

## Extruded Base and Extruded Cut

### 1. Start SolidWorks and Create a Part Document

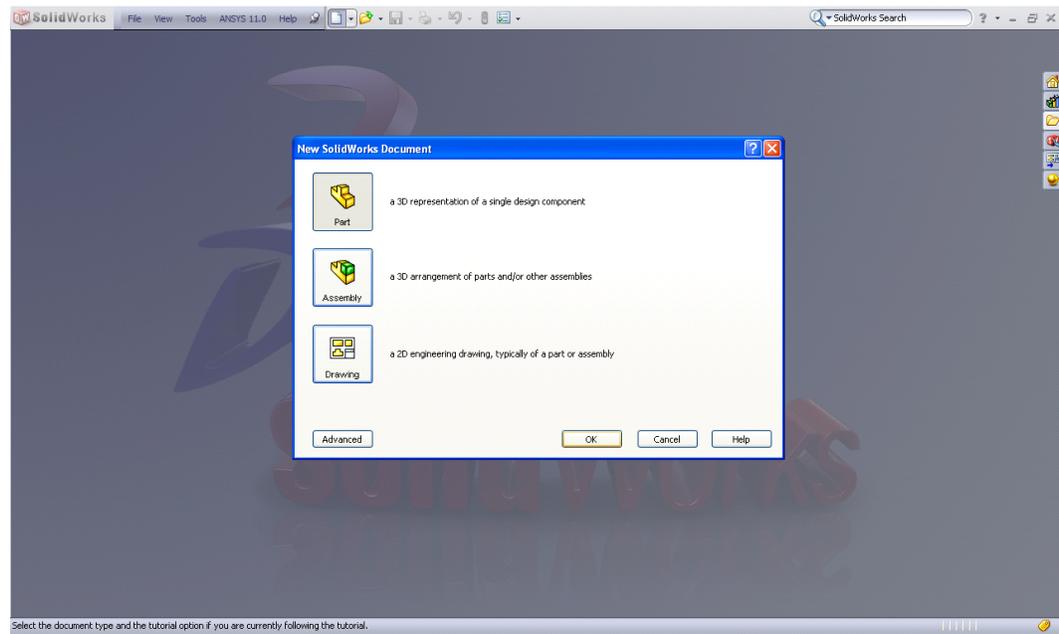
#### Open SolidWorks

From Desktop, select Start | All Programs | SolidWorks 2008 | SolidWorks 2008 SP2.1 | SolidWorks 2008 SP2.1

#### Create a Part Document

From Menu, select File | New ... or click on New button in the toolbar.

In the New SolidWorks Document window, select Part, and then click OK.



## Set Units

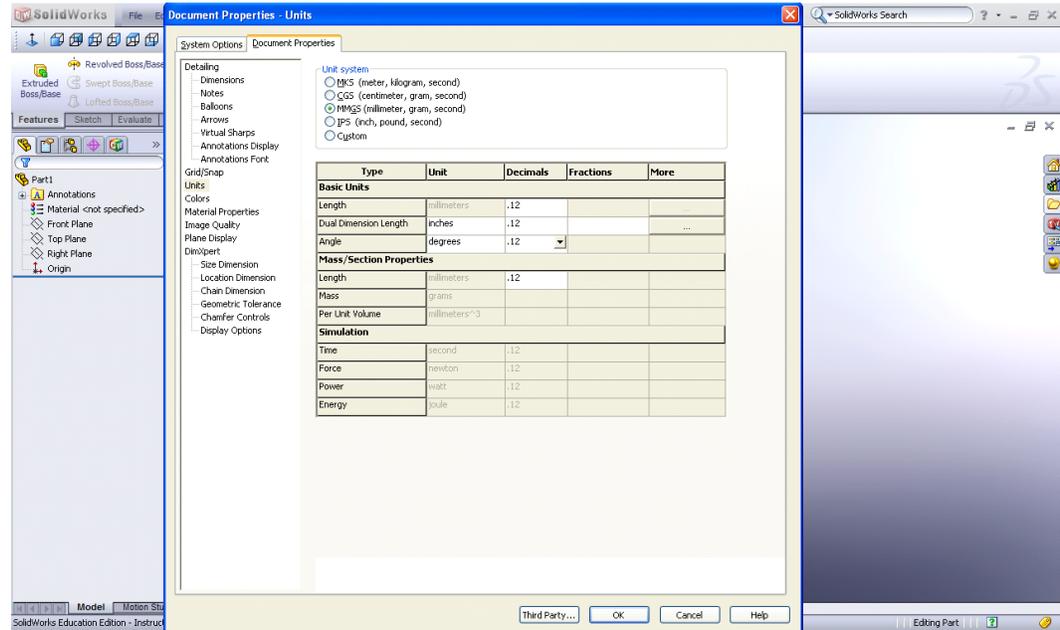
From Menu, select Tools | Options ...

Select the Document Properties tab from the Document Properties window.

In the Detailing window, select ISO from the pull-down list (default).

Click Units to show the Units window.

In the Units window, choose MMGS (Millimeter, Gram, Second), set Decimals to .12 (default), and then click OK.



## Save the Part Document

From Menu, select File | Save As ...

In the Save As window, browse to a folder, type a part name in the File name ("TEMP", use the default extension .SLDPRT by selecting Part(\*.prt;\*.sldprt) in the Save as type, then click Save.

## Close the Part Document

From Menu, select File | Close to close the part document.

## Open the Part Document

From Menu, select File | Open... or click on Open button in the toolbar.

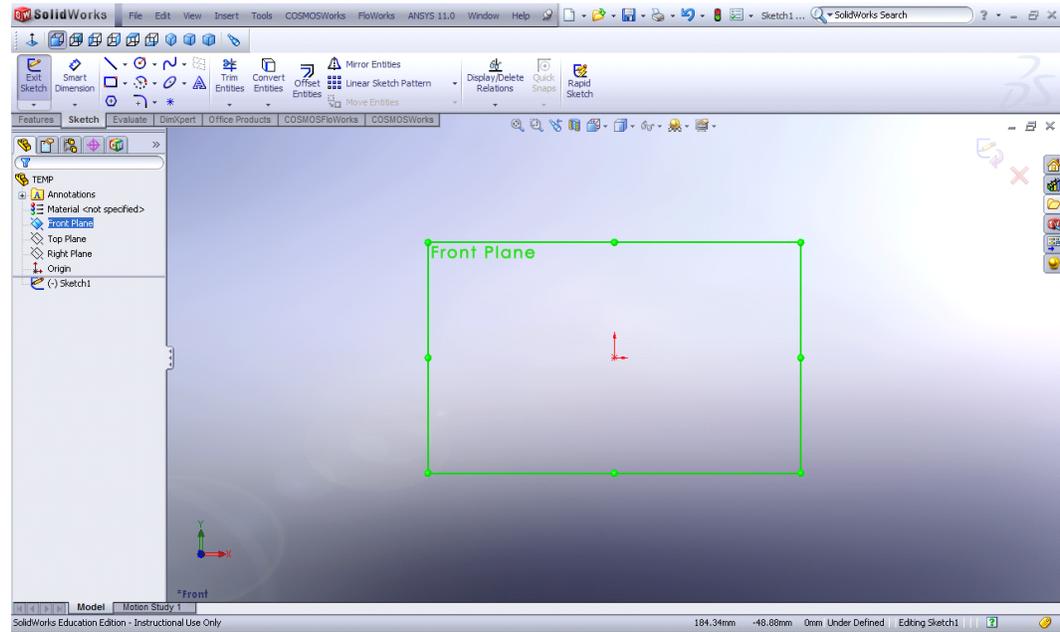
In the Open window, browse to the folder, select the file, and then click Open.

## 2 Create a Bolt Using Extruded Base Feature

### 1) Sketch a Profile

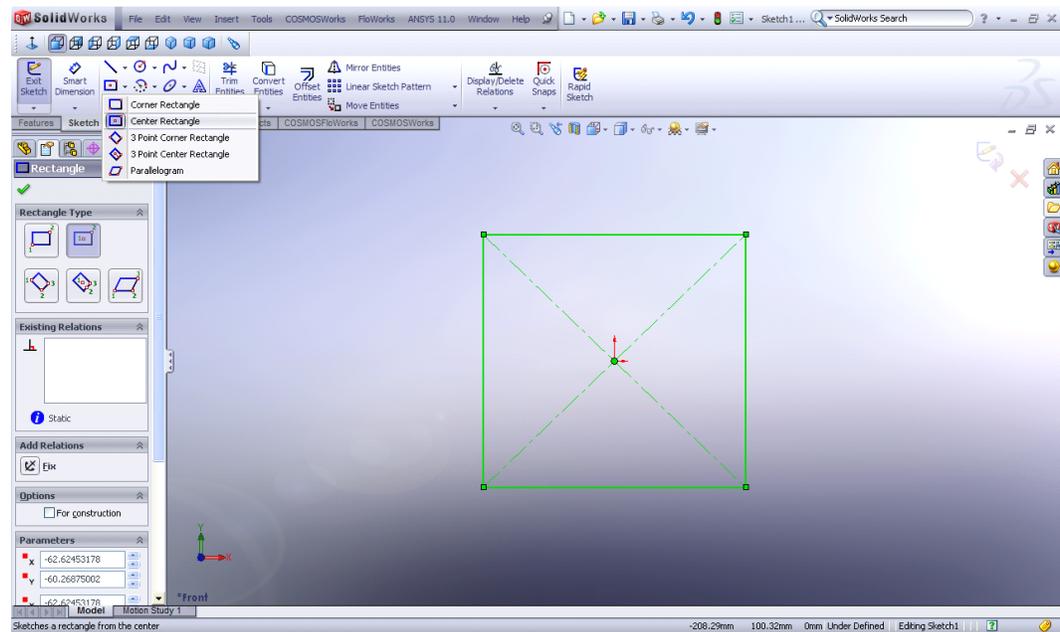
#### Select a Sketch Plane

Click Sketch tab. Select Front Plane as a sketch plane from the FeatureManager design tree. Click Sketch from the CommandManager. Now it is in the Sketch module.



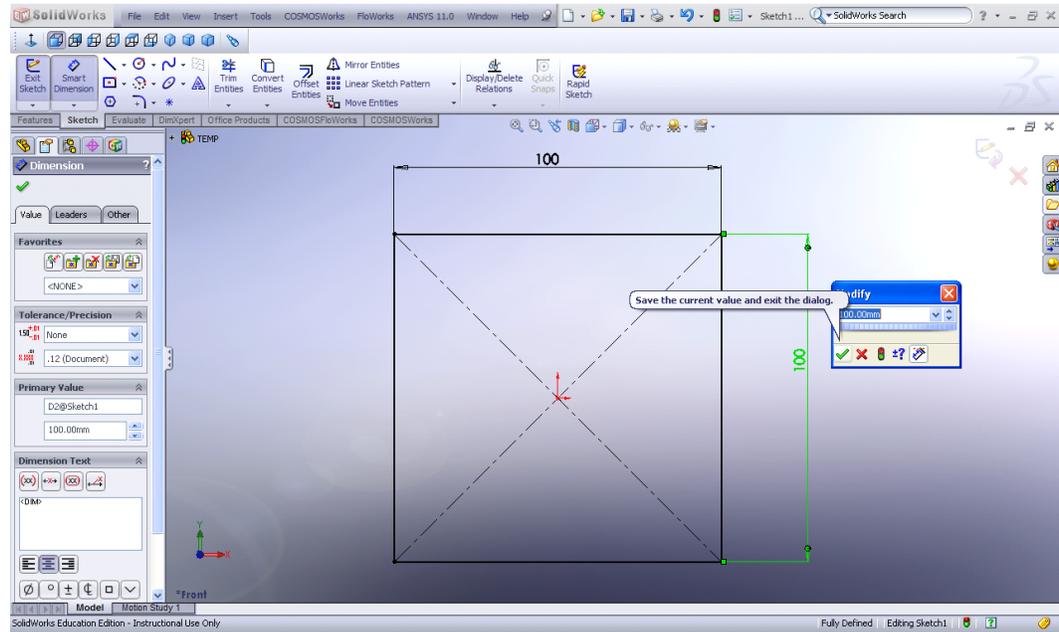
#### Sketch a Square

Click the pull down menu of Rectangle from CommandManager (Sketch toolbar), and select Center Rectangle. Draw a rectangle by clicking the origin and then click any other place on the sketch plane.



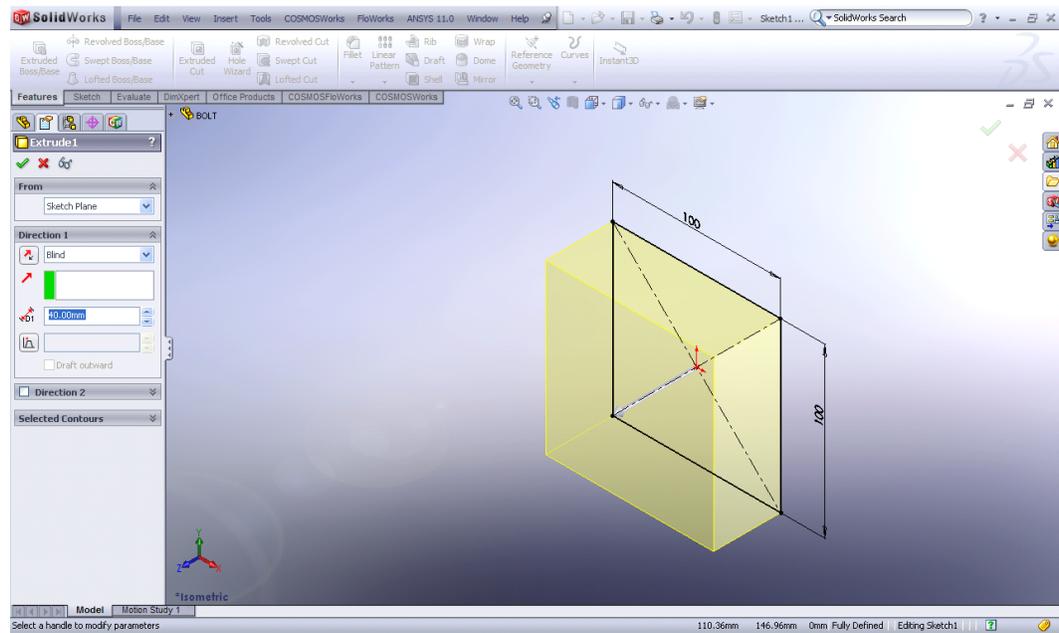
## Add Dimensions

Click Smart Dimension from CommandManager. Click the horizontal line of the rectangle. In the pop-up Modify window, enter 100 (mm) as the length in the edit box, and then click the checkmark. Add a dimension of 100 (mm) for the vertical line. Click Zoon to Fit.



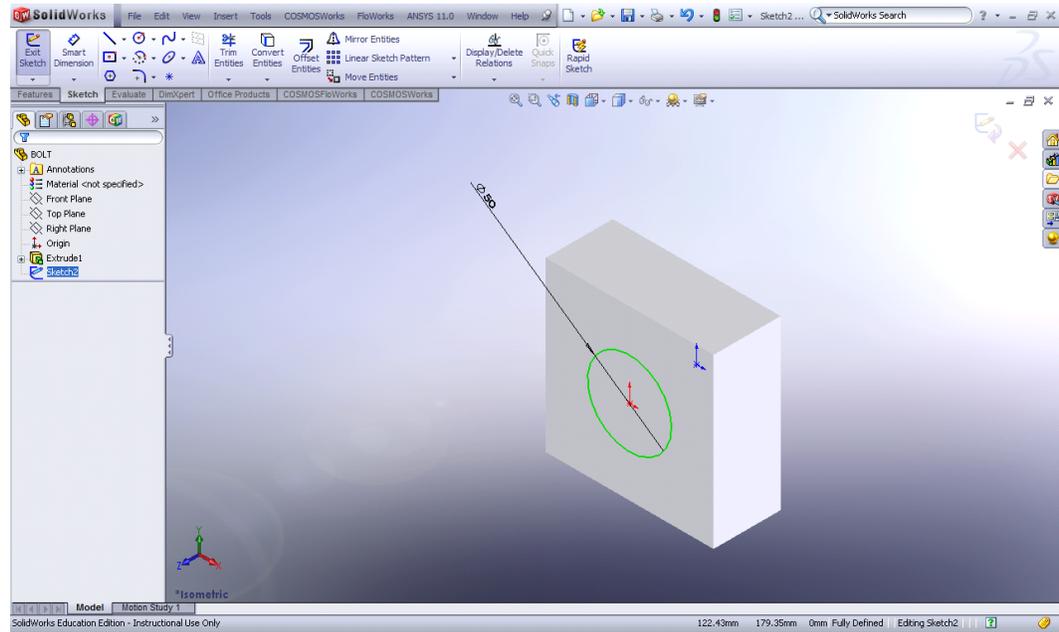
## 2) Create a Solid Model (the Head of a Bolt)

Click Features tab. Click Extruded Boss/Base from the CommandManager (Feature toolbar). Select Blind in the Direction 1 field (default) and enter 40 (mm) as the depth in the edit box, and then click the checkmark.



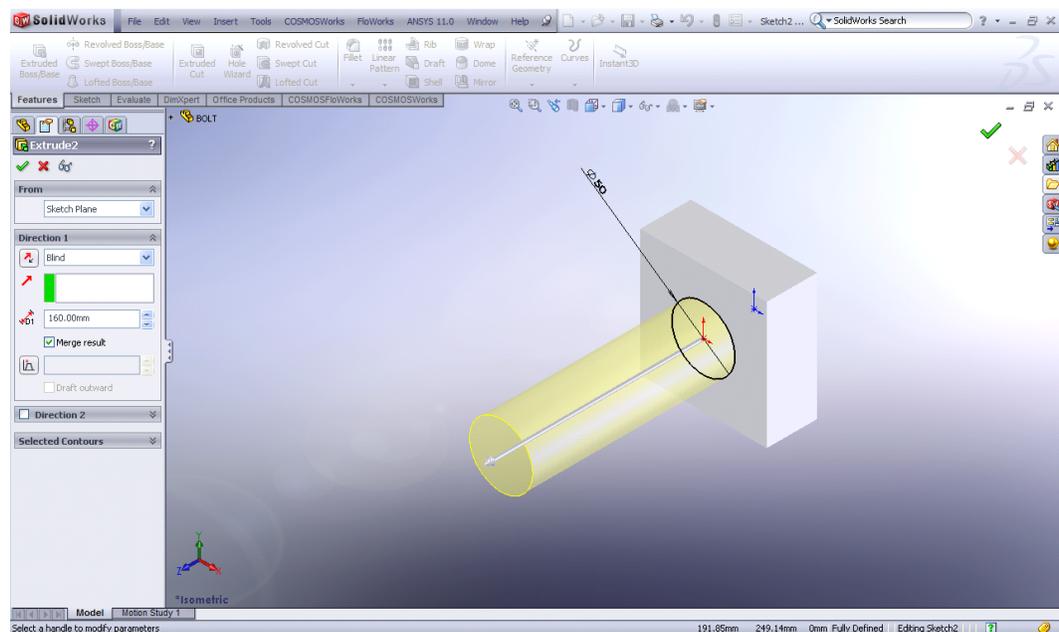
### Sketch a Circle

Click Sketch tab again. Click the front face of the rectangular prism as a sketch plane. Click Sketch from the CommandManager. Click Circle from the Sketch toolbar. Draw a circle by clicking the center of the rectangle and then click any other place on the sketch plane. Add a dimension of 50 (mm) for the diameter of the circle.



### Add the Body of the Bolt

Click Features tab. Click Extruded Boss/Base from the CommandManager. Select Blind in the Direction 1 field (default) and enter 160 (mm) as the depth in the edit box, and then click the checkmark.



### Rename the Created Features

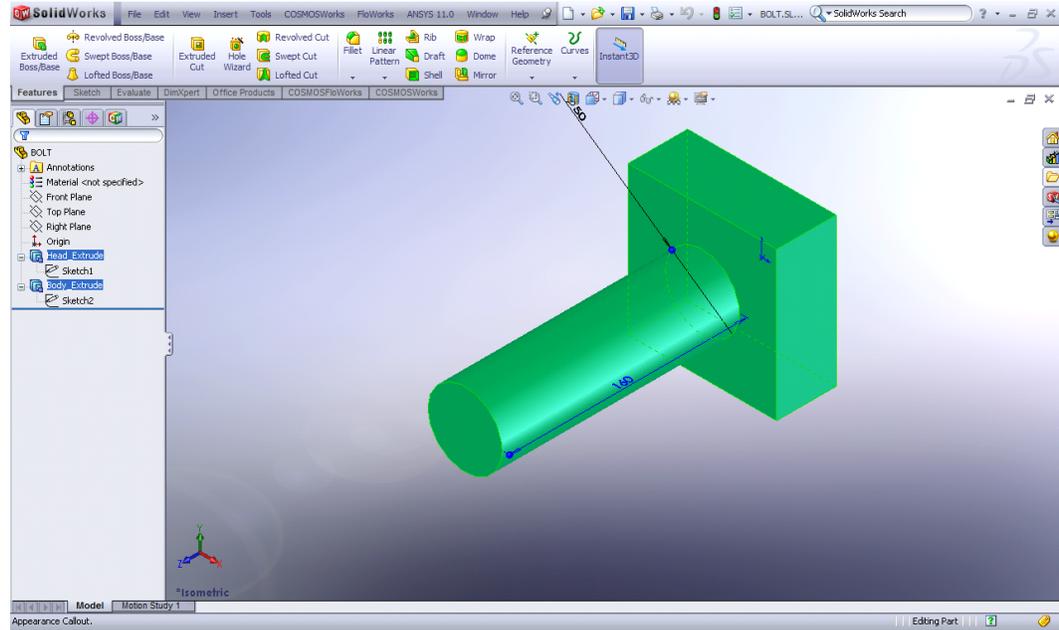
Select Extrude1 from the FeatureManage design tree, click on it again to rename it as “HEAD\_Extrude”.

Select Extrude2 from the FeatureManage design tree, click on it again to rename it as “BODY\_Extrude”.

### Save the Part

From Menu, select File | Save As ...

In the Save As window, browse to a working folder, enter a part name (BOLT) in the File name edit box, and then click Save.



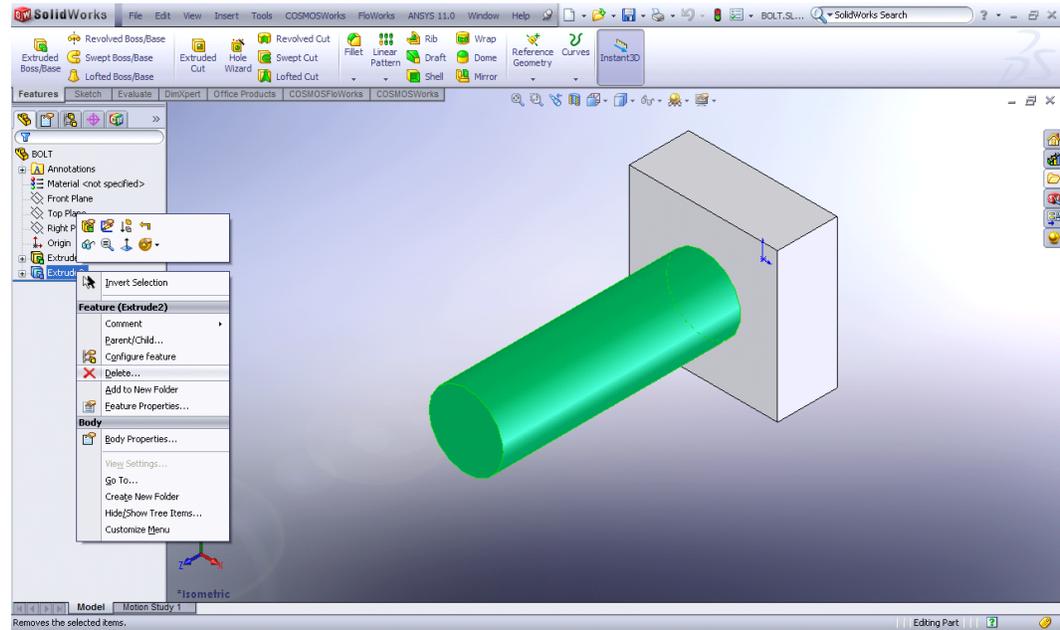
### 3 Create a Nut Using Extruded Cut Feature

#### Use the Exist Part Document

Open the “BOLT.SLDPRT”, and save it as a new part document “NUT.SLDPRT”.

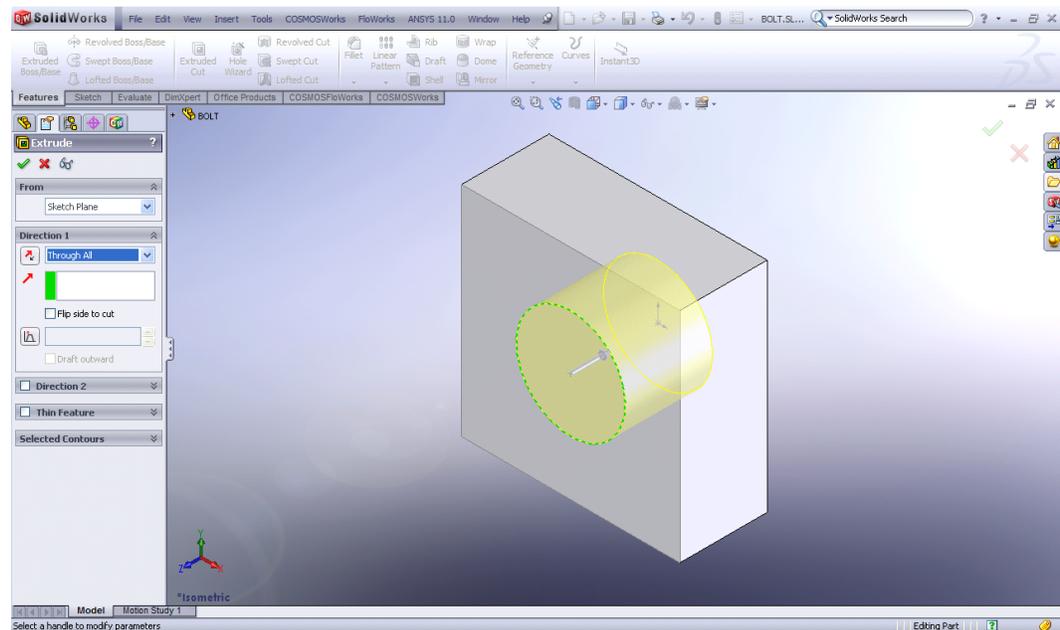
#### Delete a Feature

Right click on BODY\_Extrude from the FeatureManage design tree, and select Delete...  
In the Confirm Delete window, click Yes.



#### Use the Exist Sketch to Create a Extruded Cut Feature

Select the Sketch2 (Circle) from the FeatureManage design tree. Click Features tab. Click Extruded Cut from CommandManager. Select Through All in the Direction 1 field, then click the checkmark.



Select Cut-Extrude1 from the FeatureManager design tree, click on it again to rename it as “HOLE\_ExtrudeCut”. Click Save to save the part.

