

Sketching Tips for New Users

By Neil Munro

One of the more challenging hurdles for new Autodesk Inventor™ users is mastering the concept of using constraints and dimensions to control the size and shape of feature sketches in the part-modeling environment. In this month's tutorial, we'll look at some techniques to help you break out of the "beginner" category. Although the tutorial is geared towards new users, we'll discuss a few techniques that might be new to intermediate-level users as well. If you are completely new to feature-based modeling, review the introductory [tutorial](#) on Autodesk Inventor before proceeding.

A couple of points from those earlier lessons bear repeating.

To fully define a part sketch, you must locate the sketch relative to existing geometry, which means you must locate the first sketch with respect to the origin reference geometry. A good technique is to use the Project Geometry tool to project the Origin Center Point into your sketch and then reference this point either with geometric constraints or dimensions to define the XY position of a sketch point (see Figure 1). Alternatively, apply a Fix constraint to a single point in the sketch. However, adding more than one Fix constraint defeats the idea of using dimensions and geometric rules to control the shape and size of the sketch, and thus the feature.

Note: Avoid using the projected origin point or a Fix constraint if the part is to be made adaptive.

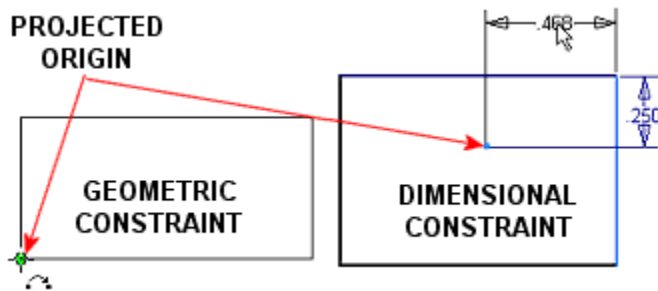


Figure 1: Sketch references to projected origin.

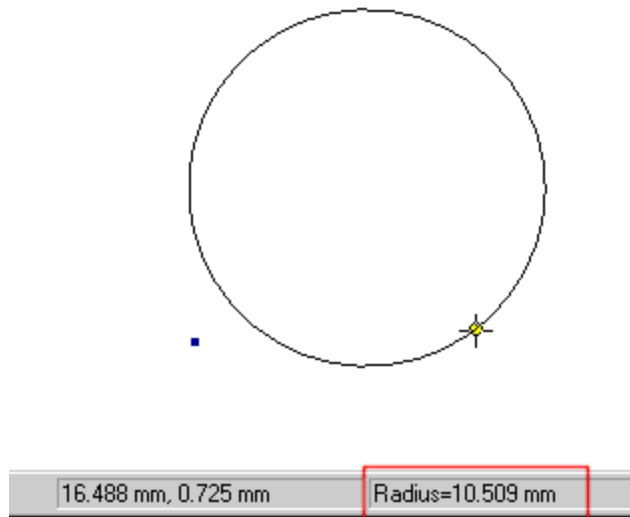




Figure 2: Projected center point and circle.

The first object that you sketch should be roughly to scale. A line, circle, or rectangle is usually that first object. With one entity roughly to scale, the approximate size and position of other objects are much easier to judge. Although precise sketching is not required, sketches that are close to the desired size react predictably when you finalize entity size and orientation.

OK, let's get on with it.

Starting a New Part

Certain geometric constraints (rules) are added as you create sketch geometry. Let's get started on a part and examine some of these rules.

1. Start a new part based on the Standard(mm).ipt template.
2. From the Sketch panel bar, click the Project Geometry tool .
3. Expand Origin in the browser and click Center Point.
4. From the Sketch panel bar, click the Center Point Circle tool .
5. Click near, but not on, the projected center point to define the center of the circle. Move the cursor and click again when the radius value shown in the status bar is approximately *10mm* (see Figure 2).

Note: You would typically center the circle on the projected center point. We've deliberately placed it away from the projected point so that we can demonstrate another sketch technique later in the exercise.

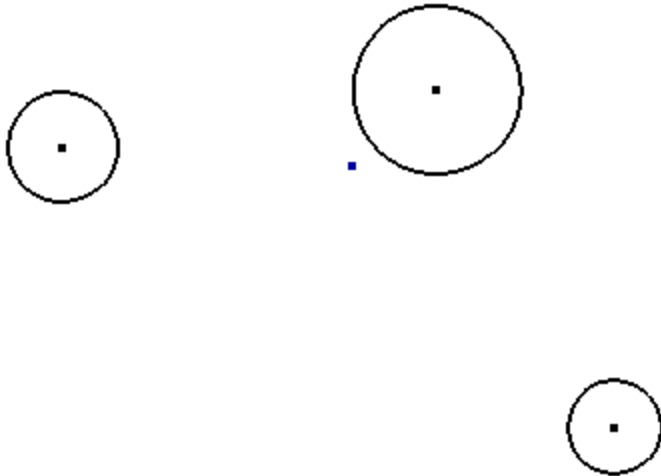


Figure 3: Additional circles.

6. Place two additional circles of smaller diameter as shown in Figure 3.

Inferring Constraints During Sketching

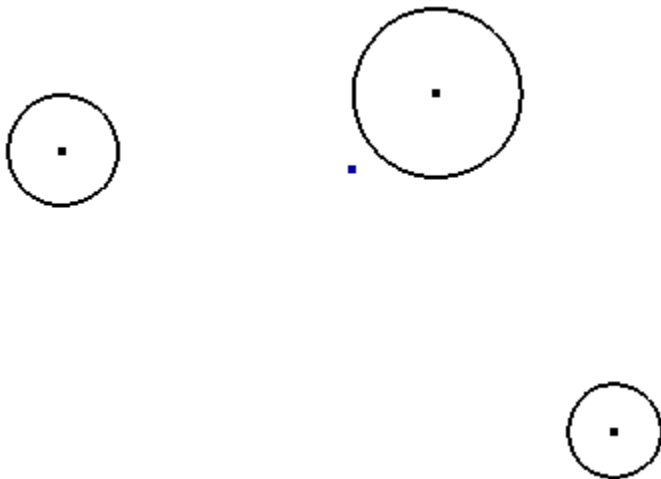


Figure 4: Click and drag tangencies.

In our design, the exterior edges of the part must be tangent to the sketched circles. Next, we'll use a technique to capture both tangencies at once.

1. Press the L key to start the Line tool.
2. Click the leftmost circle near the point shown in Figure 4 and drag towards the larger circle. You must click and drag, not click and release. Move the cursor over the larger circle until a second Tangent icon appears. Release the mouse button to complete the line.

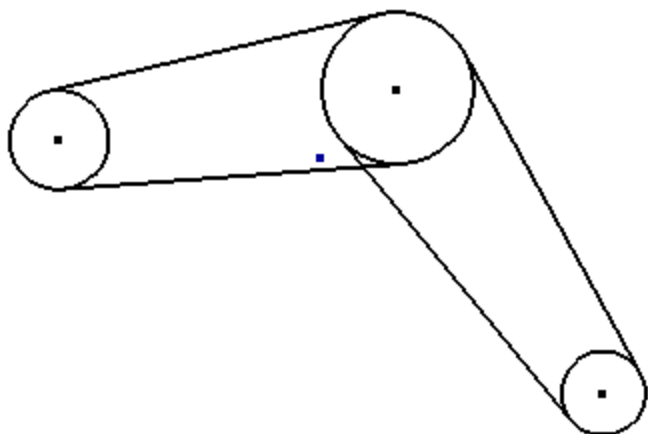


Figure 5: Additional tangent lines.

3. Use the same technique to add three additional tangent lines as shown in Figure 5.

Construction Geometry

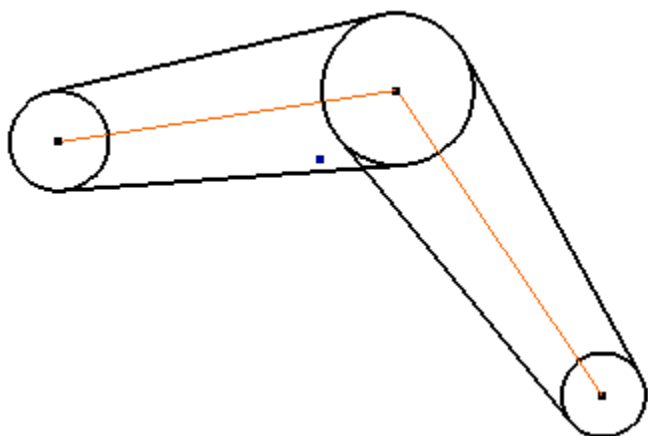


Figure 6: Construction lines.

You can use construction geometry to reduce the number of dimensions required to fully constrain a sketch and to improve its "readability." We'll add construction lines between the circle centers to demonstrate this.

1. Start the Line tool again if it is not currently active.
2. From the Style list on the Command bar, select Construction.
3. Sketch lines between the three circle center points as shown in Figure 6.

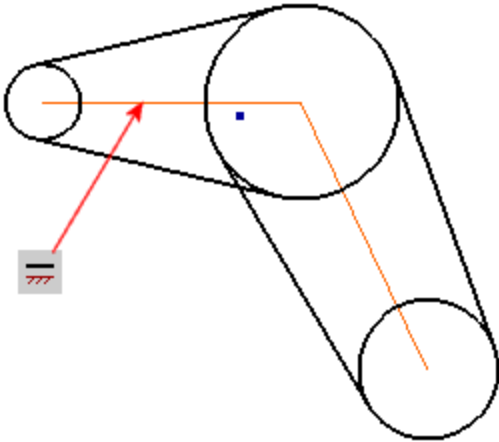



Figure 7: Horizontal constraint added.

4. From the Sketch panel bar, click the arrow next to the Perpendicular constraint tool, and then click the Horizontal constraint tool .
5. Click the construction line connecting the two leftmost circles. The sketch should now resemble the one shown in Figure 7.

Tip: Try drawing the construction lines first to act as a skeleton on which you can hang the remainder of the sketch.

You can also apply a horizontal constraint between points. Selecting the center points of the two circles would result in the same sketch state as applying the horizontal constraint to the line and would not require the construction line. However, the line imparts a visual clue that the two circle centers are aligned. In addition, the construction line enables us to dimension the angle between the two arms.

But first, let's add some additional design intent with geometric constraints. Note that the rightmost circle changed diameter when the horizontal constraint was applied.

6. From the Sketch panel bar, click the Equal constraint tool . Click the leftmost circle and then click the rightmost circle.

Use equal constraints whenever possible to capture design intent. You can equate circle or arc radii, or equate the length of two line segments. If the two circles in our sketch are independently dimensioned, the chance for error increases. If the diameter is changed later in the design process, both dimensions must be modified.

Adding Dimensions

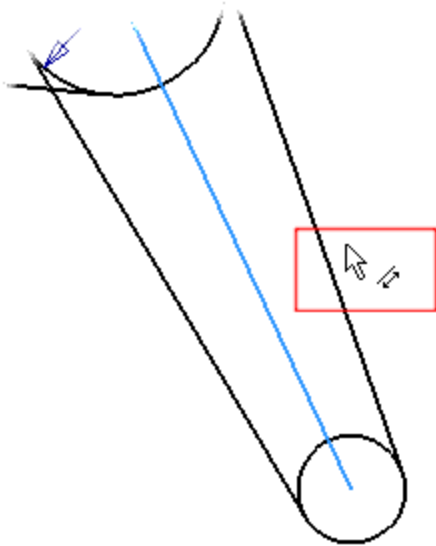


Figure 8: Aligned dimension symbol.

Another sketch technique is to create a mathematical relationship between dimension values.

1. Press the D key to start the General Dimension tool.
2. Dimension sketch entities in the following order:
 - The leftmost circle to a 10mm diameter.
 - The center circle to a 20mm diameter.
 - The horizontal construction line to 50mm.

Now dimension the other construction line.

3. Click the construction line connecting the two rightmost circles.

You get a preview of a vertical or horizontal dimension as you move the cursor to position the dimension. To place an aligned dimension, right-click and select Aligned, or use the following technique.

4. Move the cursor near the construction line until the cursor changes to an aligned symbol (see Figure 8). Click and then move the cursor to preview an aligned dimension. Click to place the dimension.

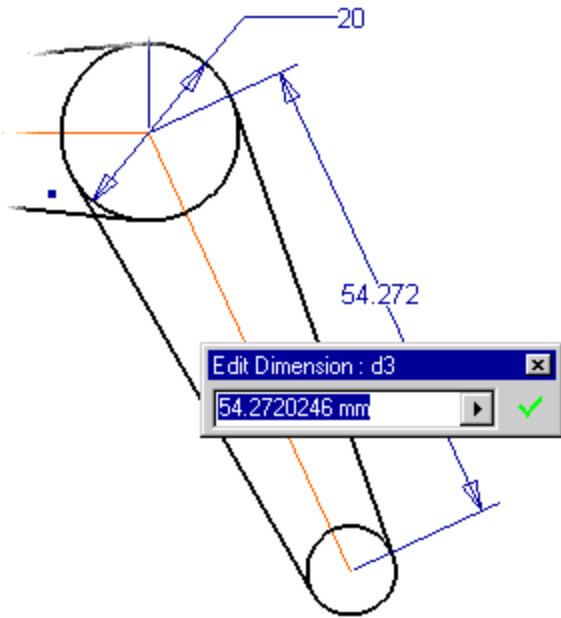


Figure 9: Edit dimension.

Tip: To edit dimensions as they are placed, right-click before placing the dimension location (after selecting sketch entities), and select Edit Dimension from the shortcut menu. The behavior toggles off and on each time you repeat this action.

5. If the Edit Dimension edit box is not shown, click the dimension text to edit the dimension value (see Figure 9).

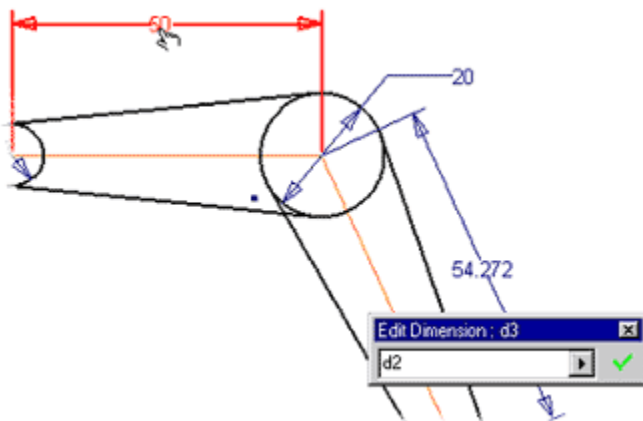


Figure 10: Creating a relationship between dimensions.

6. Highlight the dimension value in the Edit Dimension box.

7. Click the 50mm dimension of the horizontal construction line as shown in Figure 10.

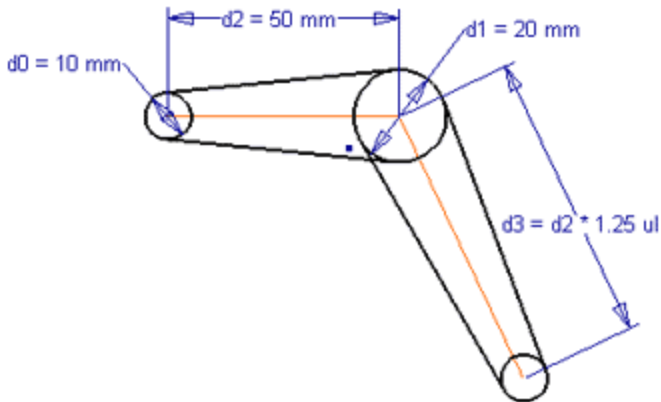


Figure 11: Dimensions shown as expressions.

Autodesk Inventor software tracks all sketch dimensions and other values such as extrusion depth as a series of parameters. The title of the Edit Dimension box indicates you are editing the value of the model parameter named $d3$. You just added a relationship between the current dimension ($d3$) and the selected dimension ($d2$). Let's complete the equation by including a ratio between the two lengths.

8. In the Edit Dimension box, change the equation to $d2 * 1.25$. The equation is evaluated after each keystroke and is shown with red text when the expression cannot be evaluated. Valid expressions are shown in black text.

9. Click the green check mark in the Edit Dimension box to accept the equation.

10. Right-click and select Done.

11. Right-click any one of the dimensions and select Show Expression. The dimensions should match those shown in Figure 11.

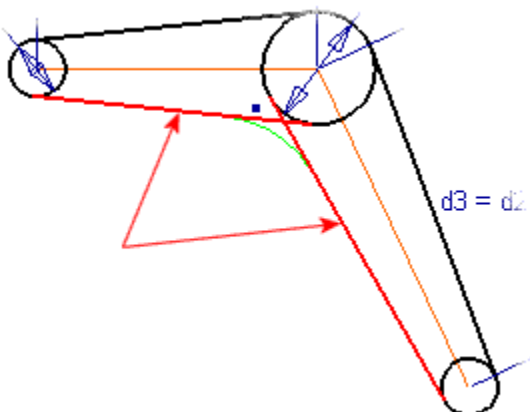



Figure 12: Fillet selections.

Note: You must be running Autodesk Inventor 5 to show dimensions as expressions.

Sketch Fillets

A common modeling decision is whether to include a fillet in the sketch or to add the fillet later as a feature. There is no hard-and-fast rule here, but it is good practice to add cosmetic or small fillets as features near the end of the modeling process. Our part requires a large fillet at the intersection of the two legs. We'll add the radius as a sketch fillet.

1. From the Sketch panel, click the Fillet tool .
2. Enter *20mm* in the Fillet Radius edit box.
3. Click the two lines shown in Figure 12.

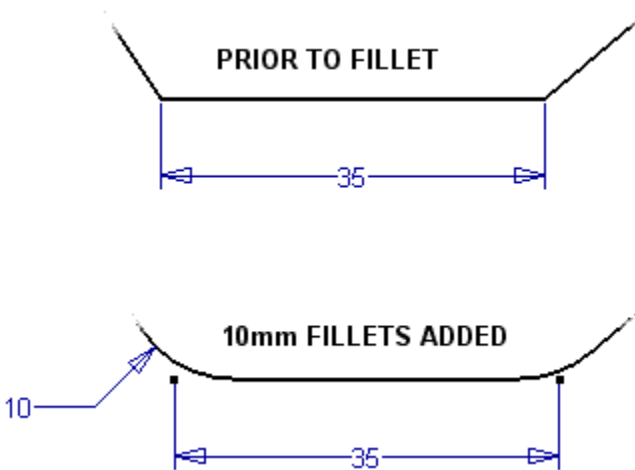


Figure 13: Sketch fillet retains dimensions and constraints.

Note that two sketch points have been added at the original intersection points between the lines and the center circle. Each point is coincident to the shortened line and the circle. Also, the tangent constraints between the lines and the circle are retained. In general, the Fillet tool retains any existing geometric or dimensional relationships. For example, if you dimension the length of a line, the dimension is retained to a theoretical intersection (see Figure 13).

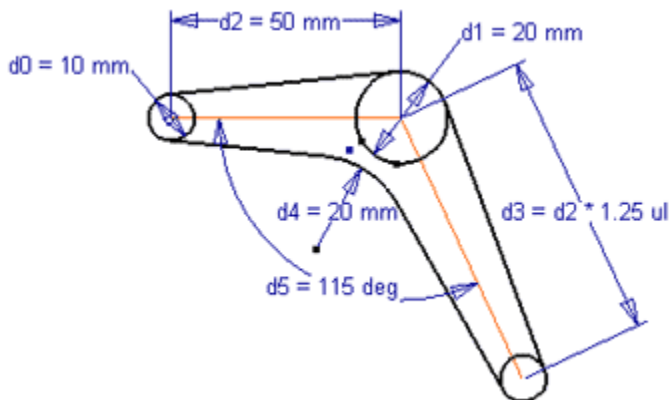


Figure 14: Angle dimension added.

4. Complete the dimensions by adding a 115-degree angle dimension between the two construction lines (see Figure 14).

Adding Constraints by Dragging

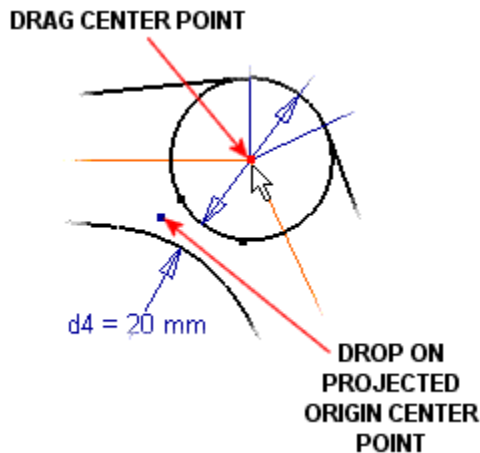


Figure 15: Drag-and-drop point and target.

Sketch entities that are fully defined as to their position are displayed in a different color than positionally underconstrained geometry. To determine the required dimensions or constraints for a sketch entity, click and drag the entity or its endpoints. The geometry will only move in unconstrained directions, giving you visual clues as to which additional constraints and dimensions are required.

All sketch geometry in our current sketch is fully constrained, but the location of the sketch on the sketch plane is still missing. We'll complete this by anchoring the sketch to the projected origin center point.

1. Move the cursor over the center point of the center circle; it will highlight in red.
2. Click and drag the sketch. In Autodesk Inventor 5, the sketch dimensions move with the sketch.
3. Drag the cursor over the projected center point. A green dot indicates that a coincident constraint will be applied between the two points.
4. Release the mouse button to apply the constraint (see Figure 15).

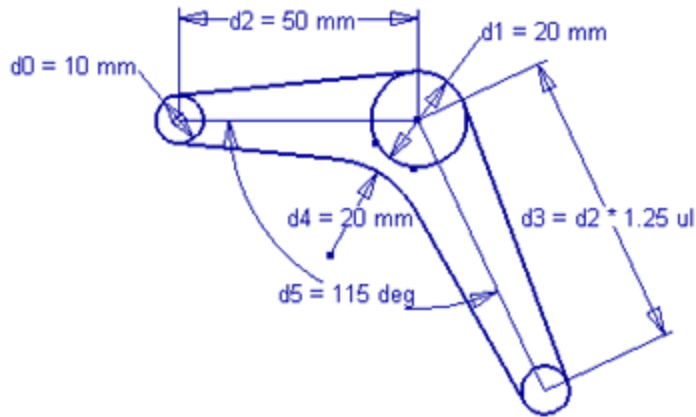



Figure 16: Fully defined sketch.

The sketch is now completely constrained (see Figure 16). Unfortunately, creating the coincident constraint using the drag method doesn't always update the color of the sketch entities to indicate they are fully constrained. Click the Update tool to see the fully defined colors. I'm using the Presentation color scheme; fully defined sketch entities are shown as dark blue; underconstrained sketch entities (Normal linetype) are black. Alternatively, you can use the Coincident constraint tool  to add the constraint (this would update the colors immediately).

Shared Sketches

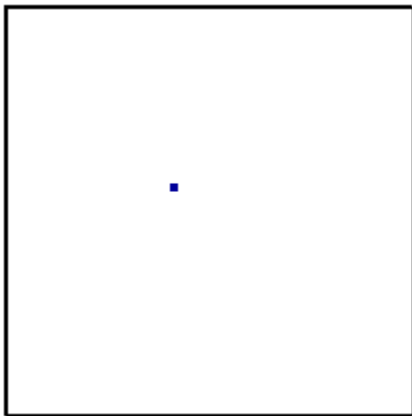


Figure 17: Sketched rectangle.


Part models contain a series of features that build on existing geometry. In some parts, you might project the same or similar geometry into the different sketches to help create closed profile(s). Autodesk Inventor enables you to share a consumed sketch (a sketch that has been used to create a part feature) and make it available as the basis for any number of features. You can use this technique to improve the efficiency of your part designs. By containing the design intent within a single sketch, rather than depending on projected geometry in multiple sketches, you can reduce the chance of errors in the modeling process.

Note: Shared sketches cannot be used in all modeling situations. Use this exercise as an example of how you might use shared sketches.

1. Start a new part using the Standard(mm).ipt template.

2. From the Sketch panel bar, click the Project Geometry tool , and project the origin center point into the sketch.

Tip: If you consistently project the same origin geometry into the first sketch, edit a template file and project that geometry into a sketch. Parts based on this template will automatically contain the projected geometry in the initial sketch.

3. From the Sketch panel bar, click the Two-Point Rectangle tool .

4. Sketch a rectangle approximately 60mm square, centered roughly on the projected origin (see Figure 17).

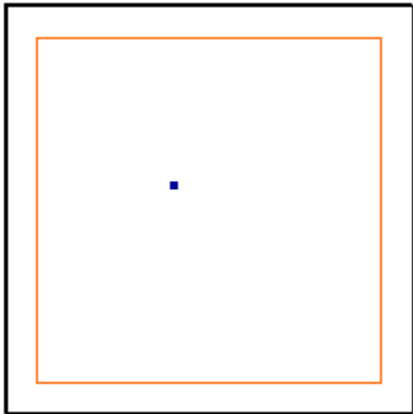


Figure 18: Offset construction rectangle.

5. From the Sketch panel bar, click the Offset tool .

6. Change the line style to Construction.

7. Click an edge of the rectangle and then move the cursor inside the rectangle. Click again to offset the rectangle loop as shown in Figure 18.

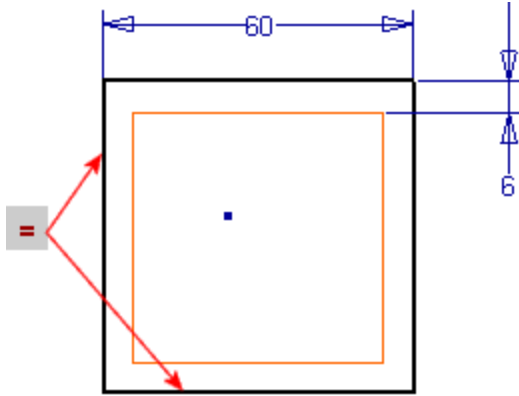


Figure 19: Dimensions and equal constraint.

The Offset tool generates appropriate constraints to locate all offset edges with a single "offset" dimension.

8. Press the D key to start the General Dimension tool.
9. Place a 6mm dimension between the two edges shown in Figure 19. Add a 60mm dimension to one edge of the outer rectangle.
10. Add an Equal constraint between two adjacent edges of the outer rectangle.

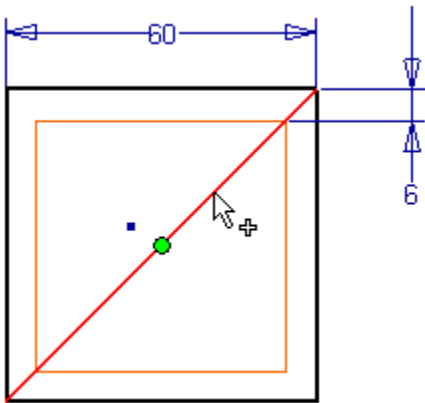



Figure 20: Sketch point at midpoint of line.

11. Draw a construction line between the lower-left and upper-right corner of the outer rectangle. Ensure the endpoints of the line are coincident to the rectangle corner points.
12. From the Sketch panel bar, click the Point, Hole Center tool .
13. Select Sketch Point from the Style list.
14. Right-click in the graphics screen and select Midpoint from the shortcut menu.

15. Click the diagonal construction line as shown in Figure 20.
16. Right-click and select Done from the shortcut menu.
17. Click and drag the point created in the previous step. Drop it on the projected center point. Click the Update tool. All sketch geometry is fully defined.

Additional Sketch Geometry

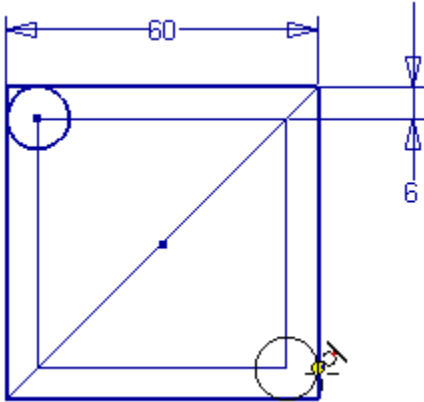



Figure 21: Two circles added to sketch.

We'll complete the rectangle sketch by adding circles that we'll use to create additional features.

1. From the Sketch panel bar, click the Center Point Circle tool .
2. Select Normal from the Style list.
3. Sketch two circles as shown in Figure 21. The center point of each circle is coincident to a corner of the construction rectangle, and the circle is tangent to the outer rectangle.

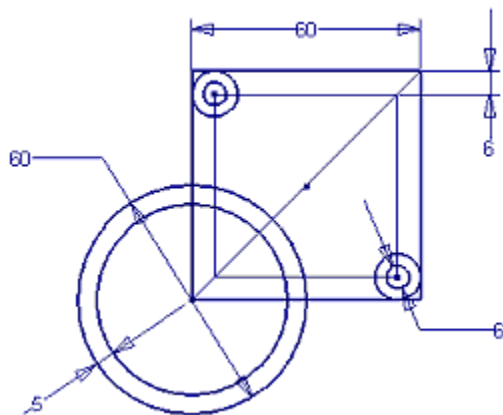


Figure 22: Completed sketch.

4. Sketch a circle centered on the lower-left corner of the outer rectangle (see Figure 22).
5. Use the Offset tool to offset the two small circles and the large circle as shown in Figure 22.
6. Add an Equal constraint between the two small circles. Add a 6mm diameter dimension to one of the small circles.
7. Add a 50mm dimension to the large circle.
8. Add a 5mm dimension between the two large circles. (Click one circle, then click the other circle. Click again to place the dimension).

Multiple Profile Feature

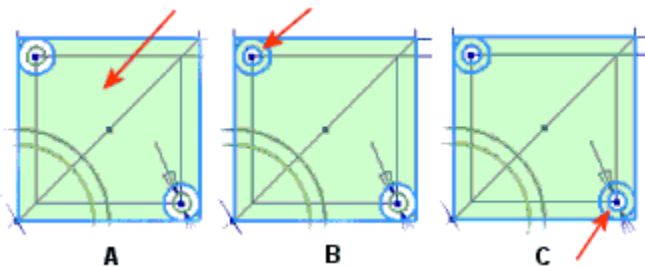


Figure 23: Three profile selections.

We now have a sketch that can be used to create a number of features. Autodesk Inventor software enables you to select multiple overlapping profiles from a sketch during feature creation.

1. Press the E key to display the Extrude dialog box.
2. Click inside the rectangle. The profile shown in image A of Figure 23 is selected.
3. Click between the two small circles in the top-left corner (see Figure 23-B).
4. Click between the other two small circles (see Figure 23-C).



Figure 24: Shared sketch in browser.

5. Enter *2mm* for the Extrusion Depth.

6. Click the Extrude-from-Midplane Extents icon .

7. Click OK.

Sharing the Sketch

We'll share the sketch so it can be used for additional extrusions.

1. Expand Extrusion1 in the browser.

2. Right-click Sketch1 and select Share Sketch from the shortcut menu.

3. The sketch is now listed above Extrusion1 in the browser, and the sketch is visible (see Figure 24). The Sketch icon indicates it is shared.

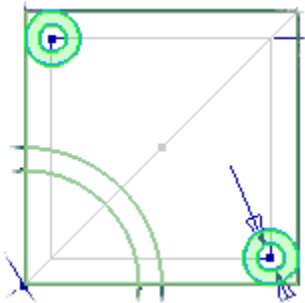



Figure 25: Profile selection for multiple features.

4. From the Standard toolbar, select the Wireframe Display mode .

5. Press the E key to display the Extrude dialog box.

6. Select the two profiles between the small circles as shown in Figure 25.

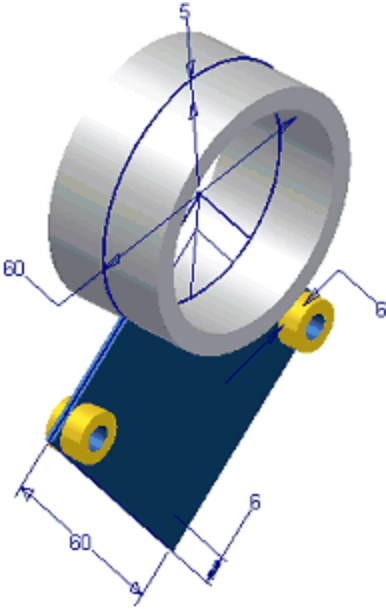
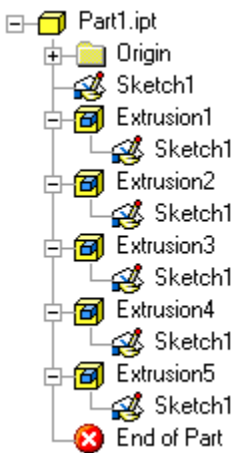


Figure 26: Completed part.



7. Enter *3mm* for the Extrusion Depth.


8. Do not change the default extrusion direction.

9. Click OK.

10. Press the E key again to restart the Extrude tool.

11. Select the same two profiles between the small circles as shown in Figure 25.

12. Enter *6mm* for the Extrusion Depth.

13. Click the Flip Extents Direction icon .

14. Click OK.

More Extrusions from the Shared Sketch

We'll complete the part by creating a pipe from the larger circles.

1. Extrude the profiles of both large circles 30mm using a midplane extrusion.
2. Finally, extrude the profile inside the inner large circle. Use the Cut option, set the Extents to All, and select the midplane extents option. The part should match the one shown in Figure 26.
3. Right-click Sketch1 in the browser and click Visibility to turn off the visibility of the sketch.

Note: Some features have been colored for clarity in Figure 26.

Summary

The Autodesk Inventor sketch environment has a rich set of tools. Learning when and where to use particular tools and techniques will enable you to create robust, easy-to-edit part models. Construction geometry and shared sketches are two of the paving stones on your road to becoming an Autodesk Inventor expert.

If you have a particular topic that you think would make a good tutorial subject, email me at neil_munro@bcit.ca.