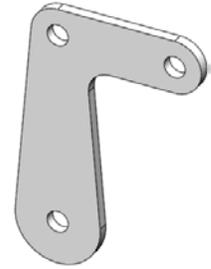
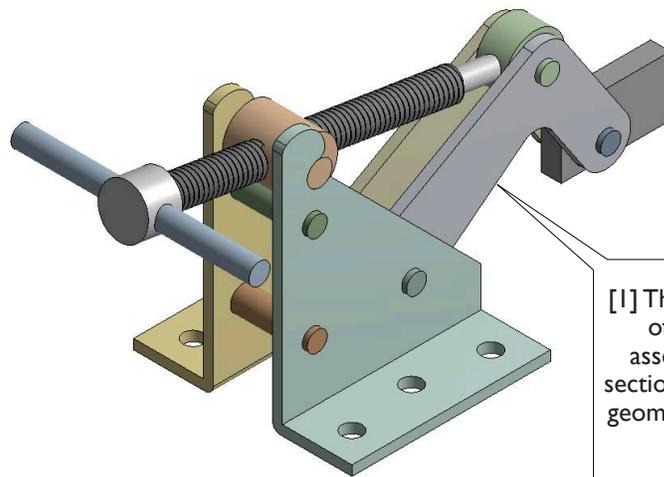


# Section I.I

## Arm

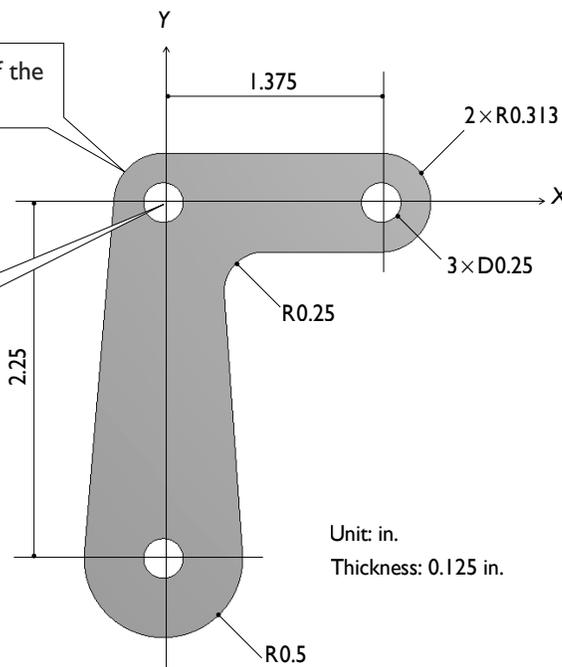


### I.I-1 About the Arm



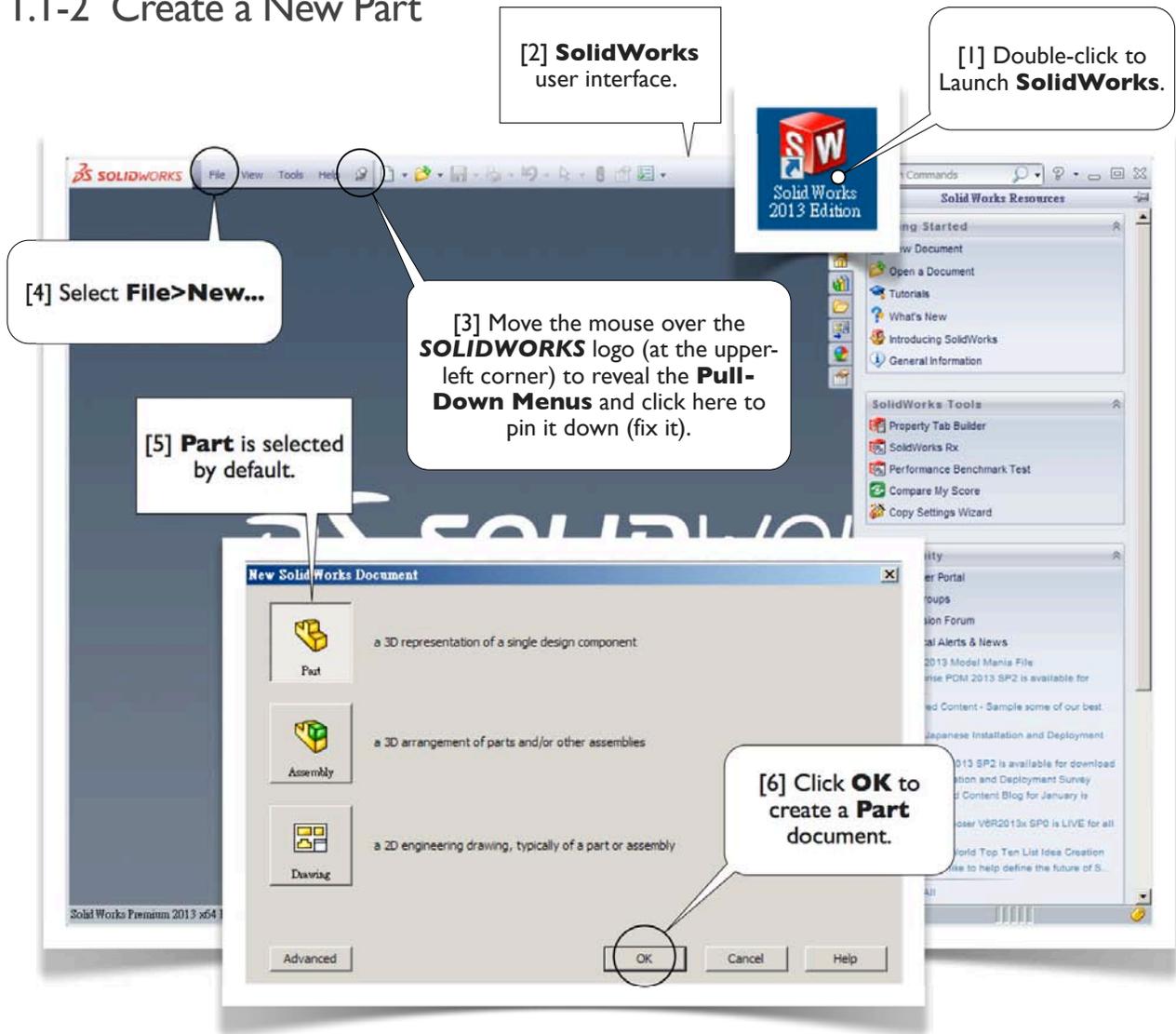
[1] The arm is a part of a clamping assembly. In this section, we'll create a geometric model for the arm.

[2] Details of the arm.



[3] Before creating a geometry model, we must set up a global coordinate system for the model. Here, the XY-plane is coincident with the back surface of the part.

## I.1-2 Create a New Part



### About the Text Boxes

1. Within each subsection (e.g., I.1-2), text boxes are ordered with numbers, each of which is enclosed by a pair of square brackets (e.g., [1]). When you read the contents in a subsection, please follow the order of the text boxes.
2. The text box numbers also serve as reference numbers when referred from other text boxes. In the same subsection, we simply refer to a text box by its number (e.g., [1]). In other subsections, we refer to a text box by its subsection identifier and the text box number (e.g., I.1-2[1]).
3. A text box is either round-cornered (e.g., [1, 3, 4, 6]) or sharp-cornered (e.g., [2, 5]). A round-cornered box indicates that **mouse or keyboard actions** are needed in that step. A sharp-cornered box is used for commentary only: no mouse or keyboard actions are needed in that step.

### SolidWorks Terms

In this book, terms used in **SolidWorks** are boldfaced (e.g., **Pull-Down Menus** in [3]).

### I.1-3 Set Up Units

[1] Select **Tools>Options...**

[2] Click **Document Properties** tab.

[3] Select **Units**.

[4] Select **IPS (inch, pound, second)**.

[5] Select **.123** (three decimal places).

[6] Click **OK**.

[7] The **Options** command is also available by clicking here.

[8] The units also can be set up from here.

Type	Unit	Decimals	Fractions	More
<b>Basic Units</b>				
Length	inches	.123		...
Dual Dimension Length	inches	.123		...
Angle	degrees	.12		
<b>Mass/Section Properties</b>				
Length	inches	.12		
Mass	pounds			
Pier Unit Volume	inches <sup>3</sup>			
<b>Motion Units</b>				
Time	second	.12		
Force	pound-force	.12		
Power	watt	.12		
Energy	BTU	.12		

## I.1-4 Draw a Circle

[1] In the **Features Tree** (on the left side of the user interface), right-click **Front** and select **Sketch**.

[2] The **Front** plane (XY-plane) is ready for sketching.

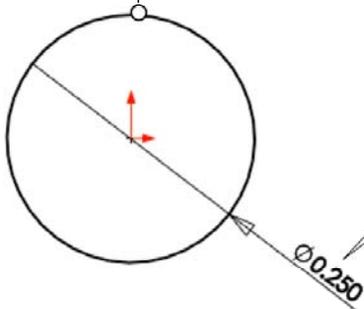
[3] Right-click anywhere in the **Graphics Area** to display a **Context Menu**, and select **Circle** command.

[4] Click at the origin and drag outward to create a circle. Press **ESC** to dismiss the **Circle** command. Then, press **ESC** again (or click anywhere in the **Graphics Area** other than the circle) to de-select the circle. Note that the circle is blue-colored (see [5]).

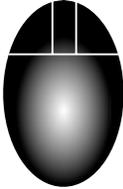
[5] **Color Codes of Sketch Entities**  
 A sketch entity is blue-colored when it is not well-defined yet [4]. A well-defined entity (i.e., fixed in the space) becomes black (see [7], next page). When over-defined, an entity becomes red.

[6] Right-click anywhere in the **Graphics Area** and select **Smart Dimension**.

[7] Click the circle and move lower-rightward to create a diameter; type 0.25 (in) for the diameter. Note that the circle now turns black. Use mouse functions to zoom in/out [8] or pan the sketch [9]. Drag the dimension to a location like this. Finally, press **ESC** to dismiss the **Smart Dimension**.



[8] Scrolling the **Mouse Wheel** allows you to zoom in/out the sketch.



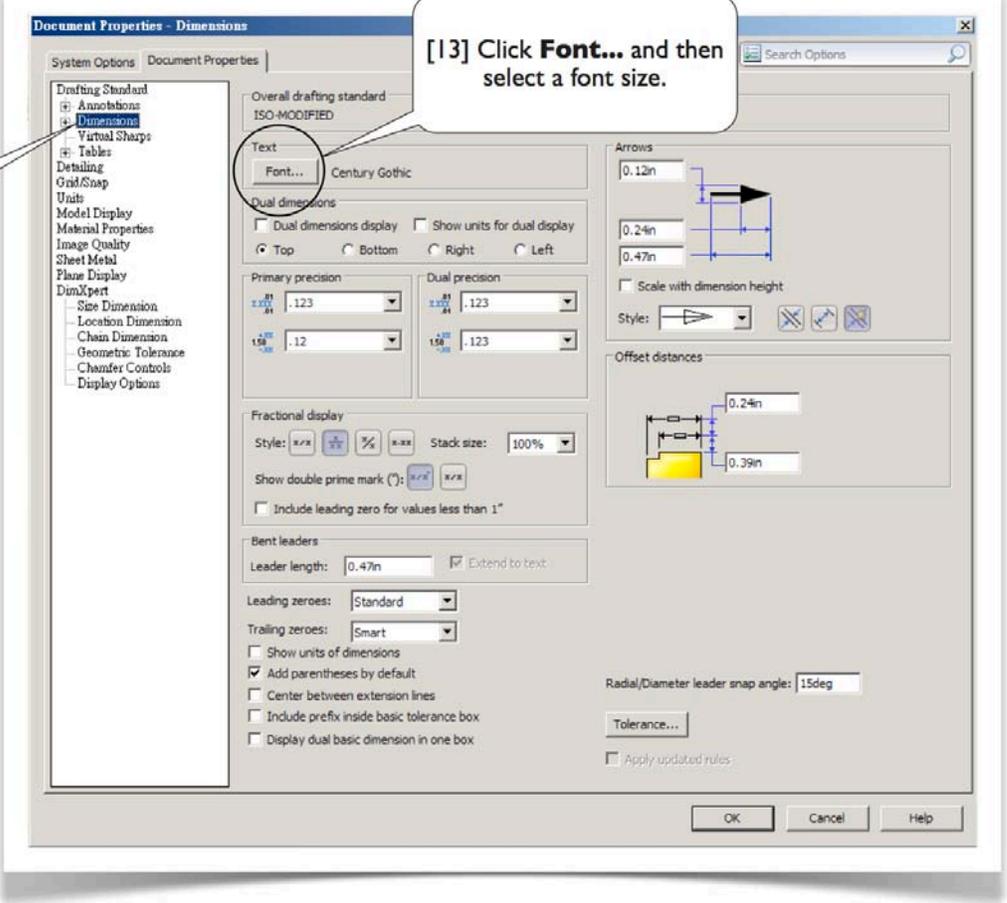
[9] Dragging the mouse with **Control-Middle-Button** allows you to pan the sketch.

[11] If you think the font size of the dimension text is too large, it can be changed (see [12]).

[10] If you made a mistake, you always can **Undo** the mistake.



[12] To change the font size of dimension texts, select **DIMENSION** in the **Document Properties** (1.1-3[1, 2]).



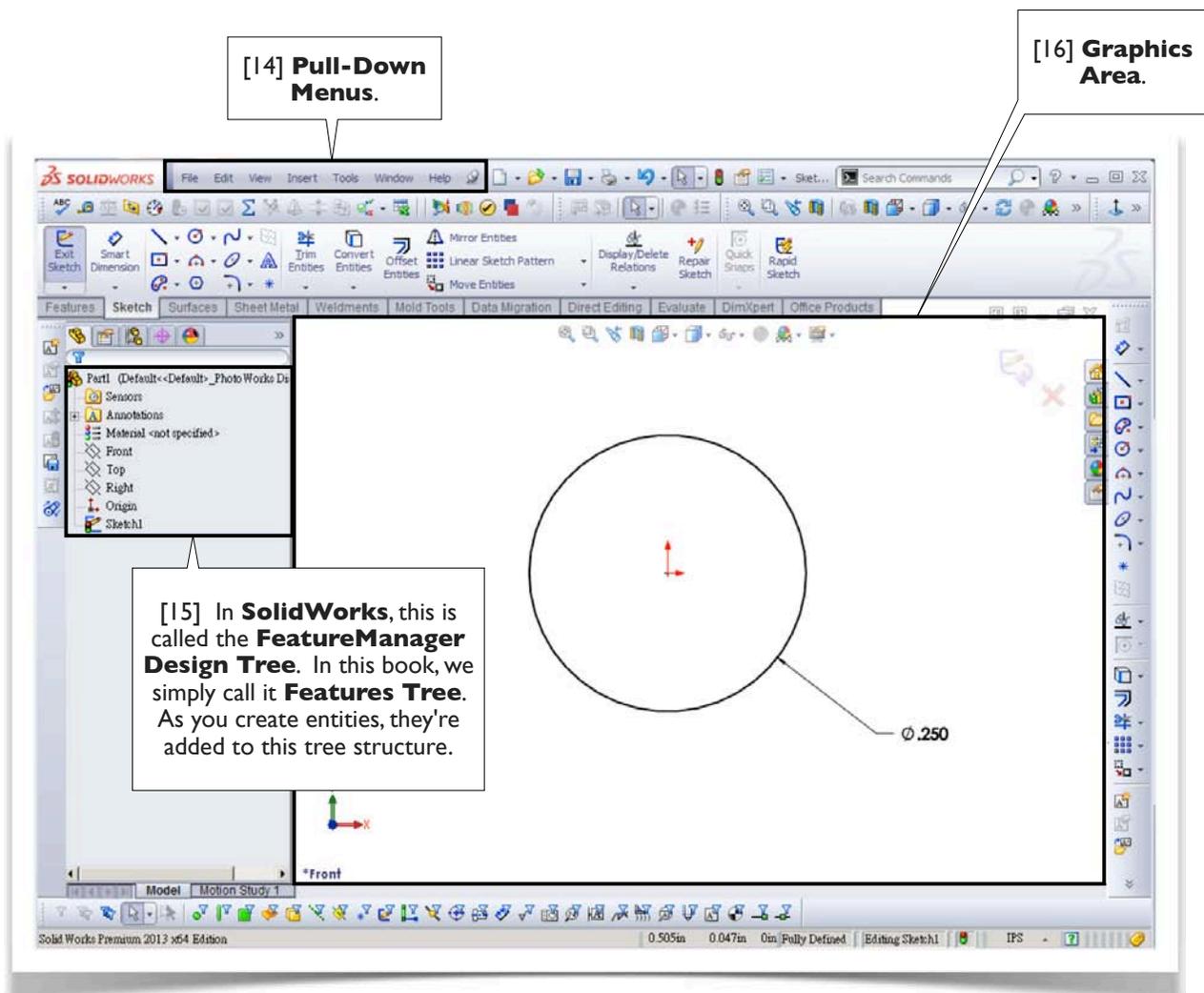
[13] Click **Font...** and then select a font size.

## SolidWorks Commands

There are literally hundreds of **SolidWorks** commands (tools). All commands can be found in the **Pull-Down Menus** [14]. Nevertheless, the most intuitive way to issue a command is through a context-sensitive menu, or simply called **Context Menu** [1, 3, 6]. To issue a command with a **Context Menu**, you right-click an object on either the **Features Tree** [15] or the **Graphics Area** [16]. The commands available in a **Context Menu** depend on the kind of object you're working on (that's why it is called a context-sensitive menu). In step [1], the object you were working on is the **Front** plane; in steps [3, 6], the object you were working on is the **Graphics Area**.

After you accumulate some experiences, you may find that a more convenient way to issue a command is simply clicking a command on a **Toolbar** (e.g., [10]). In this book, we roughly follow these rules to issue a command:

1. As novices, we issue a command through a **Context Menu**, because it is the most intuitive way.
2. If a command is not available with a **Context Menu**, we select it from the **Pull-Down Menus**, because it is the most comprehensive way (i.e., all commands can be found there).
3. As we accumulate experiences, we begin to issue a command by clicking a button in a **Toolbar**, because it is the most convenient way.



## I.1-5 Draw Another Circle

[5] Select **Smart Dimension** from the **Sketch Toolbar**.

[2] Select **Circle** from the **Sketch Toolbar**.

[1] Click **Sketch**. The **Sketch Toolbar** shows up.

[4] **Inference Line**.

[3] Move the cursor around the X-axis until an **Inference Line** [4] appears. Click to define the center then click again to define an arbitrary diameter.

[6] Click the two center points one after another and then move the mouse upward and click again to create this horizontal dimension; type 1.375 (in).

[7] In the **Head-Up Toolbar** [8], click **Zoom to Fit** to fit the sketch within the window.

[8] **Head-Up Toolbar** is at the top of the **Graphics Area**.

[16] Select **Add Relation** from the **Context Menu** again.

[9] Press **ESC** to dismiss **Smart Dimension** and select **Add Relation** from the **Context Menu** (1.1-4[3, 6]).

[20] Click **OK** to dismiss the **Property Box**.

[17] In the **Graphics Area**, click the two circles' centers.

[19] A **Horizontal** relation is added, in addition to the existing **Distance** relation.

[18] Click **Horizontal** to make the two points align horizontally.

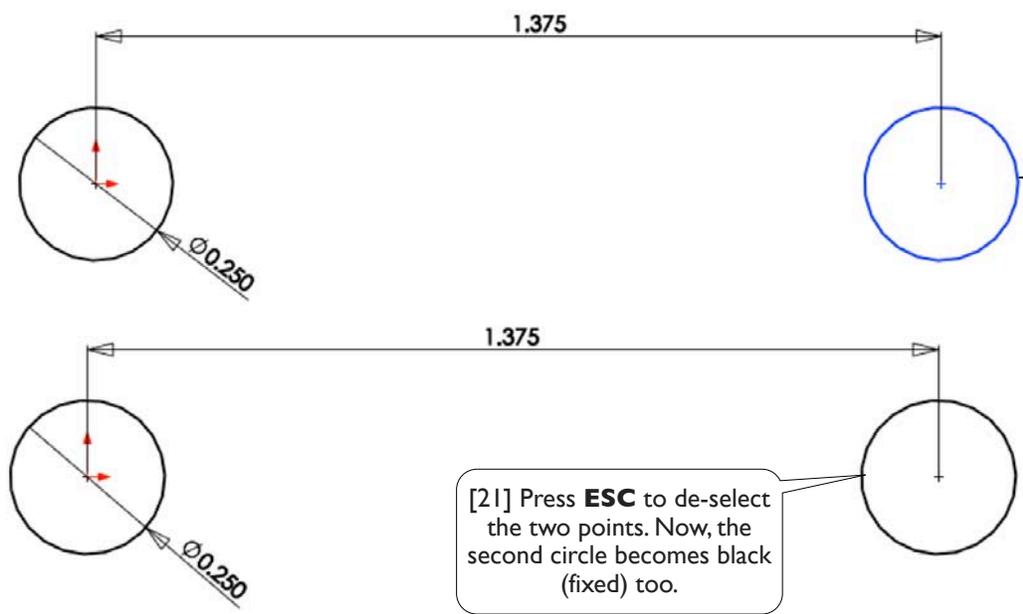
[14] Click **OK** to dismiss the **Property Box**. The **Features Tree** re-appears.

[10] A **Property Box** appears in place of the **Features Tree**.

[11] In the **Graphics Area**, click the two circles one after the other. Note that their names appear here.

[13] A relation between the two entities is added.

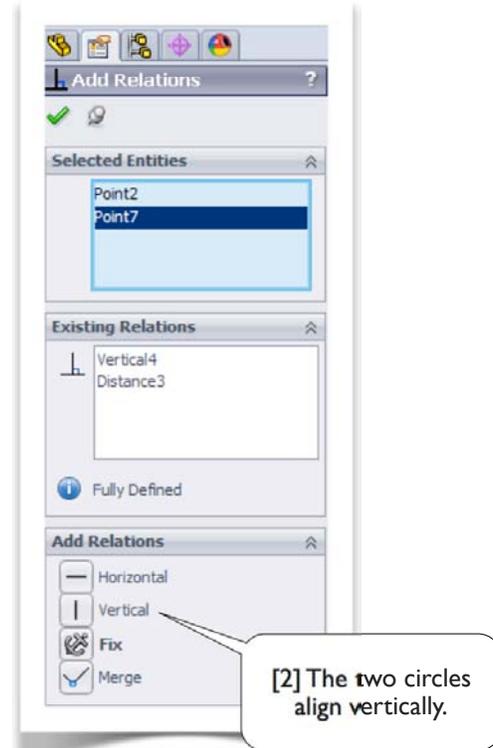
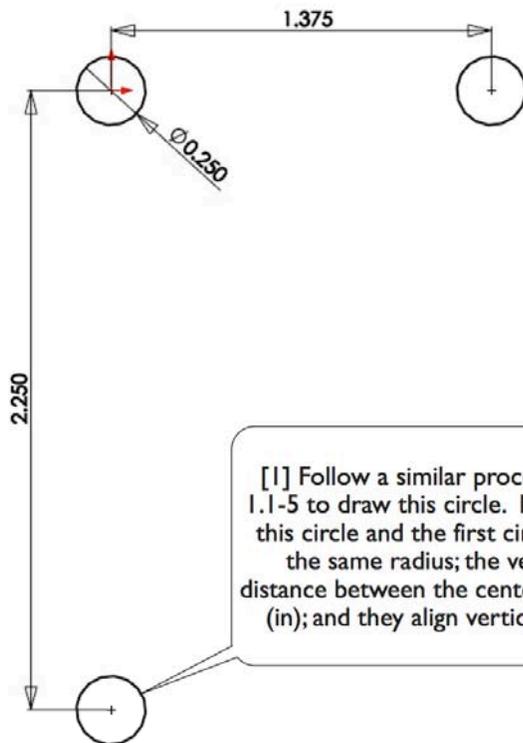
[12] Click **Equal** to make their sizes equal.



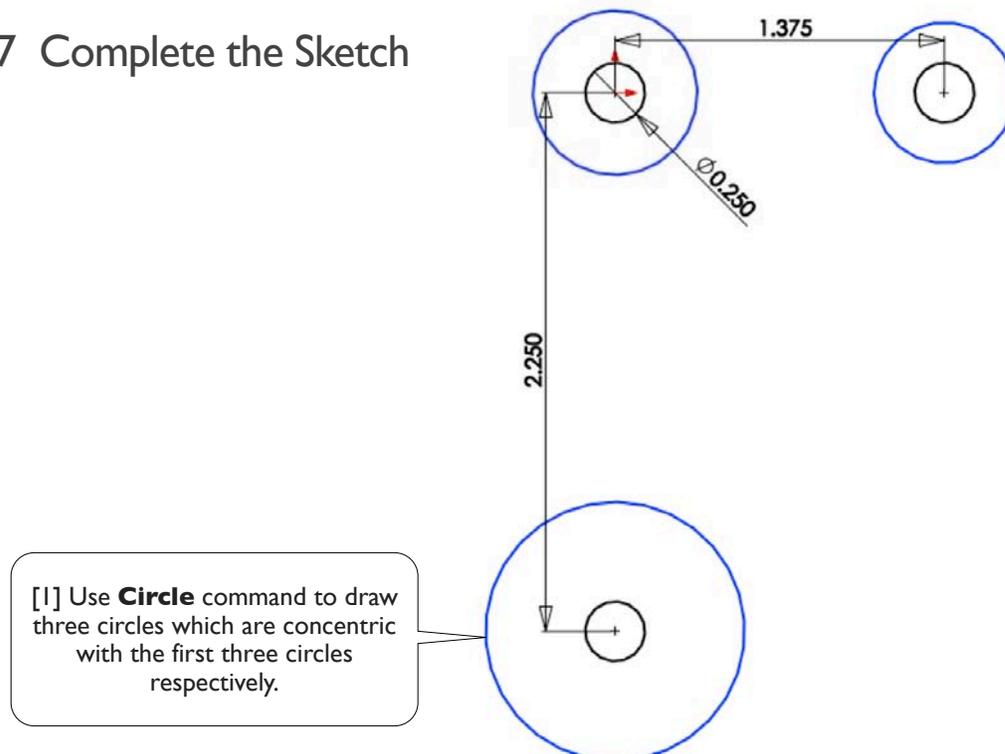
[15] De-select the two circles (press **ESC** or click anywhere on the **Graphics Area**). Now, the two circles have the same radius. The second circle is still blue-colored, meaning that it is not well-defined yet. We now impose another relation.

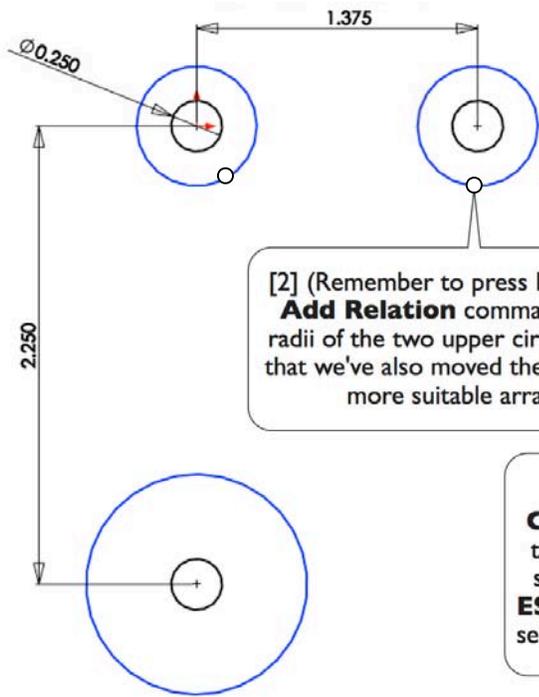
[21] Press **ESC** to de-select the two points. Now, the second circle becomes black (fixed) too.

## 1.1-6 Draw the Third Circle



## 1.1-7 Complete the Sketch



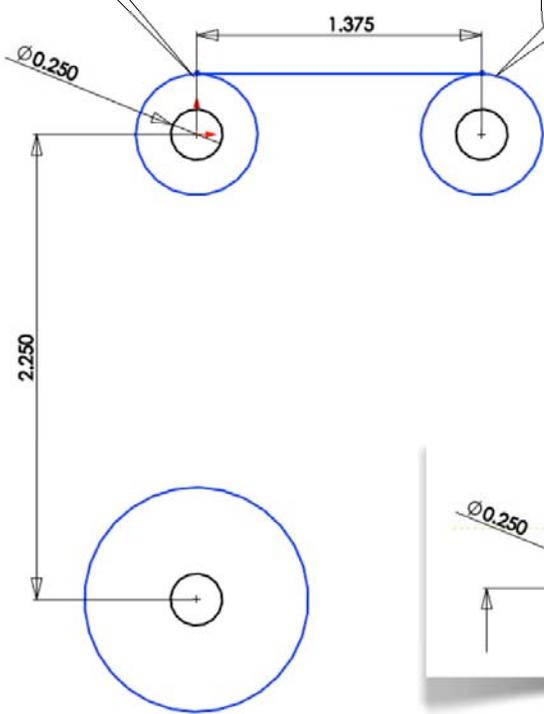


[2] (Remember to press **ESC** twice.) Use **Add Relation** command to make the radii of the two upper circles equal. Note that we've also moved the dimensions to a more suitable arrangement.

[3] Select **Line** command from the **Context Menu**. Before right-clicking to pop-up the **Context Menu**, make sure no command is active (if so, press **ESC** to dismiss it) and no sketch entity is selected (if so, press **ESC** to de-select it).

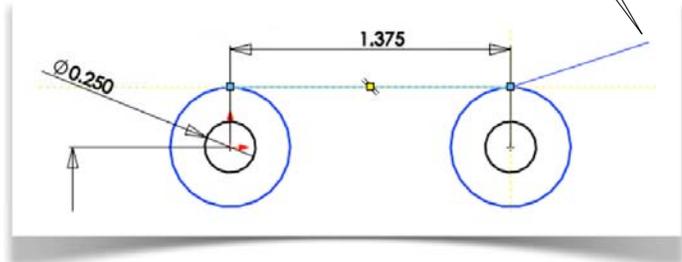


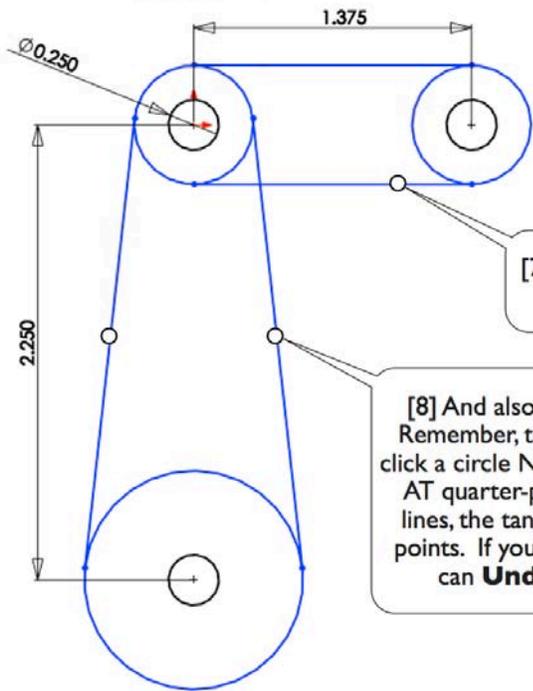
[4] Click this circle near the upper quarter-point...



[5] And click this circle near the upper quarter-point. A tangent line between the two circles is drawn. Double-click to end the line drawing [6].

[6] The **Line** command can be used to draw multiple line segments. To end a session of line drawing without dismiss the **Line** command, simply double-click.

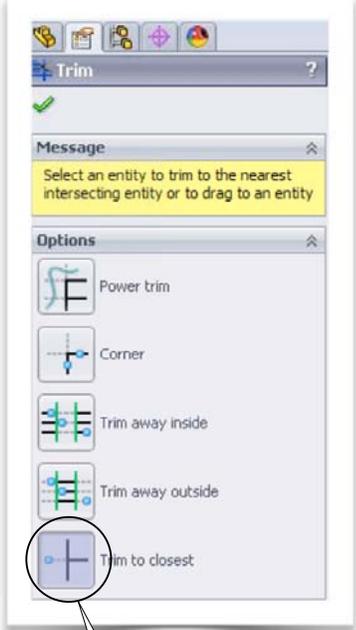




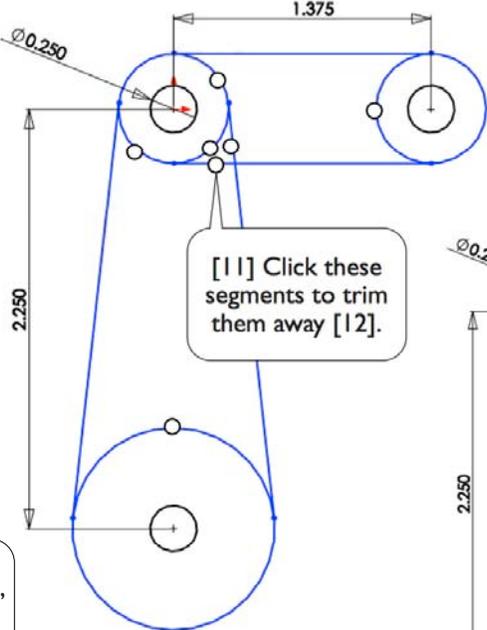
[7] Follow steps [4, 5] to add this tangent line.

[8] And also draw these two tangent lines. Remember, to draw these tangent lines, you click a circle **N**EAR quarter-points (rather than **A**T quarter-points). For these two tangent lines, the tangent points are not at quarter-points. If you made any mistakes, you always can **U**ndo the mistakes (I.1-4[10]).

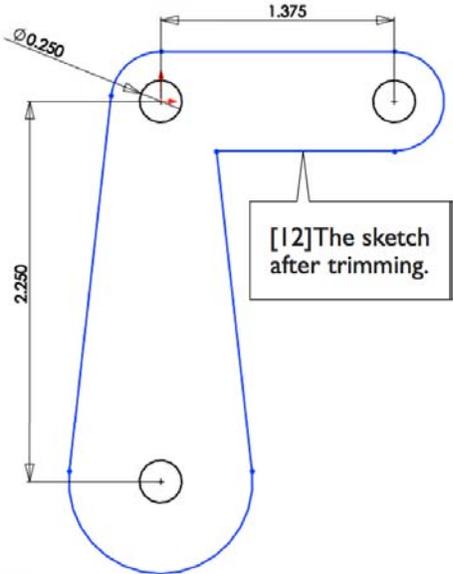
[9] Press **ESC** to dismiss the **L**ine command. Select **T**rim Entities from the **C**ontext Menu.



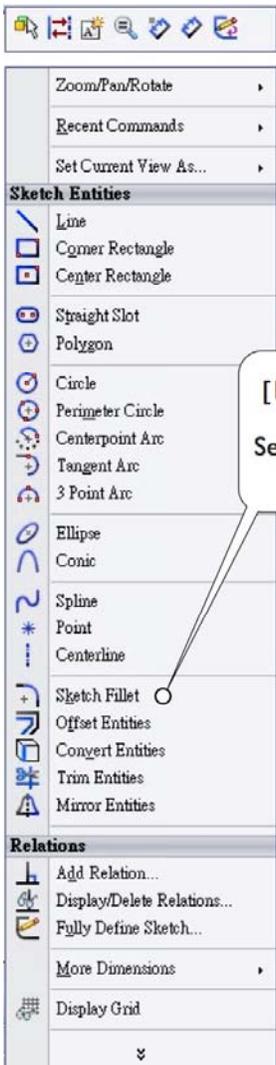
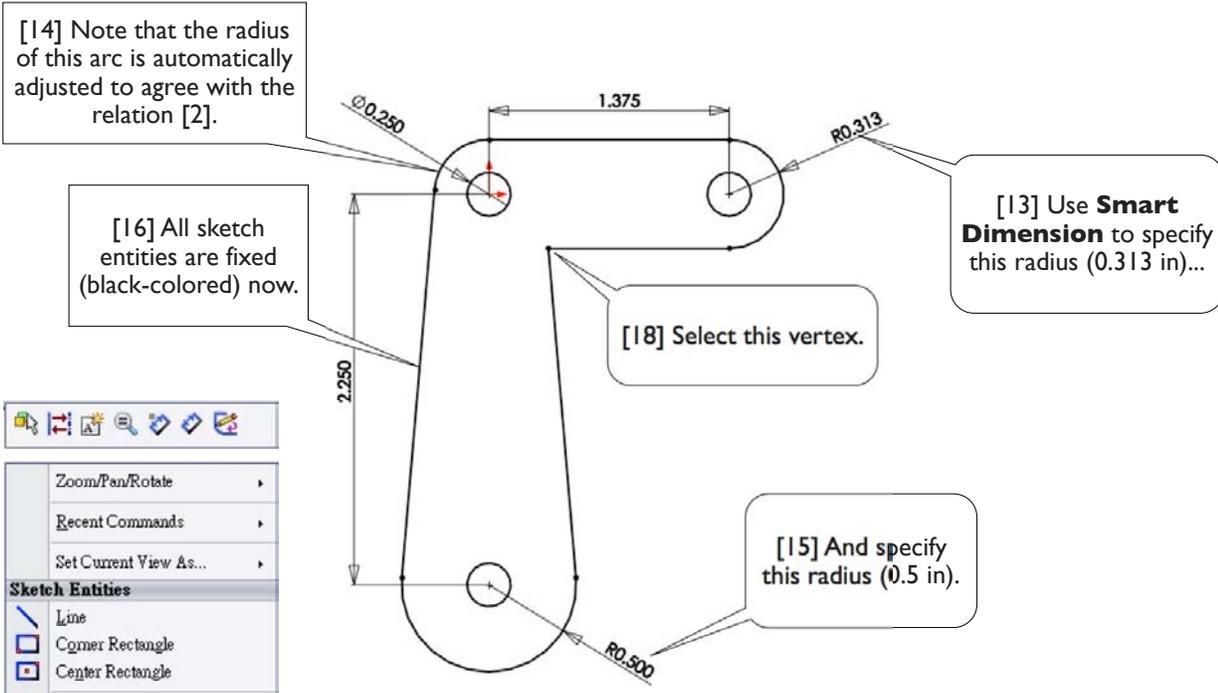
[10] In the **P**roperty Box, select **T**rim to closest.



[11] Click these segments to trim them away [12].

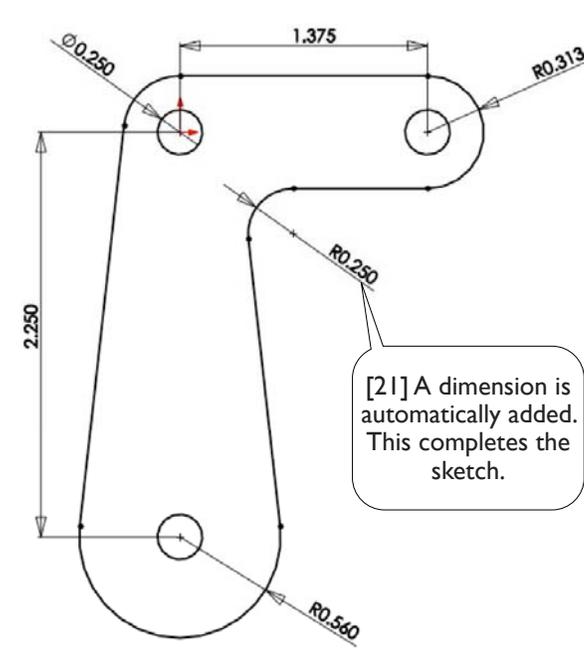


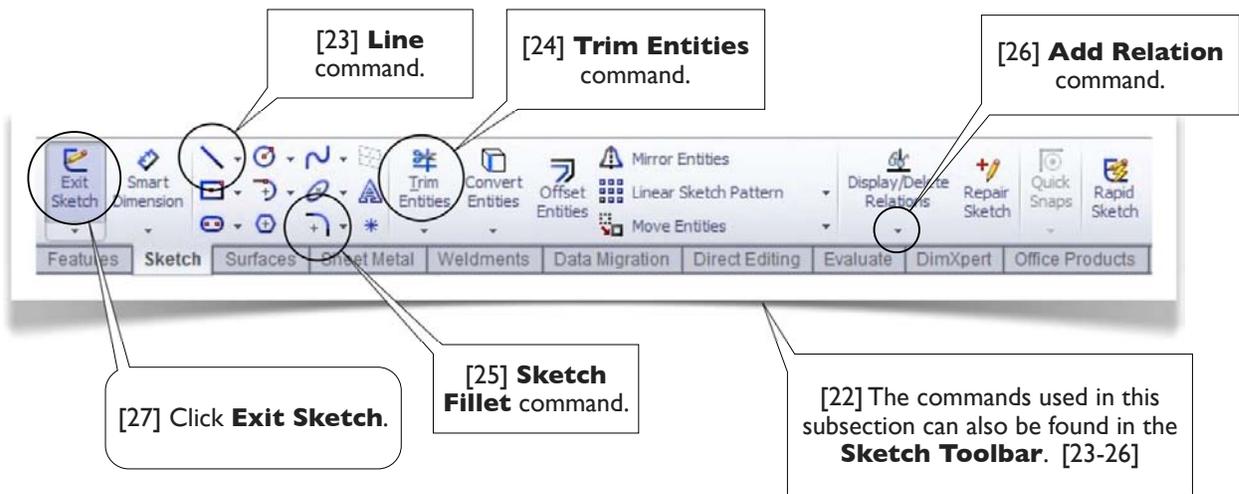
[12] The sketch after trimming.



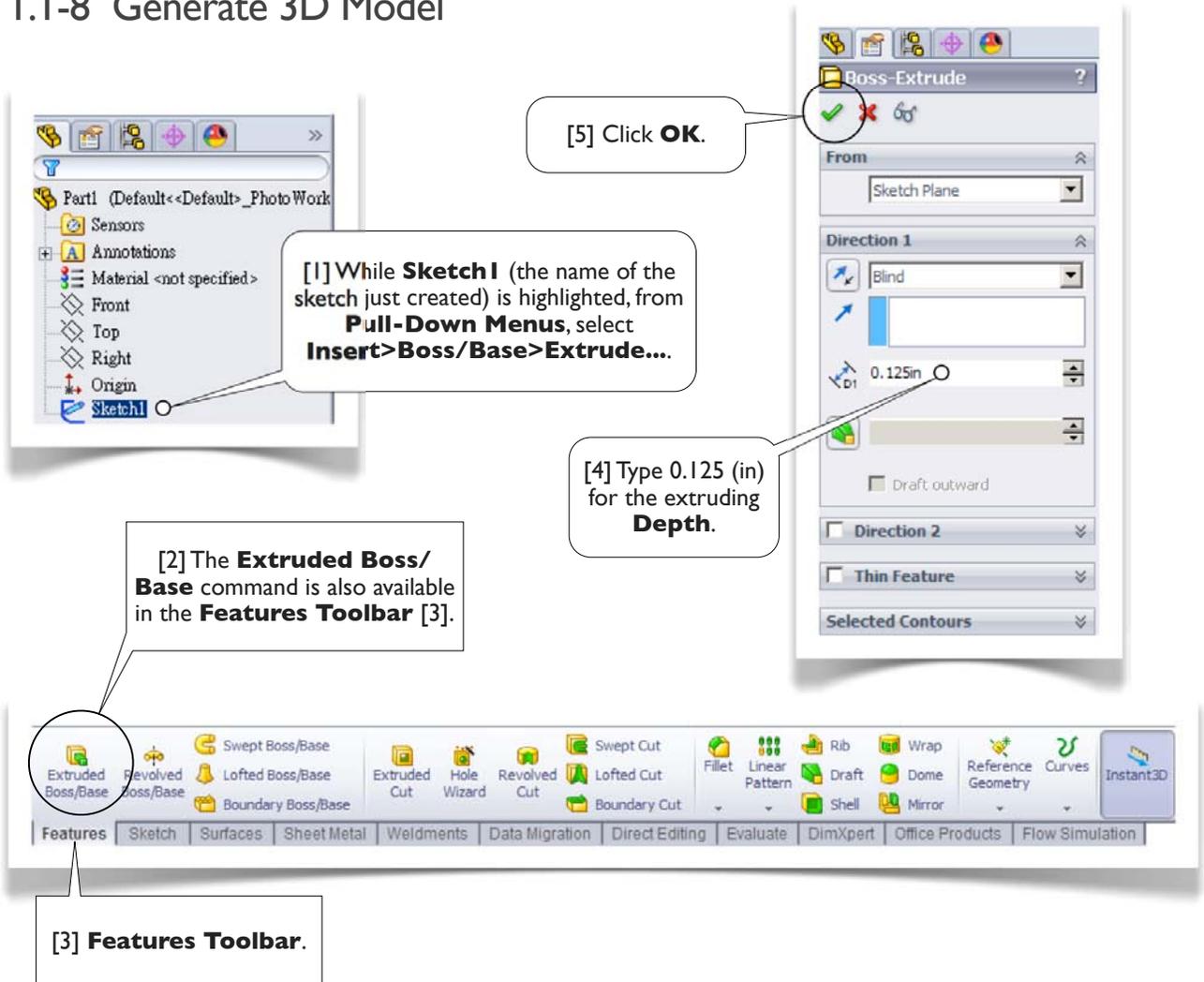
[17] Press **ESC** to dismiss **Smart Dimension**. Select **Sketch Fillet** from the **Context Menu**.

[20] Click **OK** to accept the properties.

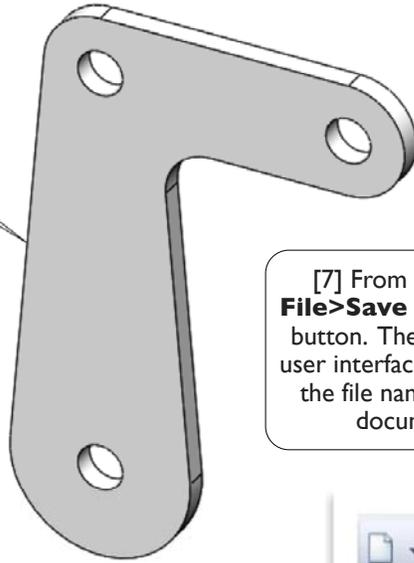




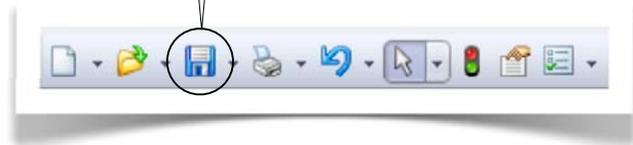
### I.1-8 Generate 3D Model



[6] The finished 3D model.



[7] From **Pull-Down Menu**, select **File>Save** or, on the **Toolbar**, click **Save** button. The **Toolbar** is on the top of the user interface. Save this part document with the file name **Arm**. The full name of the document is **Arm.SLDPRT**.



## I.1-9 Wrap Up



[1] Select **File>Close** from the **Pull-Down Menu** to close the part document. Or, you may select **Window** menu and click here.

[2] Select **File>Exit** from **Pull-Down Menu** to quit **SolidWorks**.