Dimensions and Relations

Updated 4/26/2019

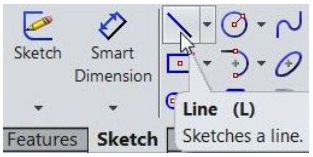
Introduction

You now have some experience in [1]**SolidWorks®** specifying sizes on model sketches using the **Smart Dimension** tool. There are a few additional remarks that will be made here which should aid you in the parametric modeling that we have been doing, which is the process of adding numbers to entities and specifying relationships between them.

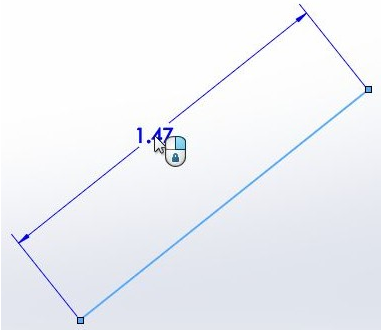
Notes on Dimensioning

In some situations, there is more than one dimension choice available when you use **Smart Dimension** and then click on an entity. The direction you move the mouse determines the choice.

Click on **New** and open a new **Part**. Select the **Front Plane** from the Design Tree and then the **Line** tool from the Sketch group.

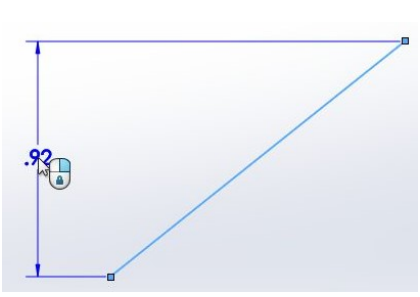


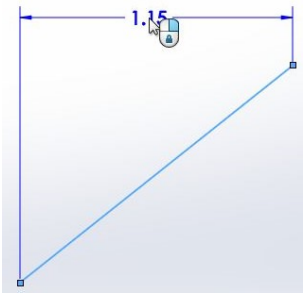
Anywhere in the Graphics Area, click and hold the left mouse button while moving the mouse in an angular direction a short distance up and to the right. Release the mouse button to complete the line. Select **Smart Dimension**, click on the line, then move away perpendicular to it – the length of the line as-drawn is shown. If you were to click now in the Graphics Area, the Modify box would open and you could enter the desired length. But don't click yet.



Move the mouse horizontally. Notice that the vertical distance between the endpoints is shown.

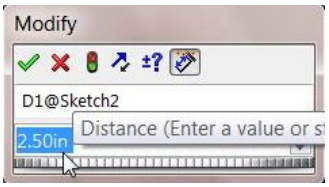
Now move vertically above the line. Notice that the horizontal distance between the endpoints is shown. So, you have three dimension choices available when clicking once on an angled line.





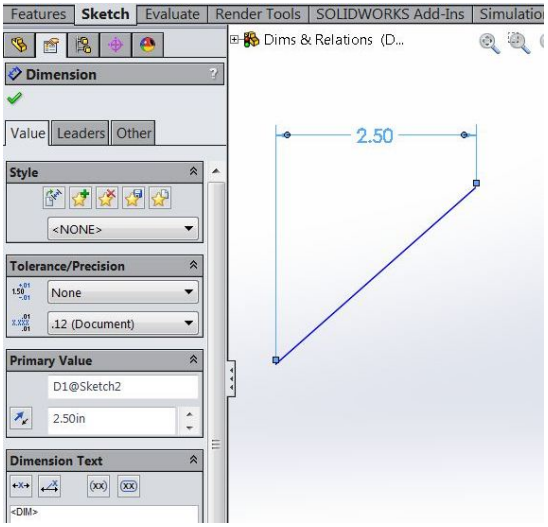
Note that a mouse icon appears at your cursor with its right button in blue. If you right-click, it will lock onto the dimension orientation that you are presently in. Then you can move the mouse in any direction you want and that orientation will stay the same.

Click to place the dimension in the Graphics Area, add a new value in the Modify box and click the green check mark.



[1]**SolidWorks** is a registered trademark of Dessault Systémes SolidWorks Corporation. Screen shots from SolidWorks are used throughout this tutorial.

In the **Dimension** Property Manager under the **Value** tab under **Tolerance/Precision**, note that in this example the precision is set at two decimal places which is the default setting for this document (the setting on your computer may be different). You may drop down the list and change the precision for this individual dimension if desired.

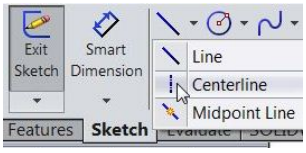


Whenever you need to change the dimension value of a line, double click on the value in the Graphics Area and change it in the Modify box. Do not change it in the **Primary Value** area of the **Dimension** Property Manager because that may cause the entire sketch to be under-defined.

Sometimes you may need to add text or symbols before or after the dimension value. This can be done in the **Dimension Text** area of the Property Manager, where <DIM> represents the dimension value in the editing window there.

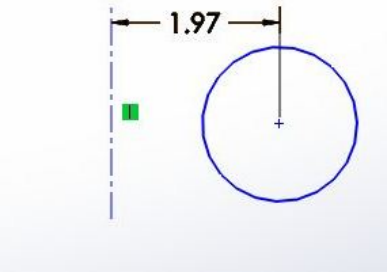
Circle Location Dimensions

The location of a circle is usually defined by where its **center** point is, as opposed to a location on the circumference such as a quadrant point or tangent point. We will now practice some dimensioning of circle locations.

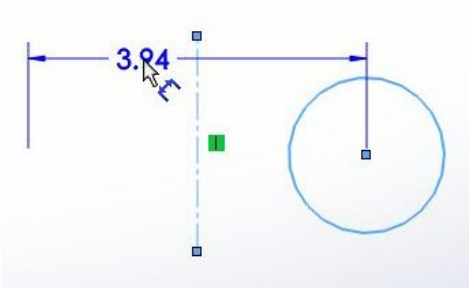


Click on the drop-down arrow to the right of the **Line** tool and select the **Centerline** tool.

In an empty space of the Graphics Area, sketch a vertical **Centerline.** Now sketch a **Circle** to the right of the centerline.



Select the **Smart Dimension** tool and click on the centerline and then click on the **circumference** of the circle. Notice that when you click on the circumference, it automatically snaps to the circle's **center point** and displays the linear dimension between the centerline and the center of the circle which is what we want.

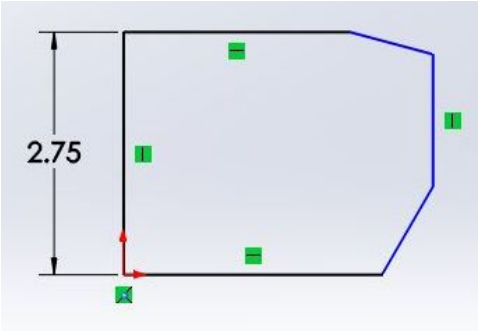


Now move the cursor to the **left** of the centerline and observe what happens: the dimension **doubles**. We will use this technique later in the course when building models by sketching **one half** of a cross section and then revolving it around the centerline to create a full 3D feature, working from model drawings that show **diameters** instead of **radii**. Okay, we're finished with this demonstration so press **Esc** twice to quit dimensioning.

We're also finished with the exercises up to this point in the tutorial. Go to the upper right-hand corner of the Graphics Area and click on the **X** to **Close** the Part but **Don't Save** it, if asked.

More on Dimensioning

Open a **New** Part, select the **Front Plane** and click on the **Line** tool from the Sketch group (a keyboard shortcut for the **Line** tool is the **L** key).



Sketch the figure approximately as shown here by creating a chain of connected lines. Start by clicking and releasing on the **origin**, move up vertically, click and release, then go horizontally to the right, and continue around, clicking and releasing at the endpoint of each line segment, ending back at the origin. The small green squares are called **sketch relations.** The ones on the horizontal and vertical line segments are automatically added as you sketch so as to lock these lines into their proper orientation. Hit **Esc** to exit the **Line** tool.

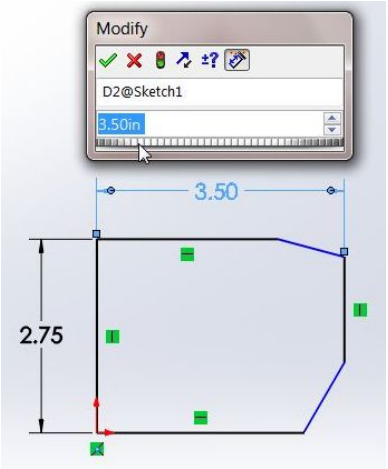
(Note that when sketching a **chain** of connected lines, the *click-and-release* method is more convenient for setting endpoints, while the *click-and-hold, drag-and-release* method is more convenient for **single** lines.)

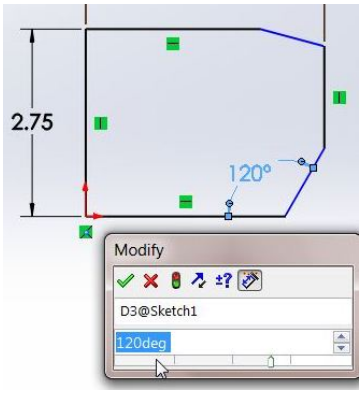
Click on **Smart Dimension** and then on the left-hand vertical line. A dimension showing the length of the line will be displayed. Move the cursor to the left and click to place the dimension and cause the Modify box to open. Enter **2.75** as the value then press **Enter** or click on the green check mark.

With **Smart Dimension** still active, click on the same vertical line again and then click on the *right-hand* vertical line.

When two parallel lines are selected, the distance between them is displayed.

Click to place the dimension and enter **3.5** on the Modify box.

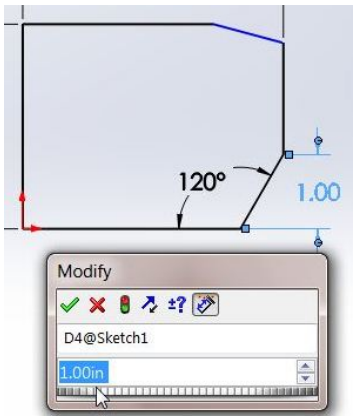




Now click on the bottom horizontal line and the angled line attached to it. When two non-parallel lines are selected, **Smart Dimension** displays the *angle* between them.

Move the cursor in a circular motion around the angle's vertex and notice that one of three different angle values appears, depending on where the cursor is. We want the one inside the figure, so click to place it there and enter the degree value of **120**.

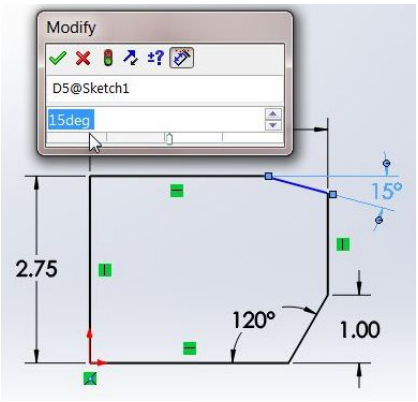
Click on the angled line and move *horizontally* away from it to the right, so as to display the vertical distance between end points. Click and enter the value of **1**.

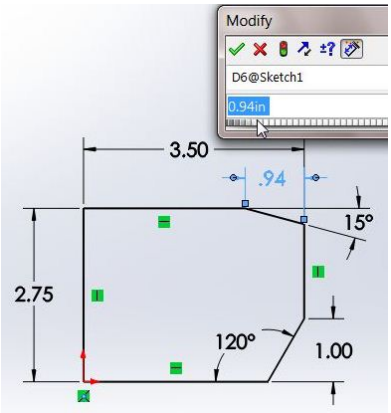


Note that if you were to move the cursor down from the line or perpendicular to it, a different dimension choice would be displayed.

Now click on the top horizontal line and the angled line attached to it.

Here again there are three choices, but position the cursor so as to display the angle as shown. Enter **15** as the degree value (if you get an error message, see if maybe you accidentally sketched the 15° line **perpendicular** to the 120° line -- see more on **Sketch Relations** below).





To fully define the sketch, one more dimension is needed.

Click on the 15° line, move straight up and click.

Enter **.94** as the horizontal distance and then hit **Esc** to finish.

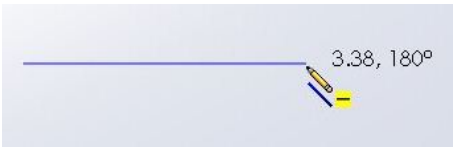
We are now finished with this exercise and need to make room in the Graphics Area for the next one. Form a selection window enclosing the entire sketch and all of its dimensions -- click-and-hold, then drag diagonally and release to capture all entities. Press **Delete** to erase all.

Sketch Relations

As you have noticed when you create sketches, little square symbols appear next to the things that you sketch. These are the automatically-added **Sketch Relations**. For example, look at the horizontal and vertical lines in the figure above and note that they have the corresponding sketch relation symbols next to them. The relations lock the orientation of the lines so that they stay perfectly aligned in the direction corresponding to the relation.

Sketch relations can also be **added** and **deleted** manually.

Select the **Line** tool (**L**) and sketch a horizontal line in the open space of the Graphics Area. Note that a temporary yellow *horizontal relation* symbol appears next to the cursor as you sketch the line in the horizontal direction. A green *horizontal relation* then gets added to the line when you are finished drawing it.



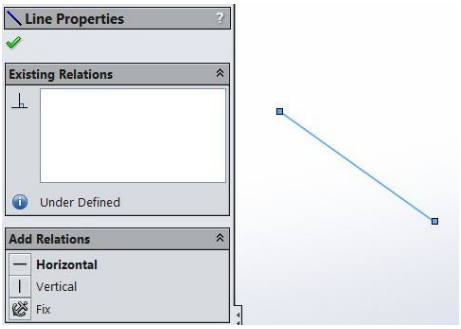
Hit **Esc** to close the **Line** tool.

Click-and-hold on an endpoint of the line and drag it around. Since we have not used **Smart Dimension** to specify a length yet, you can drag the line longer or shorter. You can also drag the line up and down, but note that it always stays **horizontal**.

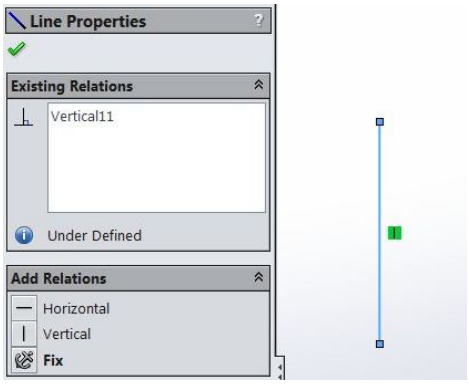
Now click on the horizontal relation symbol and hit the **Delete** key (Mac users must use the Main Menu sequence of **Edit**/ **Delete**). The line now is no longer locked horizontally.

Drag down an endpoint so that the line is at an angle.

Now click randomly somewhere on the line between the endpoints to open **Line Properties**.



In the **Add Relations** section, click on the **Vertical** relation symbol.



Note that the line rotates to the **vertical** position and now has the *vertical relation* symbol near it.

Also notice that **Vertical** has been added to the **Existing Relations** box of Line Properties.

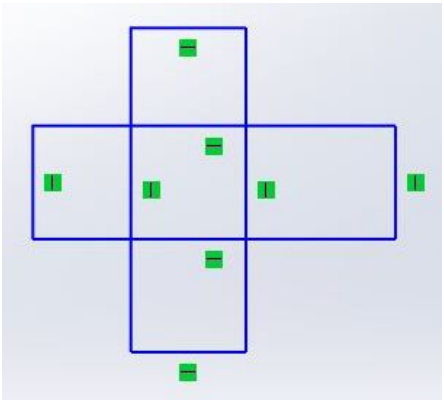
Relations can also exist **between** entities; a **Tangent** relation between a line and a circle for example.

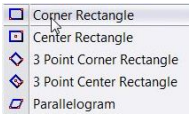
Many such relations are added automatically as you create sketches.

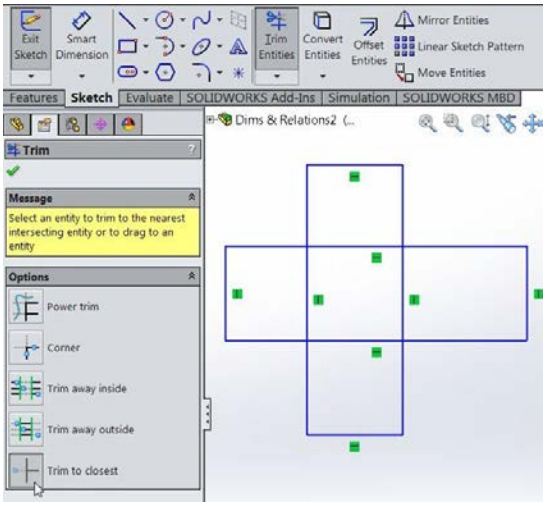
In some design situations you may want certain geometric relationships between entities to remain the same even if dimensions are modified later. Adding the appropriate sketch relations is a good way to do this.

The use of sketch relations can also reduce the total number of dimensions needed to define your model.

We will do a short exercise here to demonstrate these applications: in an open space in the Graphics Area, use the **Corner Rectangle** tool twice to create the cross shape approximately as shown here to the right.







From the Sketch group, select **Trim Entities** and then choose the **Trim to closest** option.

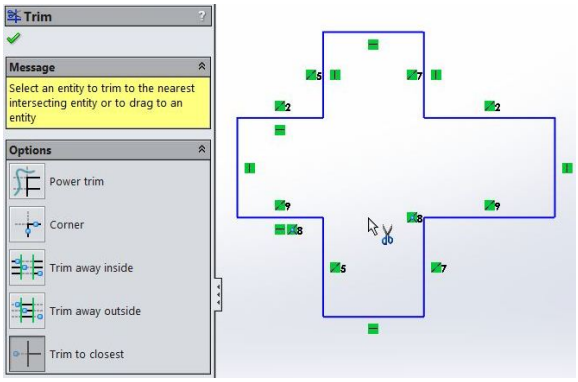
To use this option, you click on what you want to **get rid of** on the sketch.

Notice that a pair of **scissors** is attached to the cursor, indicating that the **Trim** tool is active.

Click on the **four line segments** on the **inside** of the figure where the two rectangles cross each other.

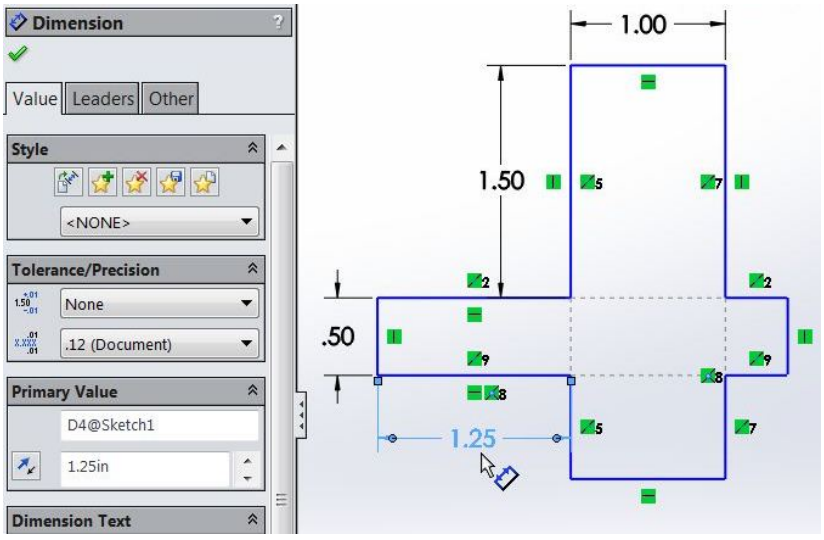
As these segments are trimmed away, **Colinear** relations are automatically added to the remaining segments of the lines.

Also, a **Coincident** relation is added where the rectangles join.



Use **Smart Dimension** to add only the four dimensions shown. The sketch may change shape as sizes adjust to the dimensions. If desired, you can click on any corner point that isn't locked into place by a dimension and drag it back to the general shape desired

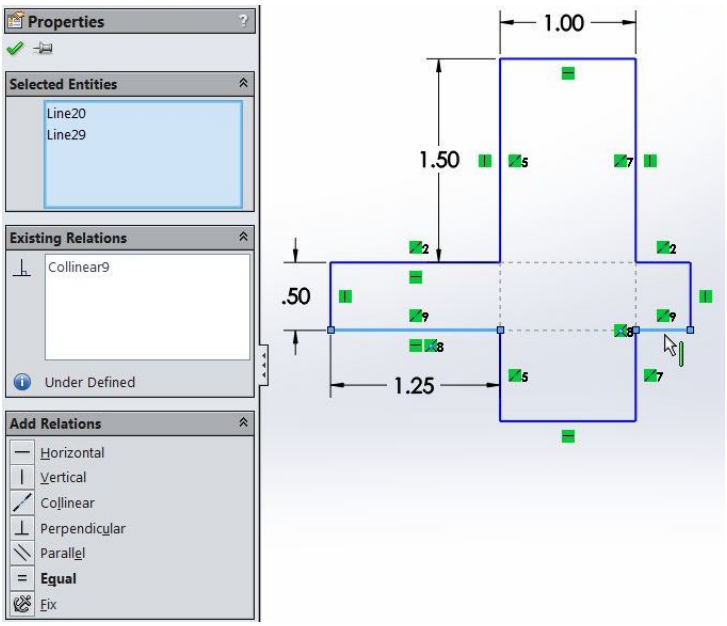
Hit **Esc** when done.



We want both the top and bottom vertical portions to be **1.50"** long, and we want both the left and right horizontal portions to be **1.25"** long.

To fully define the geometry without cluttering up the sketch with more dimensions than necessary, some **relations** can be added.

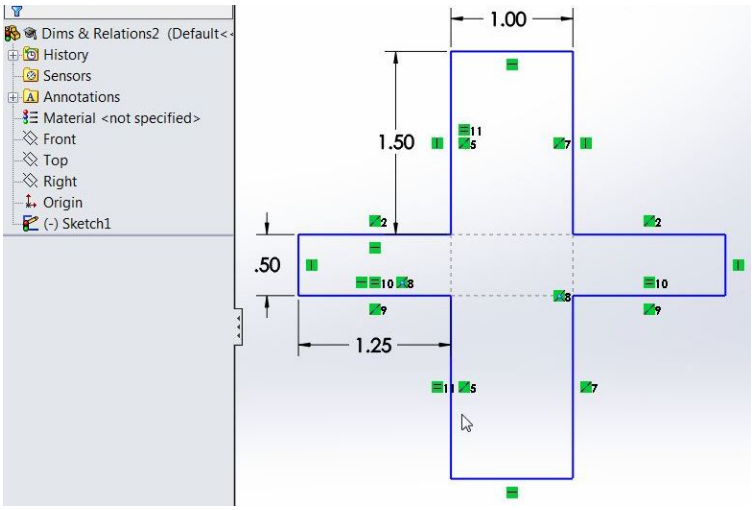
Click on the horizontal **1.25”-**long line and then hold down **Ctrl** (this allows multiple-selection of entities) while clicking on the horizontal line to the right which is colinear with it. (Note: Mac users must use the selection method at the bottom of this page instead.)



In the Property Manager, both of the selected lines are listed in the **Selected Entities** window and in the **Existing Relations** window, you see that they are colinear.

In the **Add Relations** section, click on the **Equal** icon to make the lines equal in length. Click on the green check mark to accept the changes and close the Property Manager.

Now click on the vertical **1.50”-**long line and then hold down **Ctrl** while clicking on the vertical line below it which is colinear with it.

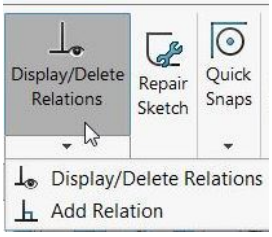


In the Property Manager, in the **Add Relations** section, click on the **Equal** icon to make these lines equal in length. Close the Property Manager.

Notice that **Equal** relation symbols have appeared on the sketch by the lines that they were applied to.

Using relations in this manner is handy, but only do so if your design intent is for those entities that you assigned to be equal are to remain so even when you modify sizes, because they will both change together.

To try this out, double-click on the **1.25** dimension and change it to **1.75**. Notice that **both** sides change to the new size. Now click **Undo**  to reverse the change.

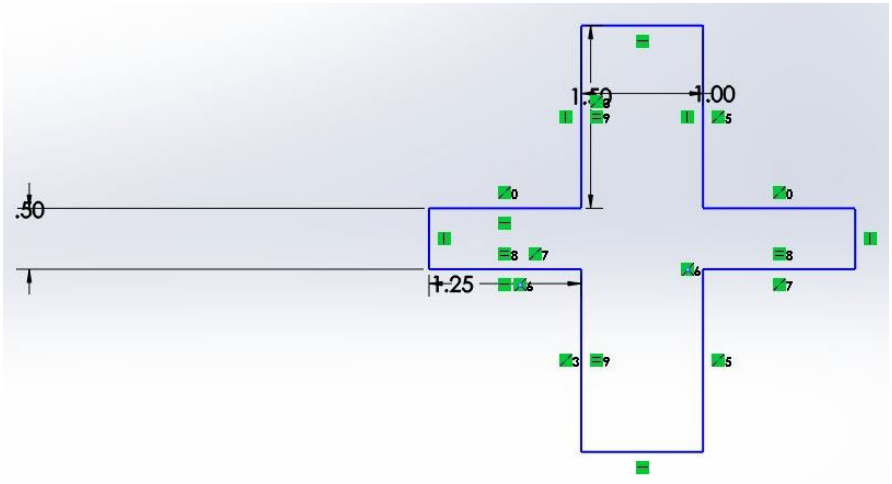


Another way to work with sketch relations is to use the **Display/Delete Relations** tool in the Sketch group. You can use its **Add Relation** option to assign relations between entities without having to hold down **Ctrl** for multiple selection.

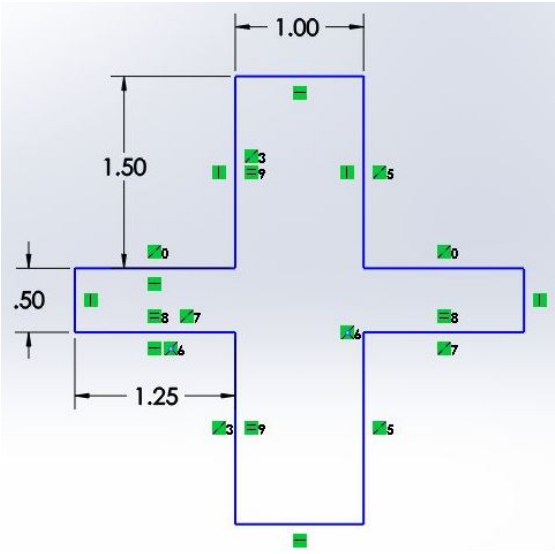
Dimension Layout and Organization

As you add dimensions to sketches, they don't always get positioned exactly where you want them to be the first time. When that happens, just drag them around and reposition them so that all dimensions are neat and legible. Get into the habit of finishing your sketches with a dimension layout that is organized and uncluttered.

Below is an example of poor dimensioning layout. Some of the numbers are obscured by lines, sketch relations, other dimensions, are too far away, or are otherwise poorly laid out.



Shown below is the same sketch after dragging the dimensions around to arrange them more neatly. In later lessons, you'll learn how to adjust arrows, extension lines, etc. to improve your dimension layout.



Note that this sketch is *Under Defined*, which is okay for this dimensioning demonstration in the open space of the Graphics Area, but for actual part models that you build in this course you want sketches to be *Fully Defined*. In the above sketch, locating the corner of a rectangle at the **origin** would fully define it.

End of Dimensions and Relations Tutorial