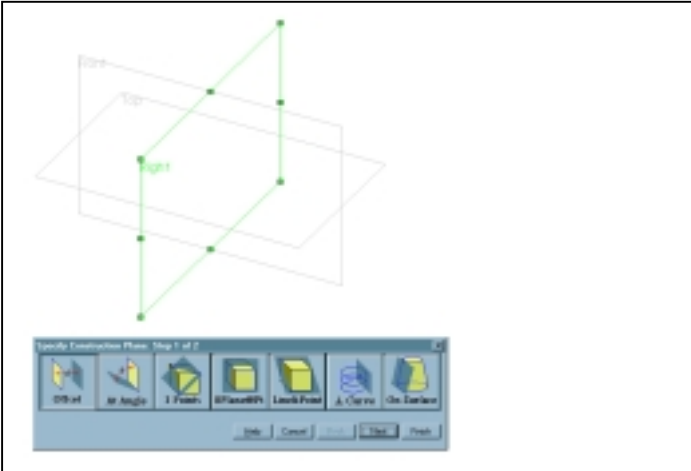


SW Tips/Tricks

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Reviewing Part Modeling Basics

Categorizing the basic commands is useful because in the SolidWorks course curriculum the material is presented by working several projects followed by lab exercises. The course presents each command as needed to create the projects and labs. By the end of the course, you have been introduced to each of the commands, but no where have the commands been listed or summarized. This is a good way to introduce or refresh your memory on the basic part modeling commands and terminology.

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Planes

Upon creating a new part, by default there are three planes in the environment. Out of the box the default planes are named Plane1, Plane2, and Plane3. Most users elect to rename these planes Front, Top, and Right. To create the base feature to a new part, first open a sketch on a plane or flat face. Since you do not have features on a new part, you must open the sketch on a plane. You can open the sketch on these existing planes, or create additional planes to open the sketch on. The existing planes are called default planes and the additional planes you create are called user-defined planes. Also once you create a feature in your part containing flat faces, you can open new sketches on those faces. You do not need to create a plane on top of the flat face, just pick the flat face and open the sketch.

Sketches

Now, you have selected the sketch location and you are ready to begin sketching. Sketches are the 2D representation to your 3D feature. You can create all the usual types of 2D entities, such as lines, circles, arcs, and ellipses, just to name a few. Centerlines and

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construction lines can be used interchangeably. They are really the same entity. Any entity can be represented as construction if needed. Any construction geometry will be for layout purposes only and will not be extruded, revolved, etc. You can also mirror the entities in a sketch. The mirror is an example of why the sketch in SolidWorks is a smart sketch. These entities will remain mirrored until the

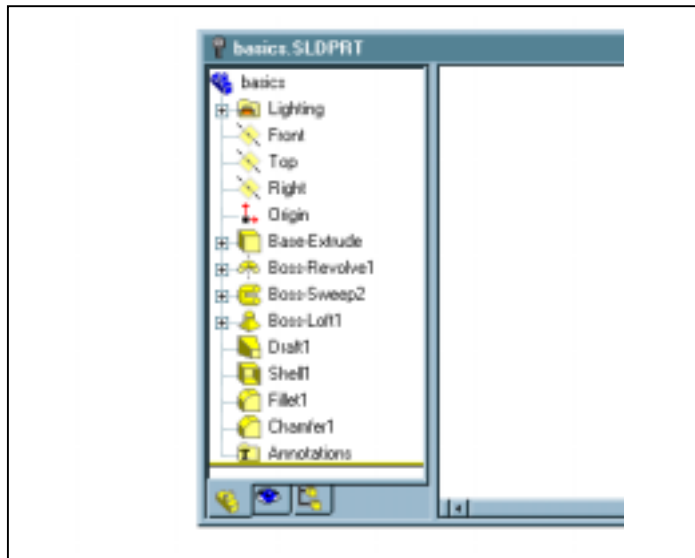
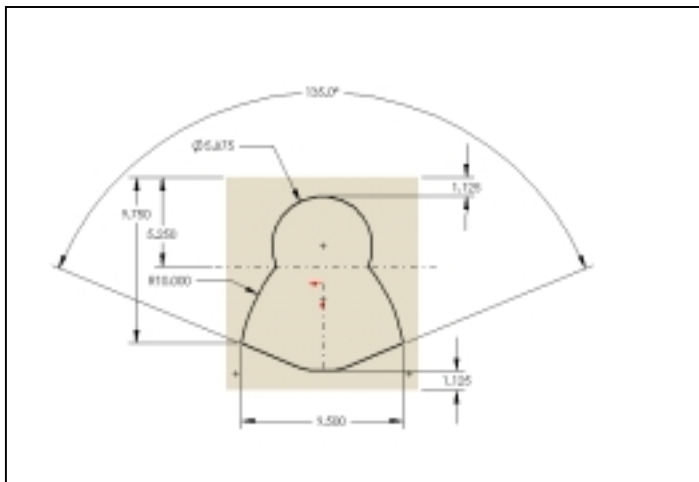
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user removes the symmetric relationship. This is the main difference between a sketch in SolidWorks and typical drafting software. Whenever you mirror, offset, convert, and dimension your sketch, the relationships will remain until removed by the user. You want to learn how to exploit these smart-sketching techniques to convey your design intent.

The Convert and Offset commands are often confused, and are sometimes difficult to recognize when to utilize the command. They are similar because convert can be thought of as an offset of zero. The confusion is that convert doesn't actually convert anything, it just takes the edge or entity in a previously drawn sketch and projects it to the current sketch. The original edge or sketch entity is still intact.

Geometric Relations

As you created the entities, you related the entities to previous geometry and also to each other. This is called Automatic relations. Typically when you are sketching lines, you see dashed lines inferring parallel, perpendicular, and tangency. Cursor feedback infers horizontal and vertical vectors. If you place the line according to these inferences, SolidWorks will add the corresponding geometric relationships. You can also add these after creating the geometry using the Add Relation command. If you are uncertain of the relations a sketch contains, you can review the existing relationships using the Display/Delete Relations command. As the name implies, you can also delete any unwanted relations through the Display/Delete Relations command. In addition to physical relations

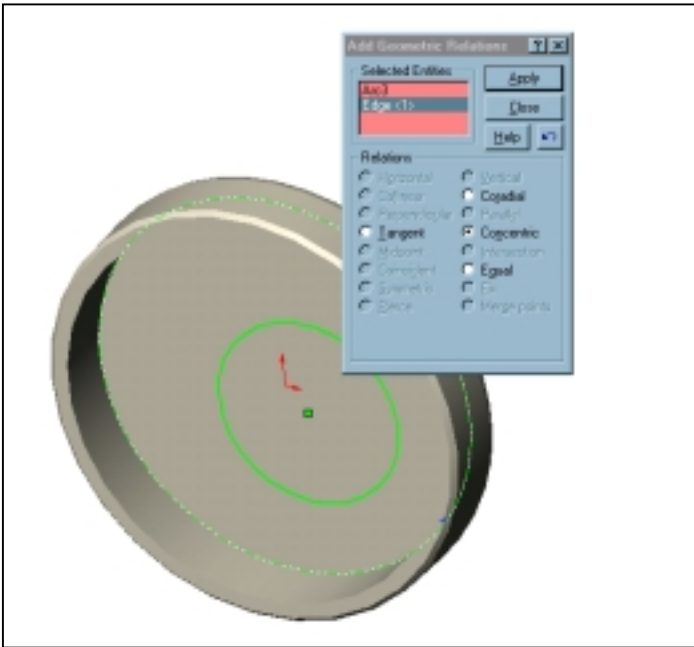


such as concentric, coradial, and collinear, you can add dimensions between entities. Once added, you can either link the two dimensions (to make both dimensions equal value), or add an equation between two or more dimensions to change the value of one dimension based on the value of the other dimensions ("Dimension A" = "Dimension B" / 2). Now you have opened your sketch, created the geometry, and related the geometry with relationships and dimension.

Features

Once this is complete, you are ready to actually make the feature. There are four basic ways to create a feature from the 2D sketches. These are Extrude, Revolve, Sweep, and Loft. You can utilize these four tools to either add or remove material. For the first feature of a part, you can only add material (because there is nothing to remove material from). This first feature is known as the base feature. Subsequently, the term Boss will be used when adding material. Removing material is called Cut. Combining the terms you end up with the default names for the features. Base-Revolve would name the first feature that was created by revolving the sketch. A Cut-Sweep would be a subsequent feature that removed material from the part by sweeping along a path. Any of these four

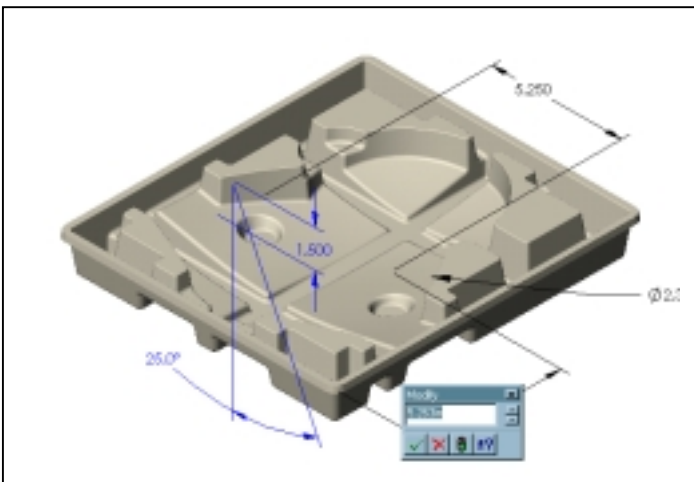
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methods require sketching and are referred to as sketched features.

There are several other methods to create what is called applied features. These include such commands as Fillet/Round, Chamfer, Shell, and Draft. These features do not require a sketch be generated. After completing the feature in a part, you can duplicate these features by creating a mirror image of the feature, or repeating the feature in a circular or linear fashion. A bolt circle of holes in a part would be a good example of repeating the individual hole in a circular pattern.



Making Changes to the Part

Now your feature has been added to your part but what if it is not quite right. Maybe the size or location is not correct, or someone has changed the design since you created the original part. You may have a tendency to use the Undo button, but before you do, consider the following commands. The easiest way to change the value of a parameter is simply double click on the feature. You can do this and the rest of these change commands from the Feature Manager or from the graphics screen. If you choose to perform them from the graphics screen, you need to know where to right select or double click on the feature in order to specify the correct feature. Right selecting or double clicking the feature from the Feature Manager may make it easier to specify the correct feature. When you double click on the feature, you see all of the dimensions that are associated with the feature. The black dimensions are associated with the sketch, and the blue dimensions are associated with the dialog box options you specified when you created the feature. Now double click on the dimension text to change the value of the dimension, and then rebuild the part model.

Alternative methods to change a part are to right click on a feature and select either Edit Sketch or Edit Definition. These two commands take you back to the sketch or the dialog box that defined the feature. You can then change the sketch or redefine the feature

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parameters. Another commonly used change tool is Properties. Just like all other Windows objects, items such as lines, arcs, dimensions, features, etc. have properties that can be modified. It is usually more efficient to change the part with one of these options rather than undo and recreate the entity or feature again.

These are the basics of part creation and by no means are all of the commands that will be used to create a part. Beginners find this summary helpful prior to learning methods of part construction to help relate all the commands together, and seasoned users find it helpful to review the basic tools in case they don't utilize all of them all the time. -(O|||O)-

If you would like to receive issues of SW Tips/Tricks please provide us the following information by:
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Calendar of Events

San Diego SolidWorks User Group

Digital Dimensions, Inc., 3934 Murphy Canyon Road
Suite B-100, 2nd Wednesday of the Month at 7:00pm

SolidWorks World - New Orleans 2000

Hilton New Orleans Riverside
January 12-15, 2000

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