L15			270		0	
L16			180			1.75
L17			270		-2	

Line has an Angle (90, 270) & LP H. Will intersect w/prev. Line. Enter

2. Choose Edit > Insert row (Alt+I).
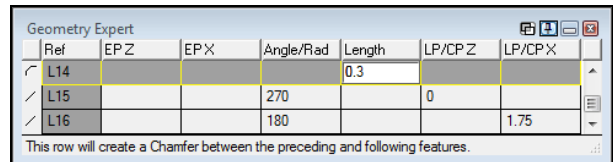
Ref	EP Z	EP X	Angle/Rad	Length	LP/CP Z	LP/CP X
L16			180			1.75
L17			270		-2	

Line needs more information, try EP, LP or Angle.

A row with no information other than the feature type, which is a line, will be inserted in the spreadsheet above the current row. Notice the new row is now the current row.

3. Select the Chamfer feature type (Alt+2).

4. Input the value shown in the inserted row and press Enter.

Ref	EP Z	EP X	Angle/Rad	Length	LP/CP Z	LP/CP X
L14				0.3		
L15			270		0	
L16			180			1.75

This row will create a Chamfer between the preceding and following features.

The chamfer will immediately be drawn because it is being created at the intersection of two features that have already been connected.
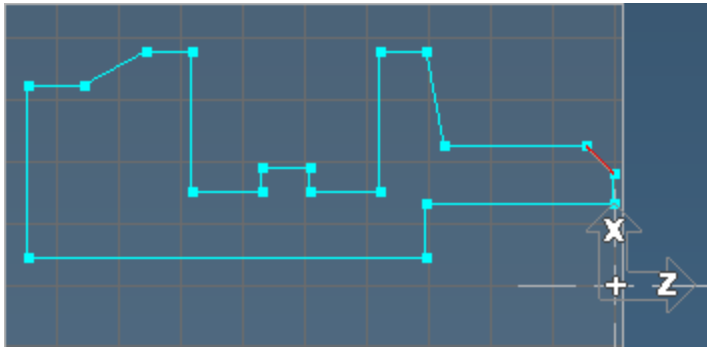
Try sketching any shape and adjusting the dimensions. If any problems occur, the Error Balloons, and Prompting information should indicate what needs to be done. You can also combine Free-Form tools with the Geometry Expert, which we will cover in the next section.

COMBINATION EXERCISES

The exercises contained in this chapter use both Geometry Expert and the free-form CAD tools to create part geometry. In the first exercise, we will demonstrate how to change shapes created in Free-form Geometry. In the second exercise, we will re-create the Mill Tutorial part using primarily Geometry Expert, but we will need to use the free-form CAD tools to create some construction geometry. For a more advanced example of construction see "Exercise 3: Gear Housing" on page 154. This can be done once you feel comfortable with both Geometry Expert and free-form CAD.

Exercise 1: Using Geometry Expert to Modify Shapes

In Part A of this exercise, we will load the Mill Tutorial part. For the part print see " Mill Tutorial" on page 159.

Mill Tutorial

Changing the Shape

1. Open the Mill tutorial part.

   > ⓘ A part file can be opened in several ways. The most simple method is using the Open command in the File menu or in the Documents dialog. More advanced methods include the shortcut `Ctrl+O` or drag and dropping files. A file can be dropped in a running version of GibbsCAM or on an icon (including a shortcut icon). See the Getting Started manual for more information on the basics of using the system.

2. Choose File > 💾 Save As... Save the file with a new name.

3. Open the Geometry Expert.

4. `Double-click` the large arc shown.

The exterior shape is loaded into the Geometry Expert. Note the features of the boss are not loaded because the boss was not part of the shape selected.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
↺	C1			2		0	0	
↻	C6			4				
↺	C2			0.5		-3	0	
/	L3			259.92134		-3.23115	-0.44336	
↺	C3			0.5		-3	-2	
/	L4							
↺	C8			2		0	0	
↻	C9			1				
↺	C4			0.5		3	-2	
/	L5			90		3		
↻	C10			0.5				
↺	C5			0.5		3	2	
/	L1							
+								

5. Select the first row, the 2" circle.

6. Change the radius and press `Enter`.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY	
↺	C1			3		0	0	
↻	C6			4				
↺	C2			0.5		-3	0	

This first circle is fully defined by a CP and Radius. Enter.

The features connected to the arc have changed. The features tangent to the arc have adjusted accordingly.

7. Select the line as shown.
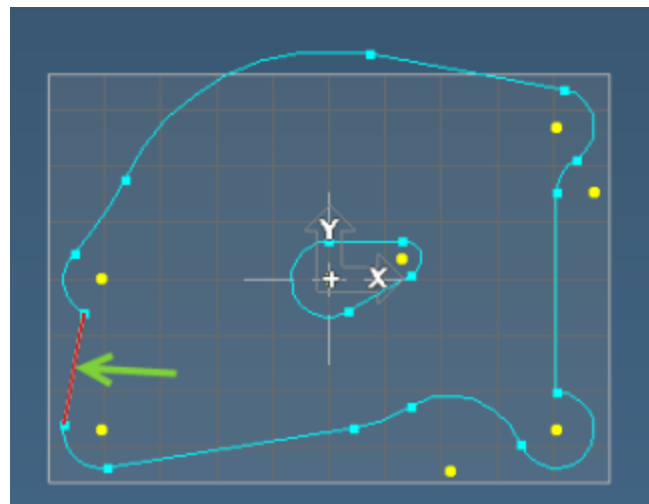
8. Change the angle value as shown and press `Enter`.

The feature changes according to the new dimension entered.

Both of the changes we have made here have maintained the continuity of the shape without breaking it into individual geometry. Shapes are broken when features do not have any points of intersection or tangency. Geometry Expert will display an error message to indicate that the feature has no intersection with the previous feature.

9. Select the arc shown.
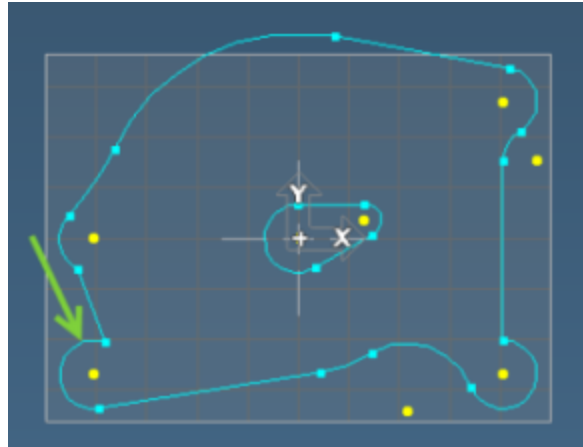
10. Change the radius value and press `Enter`.

When you attempt to enter the row, an error message will come up in the spreadsheet indicating that the previous feature cannot intersect.

The circle will be re-drawn with the new radius in yellow, indicating that it is not a fully connected feature.

11. Undo the change (Ctrl+Z).

The shape will re-connect.
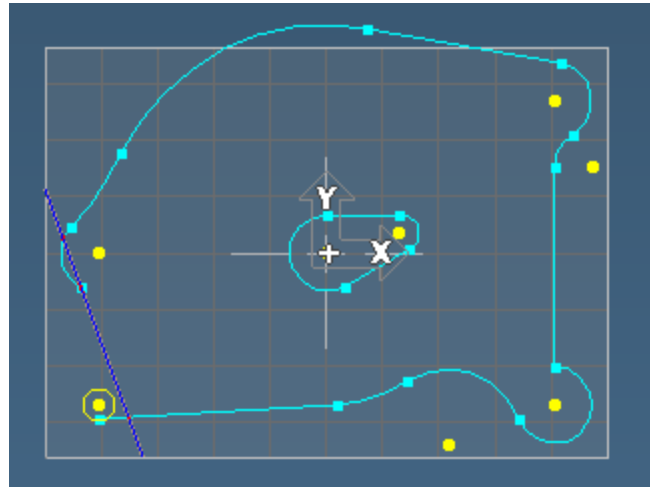
Lathe Tutorial

Changing the Shape

Open the Lathe tutorial part.

Before changes are made to the part, we will make a copy of the file. Because we are going to use this part geometry for the machining tutorial in the Lathe manual, we need to keep a copy of the file that will not be changed.

1. Choose File > Save As... and save the file with a new name.

2. Open the Geometry Expert.

3. Double-click the line shown.
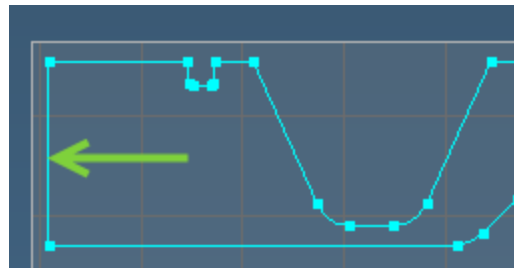
4. Locate the row shown.

This row is a 270° line at Z -4.9.

	Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
∕	L15			90		-4.9	
∕	L16			180			0.8
⤺				0.2			

This row will create a Fillet between the preceding and following features.

5. Change the Z value as shown and press Enter.

	Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
/	L15			270		-4.25	
/	L16			0			0.8
↗	C6			0.2			

This first line is fully defined by a LP H and an Angle (90 or 270). Enter.

The geometry will change according to the new dimensions entered. The features connected to that line are also changed to maintain the proper connections.

6. Select the arc shown.
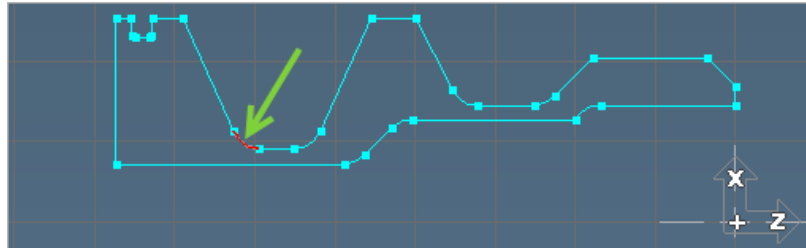
This row now becomes the selected row. Inversely, selecting the row would highlight the feature being defined on the screen.

7. Change the radius value as shown and press Enter.

	Ref	EP Z	EP Xd	Angle/Rad	Length	LP/CP Z	LP/CP Xd
↻	C2			0.6			
/	L9			114			
/	L10			180			2.8

Circle has a Radius. It w/be tan to prev. Line & to next Line or Circle. Add info or Enter.

Again, the shape will change according to the new dimensions entered.

Both of the changes we have made have maintained the continuity of the shape.
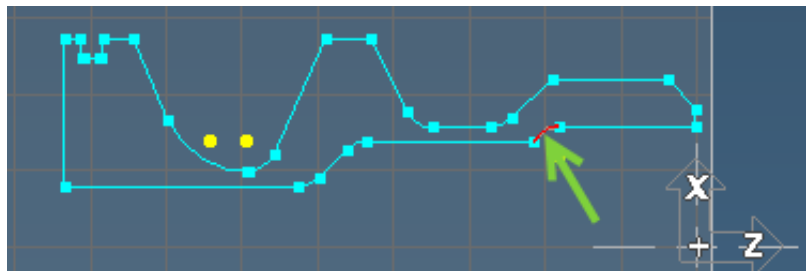
8. Select the arc shown.

9. Change the radius value as shown and press Enter.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
↻	C3			0.9		-0.9268	1.2
/	L12			0			1.6
/				90		0	

Circle has a Radius & CP. Prev. Line w/be tangent to its prior Circle and this Circle. Enter.

When you attempt to enter the row, an error message appears indicating that the previous

feature cannot intersect.

The arc will be re-drawn with the new radius value even though it cannot connect. Notice that it is now yellow, indicating that it is not a connected feature.
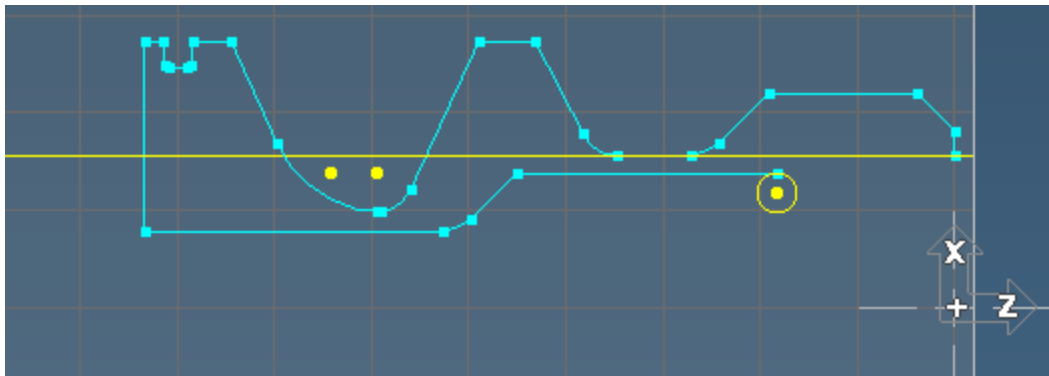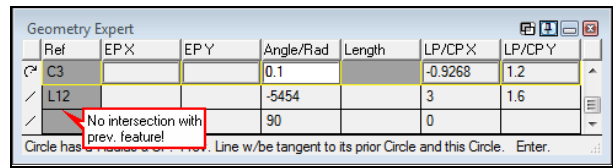
10. Undo the change (`Ctrl+Z`).

The shape will re-connect.

Exercise 2: Mill Tutorial

Mill Tutorial

For the part print for this exercises, see .

1. Create a new vertical mill part with the following workspace stock dimensions.

2. Open the Geometry Expert.

In this exercise it is important that the direction of the lines and the circles that we create follow the path of the shape correctly. We will start with the 0.5 radius circle in the upper right corner a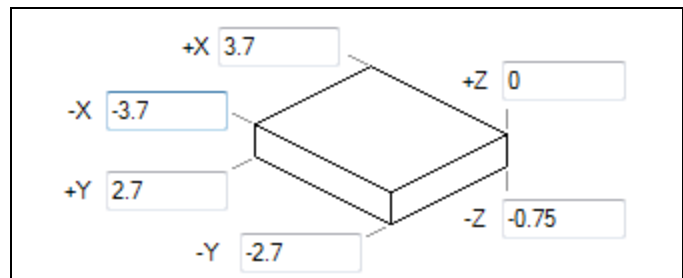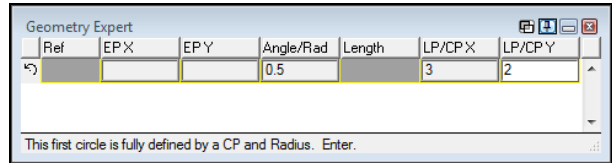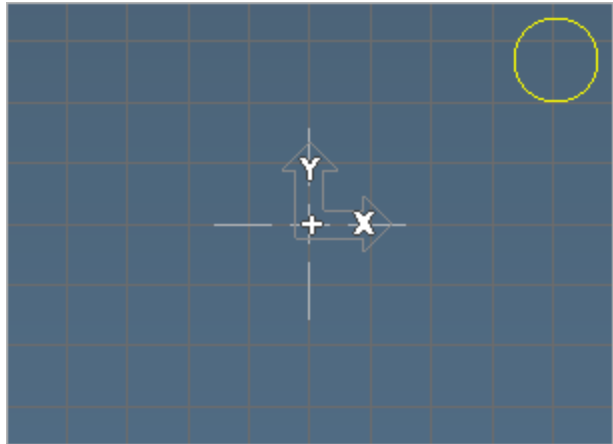nd proceed around the part in a counter-clockwise direction. Make sure that you pay close attention to the feature type selections made and the values entered.

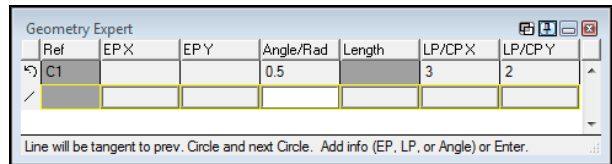3. Select the Counter-Clockwise Arc feature type (`Alt+5`).

4. Input the values shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
			0.5		3	2

This first circle is fully defined by a CP and Radius. Enter.

The only information provided on the part print about the next line is that it is tangent to the circles it is connected to. (Make sure the view is set to Top to see the geometry correctly.)

5. Delete the angle value (`Backspace`) and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
C1			0.5		3	2

Line will be tangent to prev. Circle and next Circle. Add info (EP, LP, or Angle) or Enter.

When this floating line is entered the next row will default to a clockwise arc feature type; however, we need a counter-clockwise arc.

6. Change to the Counter-Clockwise Arc feature type (`Alt+5`).

7. Input the values shown and press `Enter`.

Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
C1			0.5		3	2
L1						
C2			2		0	0

Circle has a Radius & CP. Prev. Line w/be tangent to its prior Circle and this Circle. Enter.
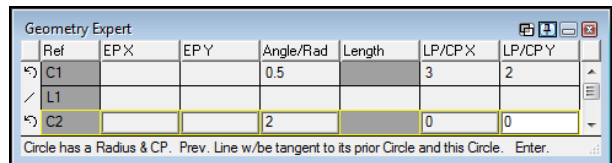
Both the circle and the line are drawn.
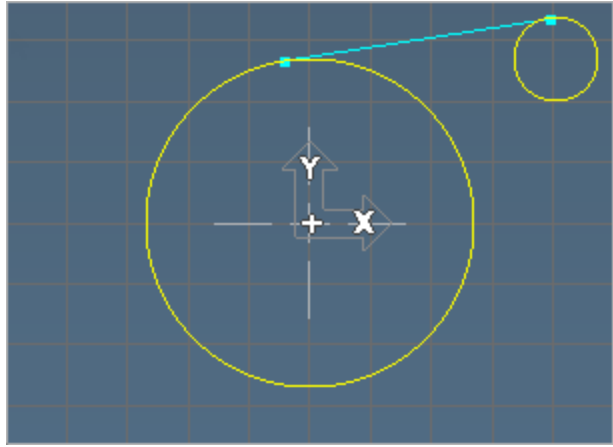
8. Select Clockwise Arc feature type (Alt+4).

9. Input the values shown and press Enter.
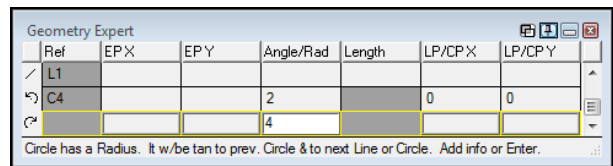
This circle is a floating feature.

10. Select the Counter-Clockwise Arc feature type (Alt+5).

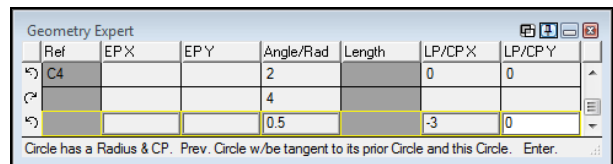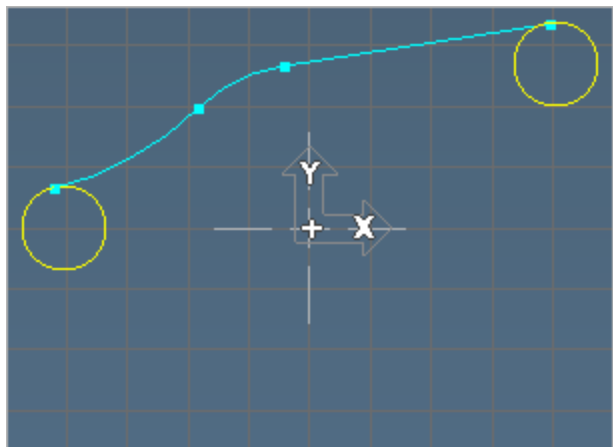11. Input the values shown and press Enter.

A center point must be entered for this circle because the preceding feature is a floating arc, and depends on this circle for its creation. Therefore, this feature must be fully defined.

At this point we need to use the free-form CAD tools to create the next line because construction geometry is required. We will create the geometry necessary using free-form CAD, and then reload the shape into Geometry Expert to complete the shape.

12. Close Geometry Expert.

13. Open the Point sub-palette. (1)

14. Select Explicit point. (1)

15. Input the values shown and press Enter.

16. Open Circle sub-palette. (3)

17. Select Radius & Center Point. (1)

18. Select the point shown.

19. Input the value shown and press Enter.
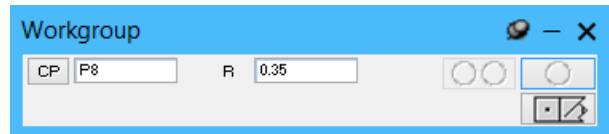
Extracting a Center Point of a Circle

20. Open the Point sub-palette. (1)

21. Select Center Point.

This option creates a point at the center of a selected circle.

22. Select the circle shown and
    press Enter.
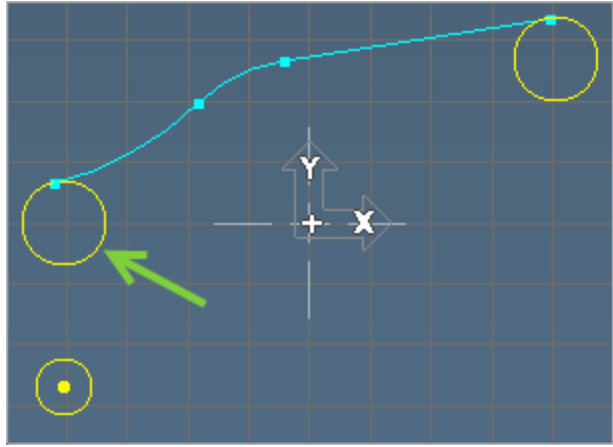
23. Open the Line sub-palette. (2)

24. Select the point and circle shown, and press Enter.

25. Select the line shown and press Enter.

26. Open the Line sub-palette. (2)

27. Select Parallel line.

28. Select the line we just created.

29. Input the value shown and press Enter.
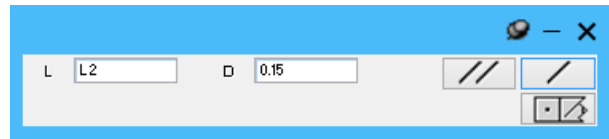
30. Select the line shown and press Enter.
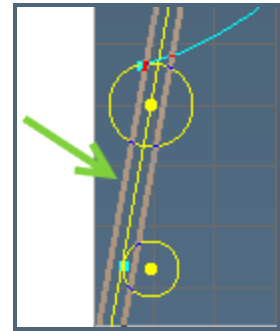
31. Connect the circle and line shown.
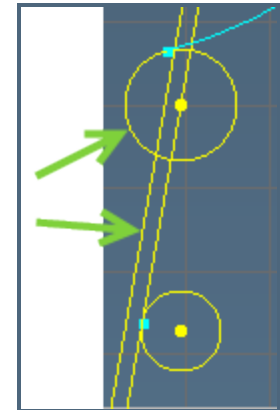
32. `Double-click` the circle shown. (This will select the circle and the line that joins onto it)
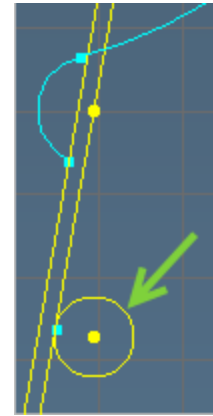
33. `Click` the Trash button.

Now we will load the shape back into Geometry Expert to create the remaining features.
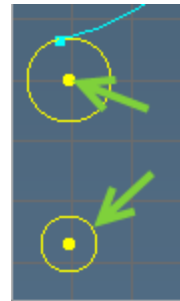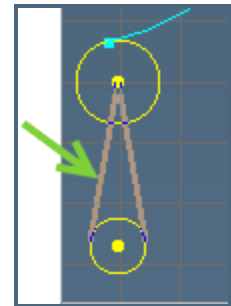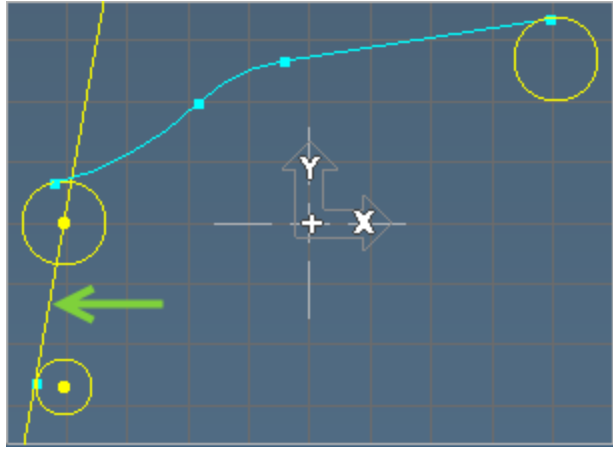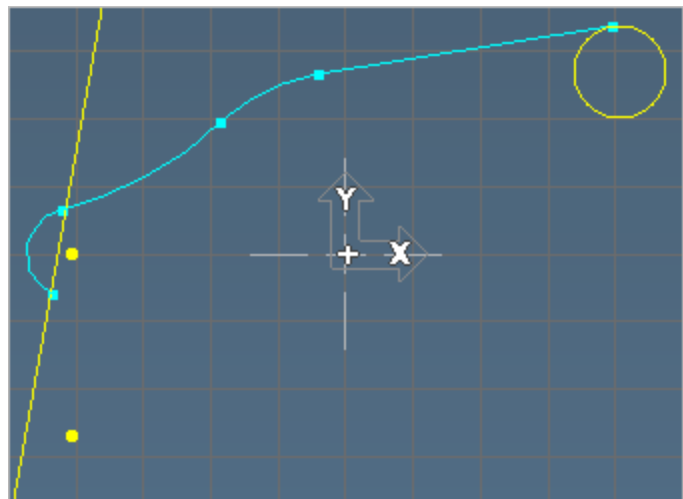
34. Open the Geometry Expert.

35. `Double-click` any feature.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
↺	C1			0.5		3	2
╱	L1						
↺	C4			2		0	0
↻	C5			4			
↺	C3			0.5		-3	0
╱	L3			259.92134		-3.23115	-0.44336
╱				90			

Line has an Angle. It will be tan. to the next Circle (Enter), or add more info (EP or LP).

Because this shape was created using Geometry Expert, it will be reloaded in the same order as when it was created.

The line we just created is the last feature.

36. Create a clockwise arc (Alt+5) with the values shown.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
↰	C3			0.5		-3	0
/	L3			259.92134		-3.23115	-0.44336
↱				0.5		-3	-2

Circle has a Radius & CP. It will intersect prev. Line. Enter.

37. Delete the angle and press Enter.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
/	L3			259.92134		-3.23115	-0.44336
↱	C6			0.5		-3	-2
/							

Line will be tangent to prev. Circle and next Circle. Add info (EP, LP, or Angle) or Enter.

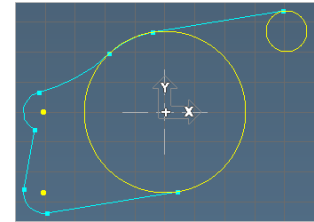This will create a tangent floating line.

38. Create a counter-clockwise arc (Alt+5) with the values shown.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
↱	C6			0.5		-3	-2
/							
↰				2		0	0

Circle has a Radius & CP. Prev. Line w/be tangent to its prior Circle and this Circle. Enter.

39. Create a clockwise arc (Alt+4) with the values shown.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
/	L2						
↰	C7			2		0	0
↱				1			

Circle has a Radius. It w/be tan to prev. Circle & to next Line or Circle. Add info or Enter.

This will create a tangent floating arc.

40. Create a counter-clockwise arc (Alt+5) with the values shown.

	Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
↰	C7			2		0	0
↱				1			
↰				0.5		3	-2

Circle has a Radius & CP. Prev. Circle w/be tangent to its prior Circle and this Circle. Enter.
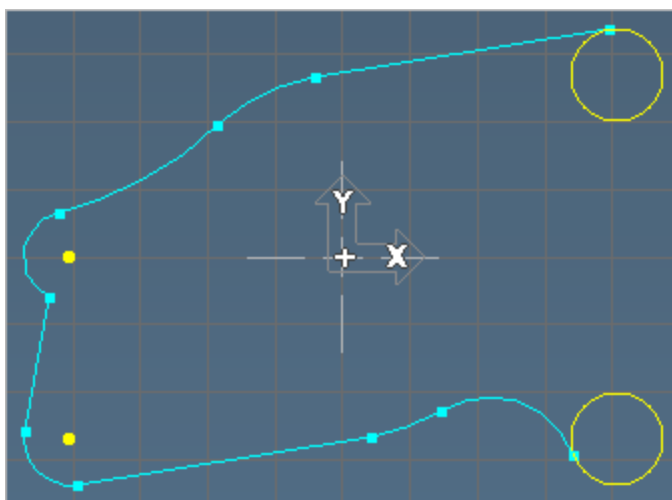
41. Create a line with the values shown connected through the point shown.

Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
C9				1		
C8				0.5	3	-2
				90	3	

Geometry Expert

Line has an Angle (90, 270) & LP H. It will intersect the prev. Circle. Enter

42. Create a clockwise arc (`Alt+4`) with the default value.

Geometry Expert

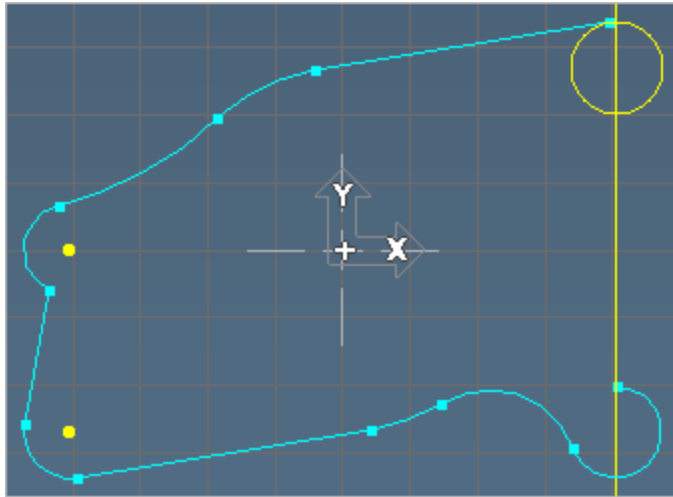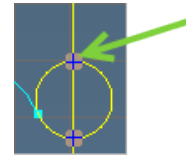| Ref	EPX	EPY	Angle/Rad	Length	LP/CPX	LP/CPY
C8				0.5	3	-2
L4				90	3	
				0.5		

Circle has a Radius. It w/be tan to prev. Line & to next Line or Circle. Add info or Enter.

43. Close the shape. (`Alt+6`)

In order to create the center boss, we will need to close Geometry Expert and start a new shape because the boss is not connected to the outside wall of the shape. The boss will also be created using a combination of free-form CAD tools and Geometry Expert.
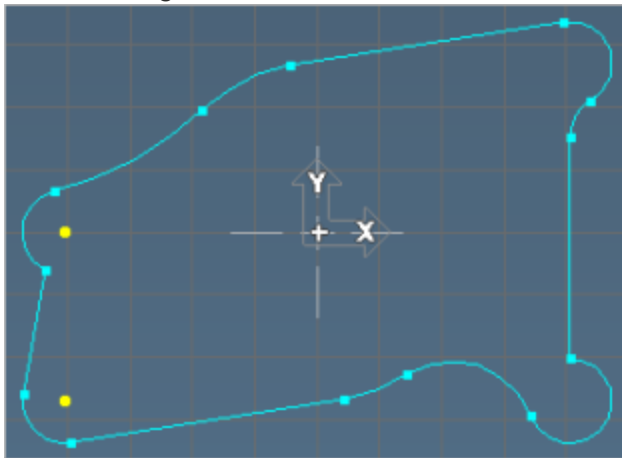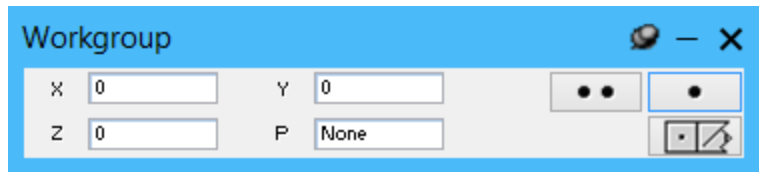
Boss Geometry

1. Open the Point sub-palette. (`1`)

2. Select Explicit point. (1)

XYZ

3. Input the coordinates shown.

Workgroup					
X	0	Y	0		
Z	0	P	None		

4. Click the Single Point button.

5. Open the Point sub-palette. (1)

6. Select Polar Point.

7. Select the point at the origin as shown.

8. Input the values shown.

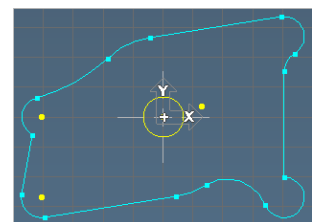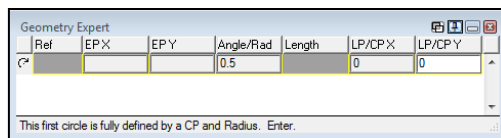| Workgroup				
P	P26	D	1	
A	15			

9. Click the Single Point button.

10. Open Geometry Expert.
.

11. Select the Clockwise Arc feature type and input the values shown and press Enter.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y	
↻				0.5		0	0	

This first circle is fully defined by a CP and Radius. Enter.

12. Delete the angle of the row and press `Enter`.

We will enter the coordinates of the centerpoint of the next arc by using the Interrogation cursor. Clicking on a feature or point while holding down the `Alt` or `Shift+Alt` keys, the appropriate value for the active cell will be entered in the spreadsheet. The `Shift+Alt` shortcut will interrogate all the properties of a given point. The `Alt` key alone will interrogate the properties for just the active cell.

13. Input the radius value shown below and then place the cursor in the CP X cell, making it the active cell.

14. `Alt+click` the point shown.

The X coordinate for that point will be entered.

15. Move to the centerpoint Y coordinate. While holding down the `Alt` key, click the point again and press `Enter`. `Enter`.

Alternatively you can use the `Shift+Alt` interrogation option. This inputs both the X and Y coordinates in one step. You can delete the X and Y values and redo the last steps using the `Shift+Alt` modifier to see how using each method may be useful.
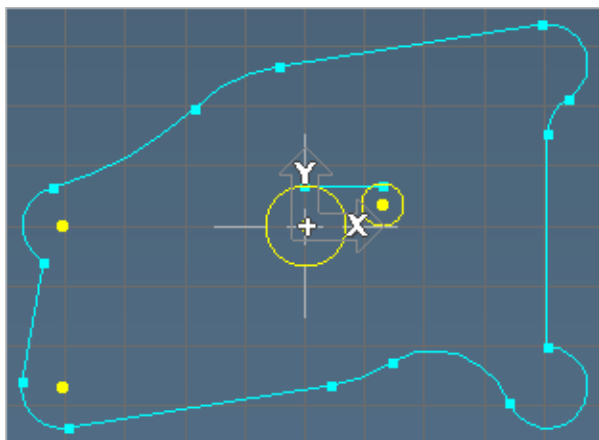
16. Delete the angle of the next row and press `Enter`.

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
/	L5						
↻	C12			0.25		0.96593	0.25882
/							

Geometry Expert

Line will be tangent to prev. Circle and next Circle. Add info (EP, LP, or Angle) or Enter.

17. Close the shape. (`Alt+6`)

	Ref	EP X	EP Y	Angle/Rad	Length	LP/CP X	LP/CP Y
↻	C12			0.25		0.96593	0.25882
/	L6						
+							

Geometry Expert

The geometry is now complete. You may use either Mill Tutorial geometry for the Mill tutorial.

Exercise 3: Gear Housing

Advanced Tutorial

About the Part

This is a rather complex part as far as geometry creation is concerned. You should only do this exercise after you have completed the exercises in both the Geometry Expert and free-form CAD chapters. Step-by-step instructions are not provided in this exercise. Instead, some very general guidelines are outlined, and problem areas are highlighted and discussed. If you find this exercise to be too difficult, we recommend that you go back and create either the Mill or Lathe part again, but don't use the step-by-step instructions. Instead, work from the part print and use the tutorial text as an aid if you get stuck.

This part provides a good example of determining when Geometry Expert or free-form CAD should be used to create certain shapes depending on how the part is dimensioned. The exercise also highlights the differences between arcs and fillets and the issues involved with forward tangencies in Geometry Expert.

For the part print for this exercise, see . The part print has four areas that are circled and labeled. These are the particularly difficult areas that will be highlighted in this exercise.

The part is composed primarily of three closed shapes—the outside wall and two inside pockets.
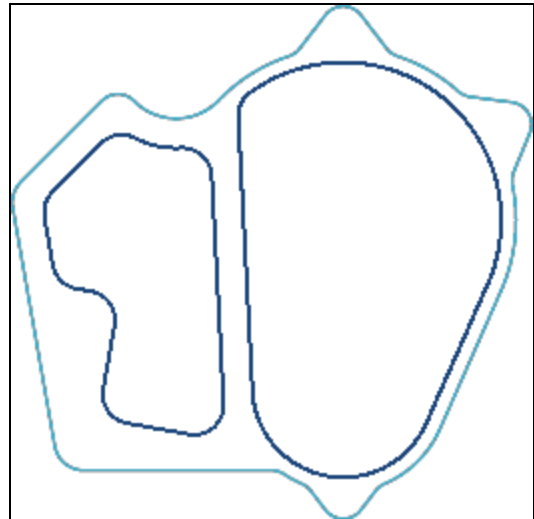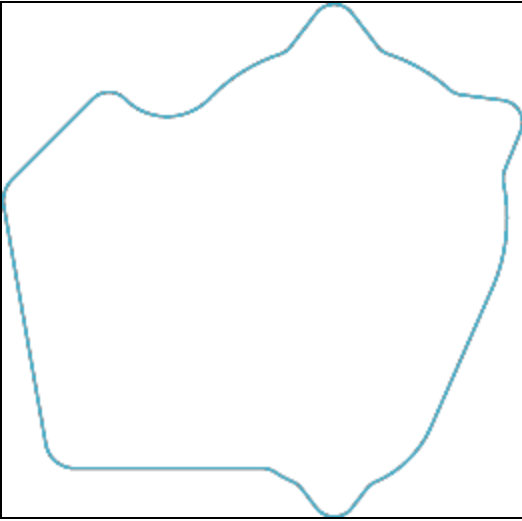
The outside wall, shown, can be created most efficiently using Geometry Expert. The entire outside shape can be created with Geometry Expert, except for one point that must be created using the free-form CAD tools.

The centerpoint for one of the outside arcs must be created in CAD; the dimensions of that point can then be interrogated.
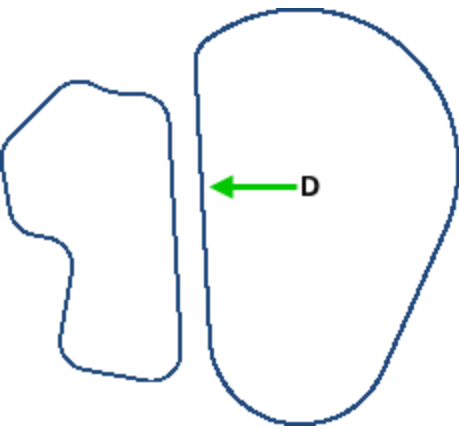
A combination of Geometry Expert and free-form CAD is the best method for creating the larger inside pocket shown.

The line labeled D must be created in CAD because it is defined as being tangent to a circle that is not part of the closed shape that contains line D.

The smaller inside pocket shown is best created using only the free-form CAD tools because most of the features that compose this shape are dimensioned with construction geometry.

For the outside shape, you should begin with the designated Start Feature (1.466" R arc), and proceed around the shape in a clockwise direction. In this exercise, it will be very important to read the prompting information, and to also make sure that the directions of the arcs and lines are correct. Remember, line direction is indicated by the angle value.

Outer Shape

Area A is highlighted as a difficult section because this portion of geometry requires that a floating line (line defined with only an angle) be tangent to the following feature, but not the previous feature. This is an example of a forward tangency.

After creating the first arc, the second feature (0.2" radius) must be created as a counter-clockwise arc rather than a fillet.

A fillet cannot be used in this case because the system cannot calculate the fillet until the intersection between the preceding arc and following line is complete.

1. Start Feature
2. 1.466 R
3. 0.200 R (TYP)
4. 0.187 DIA (TYP)
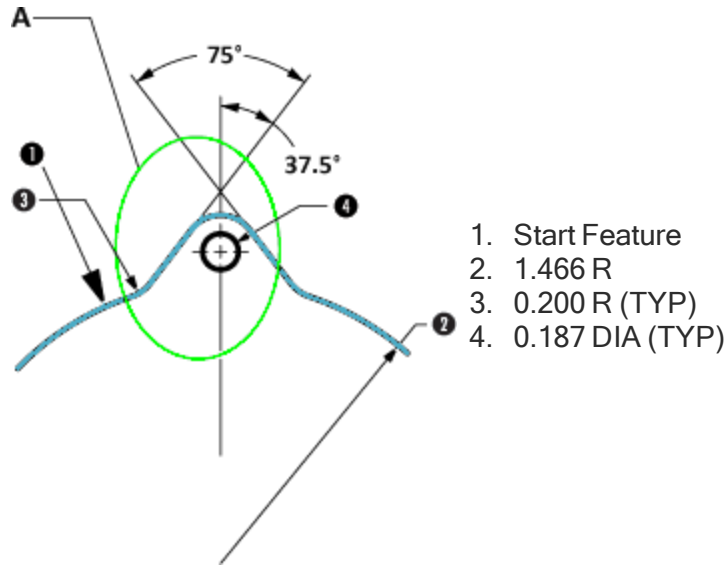
Because we only know the angle value of the following line the 0.2" radius must be defined as an arc so that the following line has the necessary tangency points for its calculation. An arc with only a radius value will be created tangent to the preceding and following features. These two features do not have to intersect, and the arc with a radius value can be used to calculate following features. If there was no radius between the first arc and the floating line, it would be necessary to enter an arc with a radius of 0 in the spreadsheet in order for the system to correctly calculate the tangency of the following line.
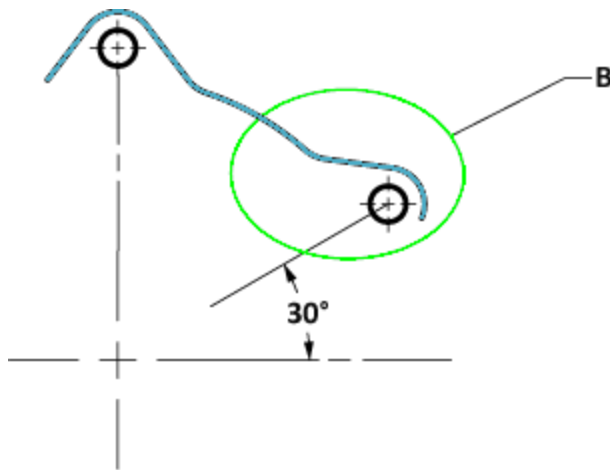
Area B is highlighted because the centerpoint for the arc must be created using the free-form CAD tools and then loaded into the Geometry Expert.

The only information that we have on that point is that it is rotated 60° to the right of the center point of the top circle. We will need to create this point using Modify > Duplicate And... > 2D Rotate. Note that the Modify menu is disabled when Geometry Expert is open.

In order to draw the center point of the top circle, you will need to extract the center point. Reload the Geometry Expert and load the shape. Interrogate the coordinates of the point as the centerpoint of the next arc.
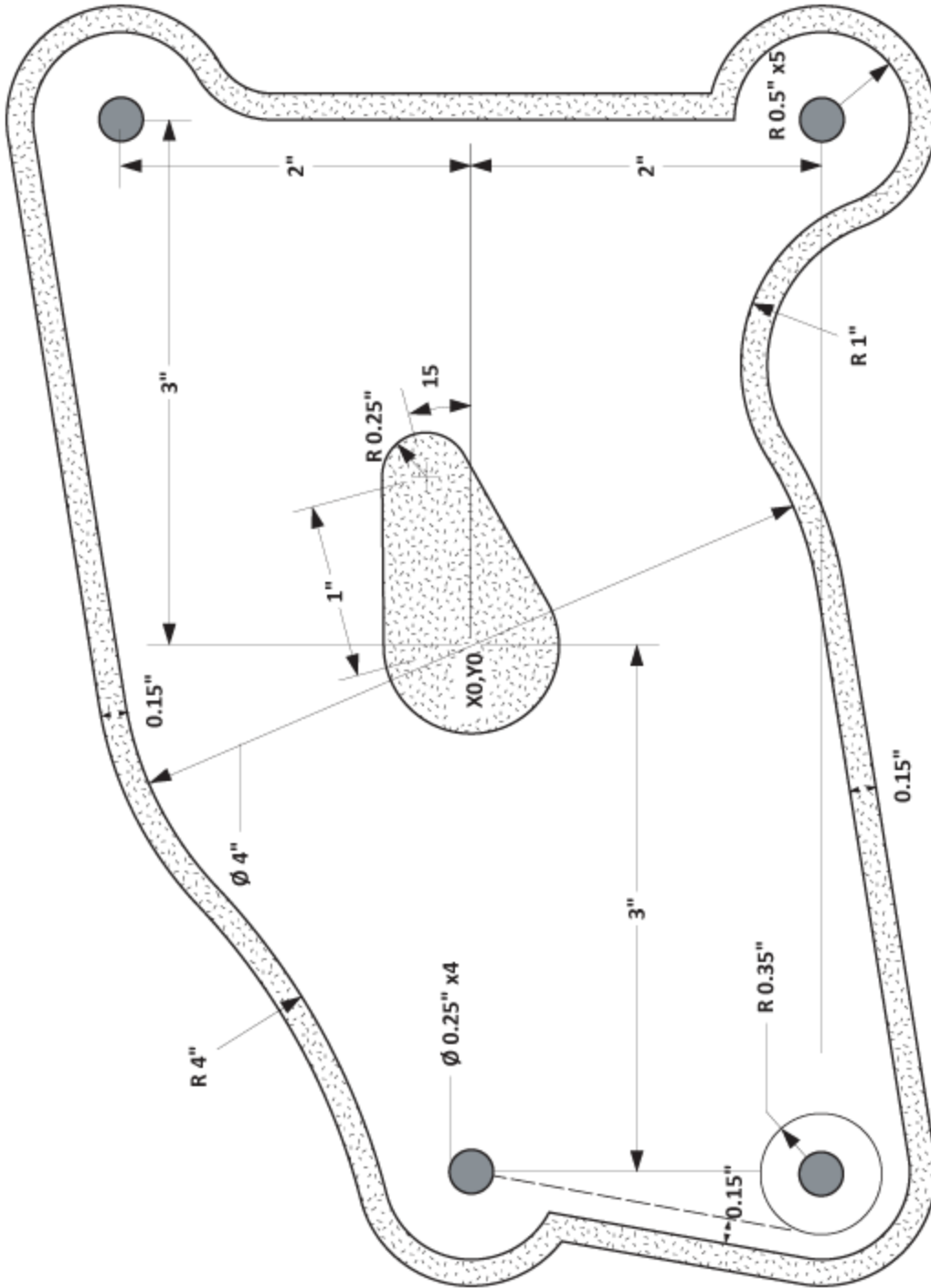
Inner Shapes

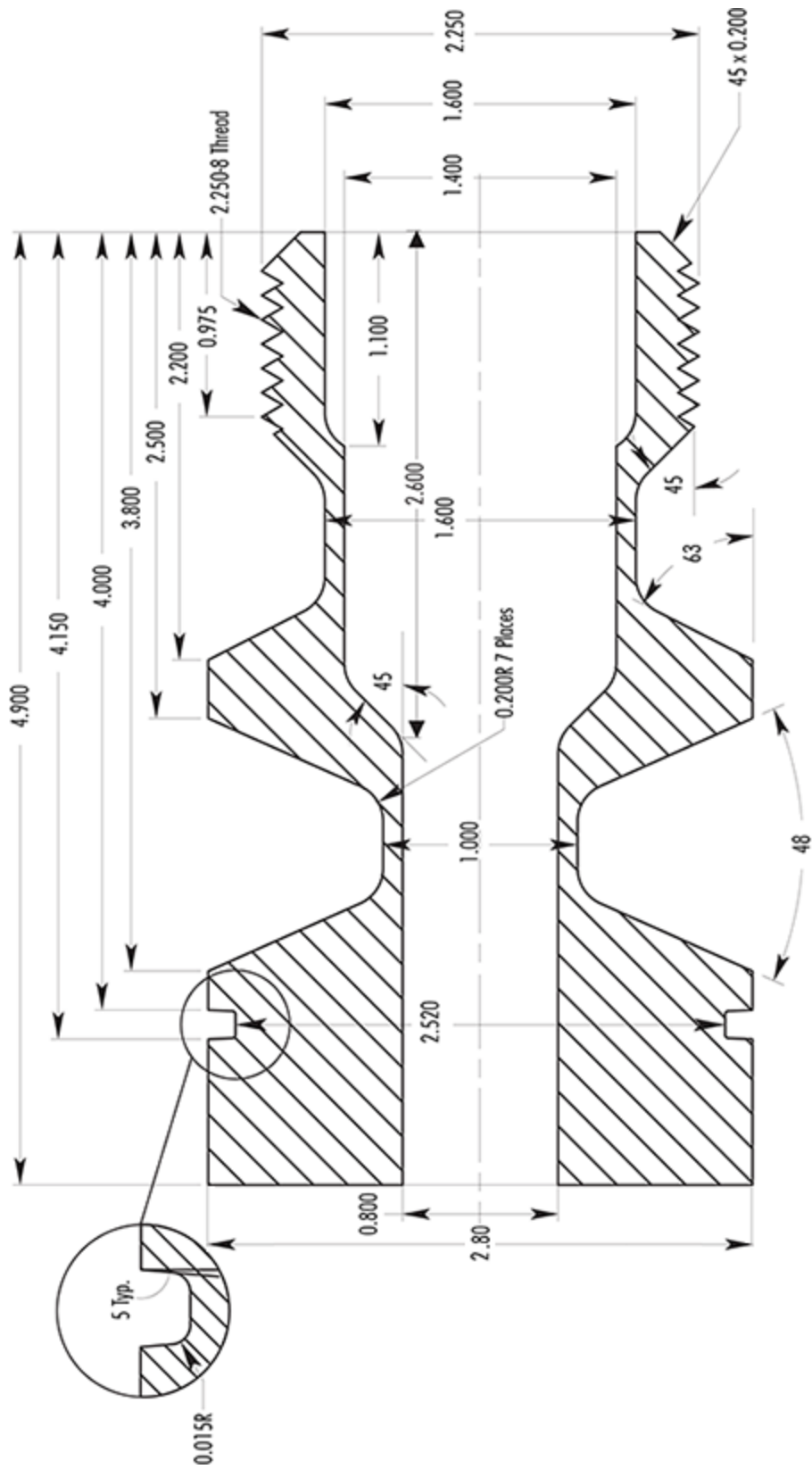Areas C and D are interrelated in that the line designated in area D must be tangent to the arc in area C.
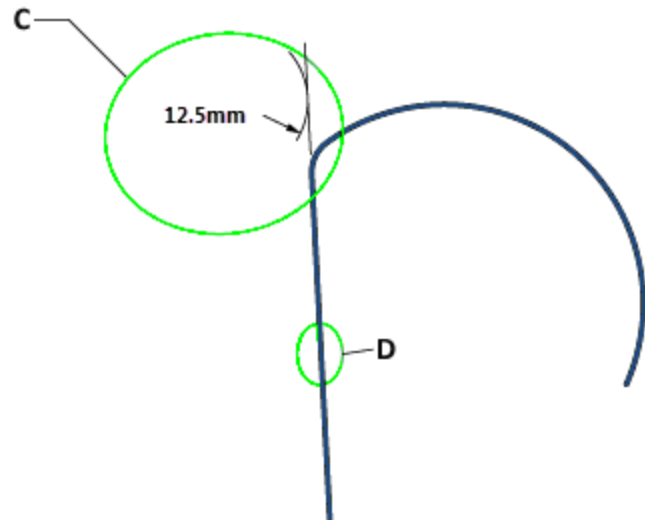
It is recommended that the larger inside pocket (of which line D is a component) be created using a combination of Geometry Expert and free-form CAD. The reason the entire shape cannot be created using Geometry Expert is that line D is dimensioned as being tangent to an arc that is not part of its shape, the circle contained in area C. It is best to create the shape in Geometry Expert starting with the 1.340 R arc, and proceeding around in a clockwise direction. When you get to line D, you will need to close the spreadsheet and use the free-form CAD tools.
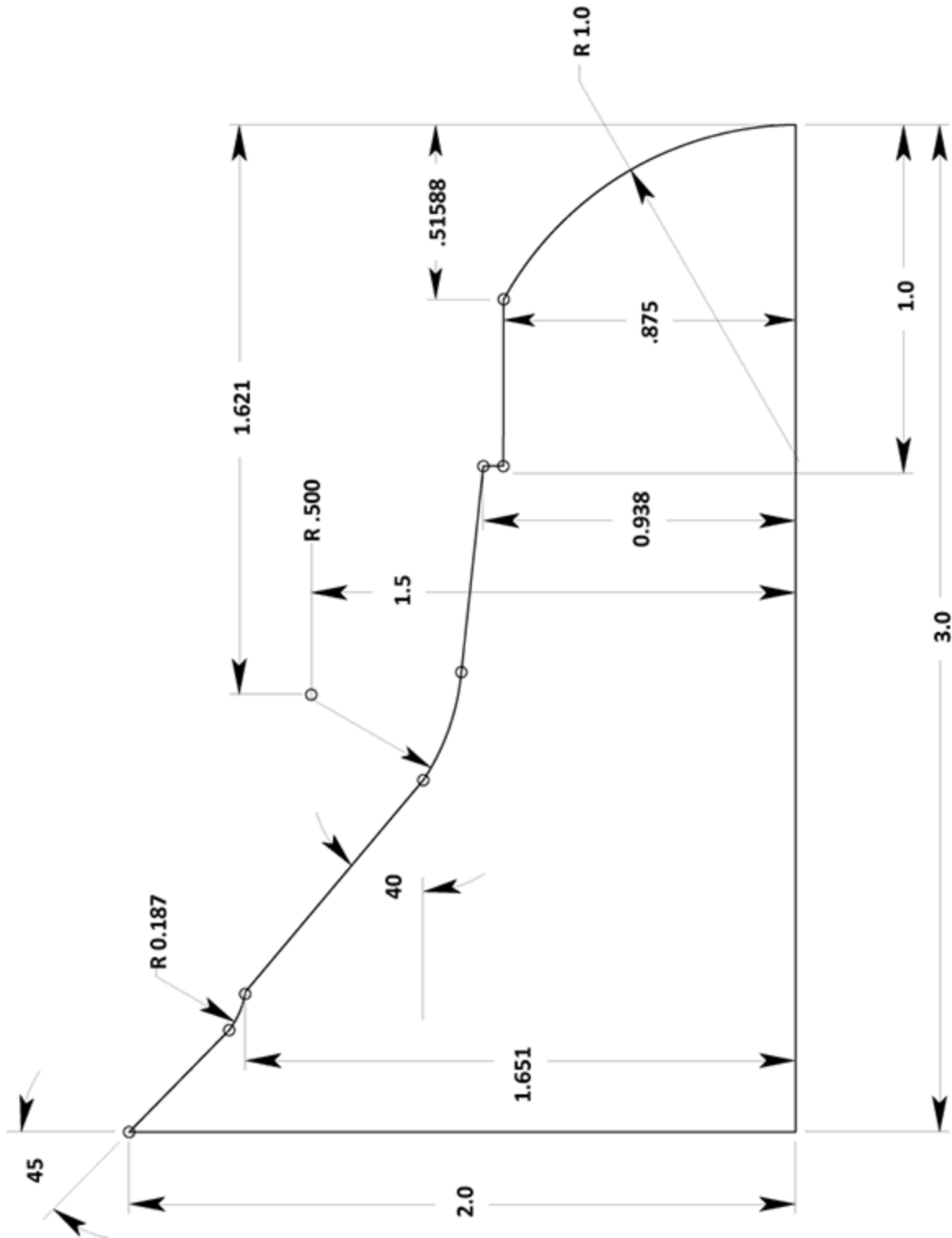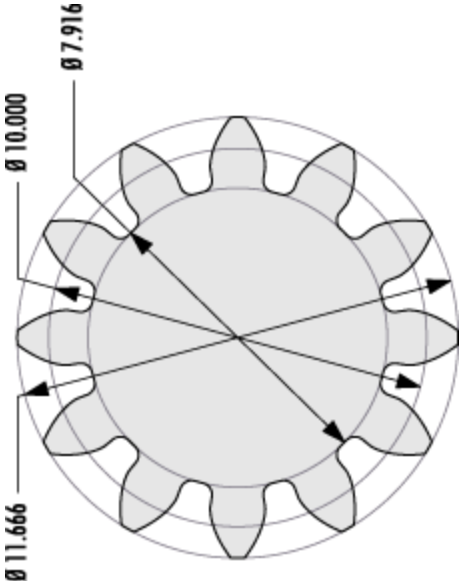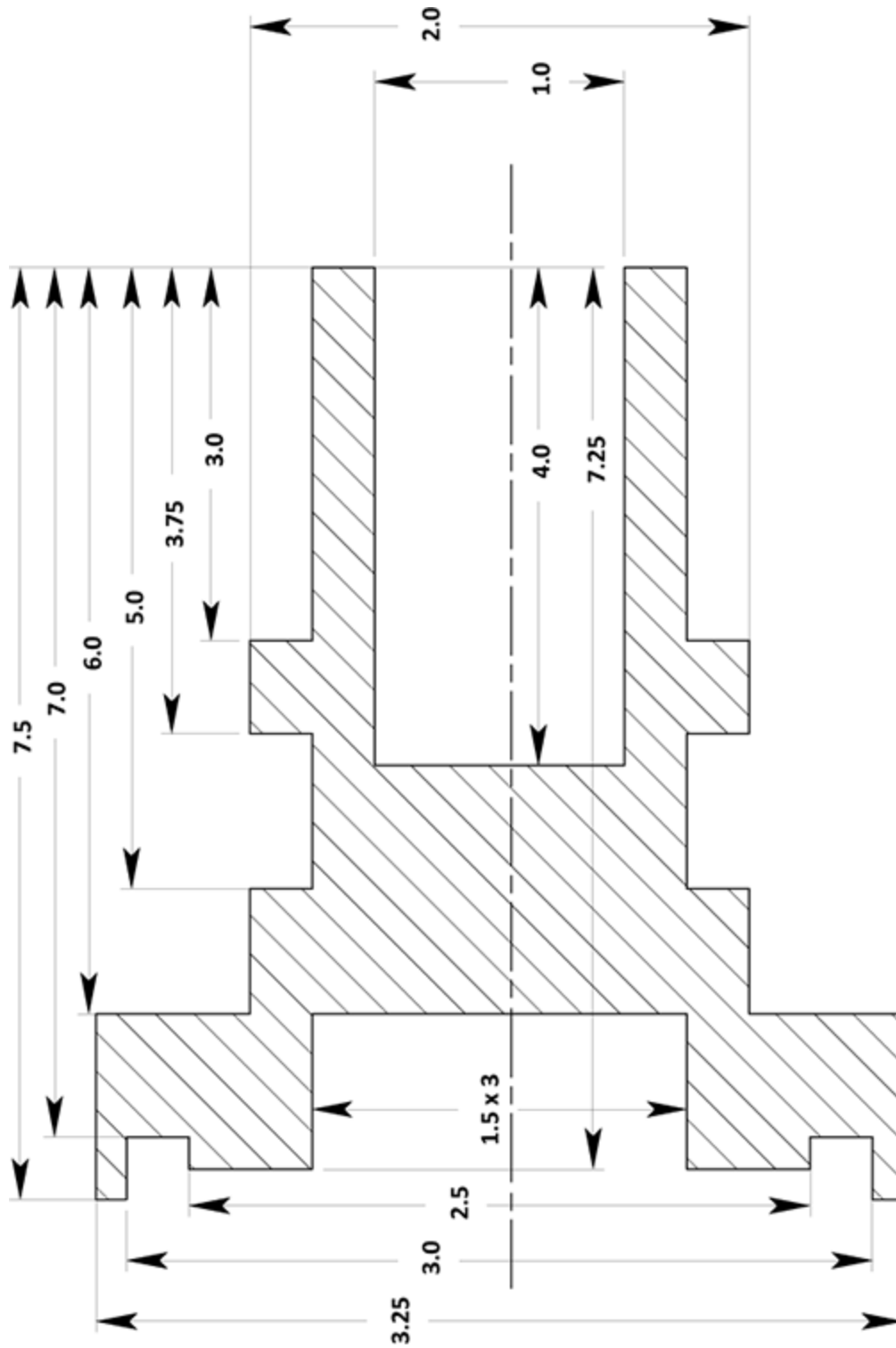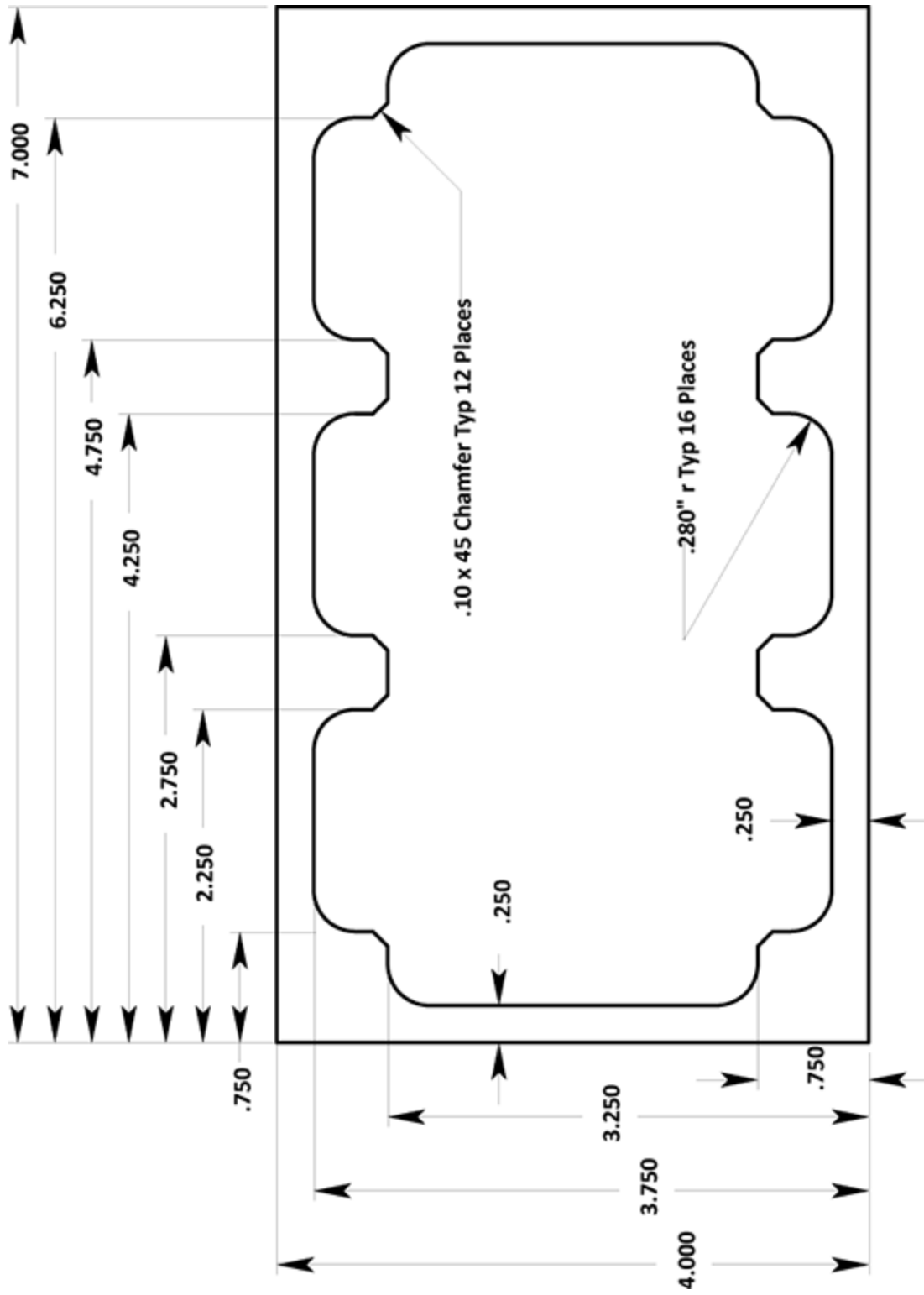
PART PRINTS

Mill Tutorial

Lathe Tutorial

Shuttle

Gear

Ø 7.916
Ø 10.000
Ø 11.666

Shaft

Chassis

.10 x 45 Chamfer Typ 12 Places

.280" r Typ 16 Places

7.000
6.250
4.750
4.250
2.750
2.250
.750

.250
.250
.250

3.250
3.750
4.000
.750

Doodle

Gear Housing