

Sketches

Introduction

Sketching is one of the most powerful tools available in UG. The basic idea of sketching is to create a 2D shape that is controlled by numerical values and geometric constraint rules. A major benefit of using sketches is that it enables you to capture your design idea quickly, without being overly concerned about the detail dimensions.

The sketch process involves creating a new sketch by choosing “*Insert / Sketch / Create*”. Sketches are *always* defined using a sketch plane, and a reference direction. Once in sketch mode, you create entities that capture the shape of a geometric feature. In Figure 1, you will find many entities that are commonly used to compose a sketch.

Once the sketch curves are drawn, and they may further defined using manually applied constraints and dimensions. You can change these dimensions and constraints until you get the desired geometry.

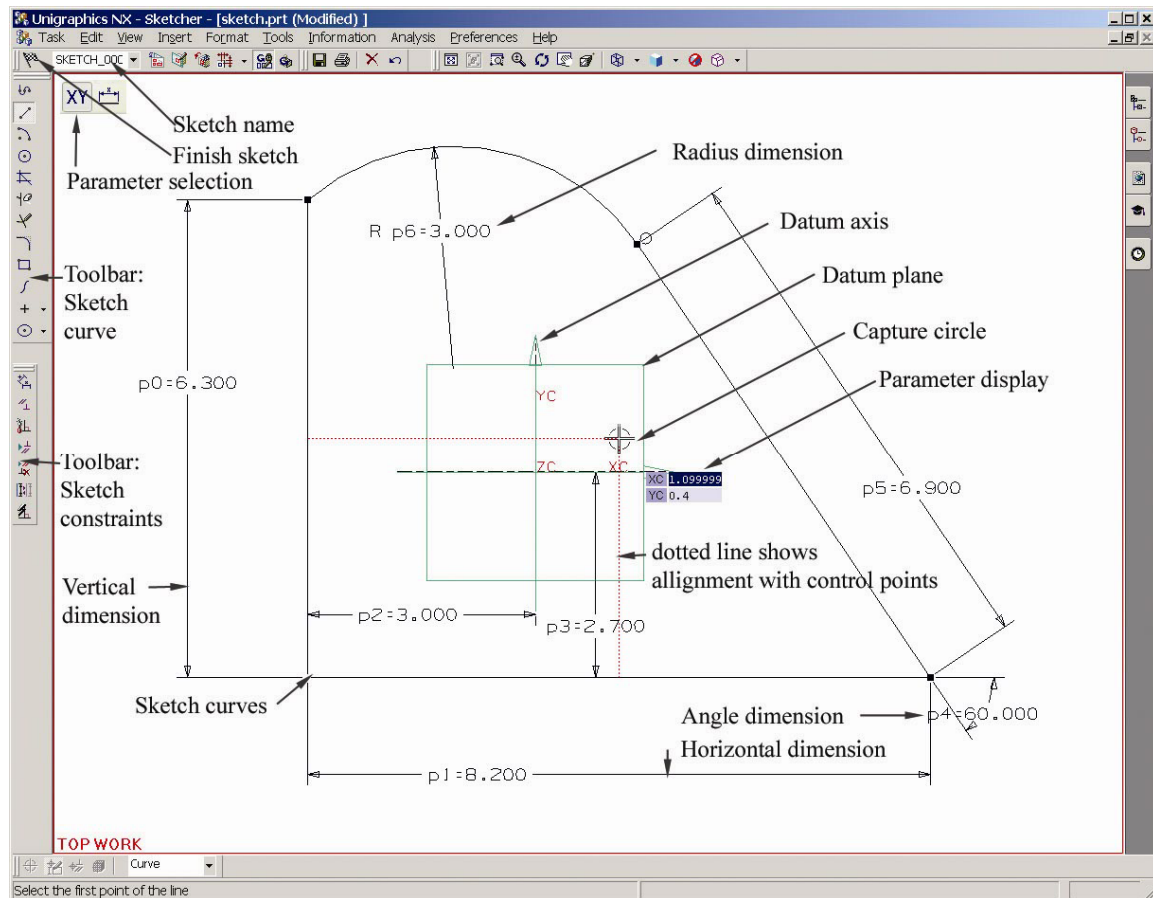


Figure 1

The final step of the sketch process is to finish the sketch and exit sketch mode. A sketch may then be used to define subsequent geometric features.

When you choose **Insert / Sketch** in a part file that contains no geometry, a datum plane and two datum axes will appear. The sketch will be located on these datum entities. All

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sketches must be placed on a datum plane or a planar part surface, and a designated reference direction.

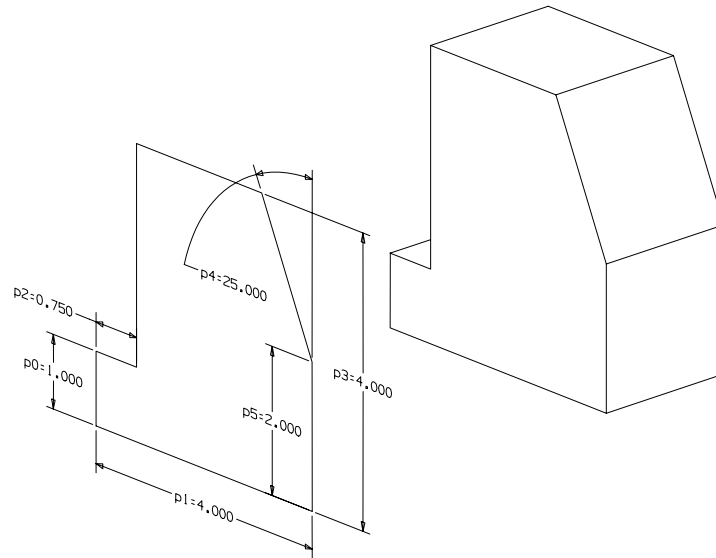


Figure 2

Figure 2 shows how a sketch can be used to create an extruded solid. The shape of the solid will change as you modify the sketch, as shown in Figure 3.

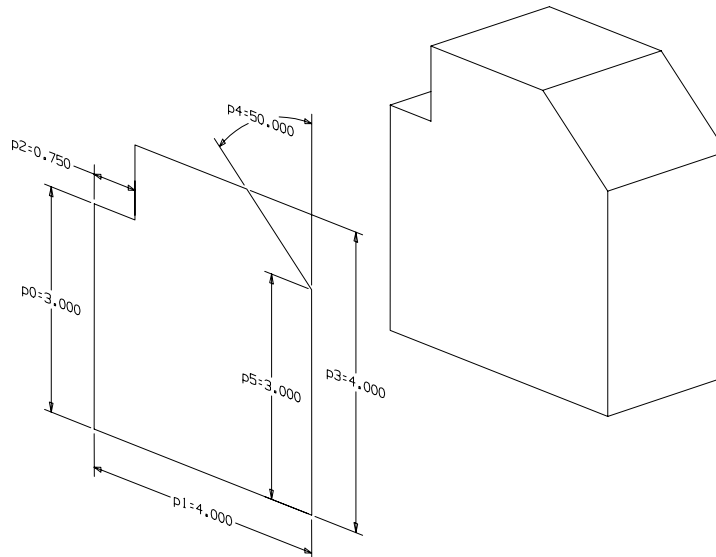


Figure 3

Topic: Creating a Sketch

To create a Sketch, choose **Insert / Sketch**. If you are working in a part file that contains no solid geometry, when you do this, UG automatically creates a datum plane and two axes that define the sketch plane. The datum plane is created on the XC-YC plane of the WCS. The datum axes are infinitely long lines that are coincident with the X and Y axes. The datum entities are displayed in UG as shown in Figure 4.

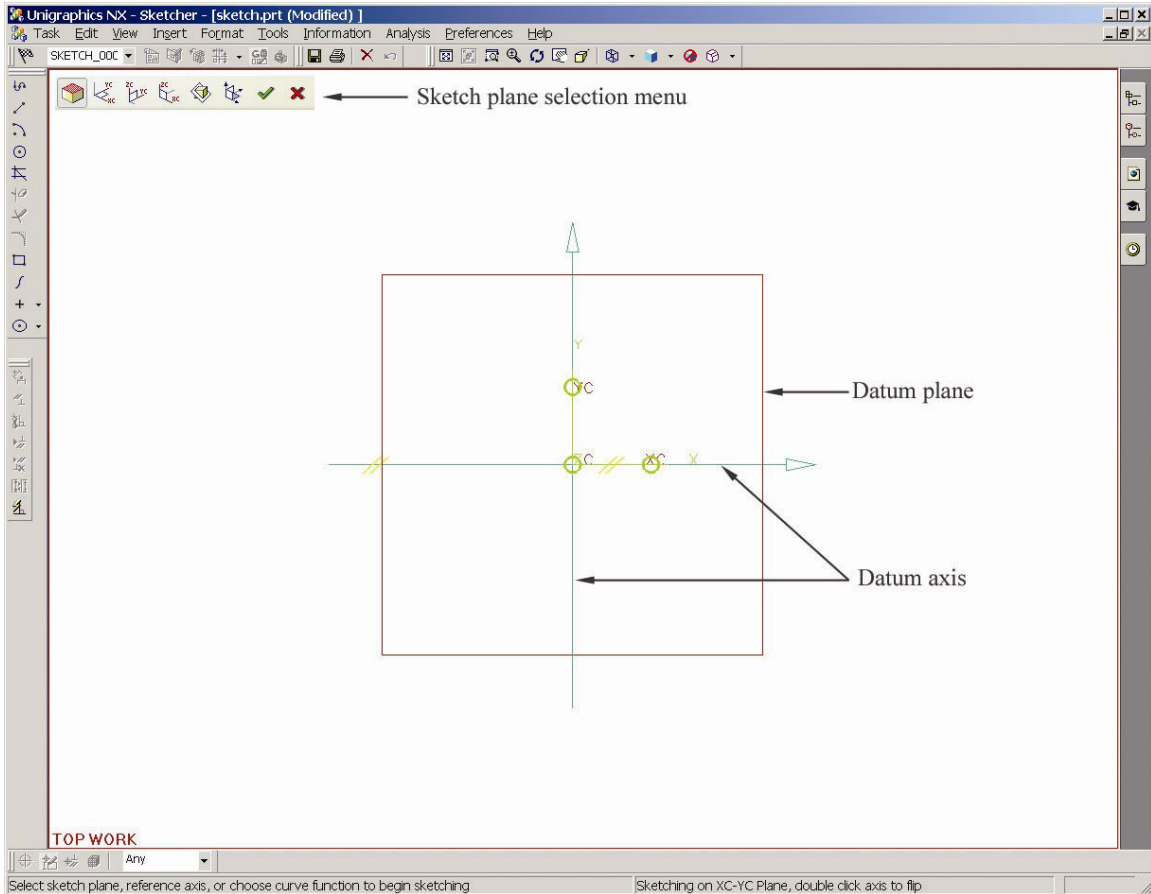


Figure 4

The next step in the sketching process is to create curves that capture the desired shape. Figure 5 illustrates some of the tools available for sketching geometry. Note that you can also use the Trim and Extend Sketcher functions to edit sketch curves.




Figure 5

The next step in the sketching process is to dimension and constrain the sketch. Figure 6 illustrates some commonly used tools for dimensioning and constraining sketch geometry.



Figure 6

The final step in the sketching process is to choose the “Finish Sketch” icon,  which is located on the upper left corner of the Graphics Window when you are in Sketch Mode. This tells UG that you are finished sketching, and are ready for bigger and better things!

To illustrate the Sketching Process, consider Figure 7, which shows a typical beverage pitcher.



Figure 7

Figure 8 illustrates the initial steps of the Sketching Process. The datum entities and the shape of the desired geometry have been created.

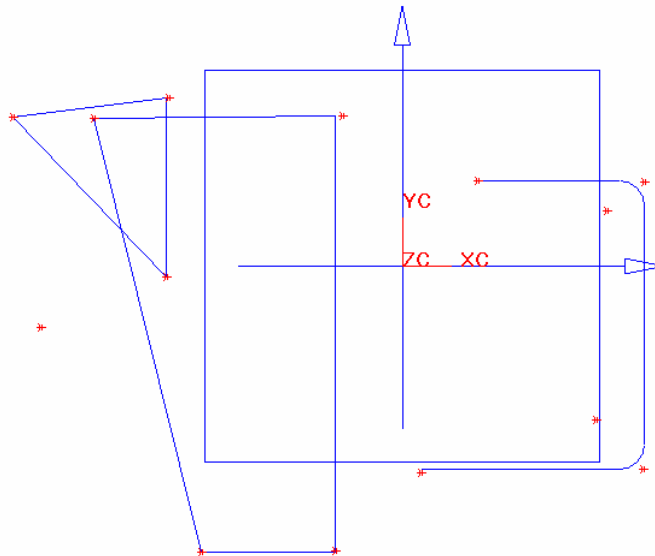


Figure 8

Once the basic shape is sketched, you add the exact dimensions and rules. In Figure 9, you can see that the pitcher has been sized and all the geometry is ready to create the solid model.

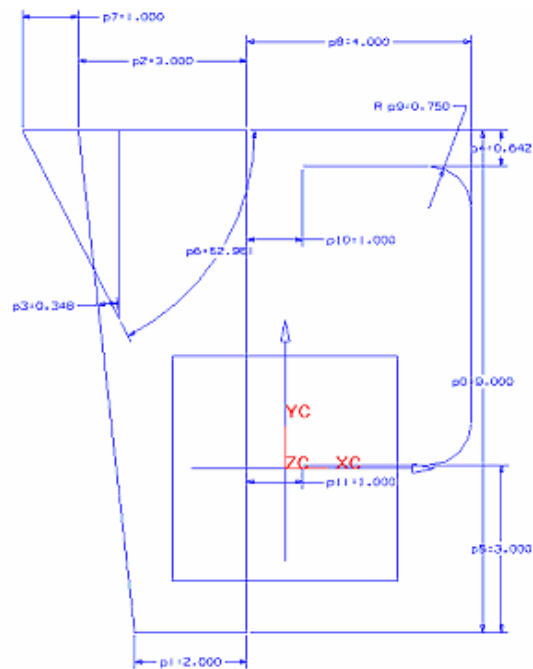


Figure 9

Note that in Figure 9 there are only four types of dimensions displayed: Horizontal, Vertical, Perpendicular, and Angular.

Figure 10 illustrates how the sketch geometry may be used to create solid geometry.

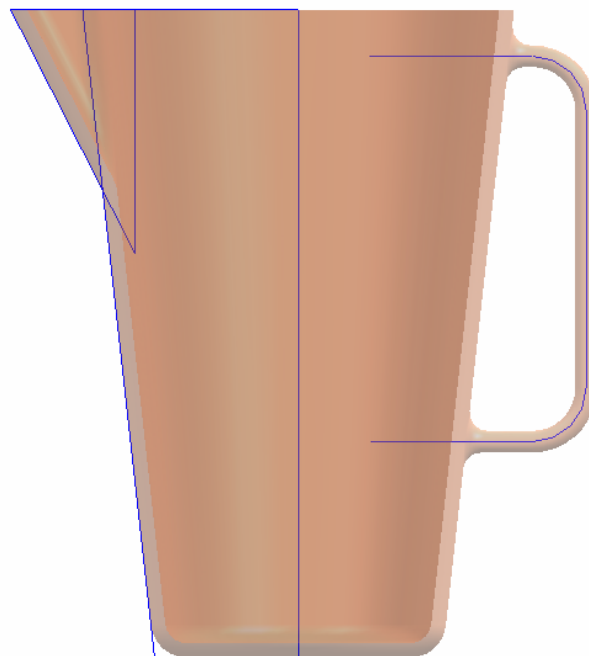


Figure 10

Detail: Adding Dimensions to a Sketch

Seven dimension types may be placed on sketch entities, using the “Dimensions” dialog box shown in Figure 11.

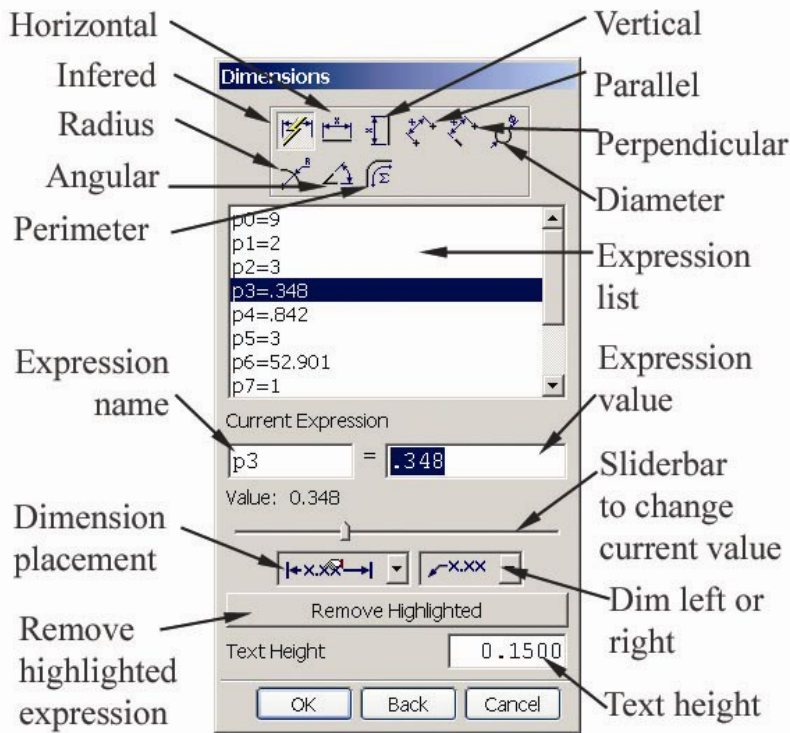


Figure 11

By default, UG uses the “Inferred” dimension tool. To use this option, select the entity or entities that you want to dimension, and UG determines the dimension type automatically, based upon the cursor position.

PROJECT:

- Create a new part file
- Select **Insert / Sketch**

A datum plane coincident with the x-y plane has been created along with horizontal and vertical datum axes, as shown in Figure 12.

- Select the ‘Profile’ icon 

- Using left hand mouse button clicks create the shape shown in Figure 12.

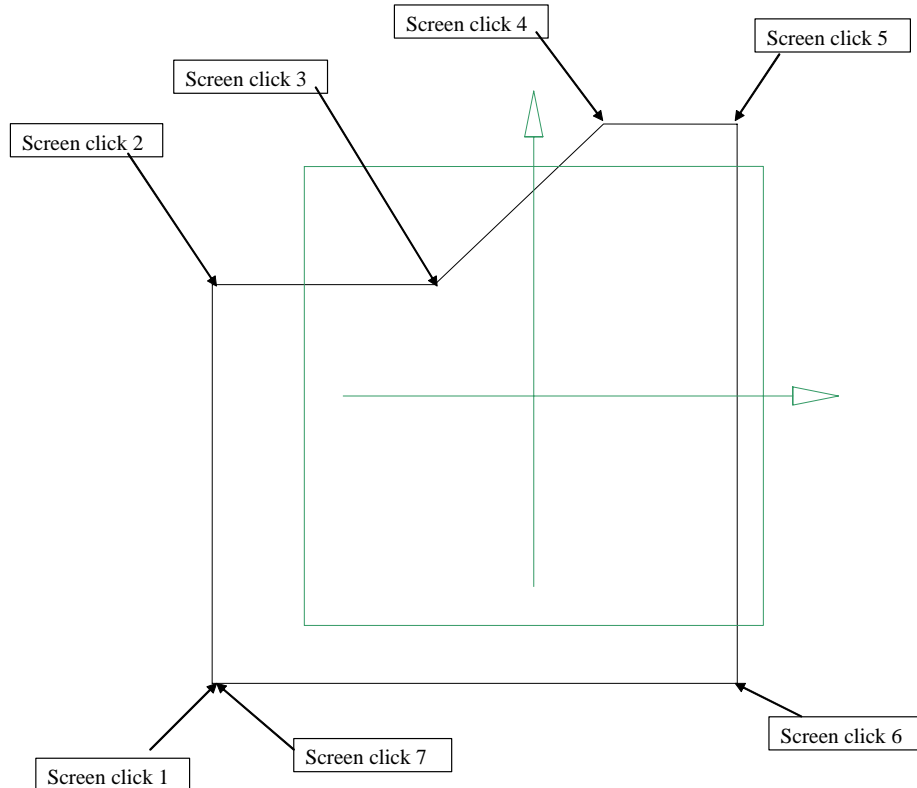




Figure 12

- You will notice that as you select screen locations, lines will be drawn between each sequential click. Make sure that when you draw the last line, you place the capture circle on top of the very first location. This will create a closed loop of curves as shown below.
- Select the 'Dimensions' icon  from the Sketch Constraints Toolbar.
- Select the 'Inferred' icon  from the 'Dimensions' dialog box.

To use the "Inferred" tool, click once on the entity that you want to dimension, and then select a screen location near the entity to place the dimension.

- Create dimensions similar to those shown in Figure 13.

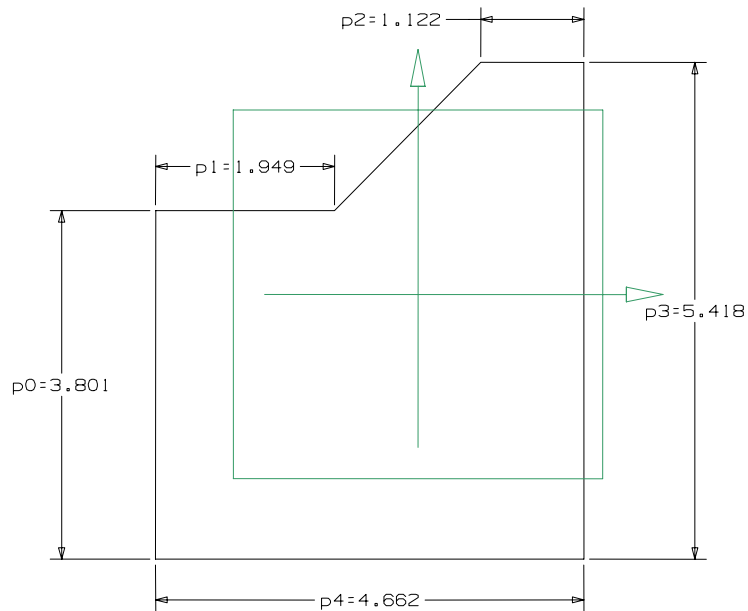


Figure 13

- Once you have created all the dimensions, you may now select on each of them and input the correct dimension value. Each time you type in a new value hit “Enter” on the keyboard to apply the change, and the shape will update immediately. When you are finished, the sketch should appear as shown in Figure 14.

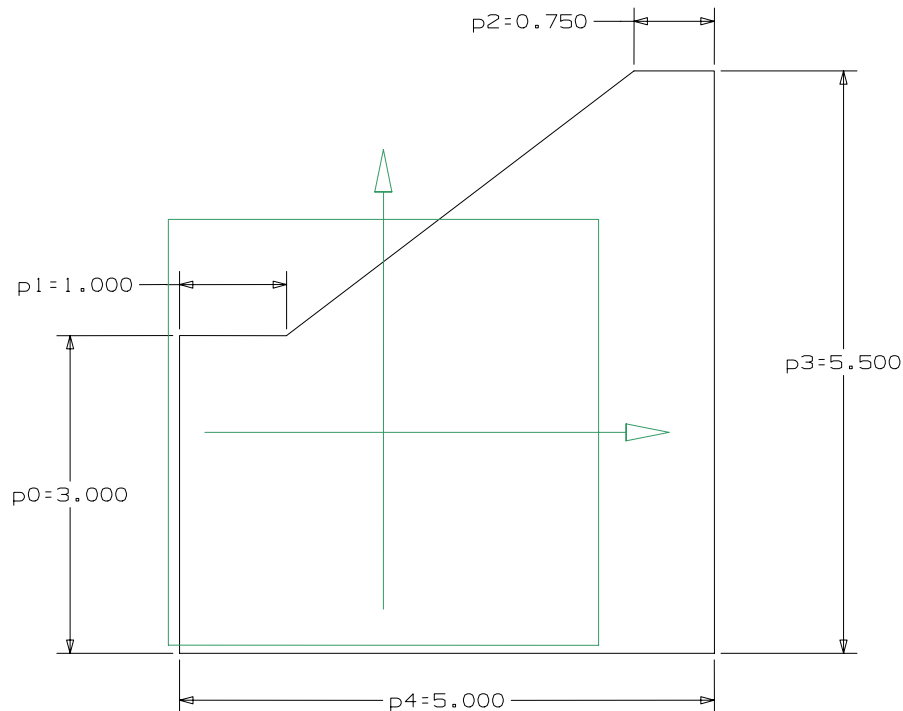


Figure 14


- End of PROJECT

Detail: Perpendicular and Parallel Sketch Dimensions

Creating perpendicular and parallel dimensions can be a little confusing. The first entity chosen when creating a perpendicular dimension *must* be a line; the dimension will be placed perpendicular to that line. The parallel dimension may be thought of as a point-to-point dimension. It is called a parallel dimension because the dimension runs parallel to a line drawn between the two points.

PROJECT

Create a sketch that resembles the one shown in Figure 15:

- Create a new part file
- Select **Insert / Sketch**
- Select the 'Profile' icon 
- Select the screen locations as shown in Figure 15.

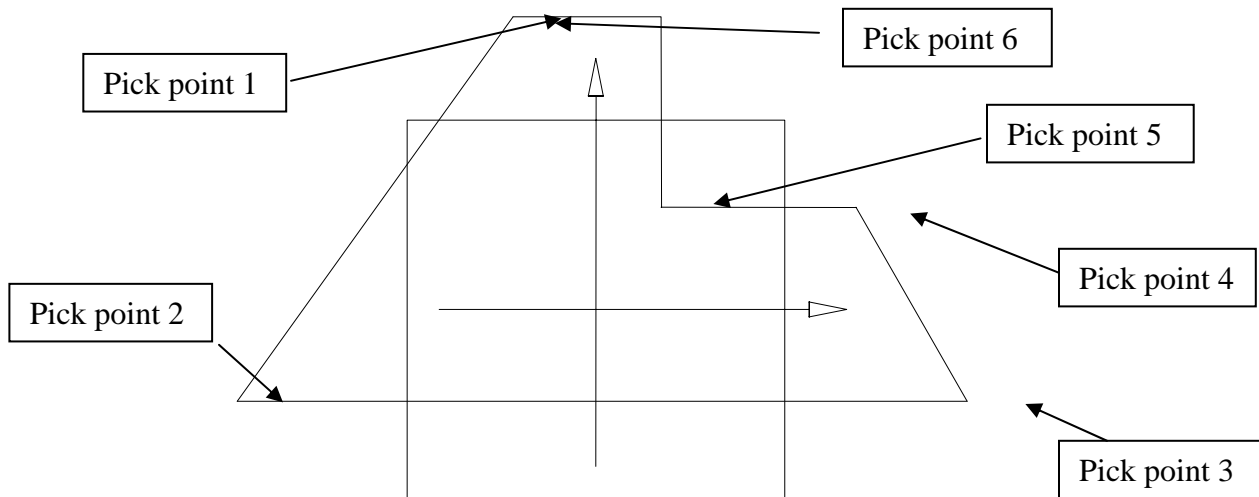




Figure 15

- Select the 'Dimensions' icon  from the Sketch Constraints Toolbar.
- Select the 'Inferred' icon  from the 'Dimensions' dialog box.

- Create the dimension as shown in Figure 16. UG creates a perpendicular dimension.

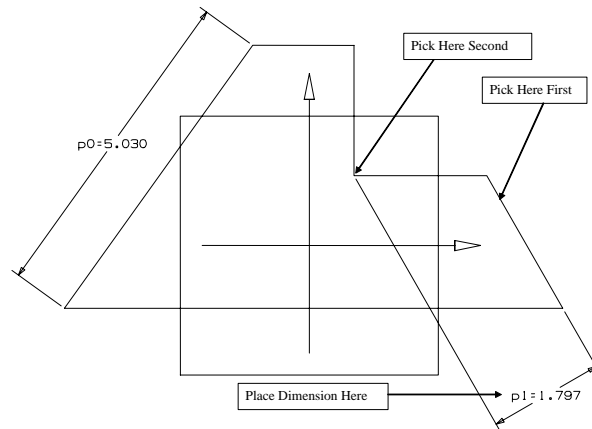


Figure 16

- Create the dimension as shown in Figure 17. UG creates a parallel dimension in this case.

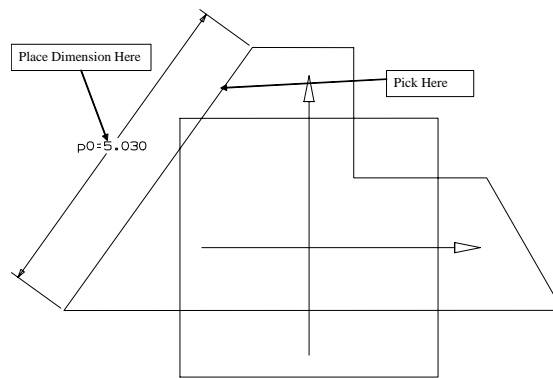


Figure 17

- End of PROJECT

Detail: When Placing Too Many Dimensions on a Sketch makes UG Unhappy

Do not place dimensions on a sketch that are conflicting, or are redundant. This results in an over-constrained condition, and causes the UG sketcher to become confused. Figure 18 shows a typical over-constrained sketch.

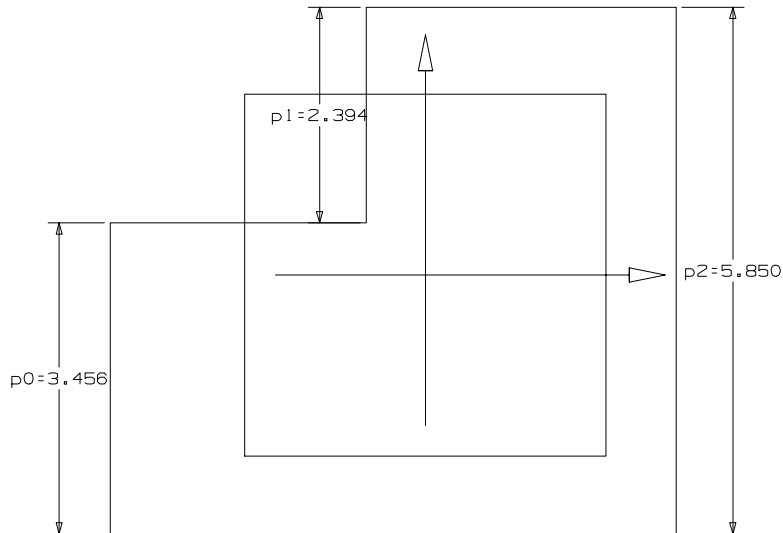


Figure 18

In this example, too many vertical dimensions have been applied. The dimensions p0 and p1 are sufficient to fully dimension the height of the sketch; thus, p1 is not required. An equally valid alternative dimensioning scheme is p2 and p0. It can be seen that the dimensioning scheme should be defined based upon the desired behavior the geometry should have when it is changed.

Detail: If There Are Too Many Dimensions on A Sketch You May Delete or Undo

“**Edit / Delete**” may be used when you have placed a dimension on a sketch that is not required. You may also use the “**Edit / Undo**” command; however, in some cases UG may undo more than you bargained for. Be careful, for there is no “**Redo**” command.

Detail: Radius and Diameter Sketch Dimensions

You may apply either a radius or a diameter dimension to sketched arcs and circles. The inferred tool creates a diameter dimension on circles, and a radius dimension on arcs (see Figure 19).

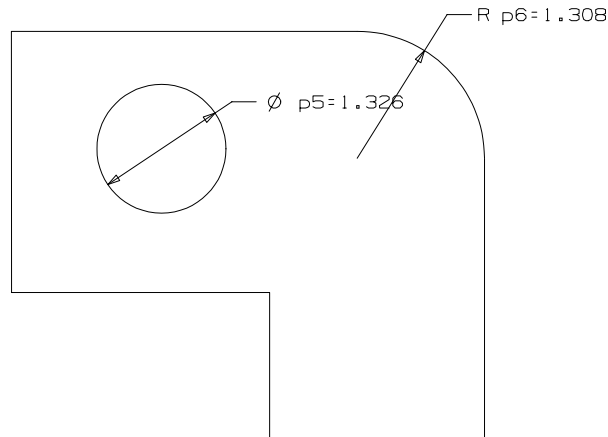


Figure 19

Detail: Moving a Previously Placed Dimension

Dimensions created with the “auto dimension” tool are automatically placed in a “good” location by UG. For horizontal and vertical dimensions, the placement is right in the middle of the span of the dimension. For diameter dimensions, the placement is usually radial to the arc.

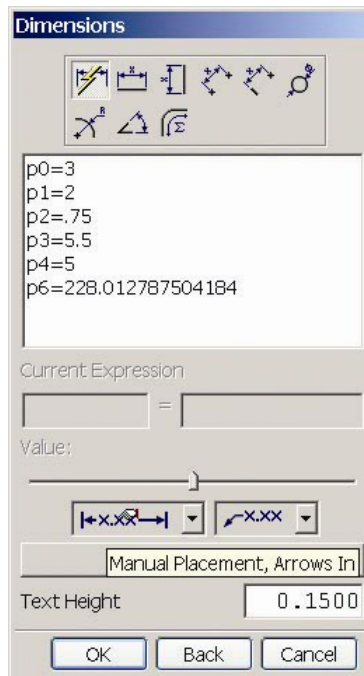


Figure 20

To change the position of a Sketch dimension, you can click on a dimension with MB1, hold down the left mouse button, and drag it to a new location. If a dimension was

created using ‘Auto Placement’, the system will limit its movement. To convert it to manual placement, select the dimension and choose one of the ‘Manual Placement’ options, as shown in Figure 20.

Topic: Defining Geometric Constraints

Creating geometric constraints is analogous to specifying a set of rules that sketch geometry must follow. For example, force the radii of all circles to be equal, or make two lines parallel. A combination of dimensions and geometric constraints gives you the ability to create very complex geometric sketches. Geometric Constraints are applied using the Create Constraints option, accessed from the Sketch Constraints toolbar.

The interaction with the geometric constraint menus can be a little confusing, since constraint options do not appear until geometry is selected. For example, if one line is selected, the “parallel” constraint will not be visible; however as soon as you select another line, this constraint option will appear. To apply a specific geometric constraint, simply choose it from the list of displayed constraints. As soon as geometric constraints are added to a sketch, they are applied to the sketch, which directly affect the shape of the geometry.

Detail: The Degree Of Freedom Indicators

When applying dimensions or constraints, you will notice yellow arrows are displayed on some of the vertices of sketched lines, or perhaps at the middle of sketched arcs. These are the degree of freedom indicators; they indicate which entities and vertices have not been fully constrained. As you place more and more dimensions and geometric constraints on a sketch, the degree of freedom indicators disappear one by one. To constrain a sketch fully, you must apply enough dimensions and geometric constraints to remove all of the degree of freedom indicators. You will also need to fix the sketch with respect to previously existing geometry. Figure 21 shows a simple sketch and its degree of freedom indicators. Keep in mind that it is not always necessary to fully constrain each sketch, but that doing it makes a sketch behave more predictably.

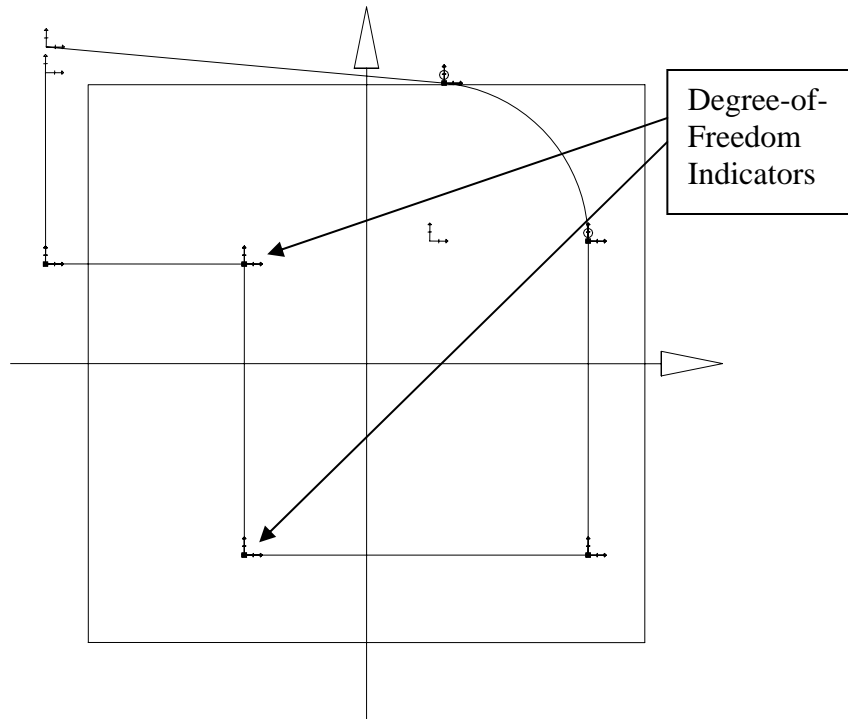


Figure 21

Detail: The Fix Constraint

One of the most important geometric constraints is the “Fix” constraint. It is used to anchor one point on the sketch, to ensure that it has a “zero point”. The Fix constraint does not fix a point in space; it fixes a point with respect to all other geometry in the sketch.

Detail: The Coincident Constraint

The “Coincident” constraint is used to place vertex (e.g. a circle center or endpoint of a line) onto another vertex (see Figure 22).

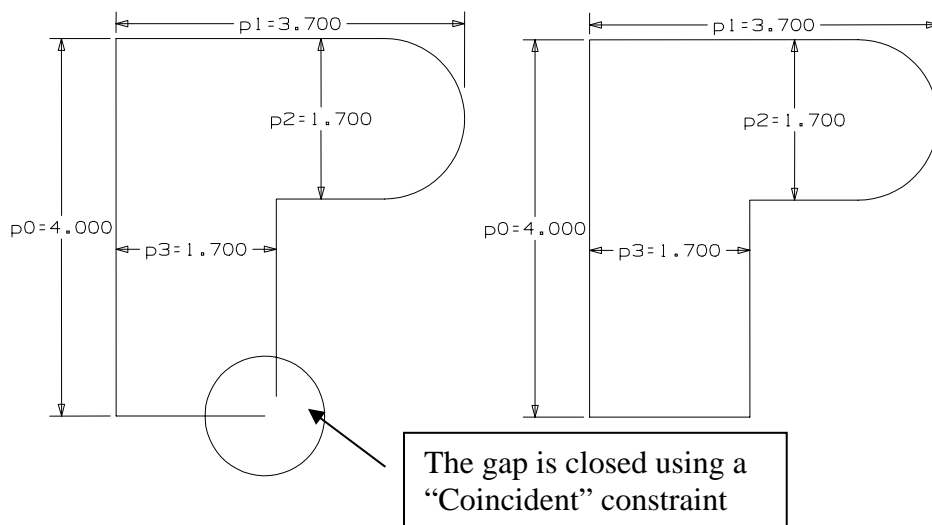


Figure 22

To use this constraint, select the desired vertices, and then select the “Coincident” constraint from the list of available constraints.

PROJECT:

Place “Fix” and “Coincident” Geometric Constraints onto a sketch.

- Construct a sketch that resembles Figure 23. Make sure it is not a closed loop.

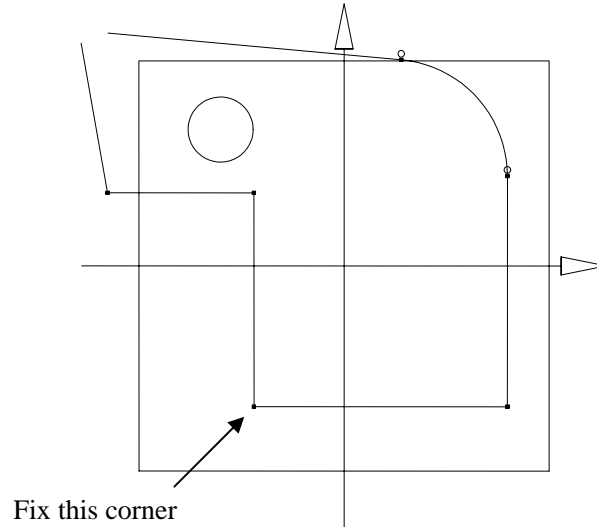




Figure 23

- Select the **Create Constraints** icon  from the Sketch Constraints Toolbar.
- Select the lower left corner of the sketch, as shown in Figure 23.
- Select the ‘Fix’ constraint icon .

Notice how the fix constraint removes the yellow degree of freedom indicators.

- Select the two points as shown in Figure 24.

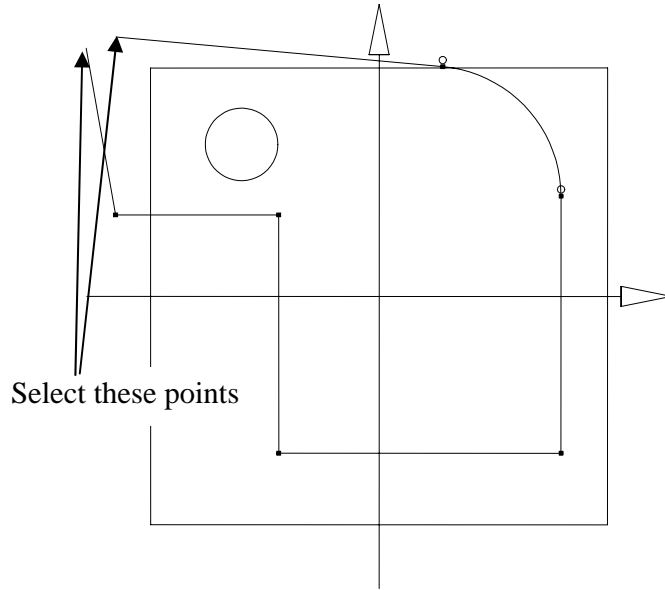



Figure 24

- Select the 'Coincident' constraint icon 
- End of PROJECT

Detail: Horizontal And Vertical Geometric Constraints

When you sketch a line that is nearly horizontal or vertical, UG will often automatically apply a horizontal or vertical geometric constraint to them. You may also place these constraints on manually, if this does not occur.

PROJECT

Create several Horizontal and Vertical Constraints.

- Sketch a profile similar to the one shown in Figure 25.

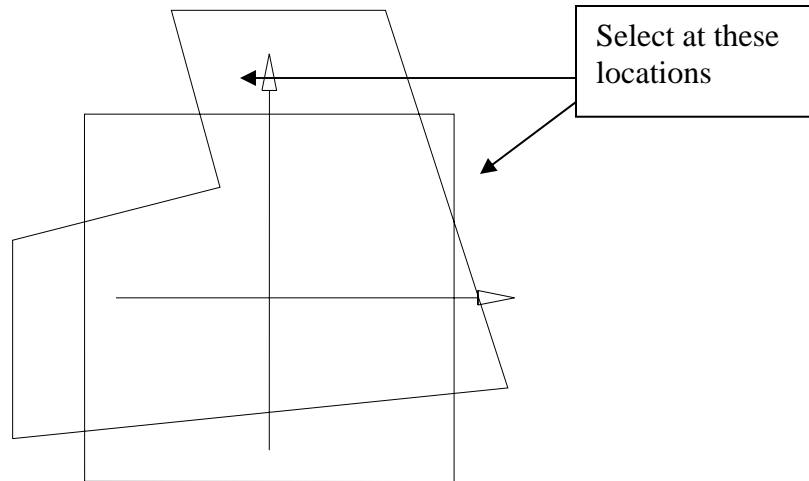



Figure 25

- Select **Create Constraints** from the Sketch Constraints Toolbar.
- Select the lines shown in Figure 25 marked “Select at these locations”.
- Select the Vertical geometric constraint 

The sketch will appear as shown in Figure 26.

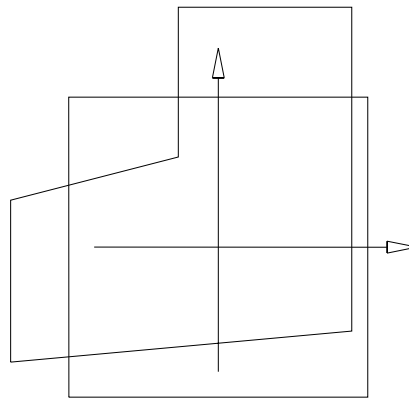



Figure 26

- Select the other lines, and apply a Horizontal constraint 

Your sketch looks like Figure 27.

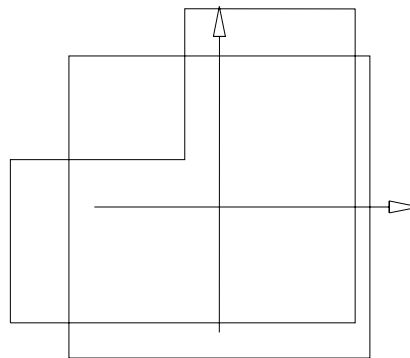


Figure 27

- End of PROJECT

Detail: The Point on Curve Constraint

The Point on Curve constraint is a very powerful constraint option. The name is a bit misleading - it should really be thought of as a point constrained to be on an infinite extension of a curve. It may be used to align points, endpoints of curves, arc center points, to lines or other curves. It is especially useful for creating geometry where elements must be aligned to each other. For example if you had a series of holes that lined up with a slot or edge, you may use the “Point on Curve” constraint.

Figure 28 contains a good example of the application for a Point on Curve constraint. The holes on the right hand portion of the shape shown in Figure 28 should be aligned with the angled edge on the left-hand side. The desired behavior is that the holes should follow the angle of the edge.

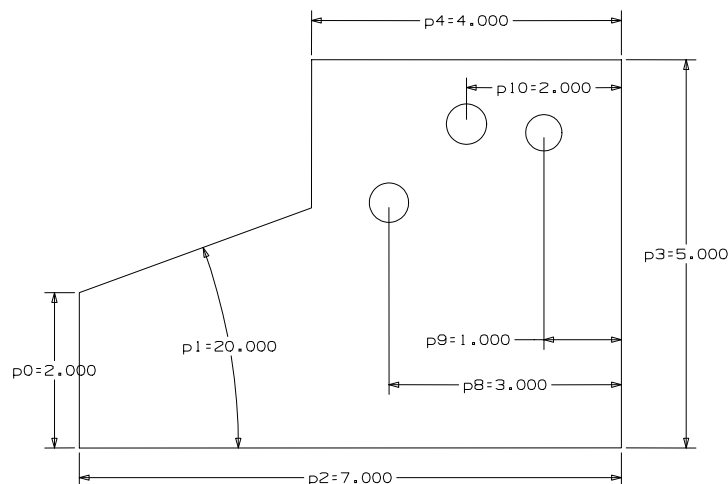


Figure 28

In Figure 29, the point on curve constraint has been applied. Each center point of the circles has been aligned with the angled edge on the left-hand side of the shape. You can see that as the angle is changed, the location of the holes follows.

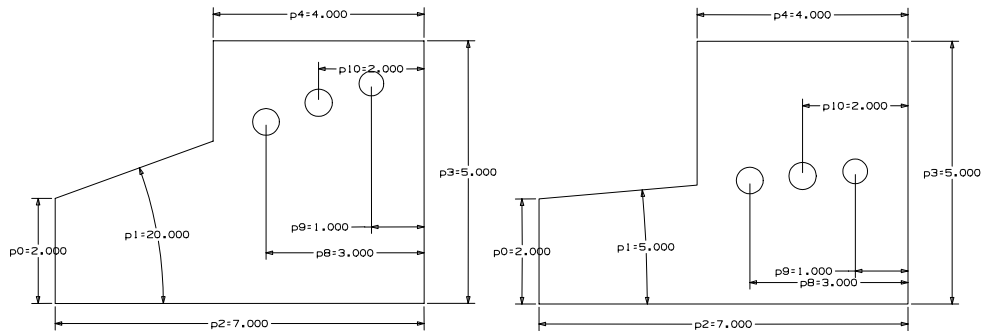


Figure 29

The following exercise illustrates the use of the Point on Curve constraint.

PROJECT

Create the sketch depicted in Figure 30.

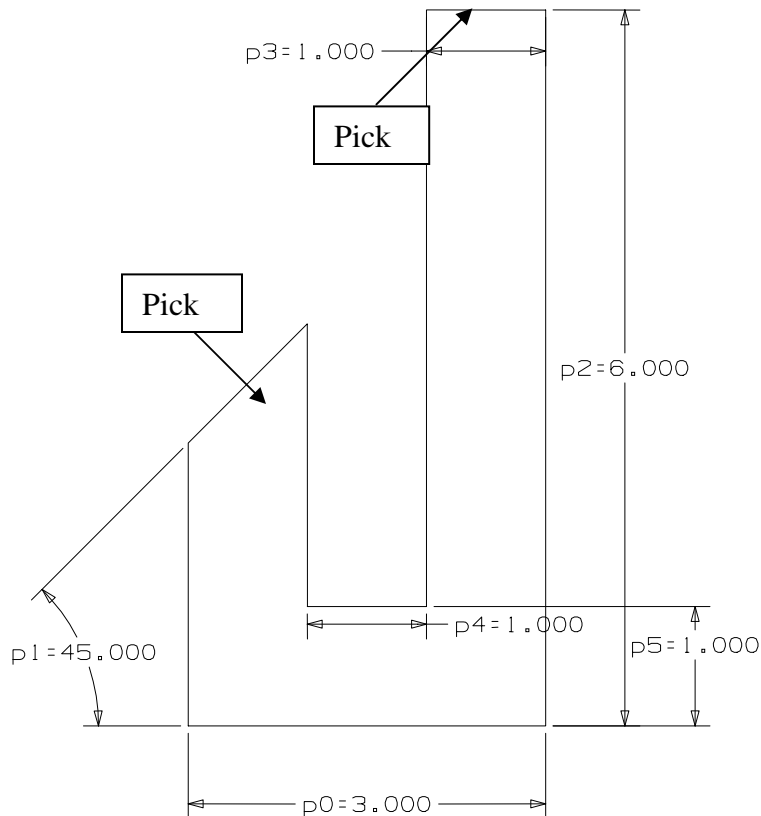



Figure 30

- Select **Create Constraints** from the Sketch Constraints Toolbar.

- Select the two entities labeled “pick here” in Figure 24. A list of constraints will appear, including the “Point on Curve” constraint. It does not matter which entity is selected first.
- Select the “Point on Curve” constraint 
- The result is shown in Figure 31.

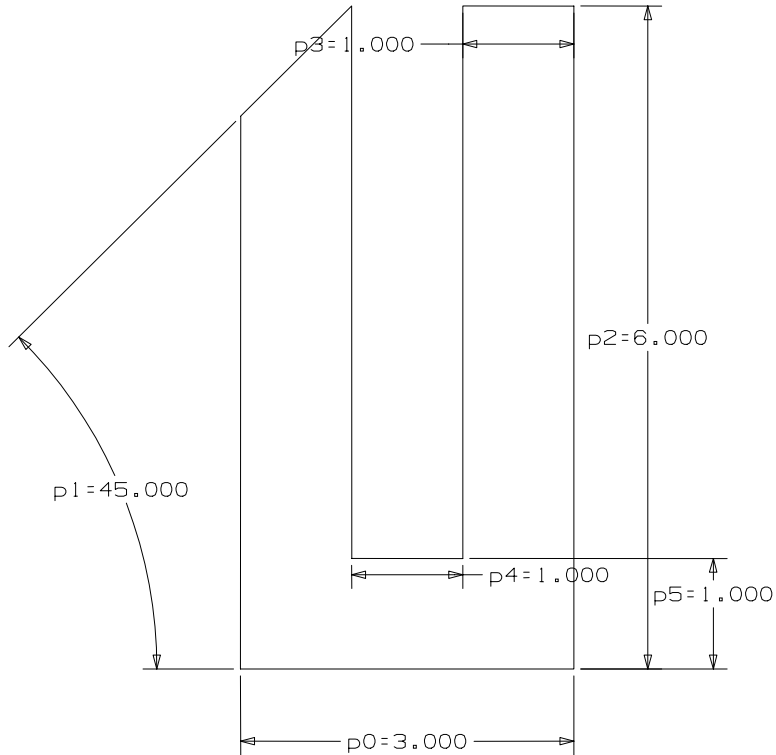


Figure 31

- End of PROJECT

Detail: The Collinear Line Constraint

The “Collinear” geometric constraint is similar to the Point on Curve constraint. The Collinear constraint aligns two or more lines to with each other, as shown in Figure 32.

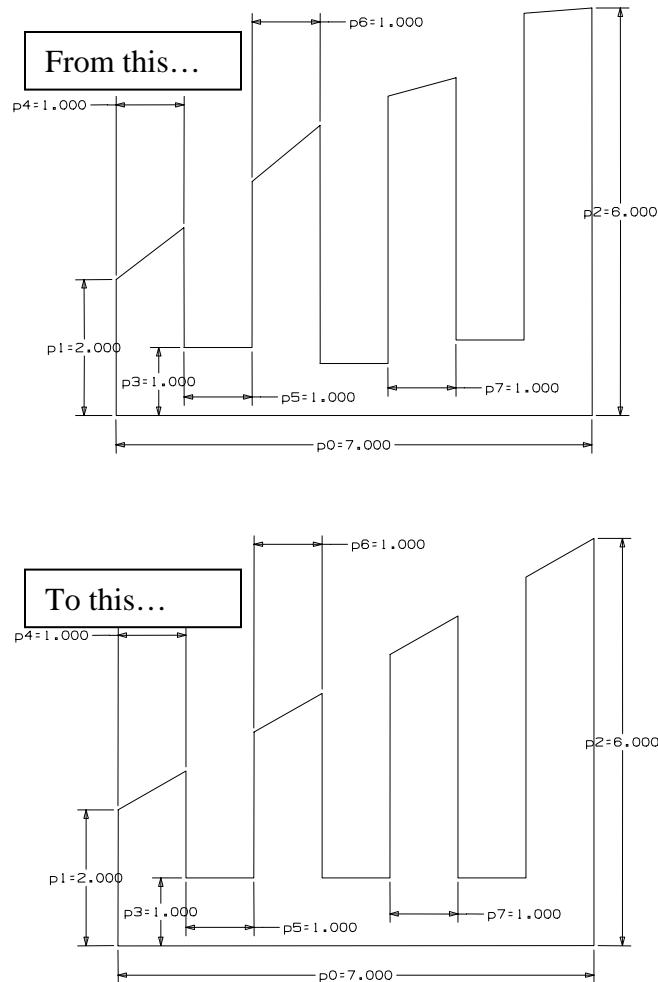
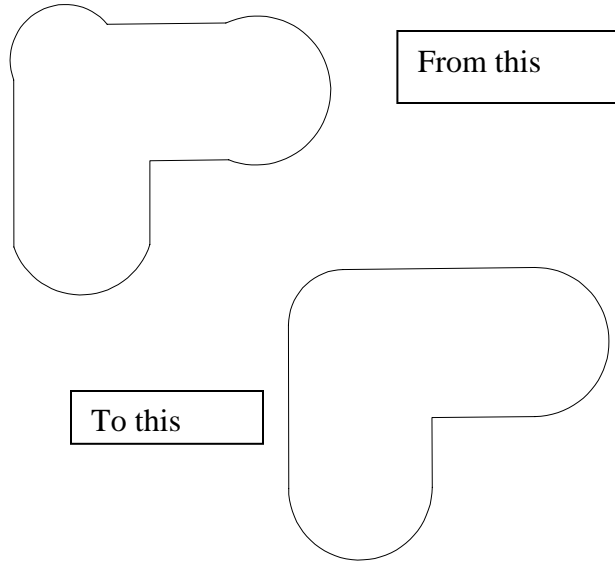


Figure 32

To apply the “Collinear” geometric constraint, select the appropriate lines, and then select the collinear geometric constraint from the list of available constraints.

Detail: The Tangent Constraint

The “Tangent” geometric constraint applies a tangency condition to the selected curves (lines or arcs) as shown in Figure 33. To apply this constraint, select the two entities near to the desired tangency location, and then select the “Tangent” constraint.

**Figure 33**