Introduction to Solidworks

OPTI 421/521

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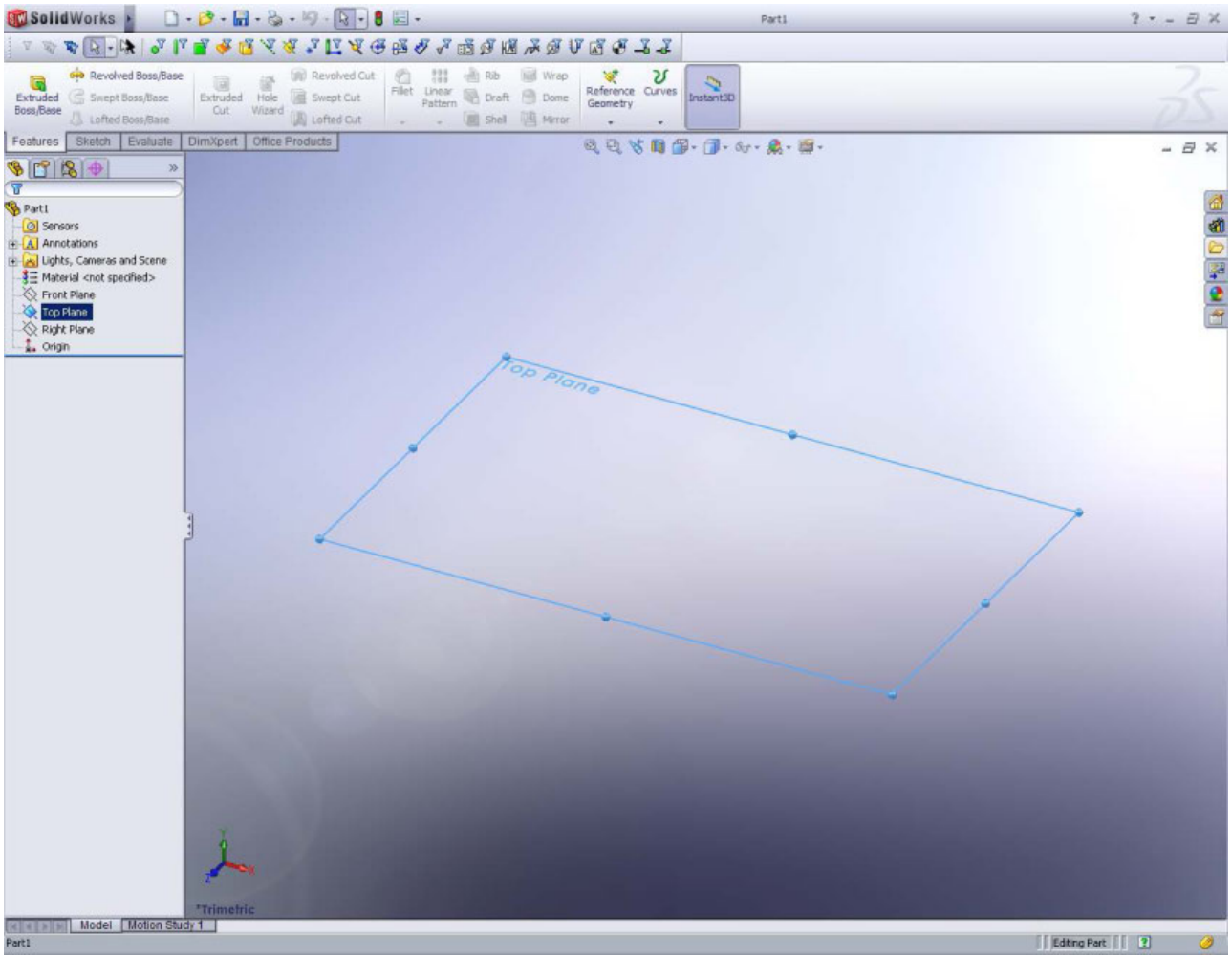
Basic order of 3D modeling

- **1. Select a plane to draw your sketch**
  - Top, bottom, right or left reference plane
  - A plane surface on your model
  - You may need to generate a reference plane
- **2. Sketch**
  - Define all dimensions properly using ‘Smart Dimension’ function in Solidworks
- **3. Complete 3D model**
  - Extruded boss, extruded cut or revolved boss

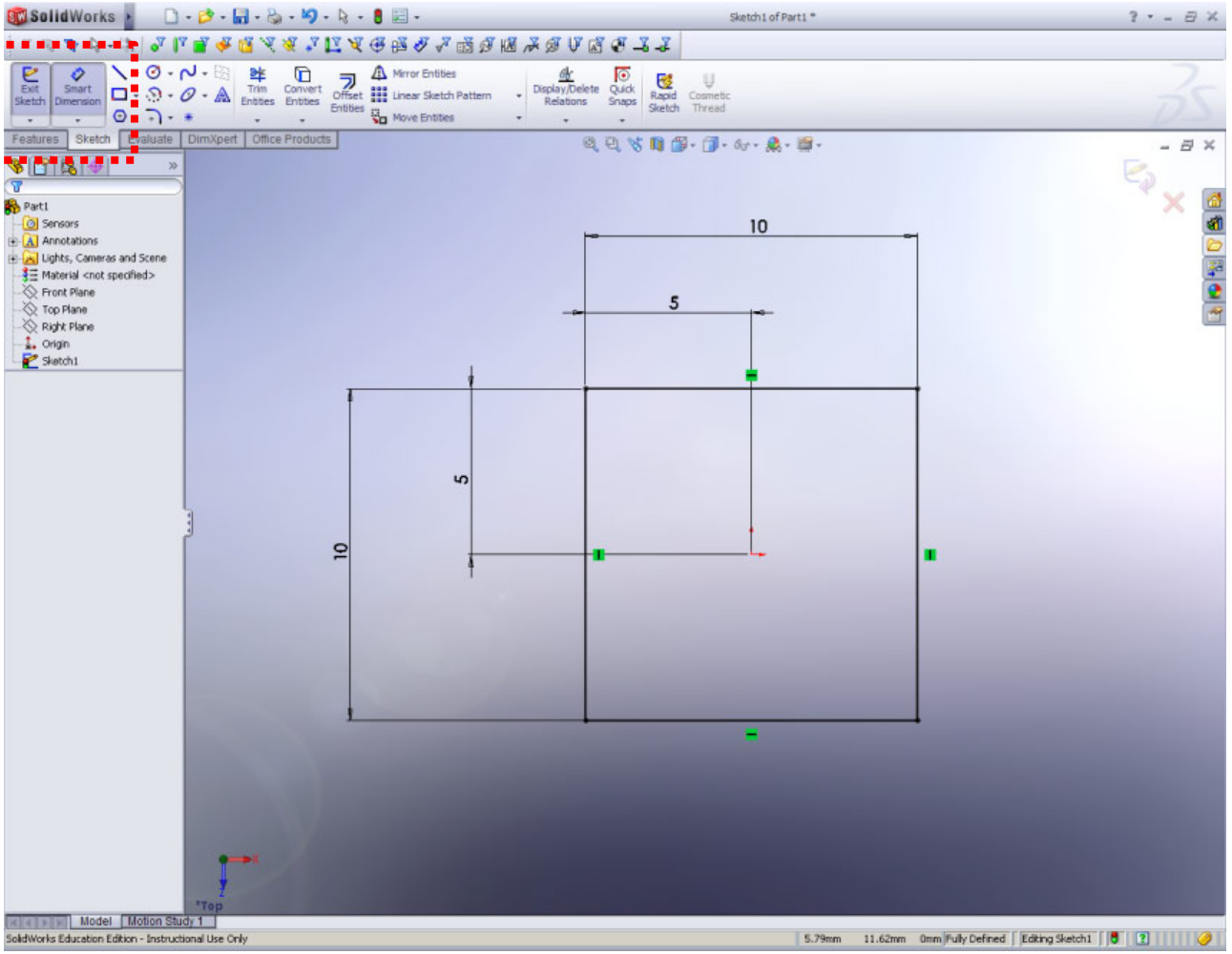
Creating a simple cube

- 1. Select a plane (Top plane is chosen as shown below)



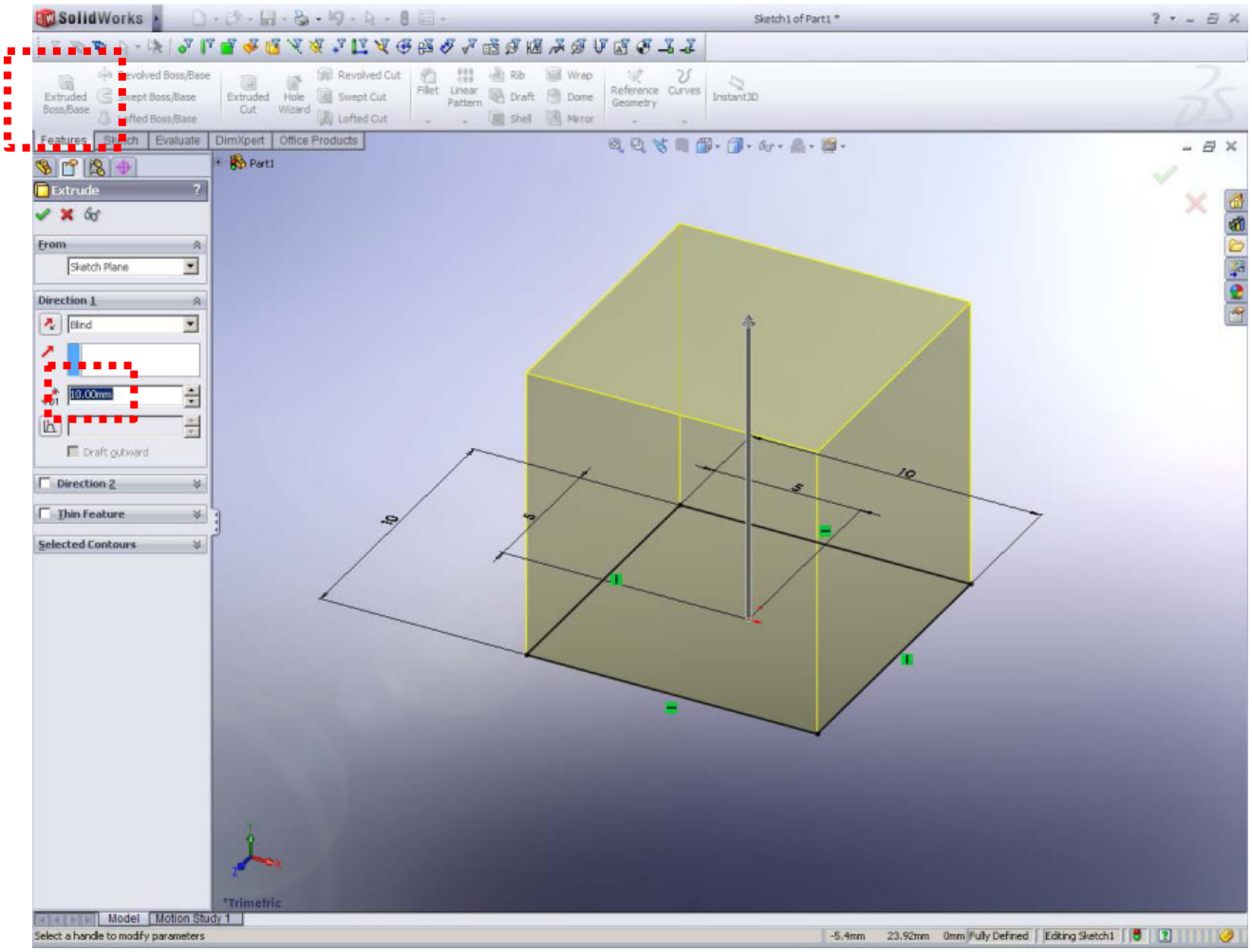
Creating a simple cube

- 2. Sketch a square and define its dimension



Creating a simple cube

- 3. Extrude the sketch to make the 10 mm cube

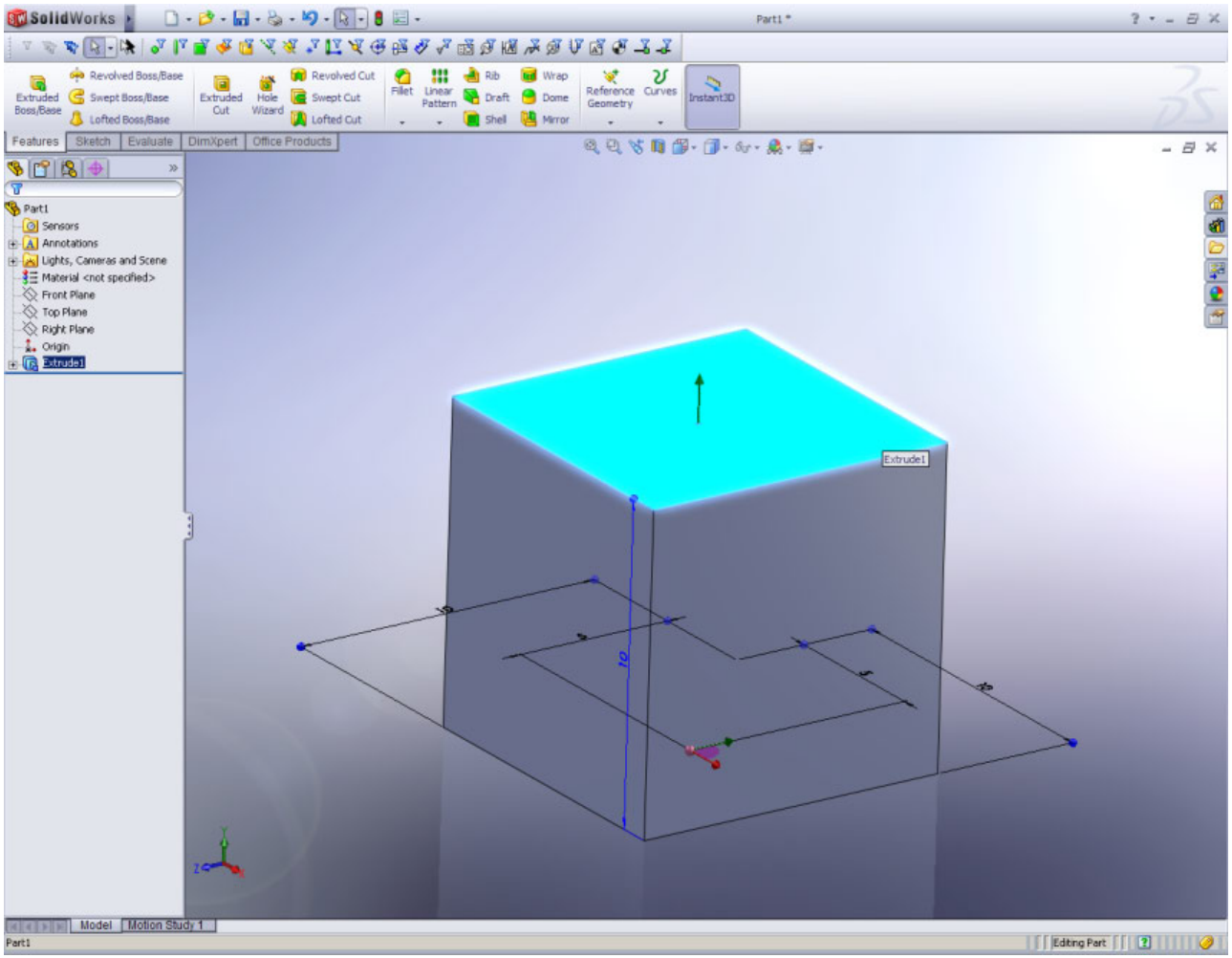


Tips

- Changing point of view with mouse control in workspace of Soldiworks
  - Mouse wheel : upward (minify) , downward (magnify)
  - Push and hold mouse wheel + mouse moving : model rotation
  - ‘ctrl’+ push and hold mouse wheel + mouse moving : model shift
  - Normal view : select a face in your model , then it will be high-lightened. Then hit ‘space bar’. Select ‘normal view’ option.

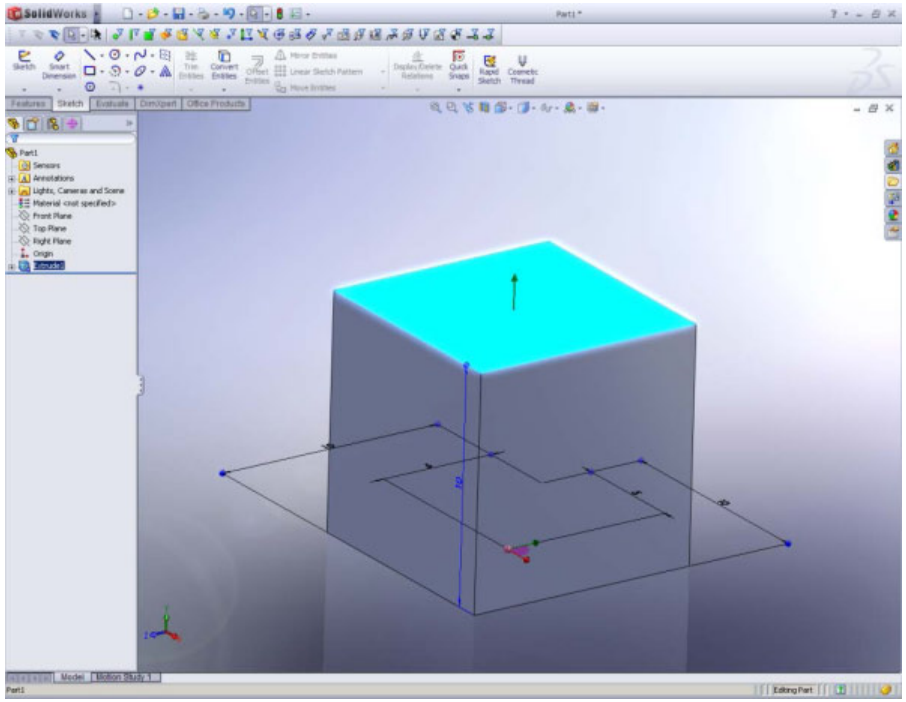
Make a hole

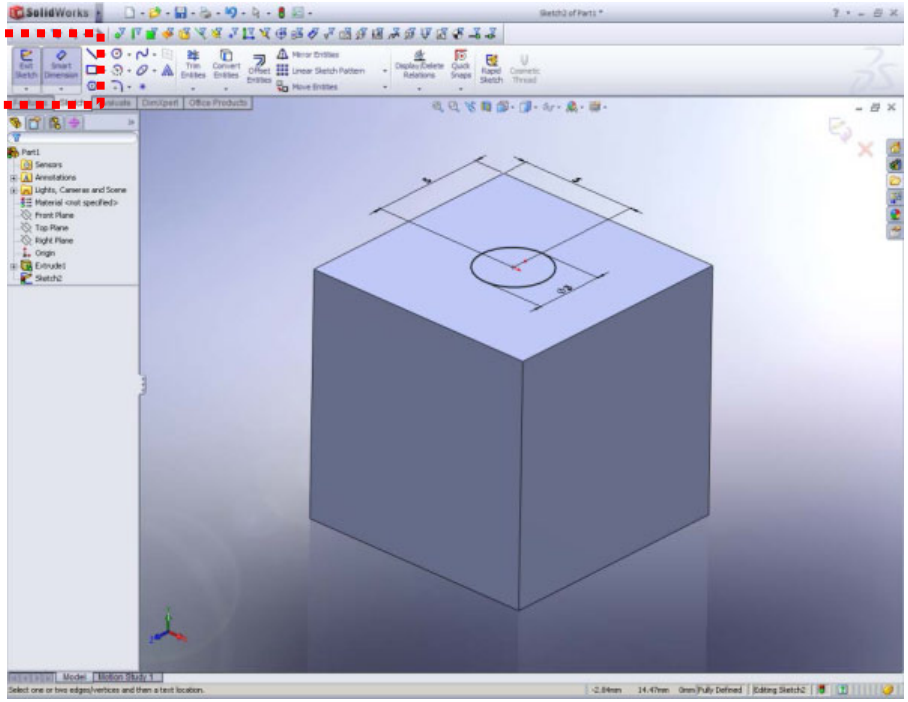
- 1. Select a surface to sketch



Make a hole

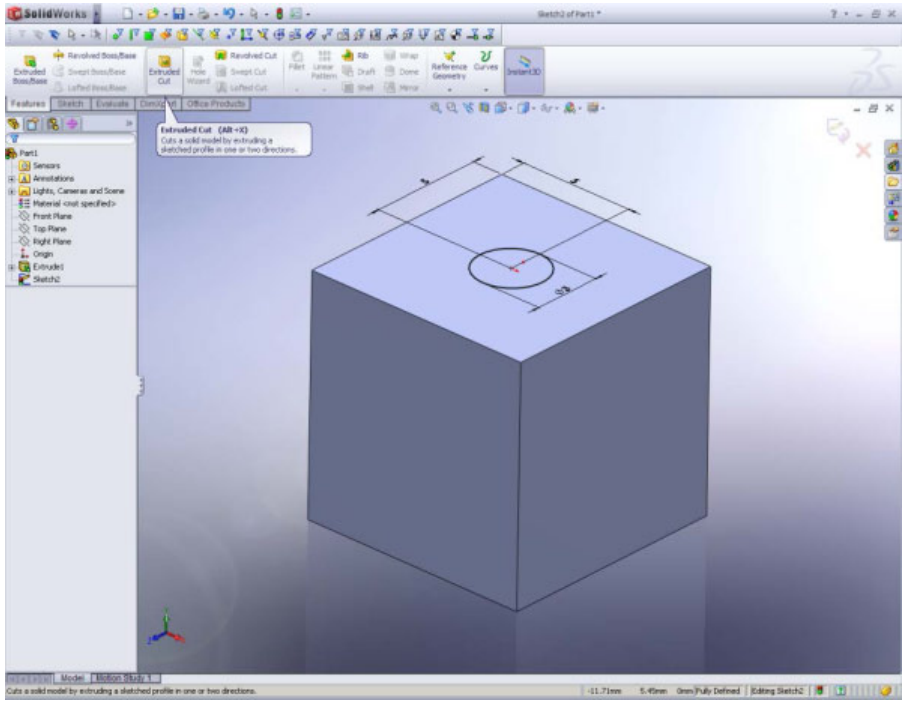
- 2. Sketch a circle on the selected plane and define its dimension

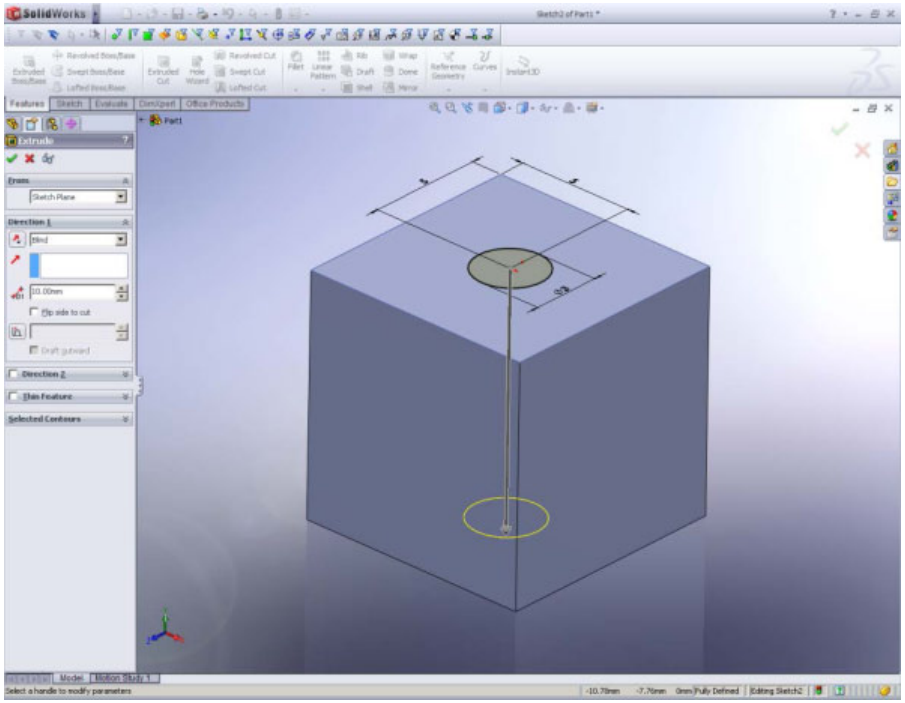




Make a hole

- 3. Extrude the sketch to make a hole through the cube





Tips

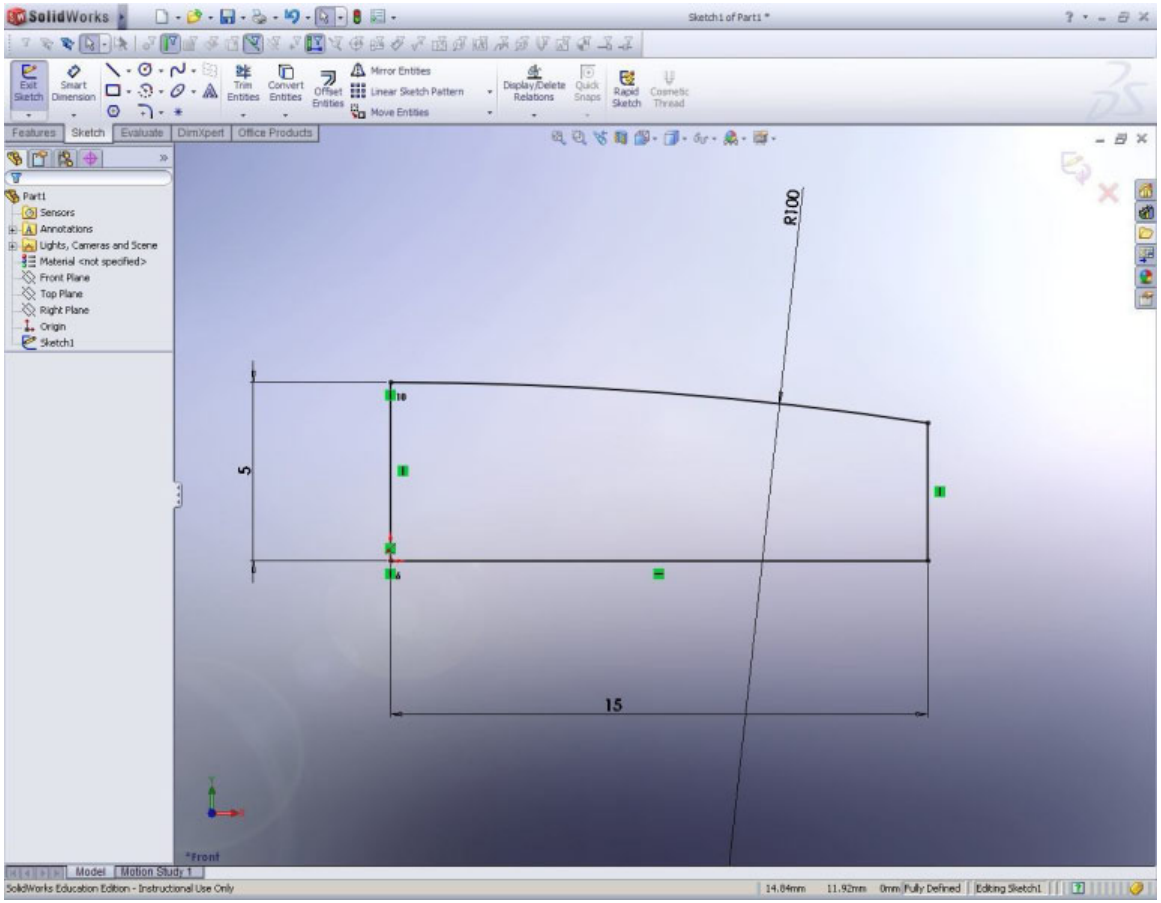
- Changing ‘Unit’ system
  - Hit ‘Soldworks button’ -> tools > options

    In the option window, there is ‘Document Properties’ tab. In the left white box, find ‘units’ and click.

    Change to your favorite unit system.
- Reference dimension
  - ‘Smart dimension’ allows you to make reference dimension that you may use for dimension check.
- Make a image file of your model
  - Save the model as ‘JPEG’ image form or just hit ‘PrintScreen’

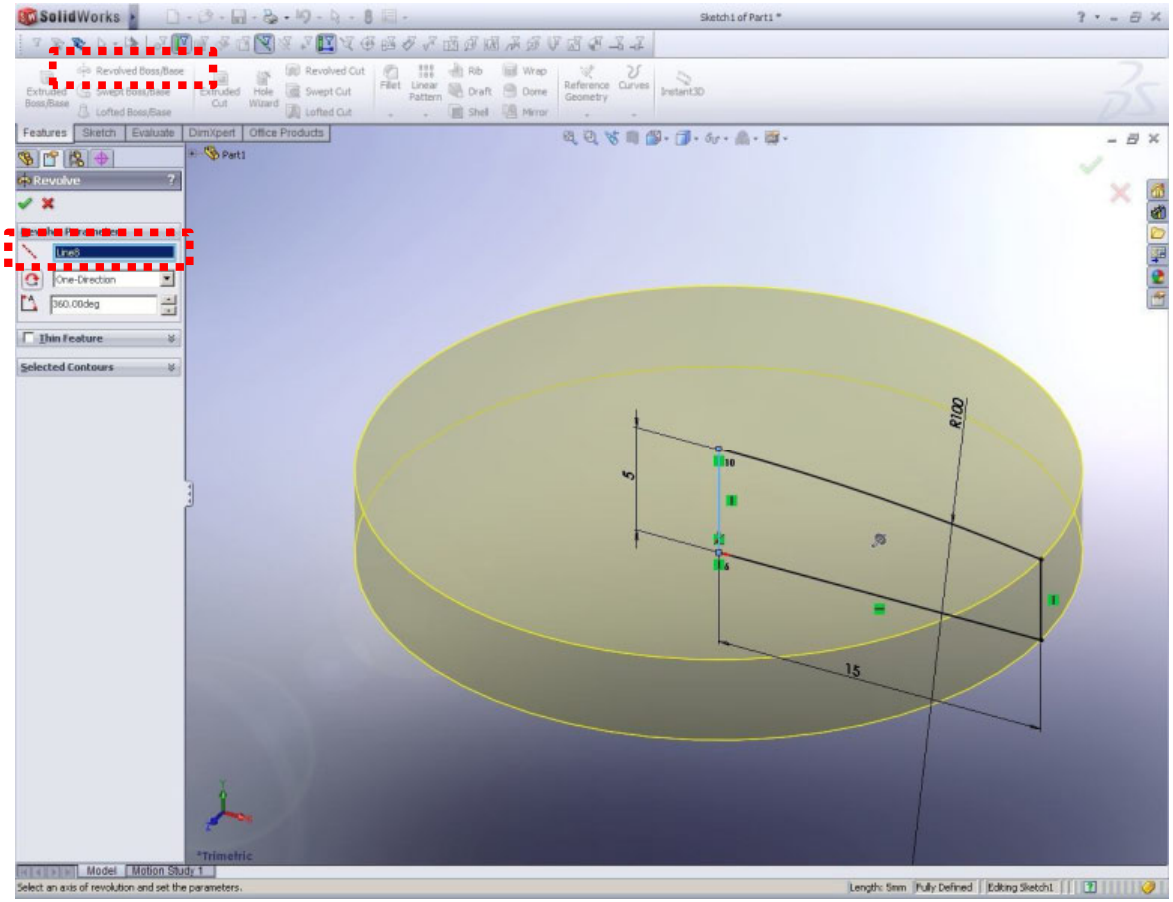
Creating a simple plano-convex lens

- 1. Select a plane anddraw half-cross section of the lens



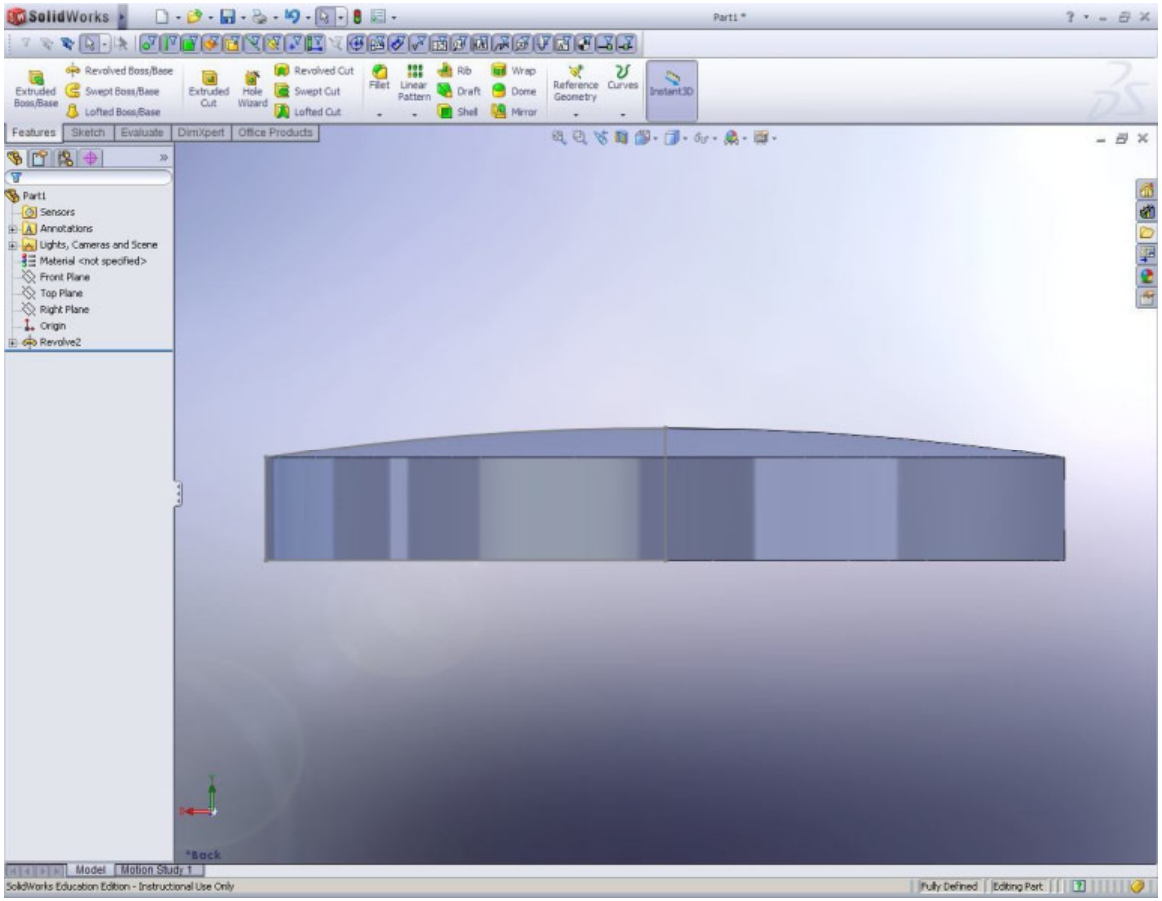
Creating a simple plano-convex lens

- 2. Set the revolving axis and revolve the sketch



Creating a simple plano-convex lens

- 3. Save the lens model

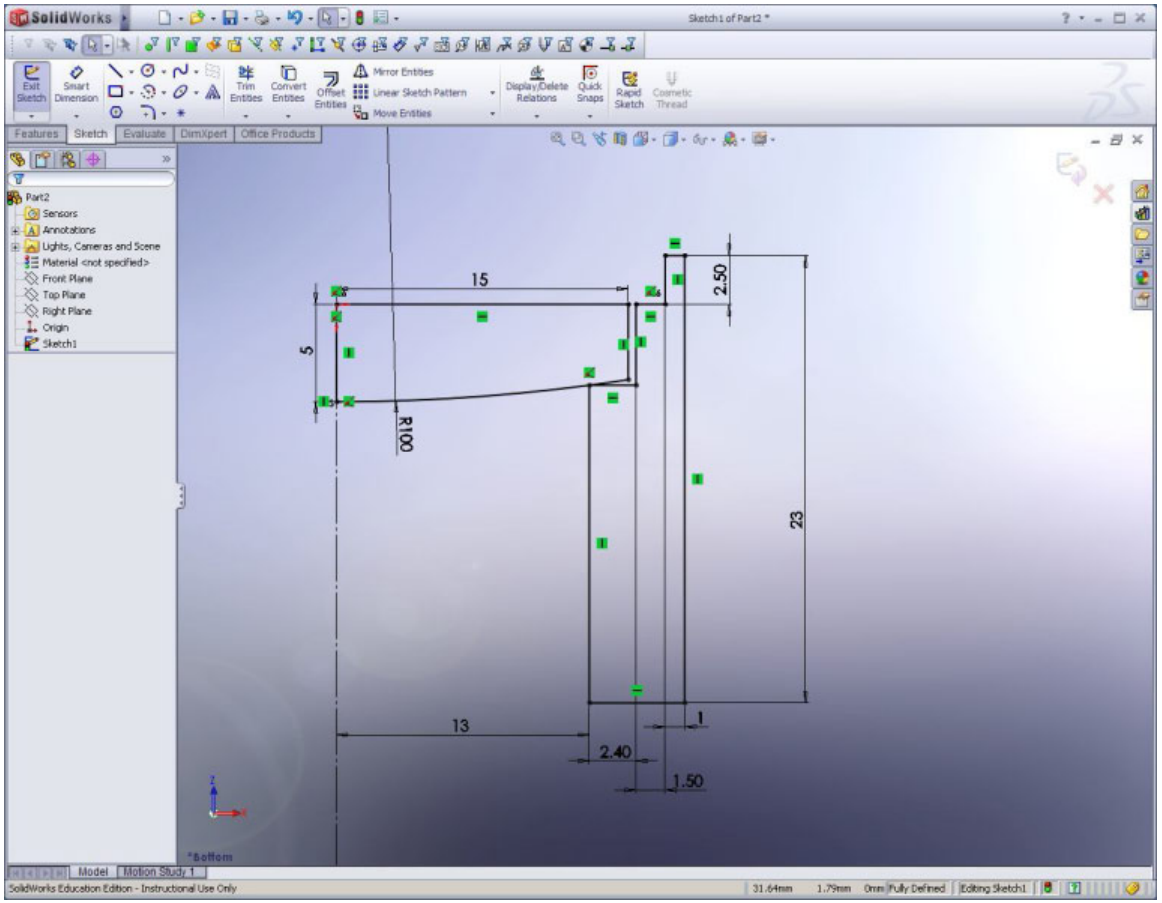


Tips

- All dimension box in Solidworks has the function of simple calculator. You can add, subtract, product and divide.
- Recommend that dots, lines and/or curves in your sketch turn to be black that means all dimensions are completely and properly defined. Blue of them still need to be confined with right dimensions.

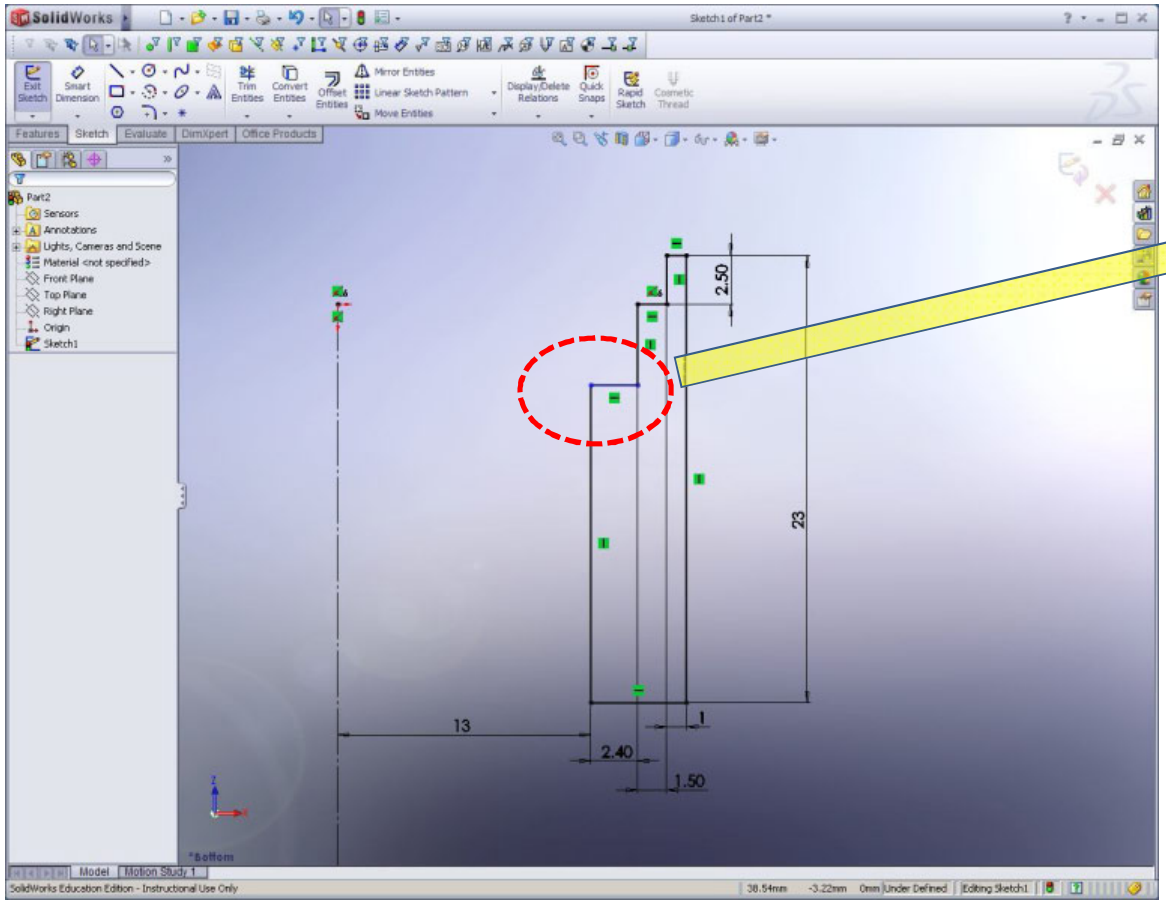
Creating a barrel for the lens

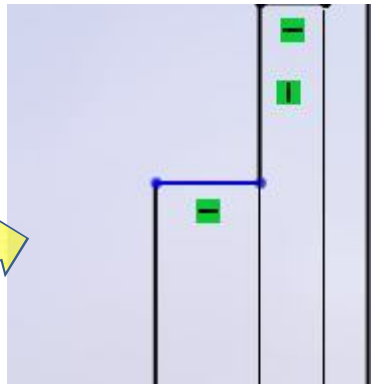
- 1. Sketch the cross section of the barrel w.r.t the lens cross section. Don’t forget the axis of revolution.

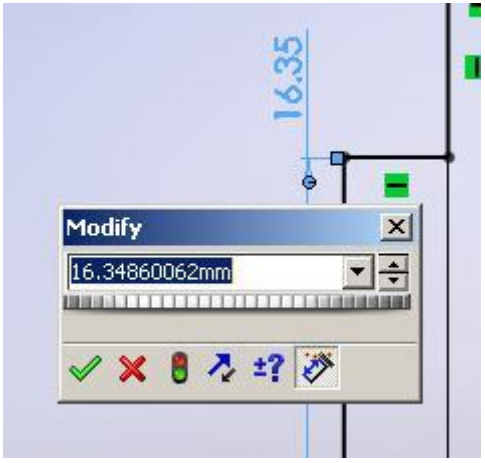


Creating a barrel for the lens

- 2. Delete the lens cross section. Then see the line that turns blue. Make the line be black using ‘smart dimension’

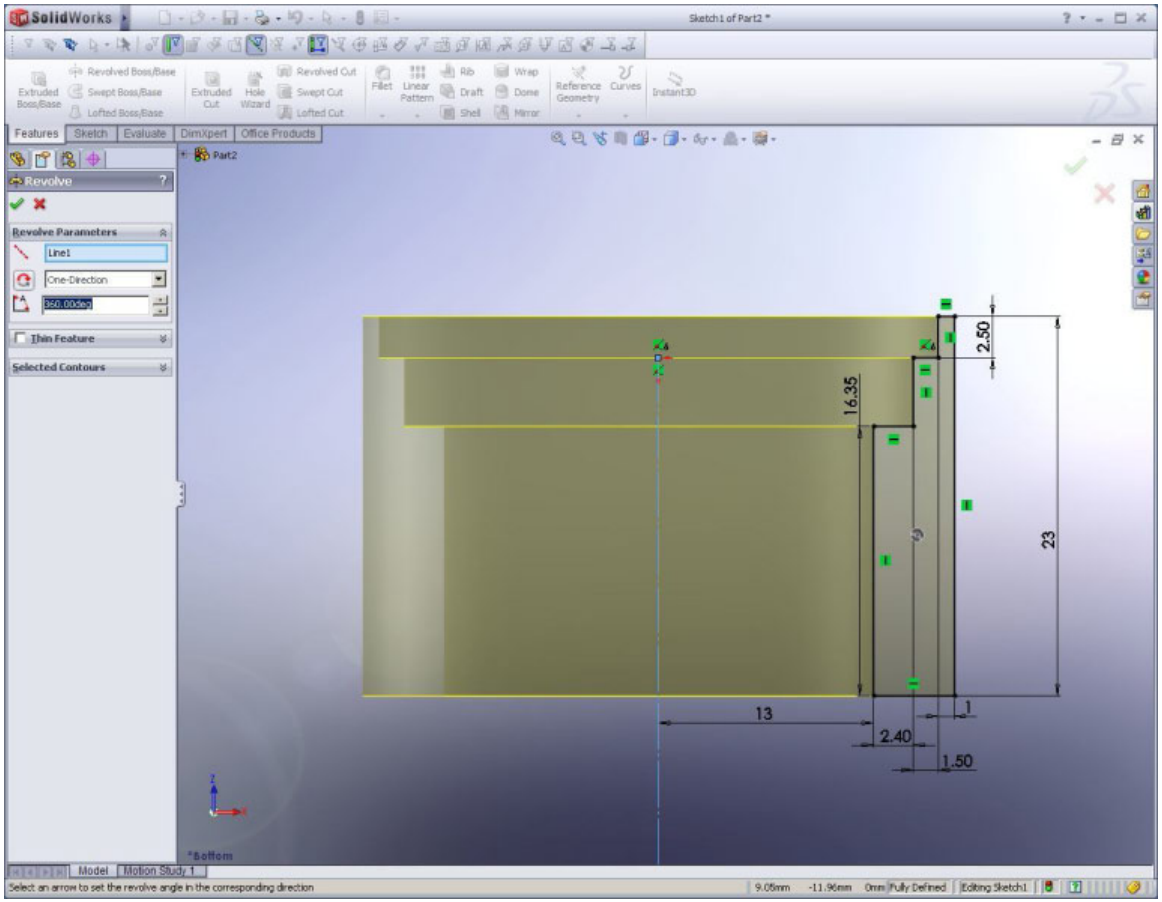




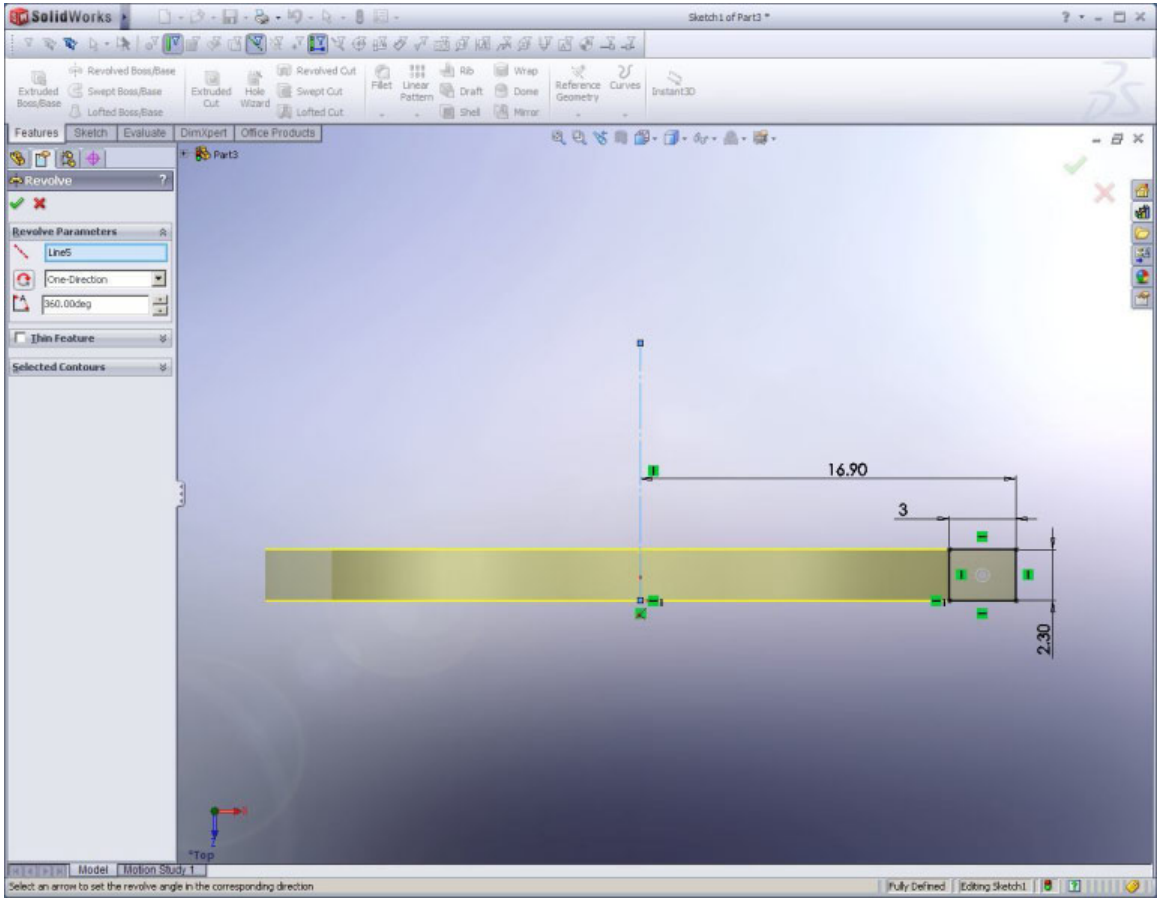


Creating a barrel for the lens

- 3.Revolve the sketch

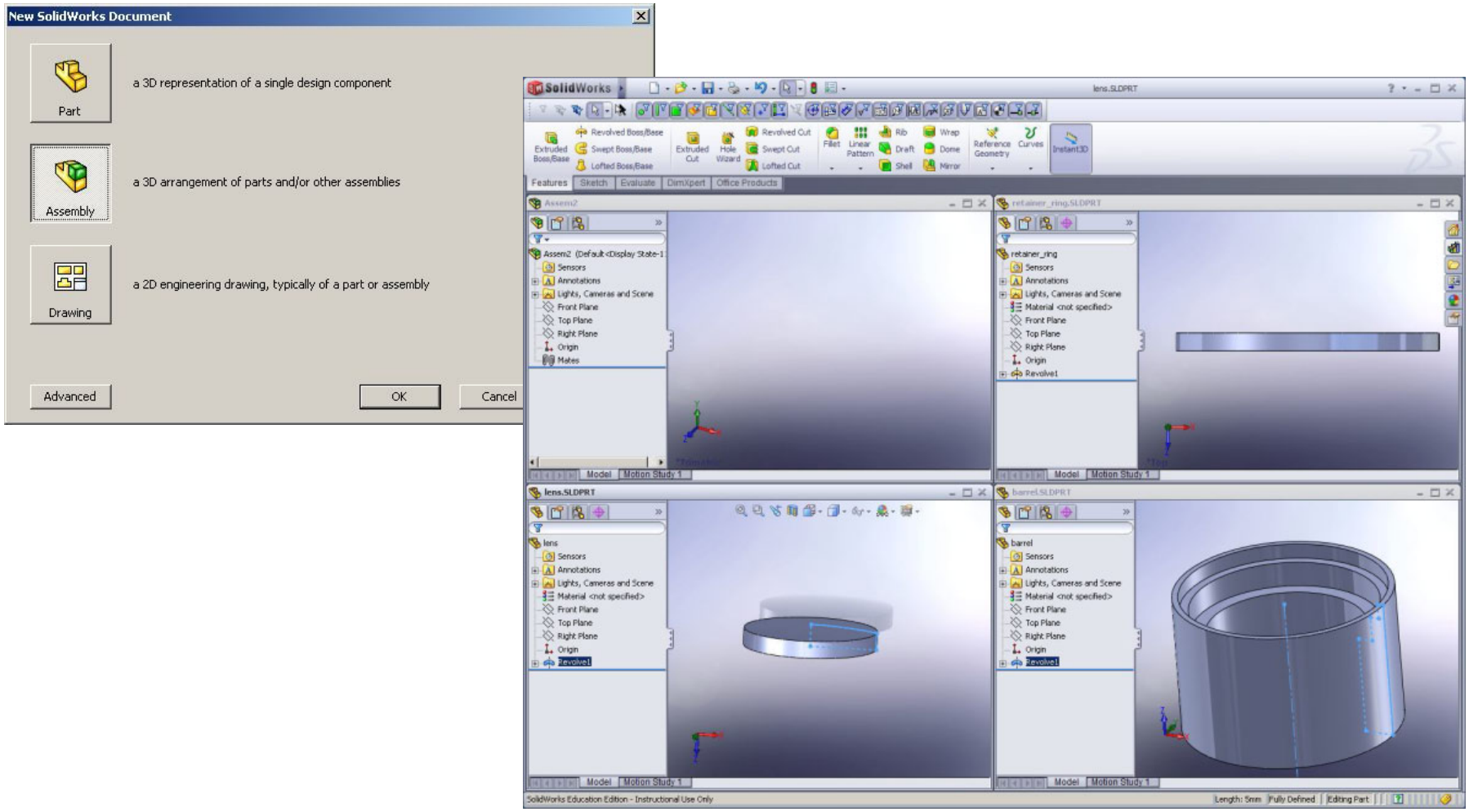


Creating a retainer ring for the lens



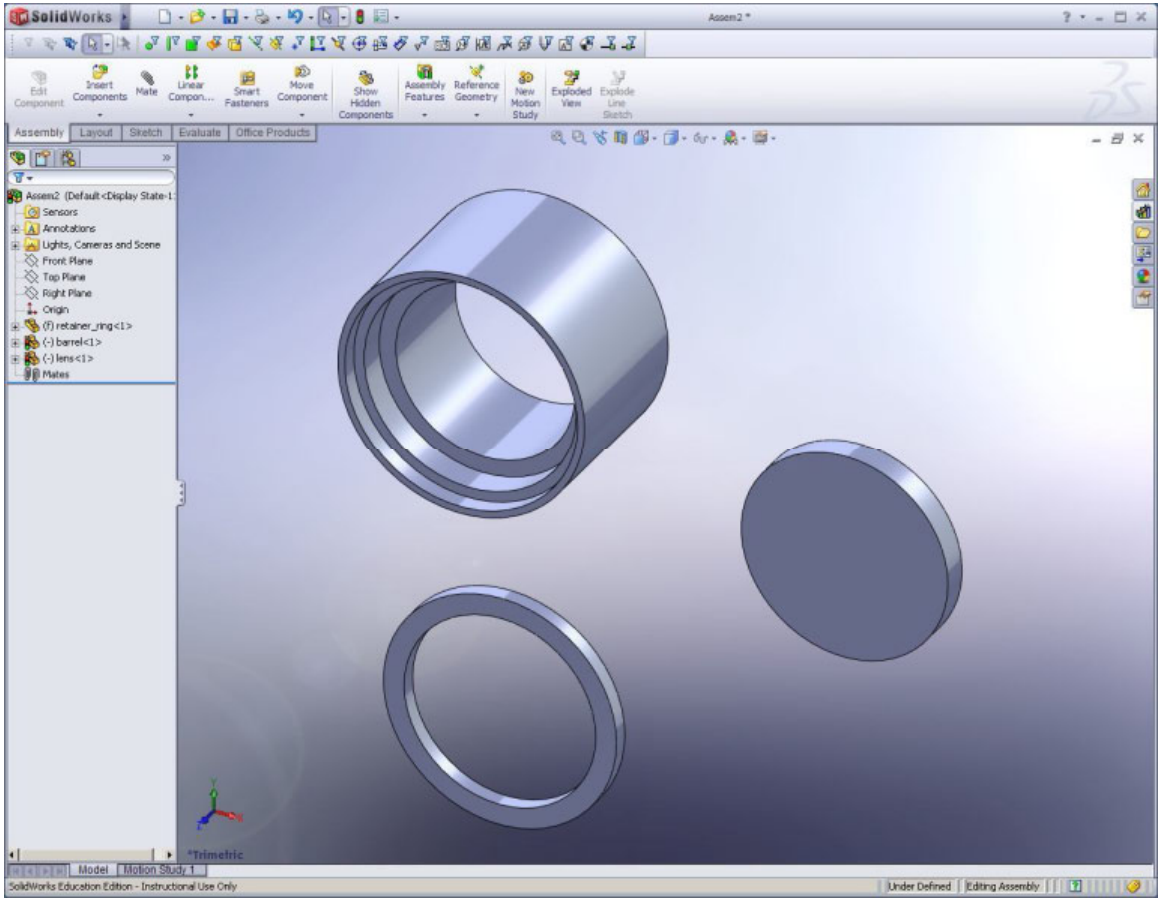
Creating a lens assembly

- 1. Open an assembly Solidworks document



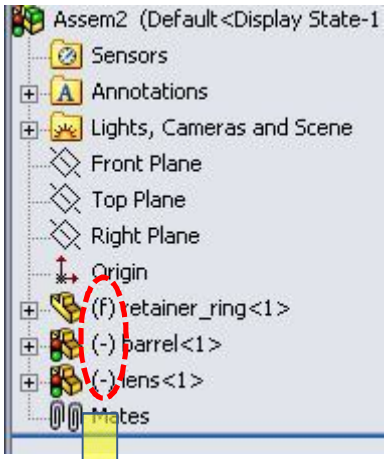
Creating a lens assembly

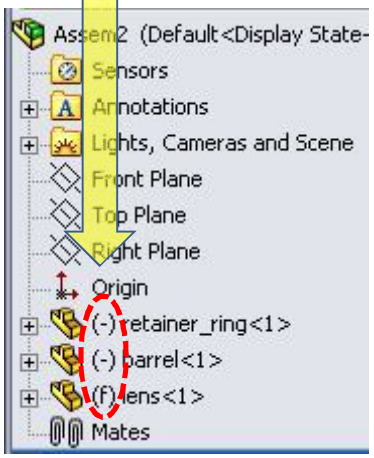
- 2. Put each component into the assembly Solidworks document.

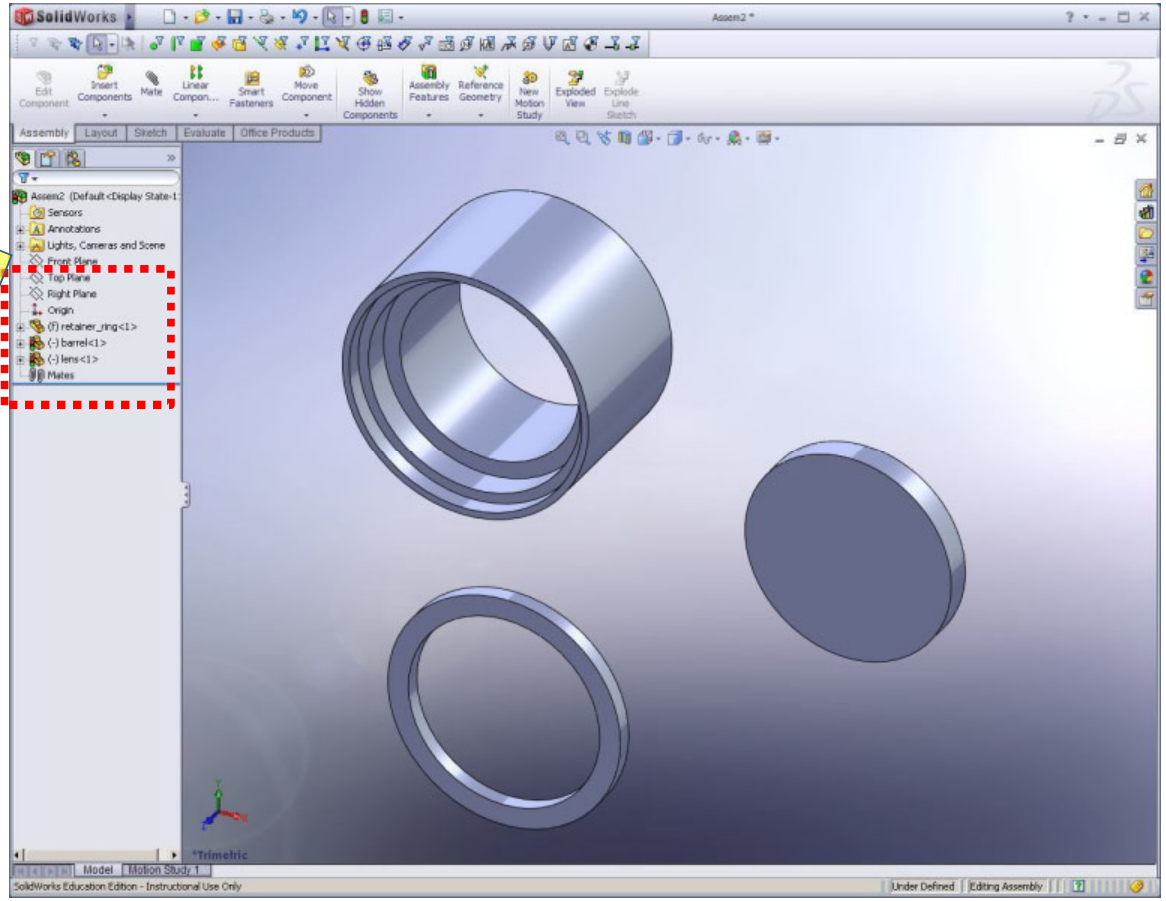


Creating a lens assembly

- 3. Fix one of them as a reference.

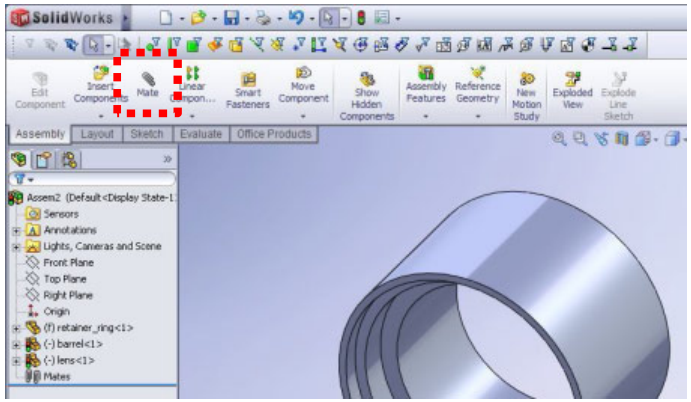




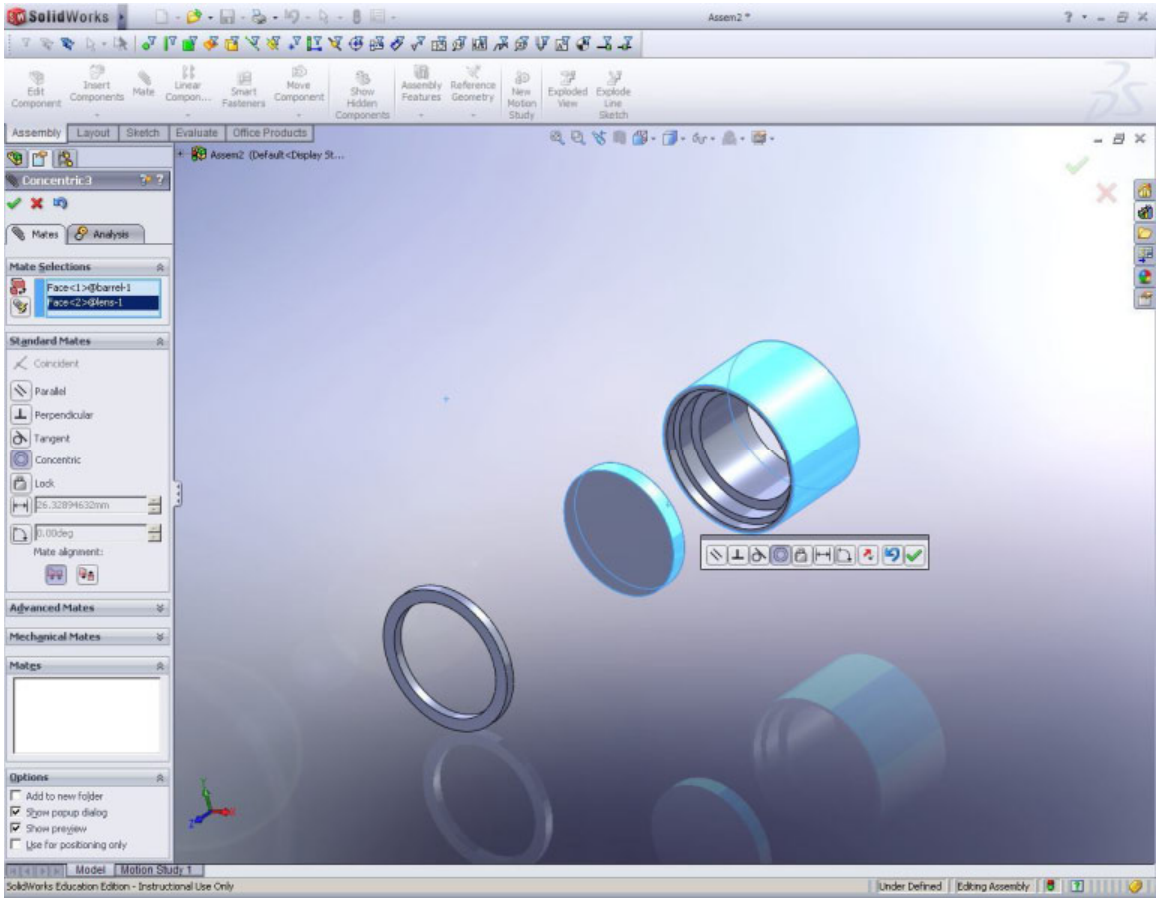


Creating a lens assembly

- 4-1. Give mating relations between them.

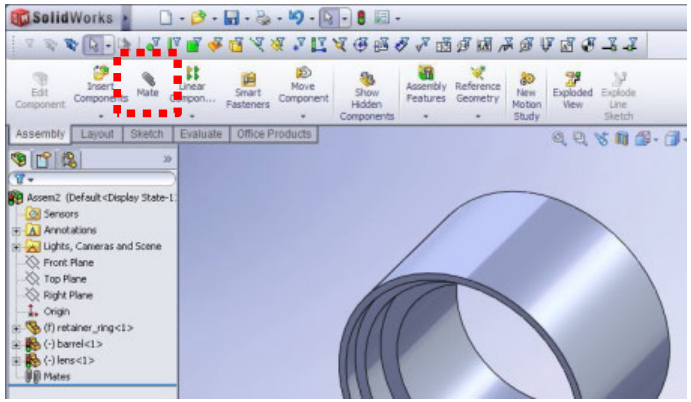


Concentric relation

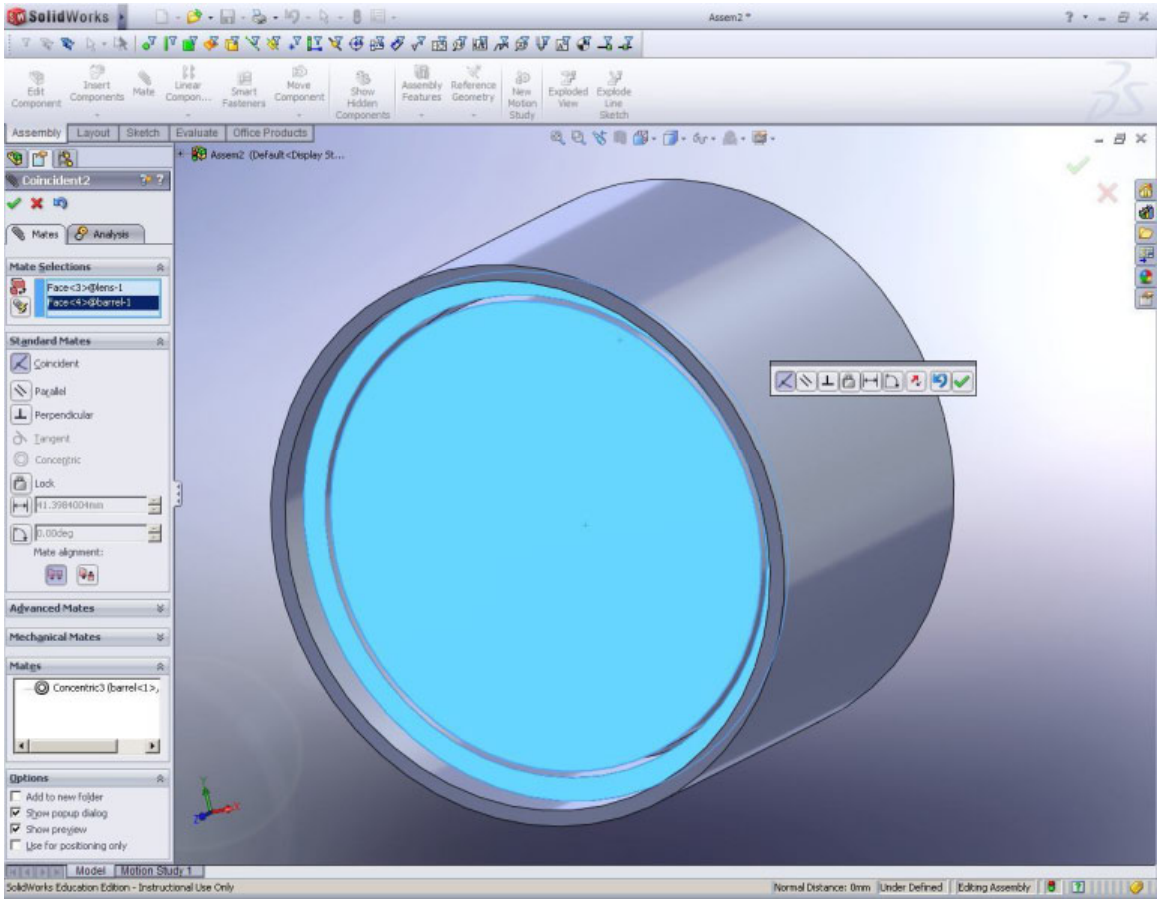


Creating a lens assembly

- 4-2. Give mating relations between them.

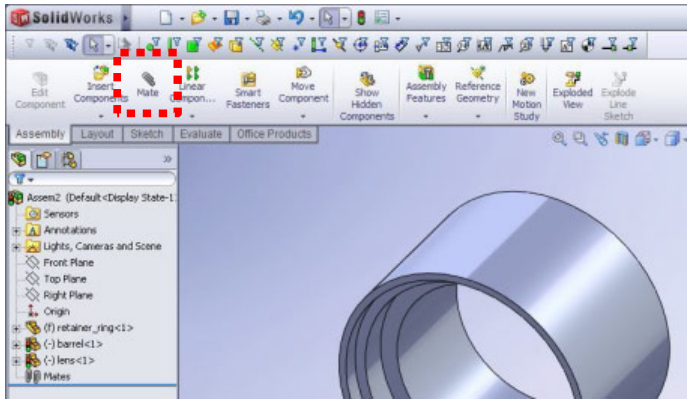


Coincident relation

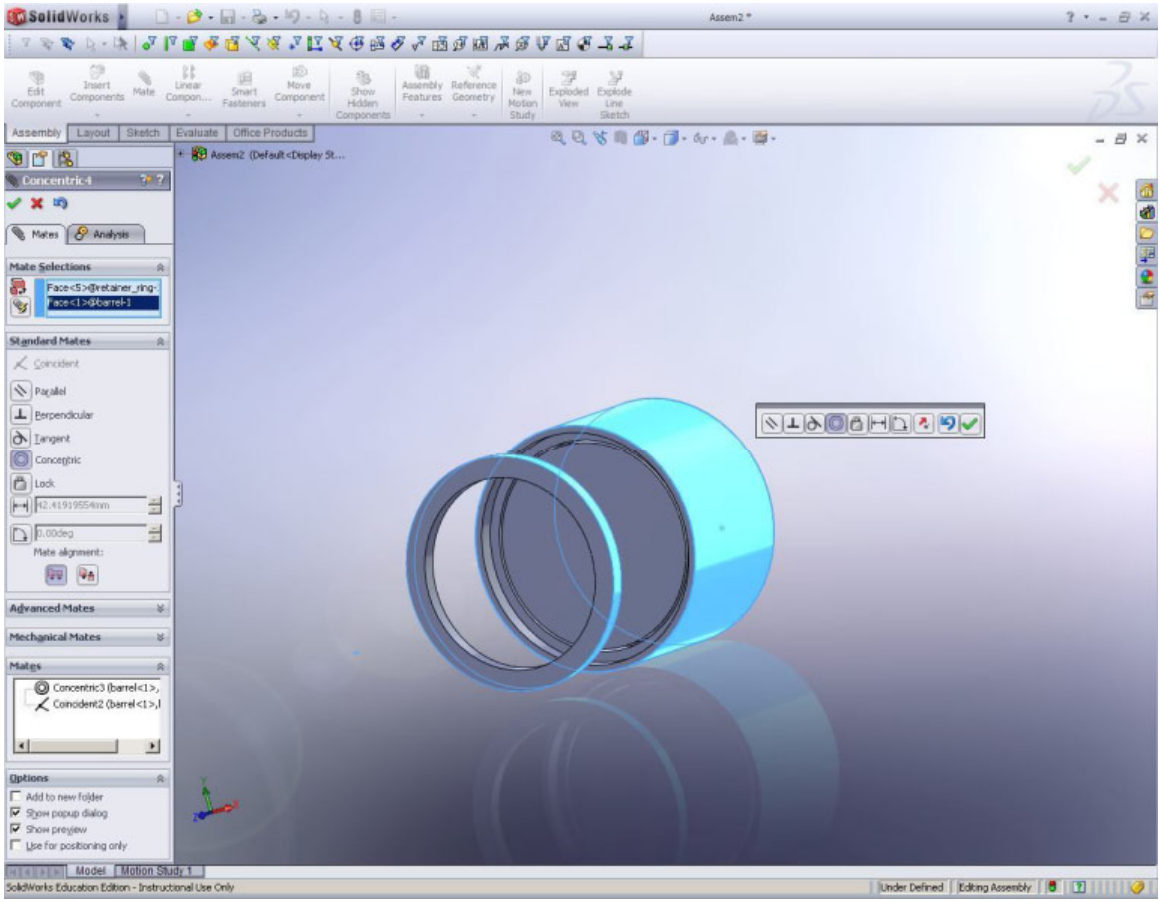


Creating a lens assembly

- 4-3. Give mating relations between them.



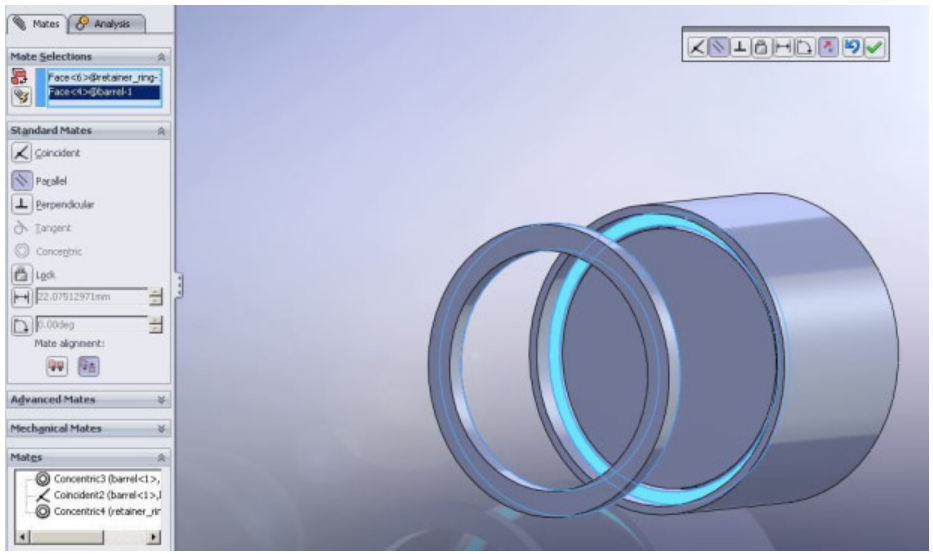
Concentric relation

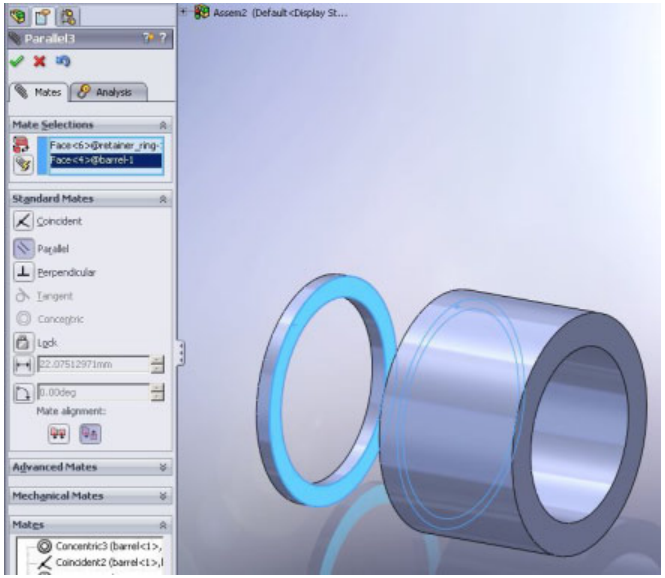


Creating a lens assembly

- 4-4. Give mating relations between them.

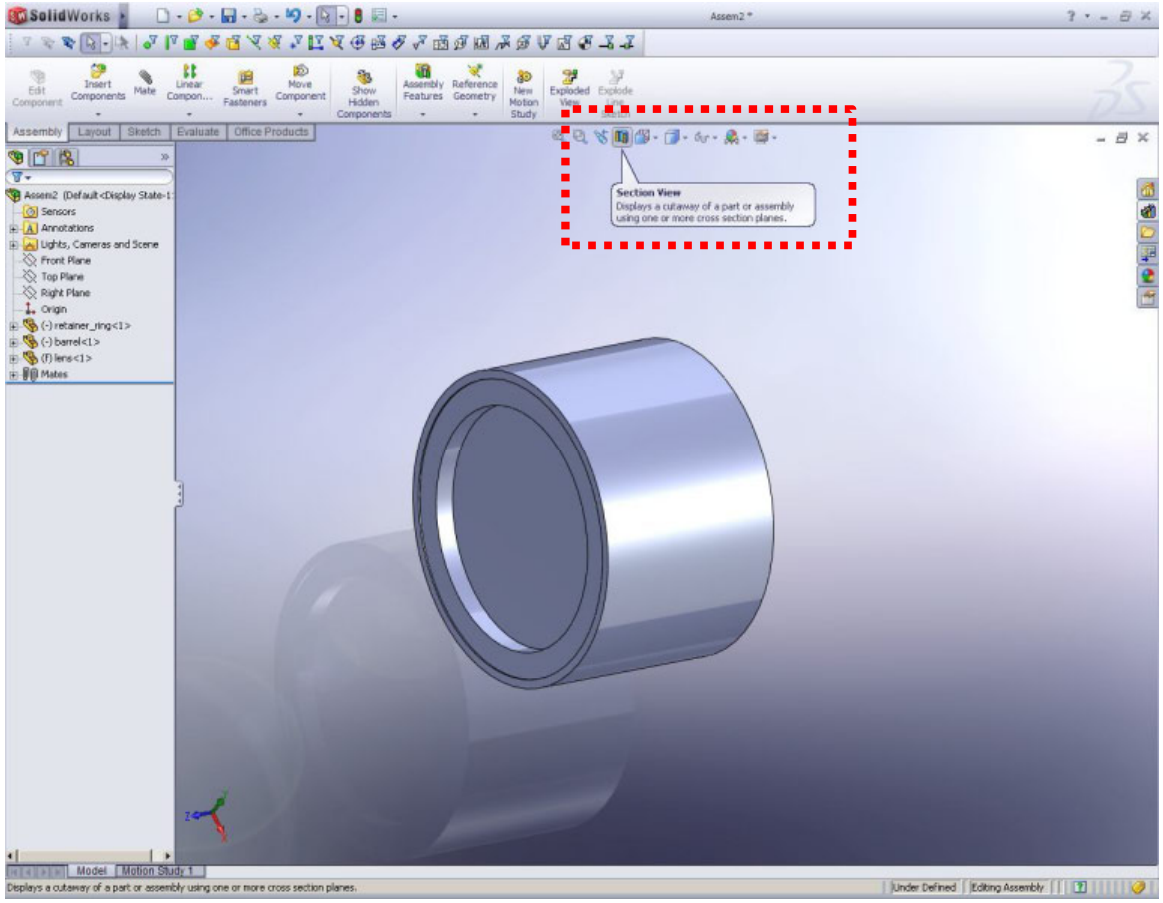
Coincident relation

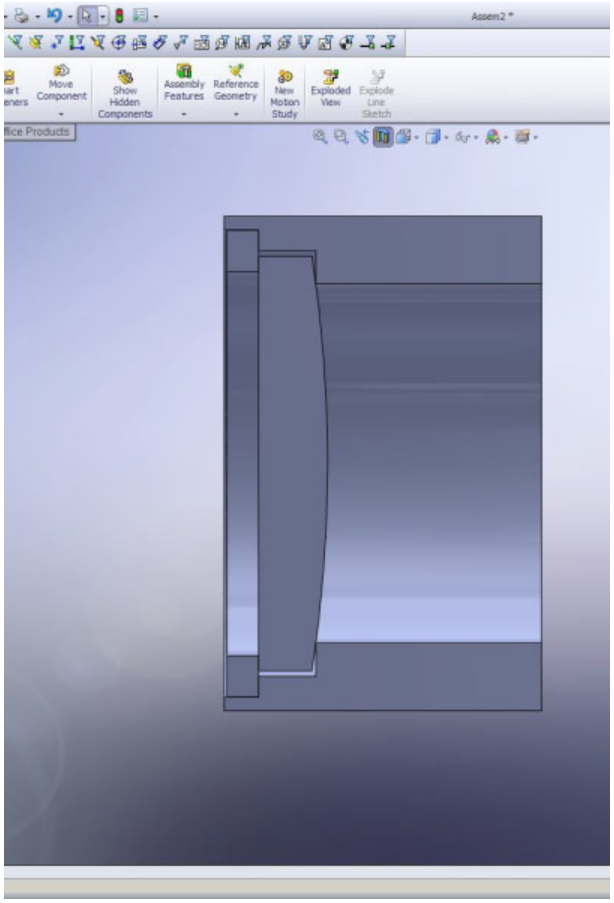




Creating a lens assembly

- 5. Cross section view

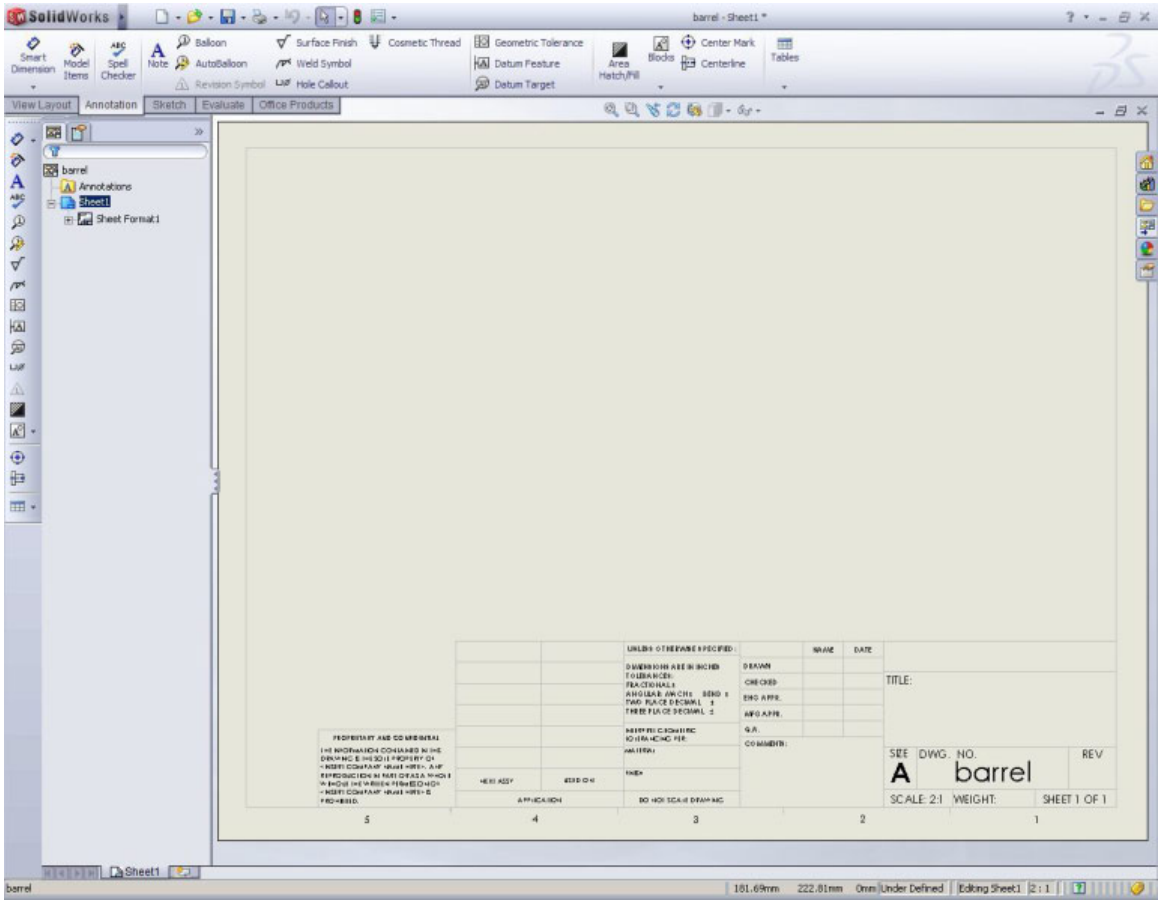




Mechanical drawing

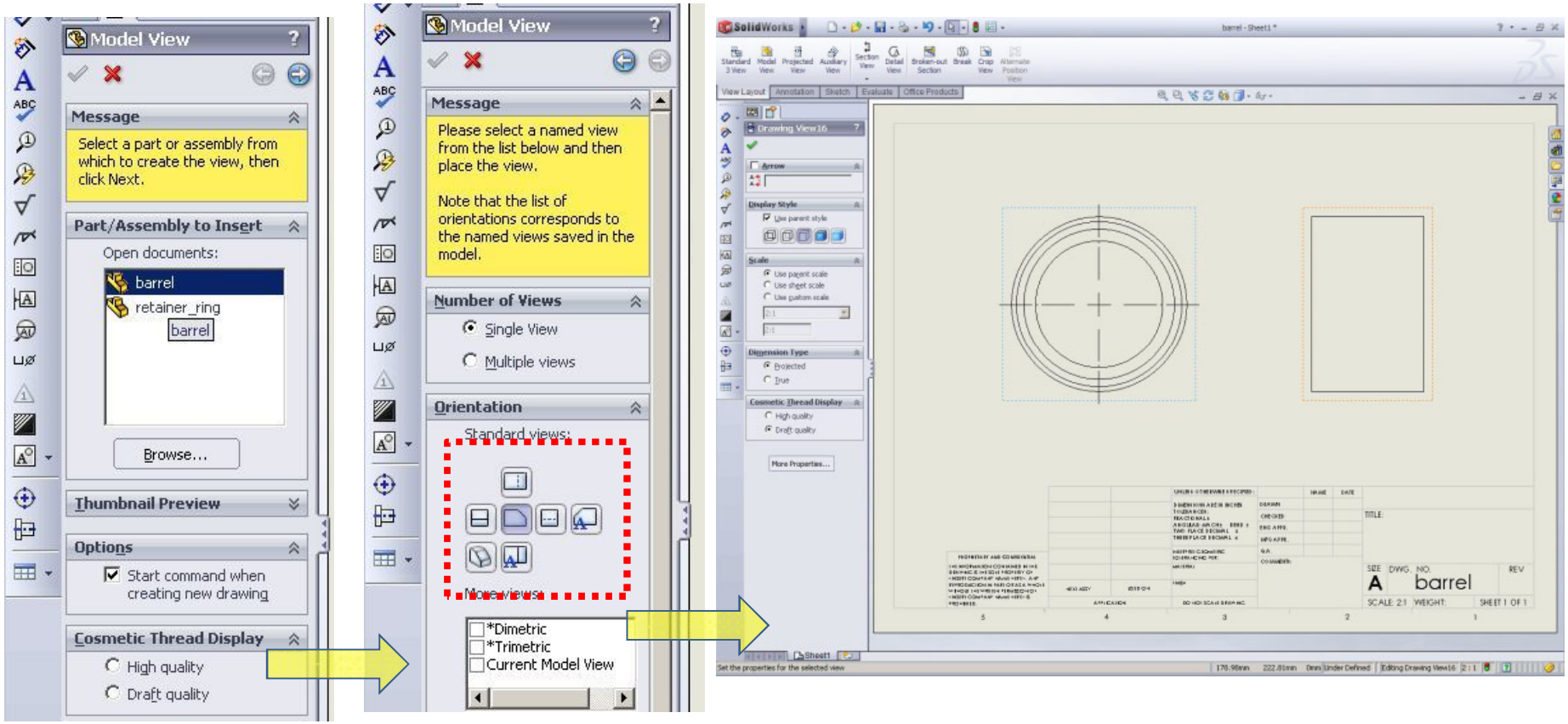
- 1. Open your part file and click ‘Solidworks’ button, then

Click FILE>MAKE DRAWING FROM PARK



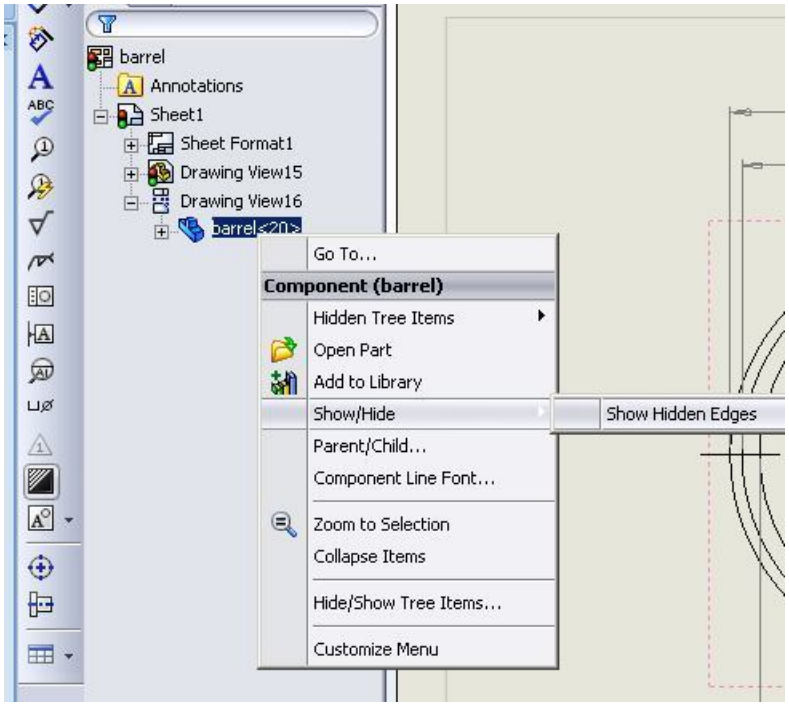
Mechanical drawing

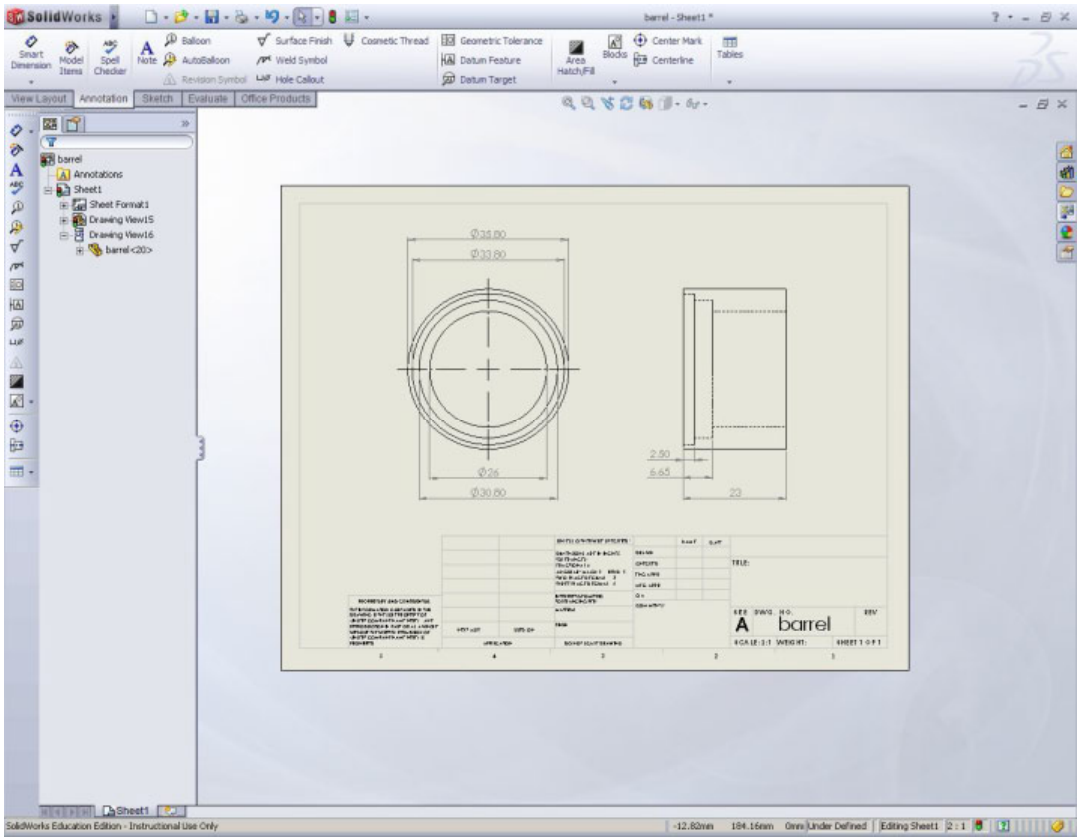
- 2. Click ‘Model view’ in the ‘View Layout’ tab. Open part document you want to draw.



Mechanical drawing

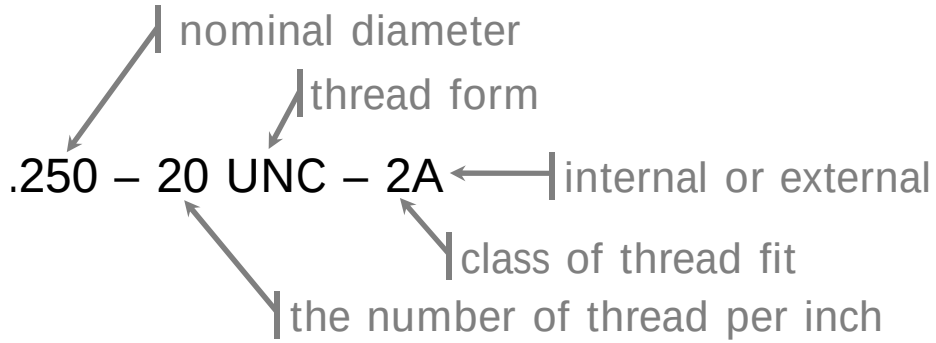
- 3. Put all dimensions in the drawing with ‘Smart dimension’.
- Before doing this, make sure that hidden lines are shown





How to define inner/outer thread

• Standard inch unit thread specification



• Thread forms

UNC: Unified Coarse, most commonly used thread in general purposes.

UNF: Unified Fine, finer threads allows better torque control and higher load tolerance.

UNEF: Unified national Extra Fine, used for ultra-precision purpose or aerospace applications.

• Class fits

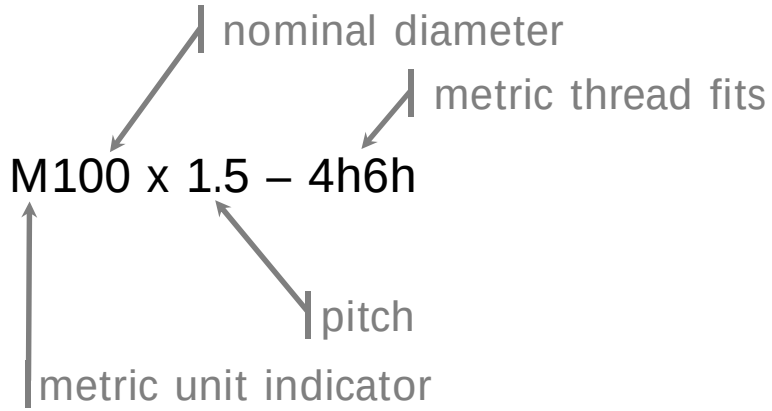
Class **1**: Loose fit, Class **2**: Standard fit, Class **3**: Tight fit

• Internal or external

A: external, **B**: internal

How to define inner/outer thread

- Metric unit thread specification



- Metric thread fits

General purpose fit : 6g (external) , 6H (internal)
Close fit : 5g6g (external), 6H (internal)

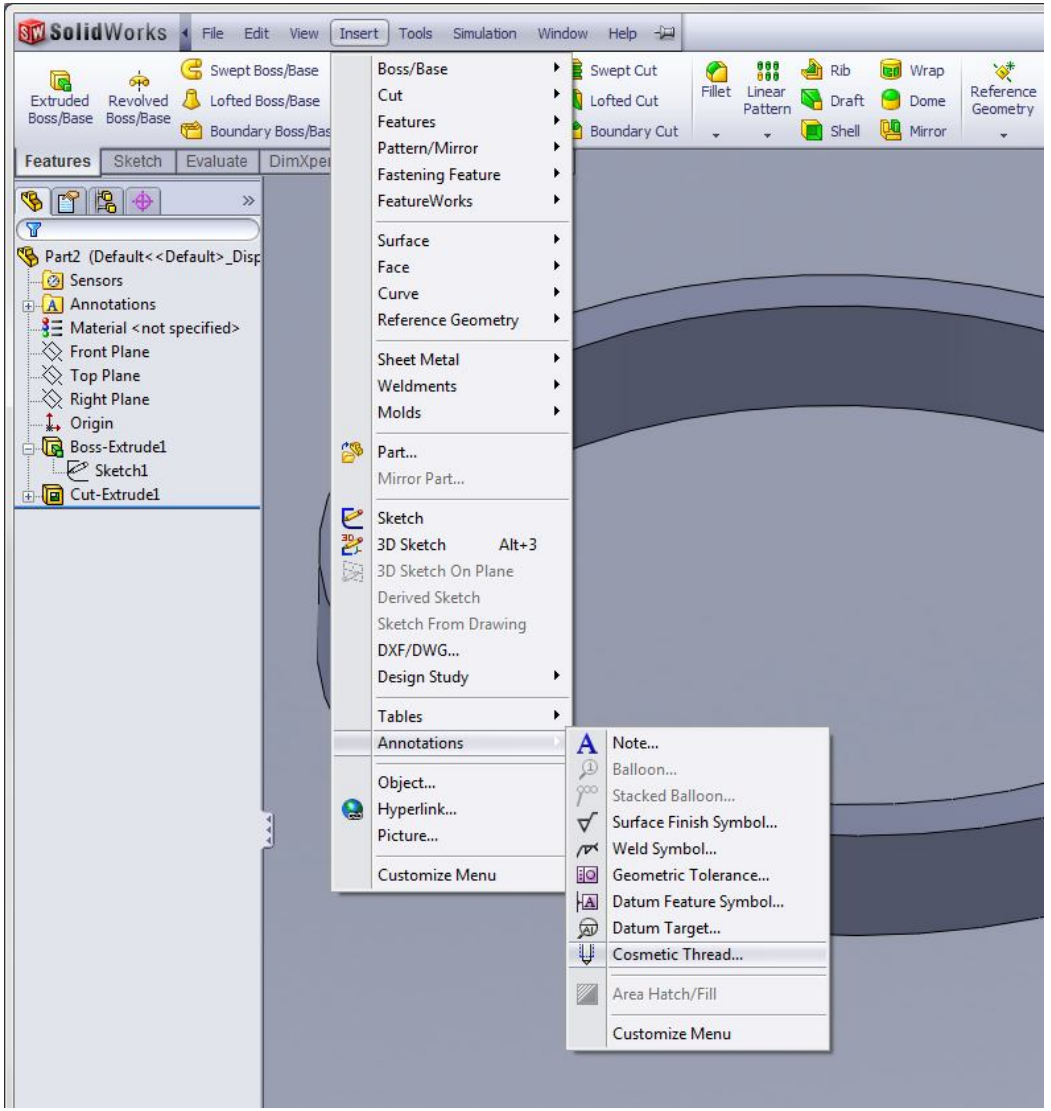
- Note: English unit equivalent

2A → 6g, 2B → 6H
3A → 4h6h, 3B → 4H5H

How to define inner/outer thread *in Solidworks*

Use ‘Cosmetic thread’ in your 3D model

Menu >
Insert >
Annotations>
Cosmetic threads...



How to define inner/outer thread *in Solidworks*

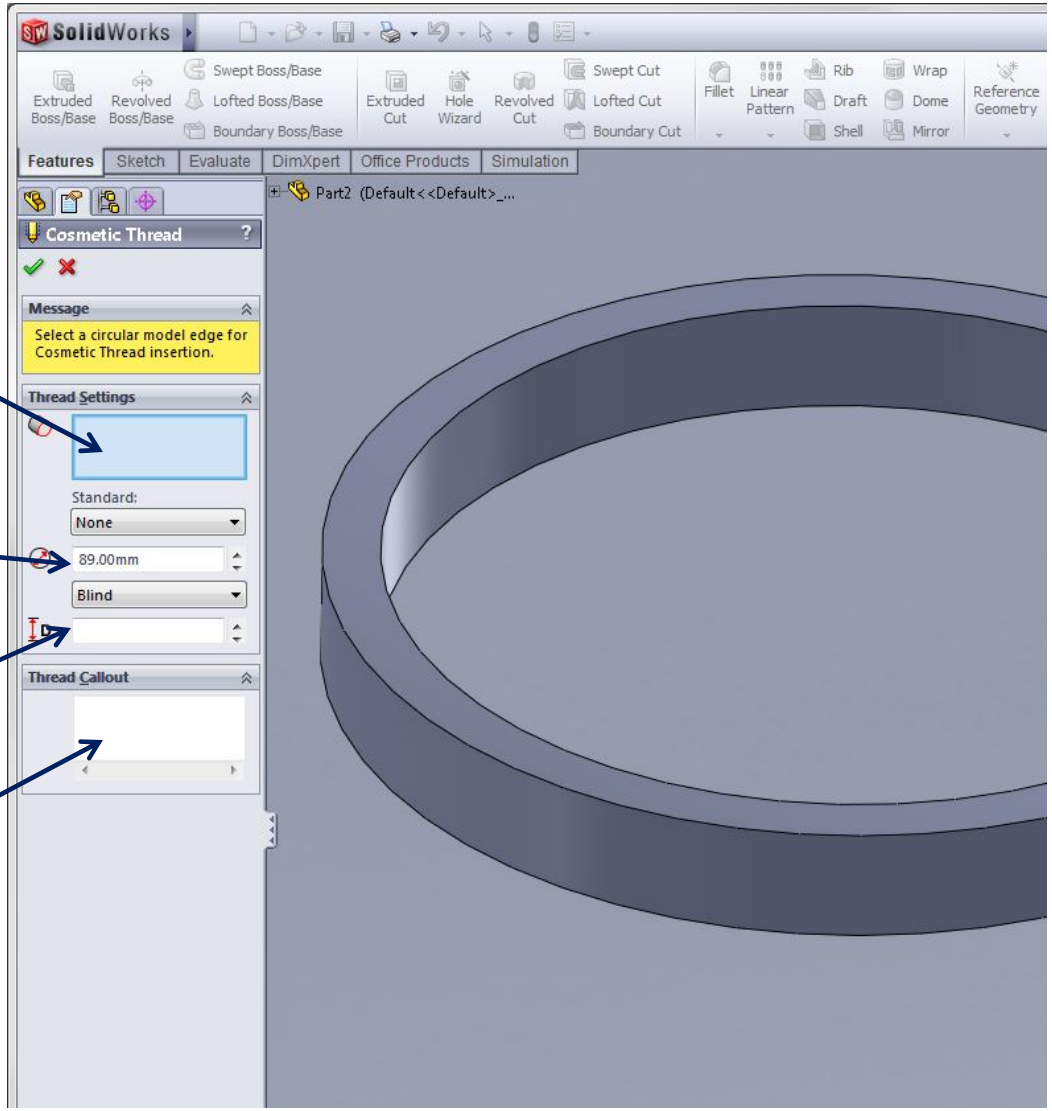
Use ‘Cosmetic thread’ in your 3D model

Select a circular edge for cosmetic thread.

Define minor diameter of the thread.

Define depth of the thread.

Type thread callout



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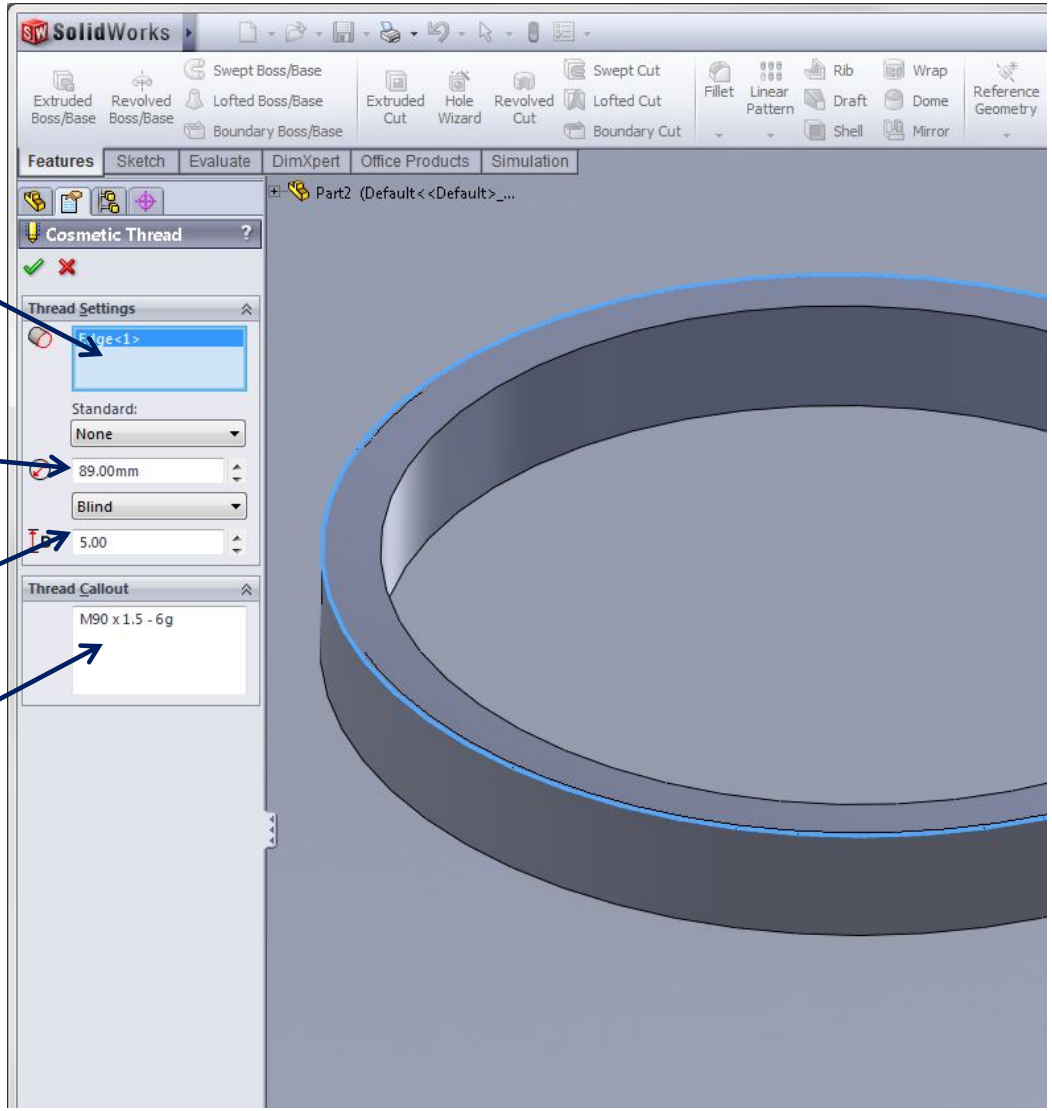
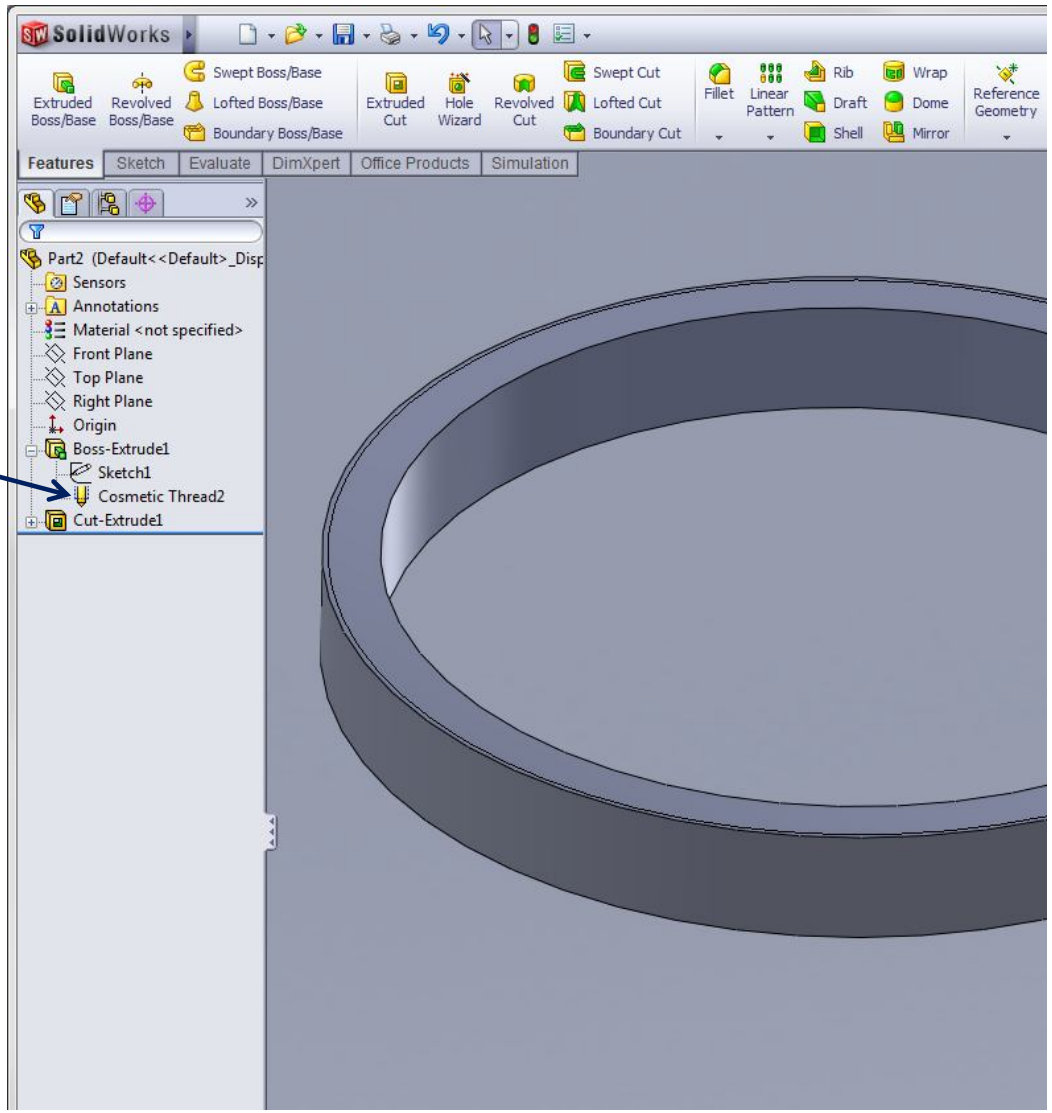


How to define inner/outer thread *in Solidworks*

Cosmetic thread icon

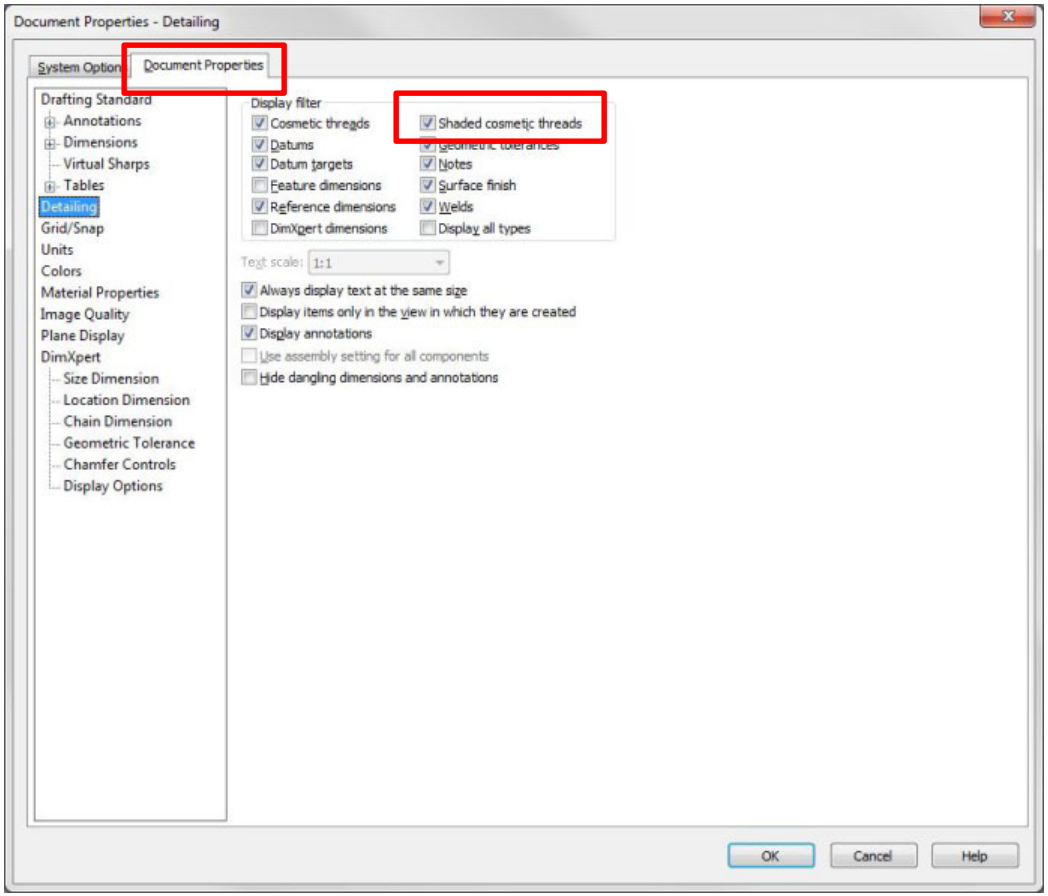


How to define inner/outer thread *in Solidworks*

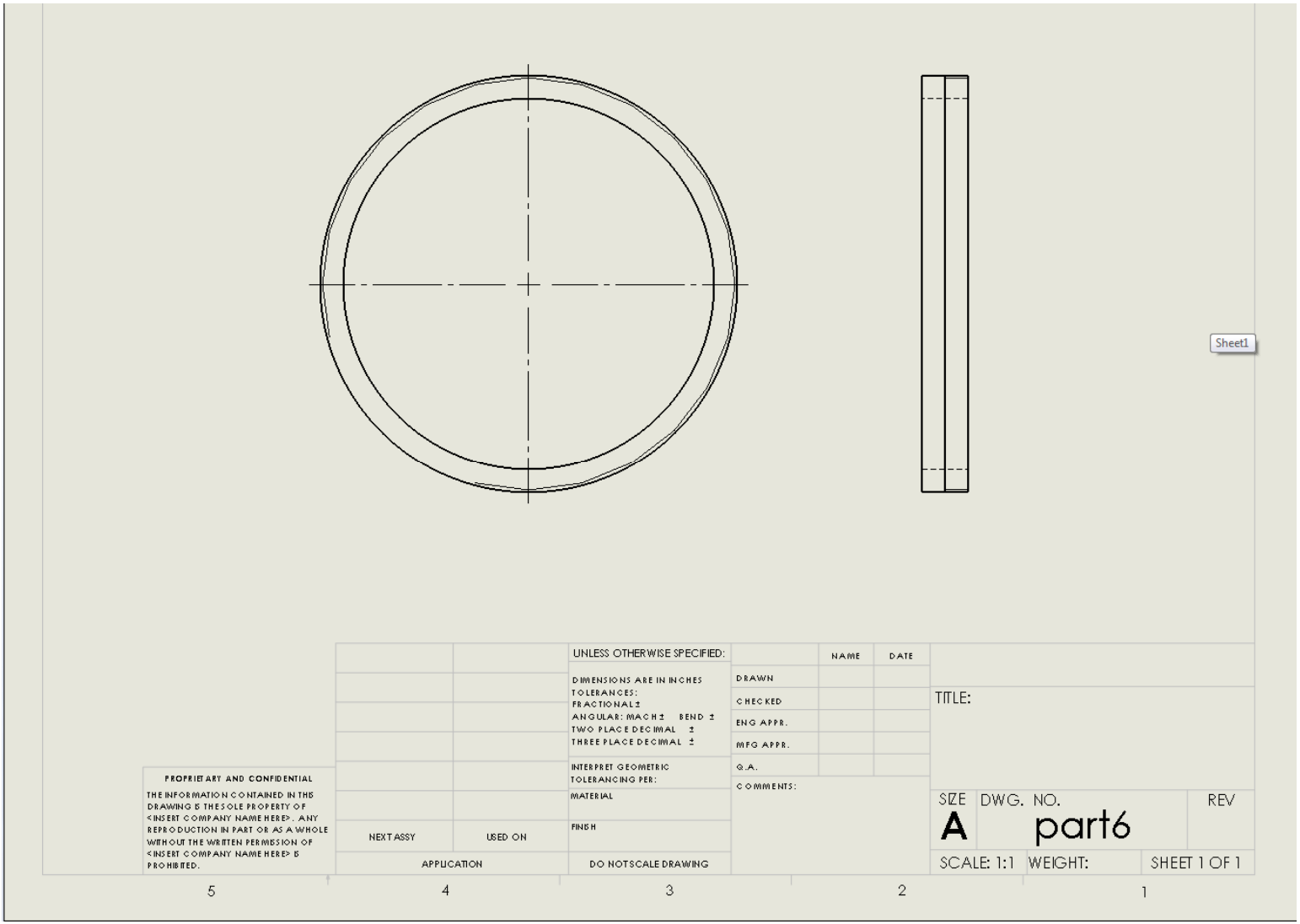
Move your mouse pointer on ‘Option’ and open the option window.

Menu >
Tools >
Options ..

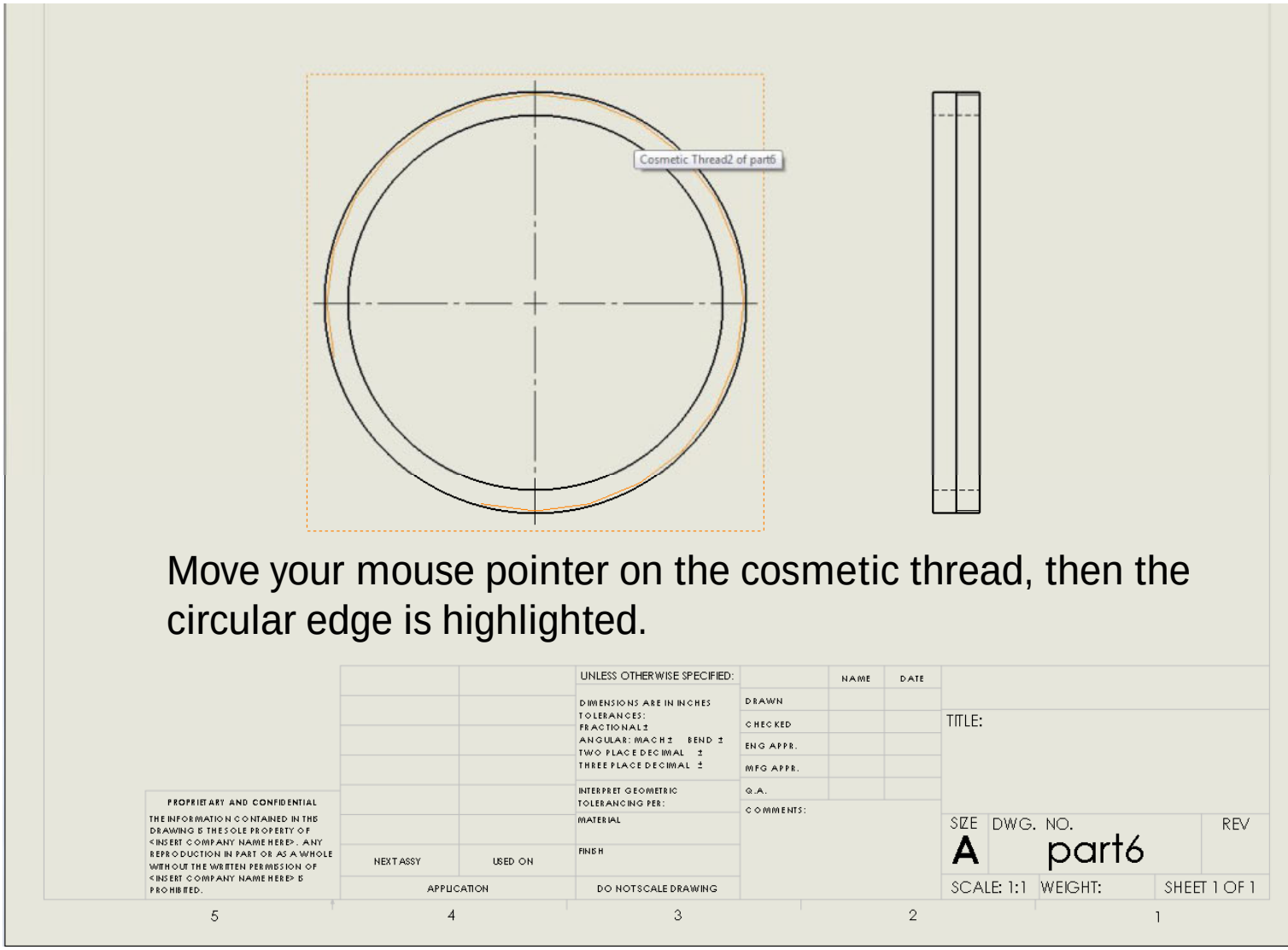
In the ‘Document Properties’ tab, there is ‘Detailing’ on the left box. Click ‘Detailing’ and find ‘shaded cosmetic threads’ in ‘display category’. Check the option.



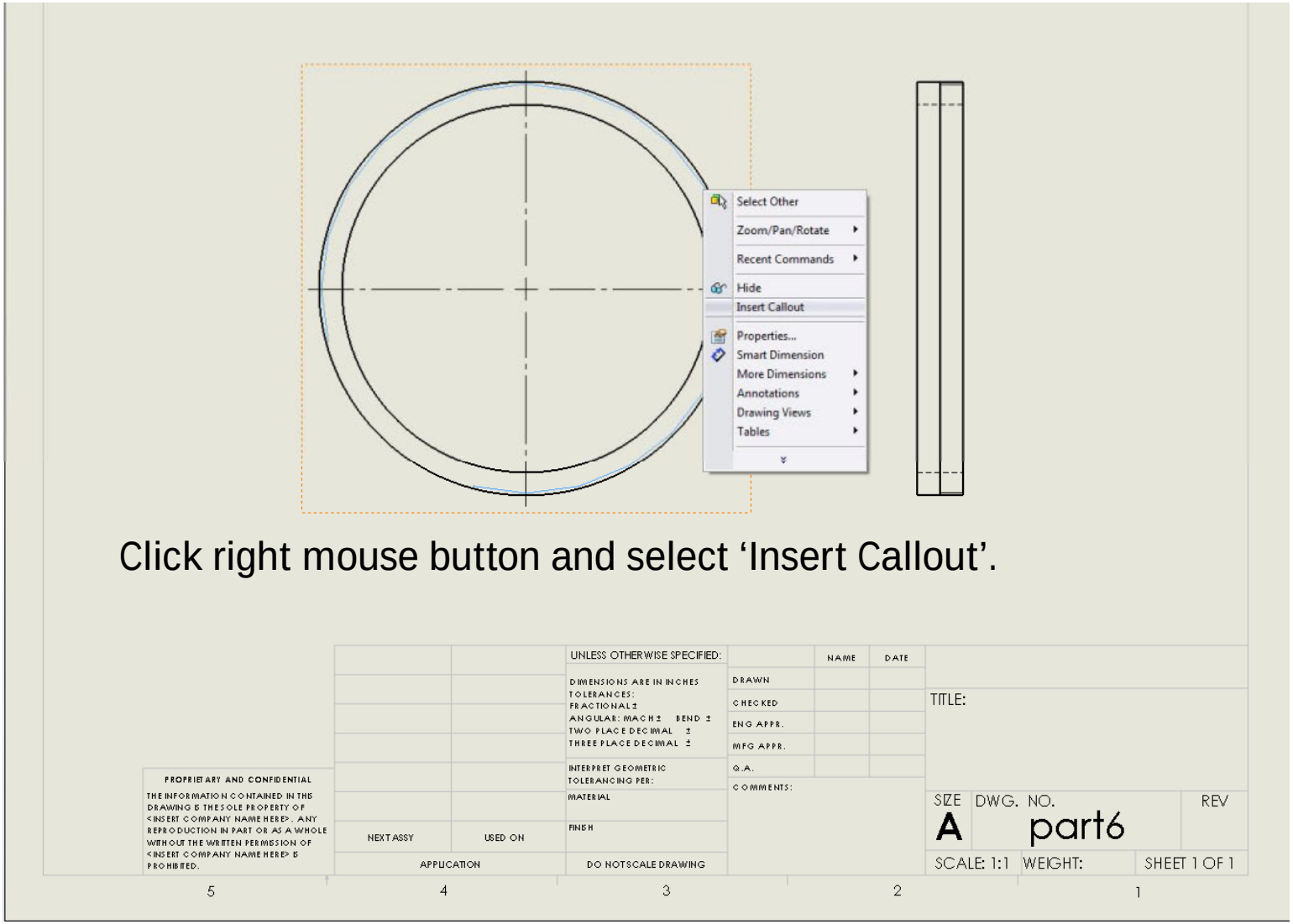
How to define inner/outer thread *in mechanical drawings*



How to define inner/outer thread *in mechanical drawings*



How to define inner/outer thread *in mechanical drawings*



Click right mouse button and select ‘Insert Callout’.

How to define inner/outer thread *in mechanical drawings*