# Module 1: Basics of Solids Modeling with SolidWorks

Introduction

SolidWorks is the state of the art in computer-aided design (CAD). SolidWorks represents an object in a virtual environment just as it exists in reality, i.e., having volume as well as surfaces and edges. This, along with exceptional ease of use, makes SolidWorks a powerful design tool. Complex three-dimensional parts with contoured surfaces and detailed features can be modeled quickly and easily with Solid-Works. Then, many parts can be assembled in a virtual environment to create a computer model of the finished product. In addition, traditional engineering drawings can be easily extracted from the solids models of both the parts and the final assembly. This approach opens the door to innovative design concepts, speeds product development, and minimizes design errors. The result is the ability to bring high-quality products to market very quickly.

CONSTRAINT-BASED SOLIDS MODELING

The *constraint-based solids modeling* used in SolidWorks makes the modeling process intuitive. The 3-D modeling begins with the creation of a 2-D **sketch** of the profile for the cross section of the part. The sketch of the cross section begins much like the freehand sketch of the face of an object. The initial sketch need not be particularly accurate; it needs only to reflect the basic geometry of the part's cross-sectional shape. Details of the cross section are added later. The next step is to **constrain** the two-dimensional sketch by adding *enough dimensions* and *parameters* to completely define the shape and size of the two-dimensional profile. The name *constraint-based modeling* arises because the shape of the initial two-dimensional sketch is "constrained" by adding dimensions to the sketch. Finally, a three-dimensional object is created by *revolving* or *extruding* the two-dimensional sketched profile. Figure 1 shows the result of revolving a simple L-shaped cross section by 270° about an axis and extruding the same L-shaped cross section along an axis.
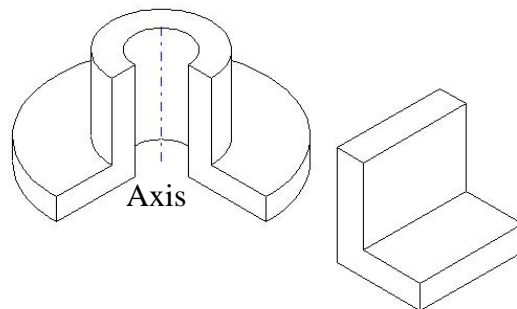
Figure 1 Revolved and extruded solid models of an L-shaped cross-section.

In either case, these solid bodies form the basic geometric solid shapes of the part. Other features can be added subsequently to modify the basic solid shape. Once the solids model is generated using SolidWorks, all of the surfaces have been automatically defined, so it is possible to shade it in order to create a photorealistic appearance. It is also easy to generate two-dimensional orthographic views of the object. Solid modeling is like the sculpting of a virtual solid volume of material. Because the volume of the object is properly represented in a solids model, it is possible to slice through the object and show a view of the object that displays the interior detail (sectional views). Once several solid objects have been created, they can be assembled in a virtual environment to confirm their fit and to visualize the assembled product. Solids models are useful for purposes other than visualization. The solids model contains a complete mathematical representation of the object, inside and out. This mathematical representation is easily converted into specialized computer code that can be used for stress analysis, heat-transfer analysis, fluid-flow analysis, and computer-aided manufacturing.

Getting Started in SolidWorks

Introduction and Reference

SolidWorks Corporation developed SolidWorks® as a three-dimensional, feature-based, solids-modeling system for personal computers. Solid modeling represents objects in a computer as volumes, rather than just as collections of edges and surfaces. Features are three-dimensional geometries with direct analogies to shapes that can be machined or manufactured, such as holes or rounds. Feature-based solid modeling creates and modifies the geometric shapes of an object in a way that represents common manufacturing processes. This makes SolidWorks a very powerful and effective tool for engineering design.

As with other computer programs SolidWorks organizes and stores data in files. Each file has a name followed by a period (dot) and an extension. There are several file types used in SolidWorks, but the most common file types and their extensions are

Part files	.prt or .sldprt
Assembly files	.asm or .sldasm
Drawing files	.drw or .slddrw

Part files are the files of the individual parts that are modeled. Part files contain all of the pertinent information about the part. Because SolidWorks is a solids-modeling program, the virtual part on the screen will look very similar to the actual manufacture part.

Assembly files are created from several individual part files that are virtually assembled (in the computer) to create the finished product.

Drawing files are the two dimensional engineering drawing representations of both the part and assembly file. The drawings should contain all of the necessary information for the manufacture of the part, including dimensions, part tolerances, and so on.

The part file is the ***driving*** file for all other file types. The modeling procedure begins with part files. Subsequent assemblies and drawings are based on the original part files. One advantage of SolidWorks files is the feature of dynamic links. Any change to a part file will automatically be updated in any corresponding assembly or drawing file.

Therefore, both drawing and assembly files must be able to find and access their corresponding part files in order to be opened. SolidWorks uses information embedded within the file and the filename to maintain these links automatically.

Starting SolidWorks

SolidWorks runs on computers running the Microsoft Windows® operating system. You open SolidWorks in the same way that you would start any other program.

Checking the Options Settings

The SolidWorks window that appears on the computer screen looks similar to the standard Microsoft Windows interface, as shown in Figure 2. The top line of the window is the Menu bar from which menus for various operations can be opened. Below the Menu bar are the toolbars which provide access to a variety of commonly used operations, or tools, with a single click of the mouse button. Toolbars can also extend down the right and left sides of the window. They may or may not be shown on your screen. At this point, most items within the toolbars are grayed out. This indicates that they are not presently available for use.
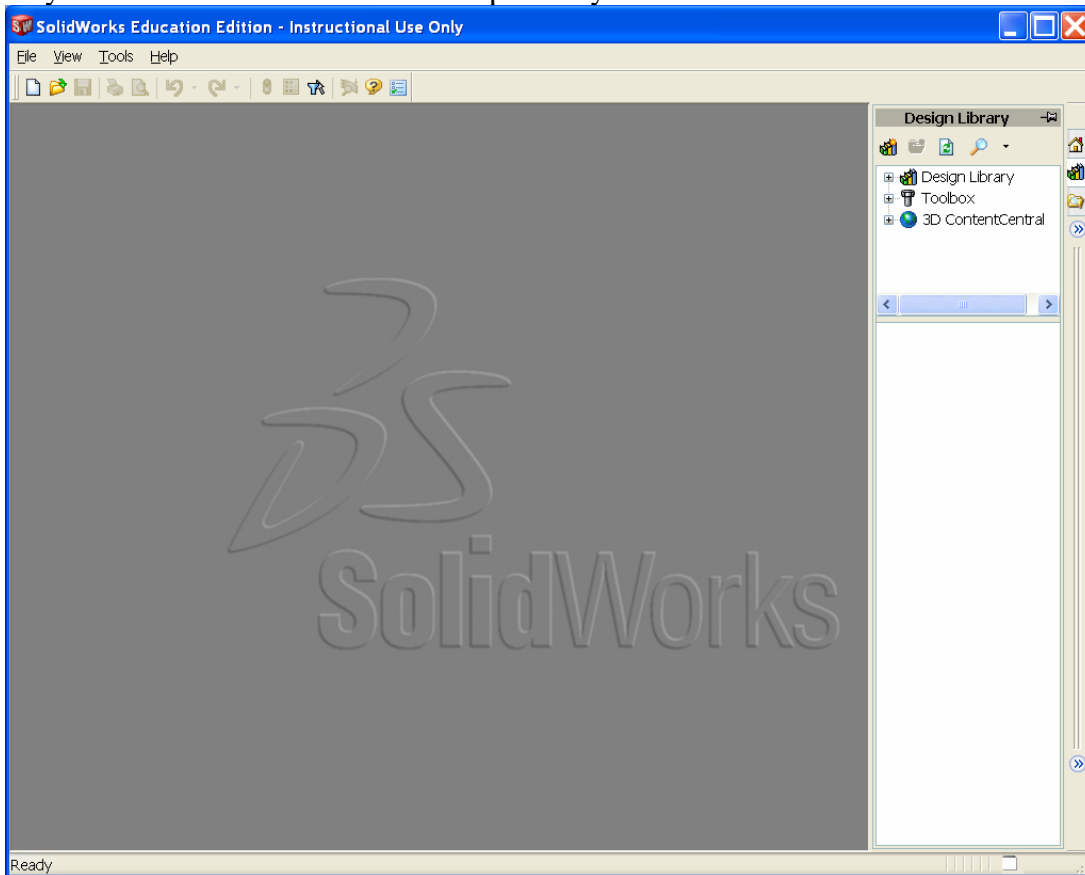
Figure 2 SolidWorks window.

Tool bars

To specify which toolbars are displayed on the screen, select View **Tool-bars** (i.e., select **Toolbars** from the **View** menu). Be sure that the **Features, Sketch, Standard, Standard Views** and **View** toolbars are checked, as shown in Figure 3. If they are not, click on each of these items until all are checked. If other toolbars are checked, click on them to uncheck them. It may be necessary to select **View,** then **Toolbars** again to display the menu after checking (or un-checking) an item to confirm that the desired change was made. The **Standard Views toolbar** may appear as a dialog box in the Graphics Window instead of as a toolbar. If so, click the blue bar at the top of the dialog box with the left mouse button and drag it to the toolbar at the upper right of the Graphics Window. Release the mouse button. The dialog box should change to a toolbar.
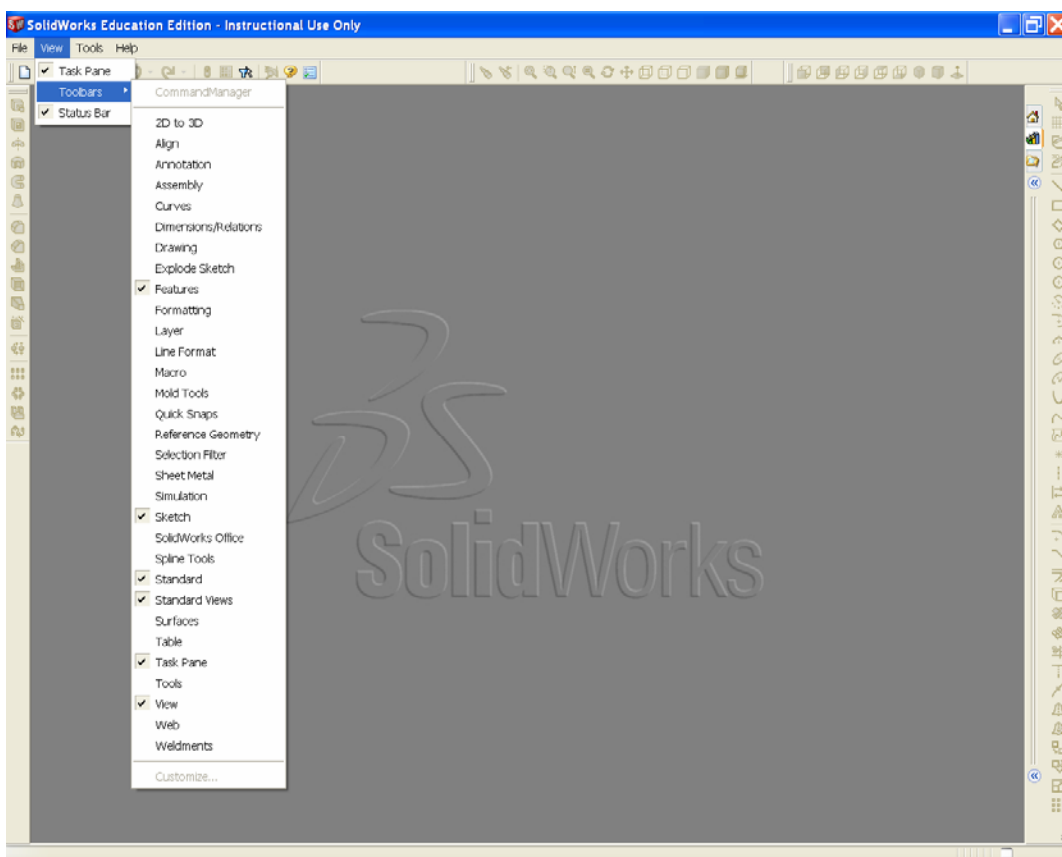
Figure 3 Toolbars menu.

- The **<u>Sketch</u>** toolbar contains tools to draw lines, circles, rectangles, arcs, and so on.

- The **<u>Features</u>** toolbar contains tools that modify sketches and existing features of a part.

- The **<u>Standard</u>** toolbar contains the usual commands available for manipulating files (Open, Save, Print, and so on), editing documents (Cut, Copy, and Paste), and accessing Help.

- The **Standard Views** toolbar contains common orientations for a model.

- The **View** toolbar contains tools to orient and rescale the view of a part.

You can find these toolbars around the Graphics Window by checking and un-checking them in the **View ⇒ Toolbars** menu. The toolbars will appear as you check them and disappear as you uncheck them. Currently, most of the items in the tool-bars are grayed out, since they are unusable. They will become active when they are available for use. Be sure that the **Toolbars** menu looks like the one in Figure 3 before continuing. Click any open spot in the Graphics Window to close all menus.

Getting Help

If you have questions while you are using SolidWorks, you can find answers in several ways:

- Click **SolidWorks Help Topics** in the **Help** Menu bar.

- Move the cursor over a toolbar button to see the ToolTip, which indicates the name of the tool.

- Move the cursor over buttons or click menu items. The Status bar at the bottom of the SolidWorks window will provide a brief description of the function.

- Click the *Help* button in a dialog box.

- Refer to the *Solid Works User's Guide,* by SolidWorks Corporation, for detailed information.

Creating a New Part

With the SolidWorks window open, select **File ⇒ New** in the Menu bar, or click the New button (a blank-sheet icon) in the **Standard** toolbar.

The *New Solid Works Document* dialog box appears as shown in Figure 4. You will be modeling a new part. If *Part* is already highlighted, click *OK.* If **it is not** highlighted, click *Part,* then *OK.*
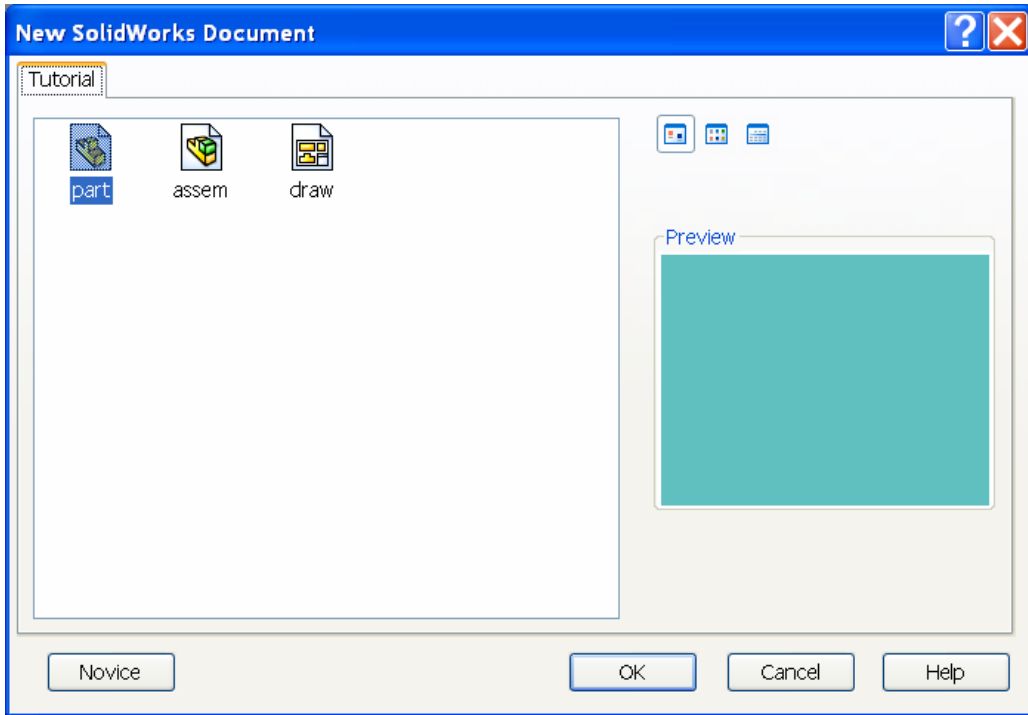
Figure 4 New document window.

A new window appears with the name Partl, as shown in Figure 5. On the left side of the window is the Feature Manager Design tree.


Sketch toolbar
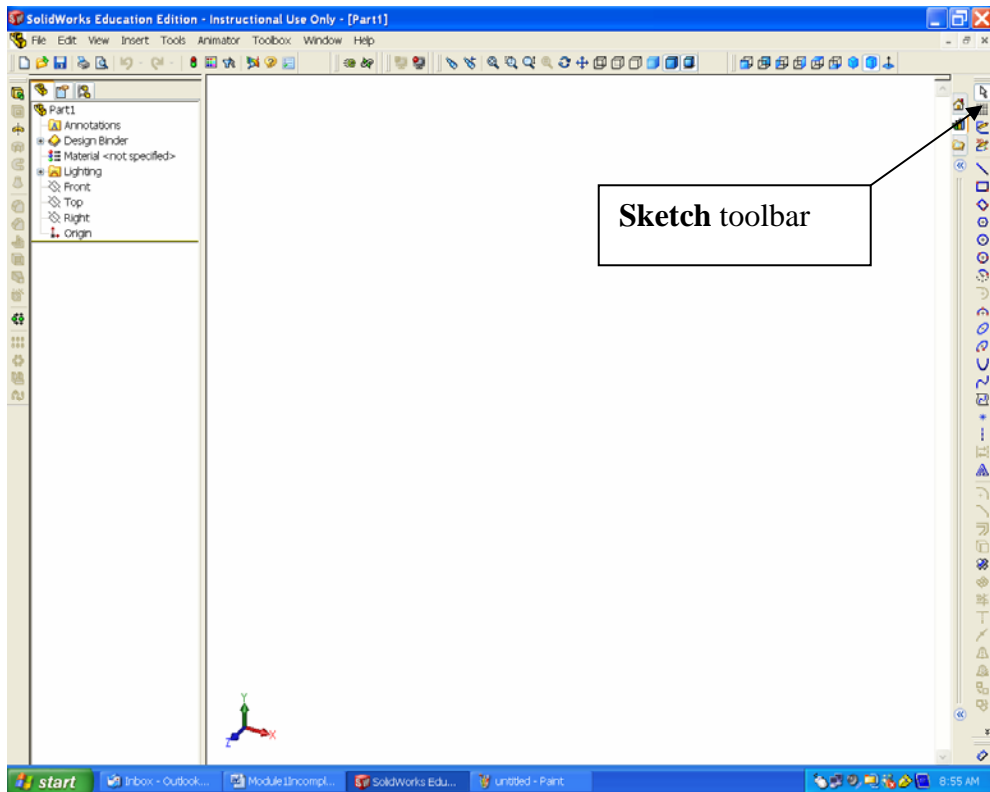
Figure 6 New part window.

It contains a list of the features that have been created so far. Every new part starts with six features: annotations, lighting, three datum planes, and an origin. The datum planes are three mutually perpendicular planes that are created in space as references for constructing features of the part that you are modeling. The three planes intersect at the origin, which is in the center of the Graphics Window. The arrows in the lower left corner of the Graphics Window show the coordinate directions. As the part is modeled, the features that are created will appear in the Feature Manager Design tree. These features can be highlighted or modified by clicking on them in the Feature Manager Design tree. For example, click on a plane or the origin in the Feature Manage Design tree to highlight these items. **Front** is the plane of the screen, **Top** is the horizontal plane perpendicular to the screen, and **Right** is the vertical plane perpendicular to the screen. Finish by clicking on Part1 in the Feature Manager Design tree, so that no plane is highlighted.

Sketching

Every part begins as a cross section sketched on a two-dimensional plane. Once a sketch is made, it is extruded or revolved into the third dimension to create a three-dimensional object. This is the base feature of the part.

The **Sketch** toolbar, shown in Figure 6, has tools to set up and manipulate a sketch of a cross section. Find the **Sketch** toolbar. Move the cursor over each of the tools, but do not click on any of the tools. The Tool Tips should appear, displaying the name of each tool.

- **Select** highlights sketch entities; drags sketch entities and endpoints and modifies dimension values.

- **Grid** activates the *Grid/Snap* field of the *Document Properties* dialogue box to change the sketching environment.

- **Dimension** adds dimensions to sketch entities.

- **Sketch** opens and closes sketches as a part is created.
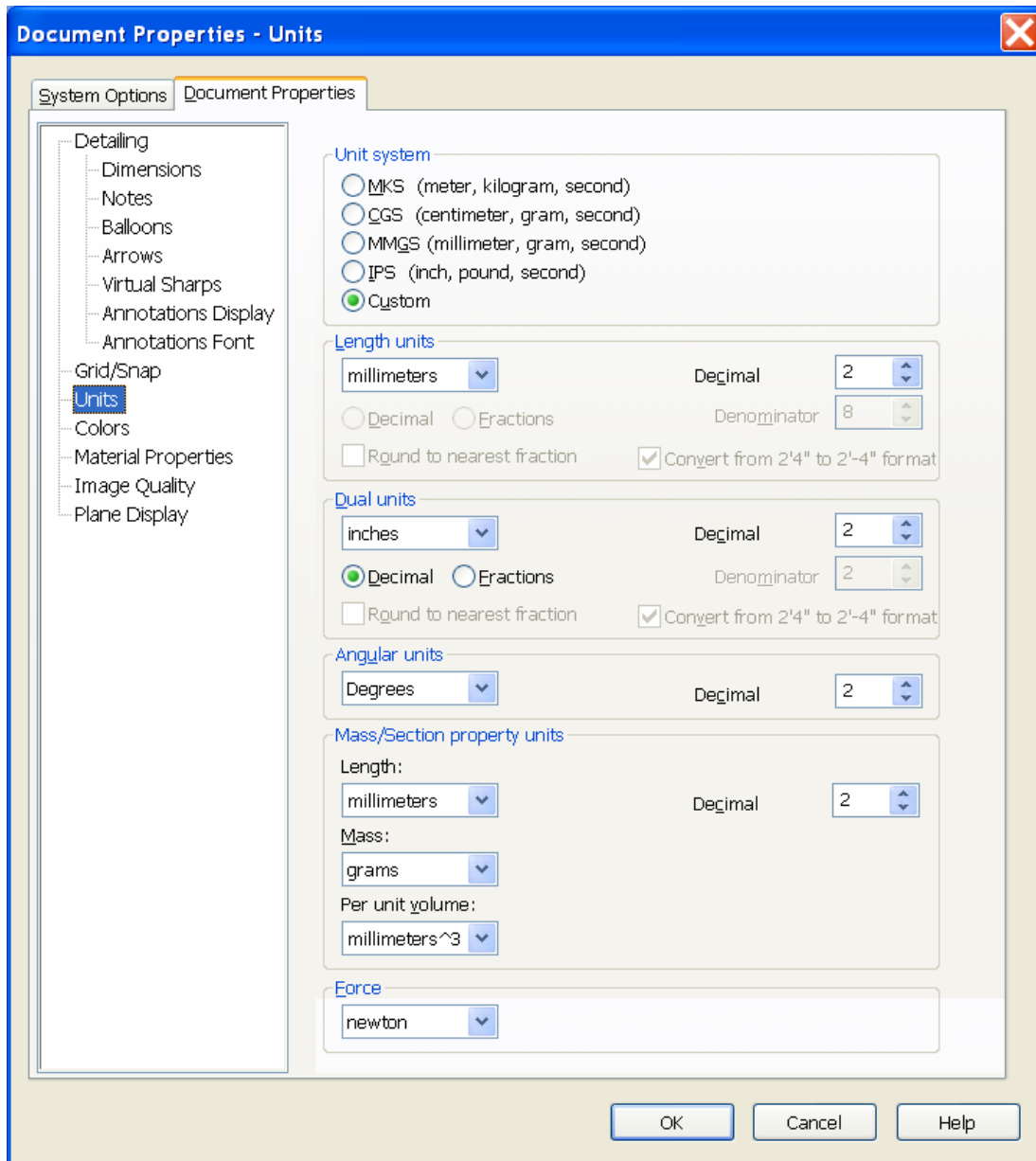
Figure 6 Document properties dialog box: *Units*

To set the units and grid size to be used, click the **Grid** toolbar button with the left mouse button. Document properties dialog box shown in Figure 6 will appear. Click *Units* on the left side of the dialog box to set the units. Setup appropriate units (*inches* or *mm*) with the desired number of decimal places or fraction denominator.

Click *Grid/Snap* on the left side of the dialogue box to control the grid that will appear on the screen when a cross-section is sketched. Be sure that all three of the boxes under *Grid* are checked, as shown in Figure 7. Adjust the grid spacing to desired values.
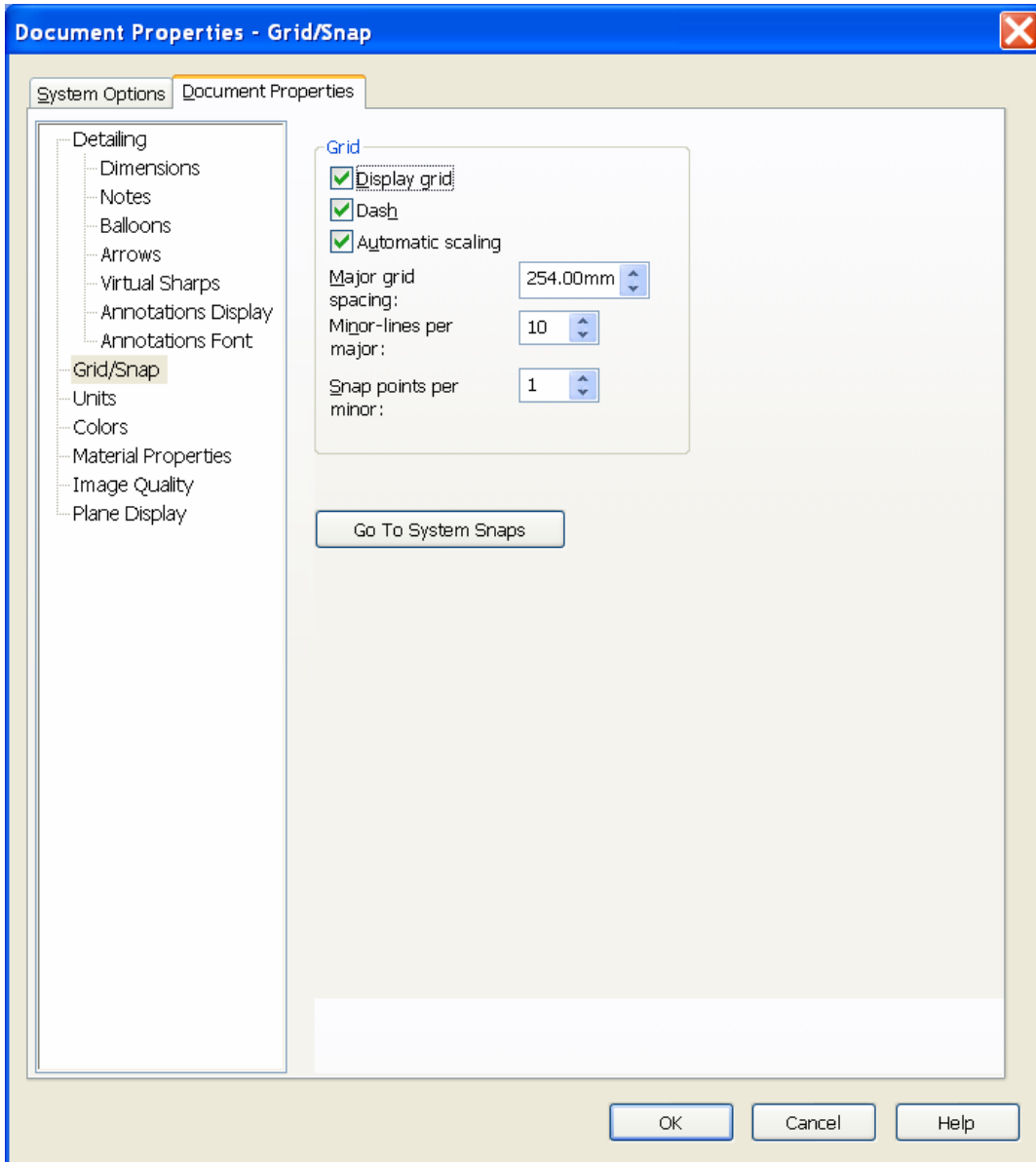
Figure 7 Document properties dialog box: *Grid/Snap*

Snap controls the way in which sketched lines are related to the grid. The points that are sketched should "snap" to the nearest intersection of grid lines when they are close. Click **Go To System Snaps**. If not, click the box **Grid** to check it as shown in Figure 8.
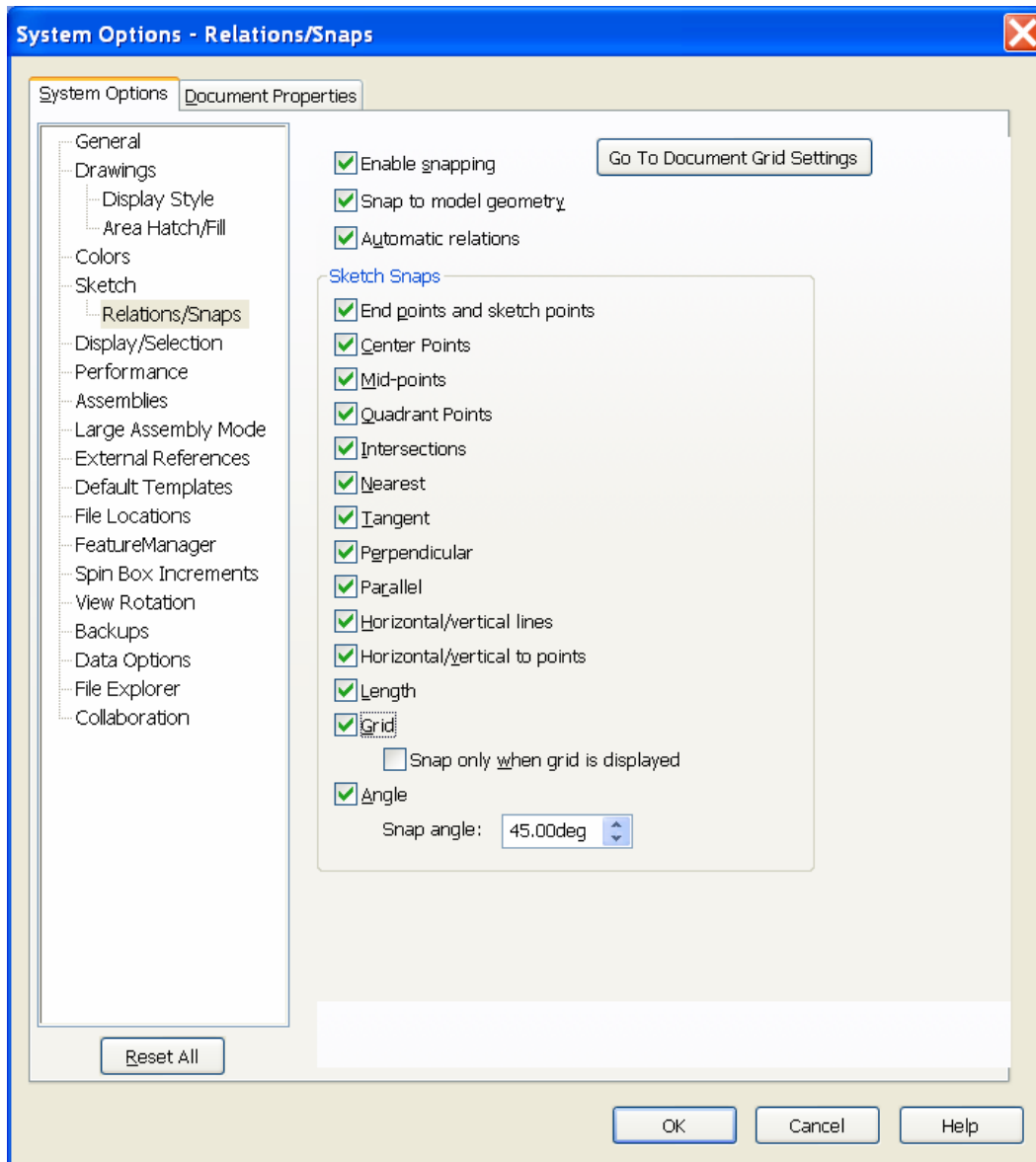
Figure 8 Document properties dialog box: **System Snap**

Click *Detailing* on the left side of tile dialogue box. If necessary, change *Dimensioning standard* to *ANSI* for inch units and ISO in case of mm units. Then, click *OK* at the bottom of the dialogue box to accept the values.

Open a new sketch by selecting **Insert Sketch,** or by clicking the **<u>Sketch</u> button (a pencil** drawing a line) in the **<u>Sketch</u>** toolbar. (Note that, for most commands in SolidWorks, it is possible to initiate the command from either the Menu bar or the toolbars.) A grid should appear on the screen, as shown in Figure 9, indicating that the sketch mode is active. The window's name changes to **Sketch of Part1.** In the bottom right corner of the screen, the Status bar reads *Editing Sketch.* You are now ready to sketch in the **Front** plane.

The **<u>Sketch</u>** toolbar, shown in Figure 9, contains tools to create and modify two-dimensional features, called Sketch Entities. Sketch Entities are items that can be drawn

on the sketch. The following Sketch Entities and Sketch Tools are available:
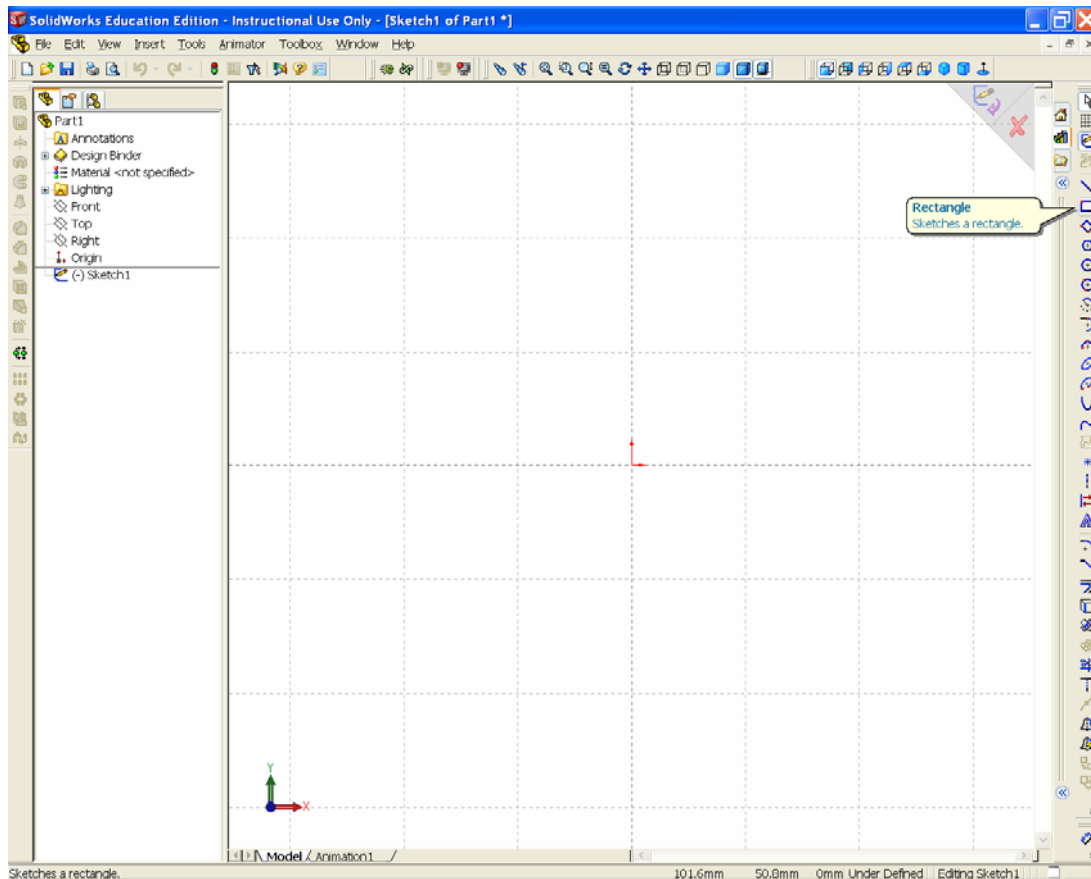
Figure 9 New sketch window.

- **Line** creates a straight line.

- **Centerpoint Arc** creates a circular arc from a center point, a start point, and an end point.

- **Tangent Arc** creates a circular arc tangent to an existing sketch entity.

- **3 Pt Arc** creates a circular arc through three points.

- **Circle** creates a circle.

- **Spline** creates a curved line that is not a circular arc.

- **Polygon creates** a regular polygon.

- **Rectangle** creates a rectangle.

- **Point** creates a reference point that is used for constructing other sketch entities.

- **Centerline** ⋮ creates a reference line that is used for constructing other sketch entities.

- **Convert Entities** ⬜ creates a sketch entity by projecting an edge, curve, or contour onto the sketch plane.

- **Mirror** ⌂ reflects entities about a centerline.

- **Fillet** ⌐ creates a tangent arc between two sketch entities by rounding an inside or an outside corner.

- **Offset Entities** � creates a sketch curve that is offset from a selected sketch entity by a specified distance.

- **Trim** ✂ removes a portion of a line or curve.

- **Construction Geometry** creates entities that aid in sketching.

- **Linear Sketch Step and Repeat** creates a linear pattern of sketch entities.

- **Circular Sketch Step and Repeat** creates a circular pattern of sketch entities.

Move and hold the cursor over each of the tools to display its function but do not click on the tool. Note the description of each tool in the Status bar at the bottom of the SolidWorks window. Some of these tools may not he included in the toolbar or other tools may be available, depending on the way in which it was previously set up. All tools are available in the **Tools** Menu.