

Lecture #9

SOLID MODELING



This week

You will learn to apply constraints and more commands in solid modeling.

The steps to follow are:

- What are Constraints?
- Types of constraints (Geometric , dimensional , ground)
- Relations in SolidWorks
- More Comands

(Hole Wizard, Mirror, Pattern, Edit Appreance, Material Specification)

- **Assignment # 8**

What is constraint?

The solid model stores more than just the final geometry— it stores the “**design intent**” rules that govern what will happen when geometry changes.

These rules are described as “**constraints**” on wireframe geometry

Types of constraints

There are three major categories of constraints:

Geometric constraints

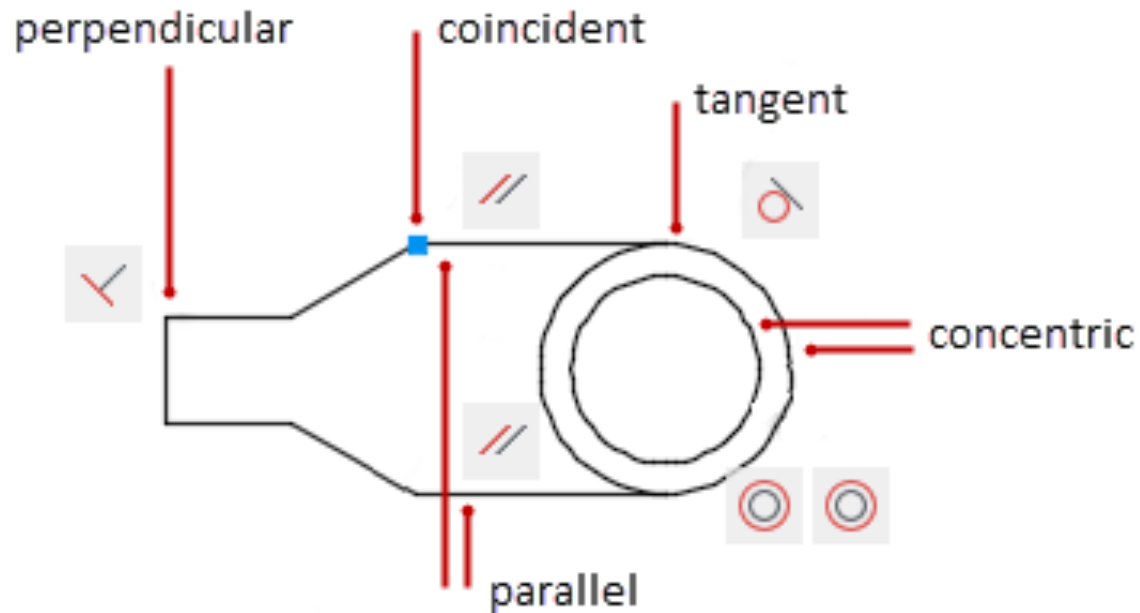
Dimensional constraints

Ground constraints

Geometric constraints

Geometric constraints control the shape of the profile and are used to define the *shape* of 2-D geometry.

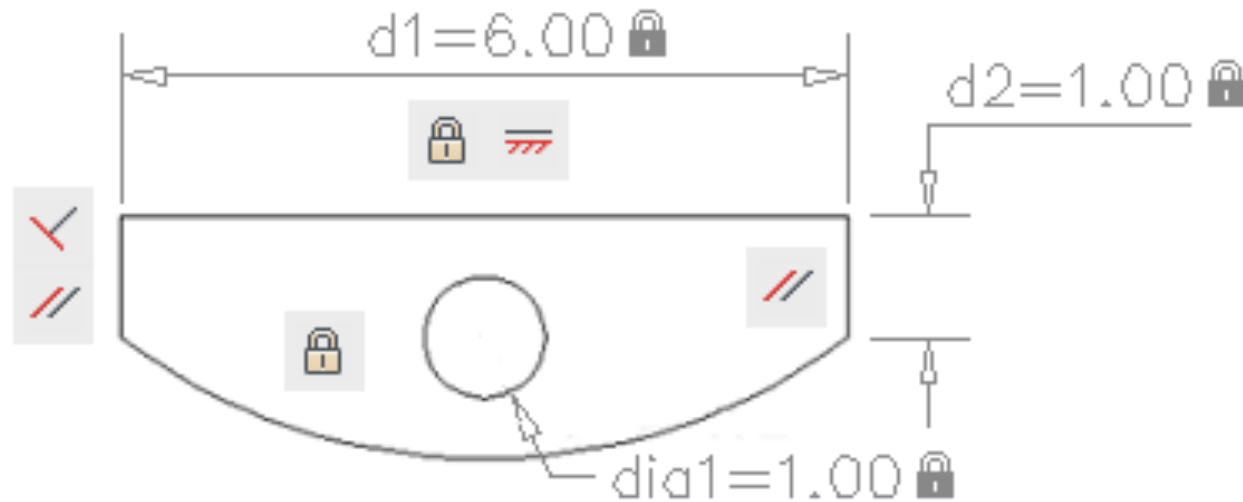
Geometric constraints force the profile to conform to a specific shape.



Dimensional constraints

Dimensional constraints add the parametric characteristic to the profile.

You can change a dimension at any time and its new value will be immediately reflected in the design.



Ground constraints

Ground constraints are added to your sketch to define which of the pair of lines involved is the movable line.

There are basically four types of ground constraints:

Vertical

Horizontal

Angular

X,Y

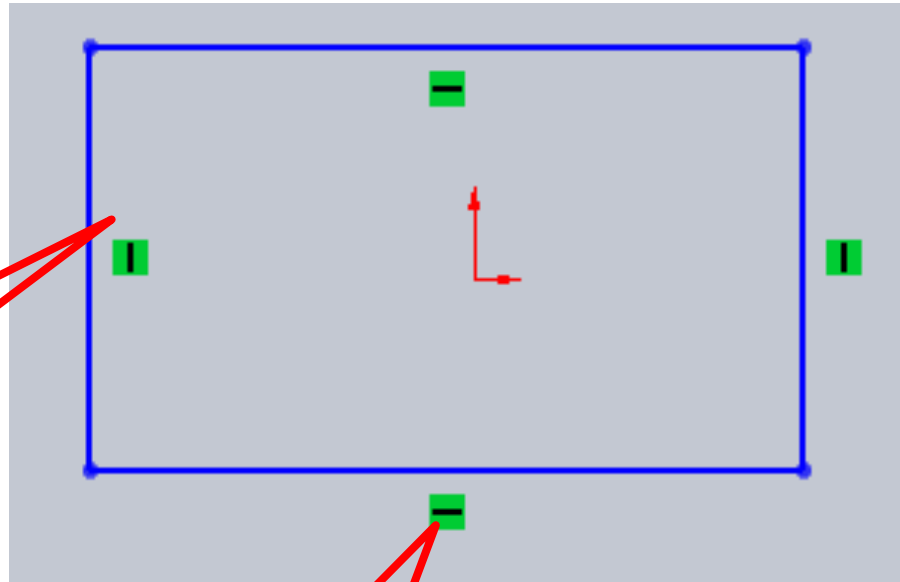
Relations

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List of Relations

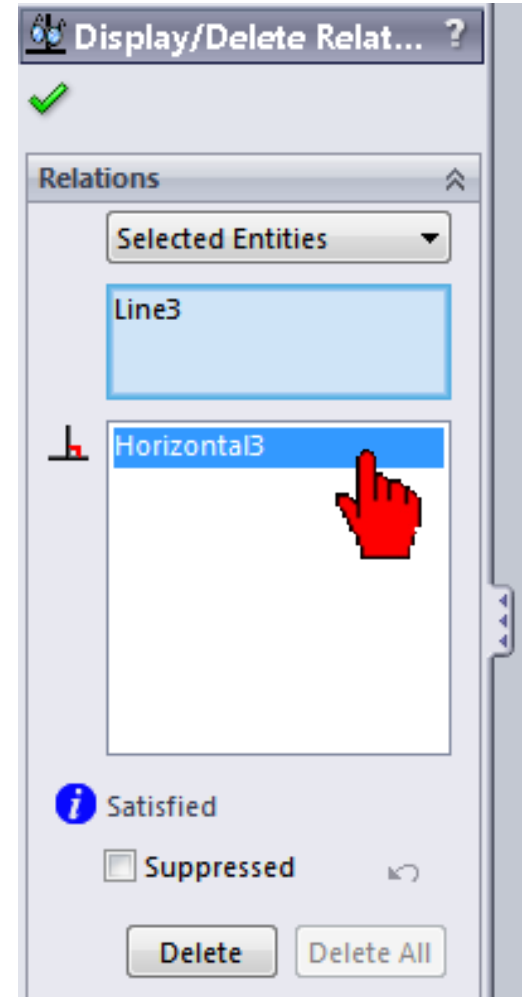
	Horizontal		Coradial
	Vertical		Tangent
	Parallel		Concentric
	Perpendicular		Fix
	Collinear		Equal

Relations

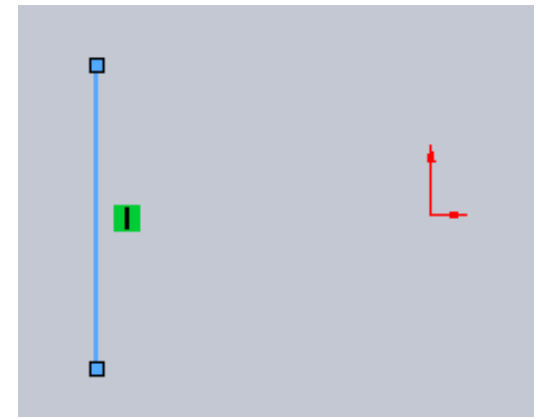
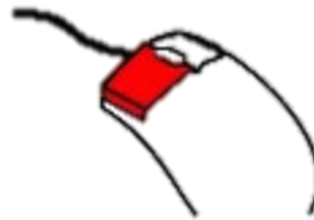
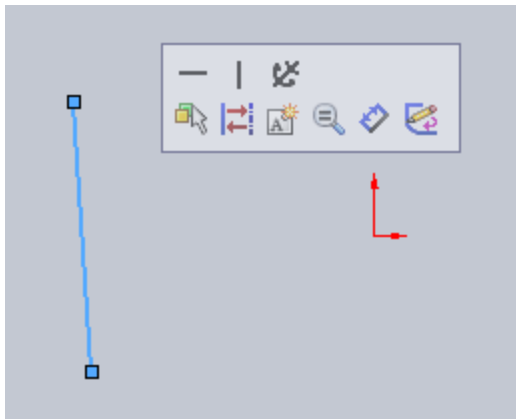
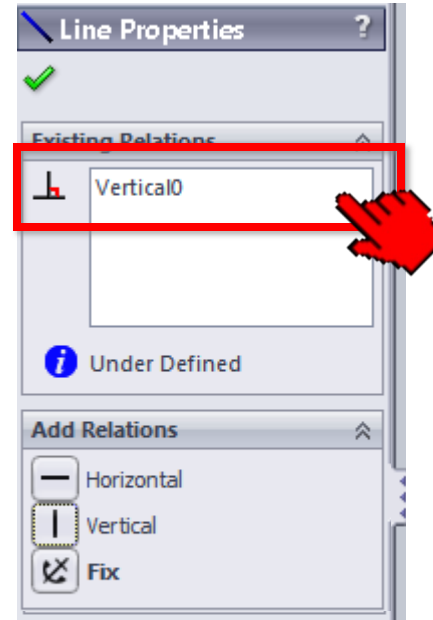
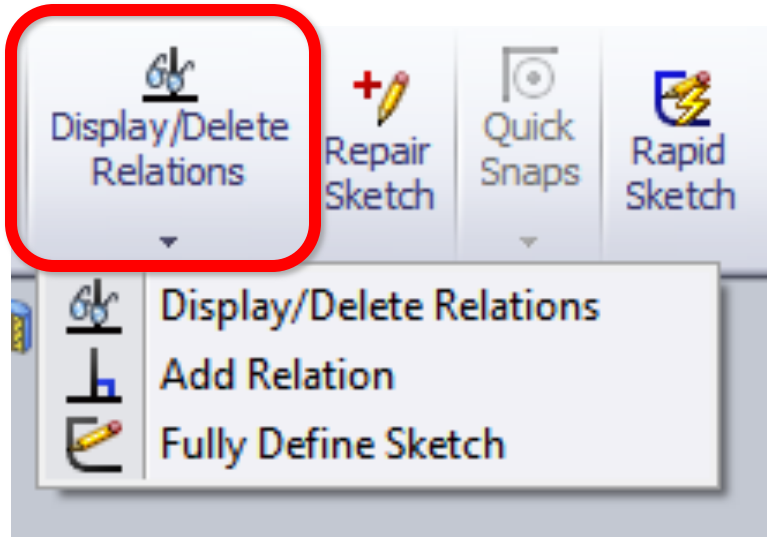


Vertical
relation

Horizontal
relation



Display and add relation



Display and add relation

The image illustrates the process of adding a collinear relation in a CAD application. It is divided into three main sections:

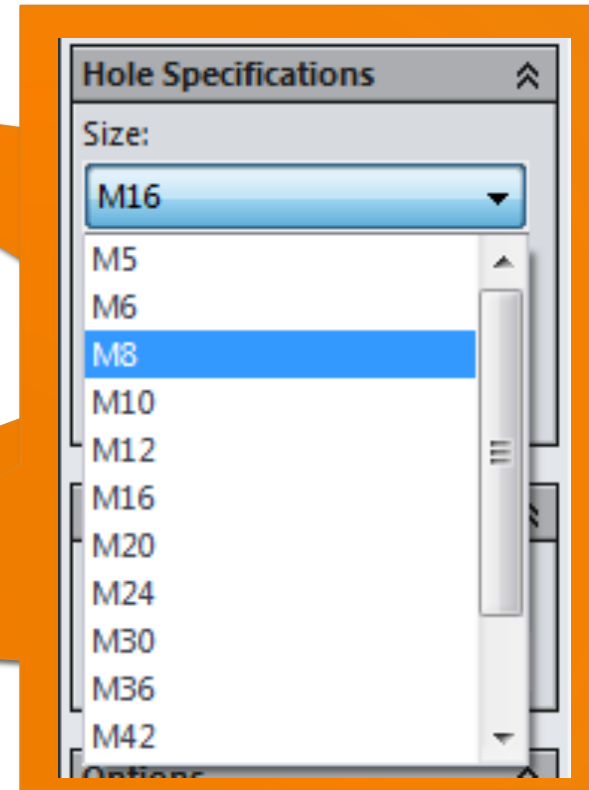
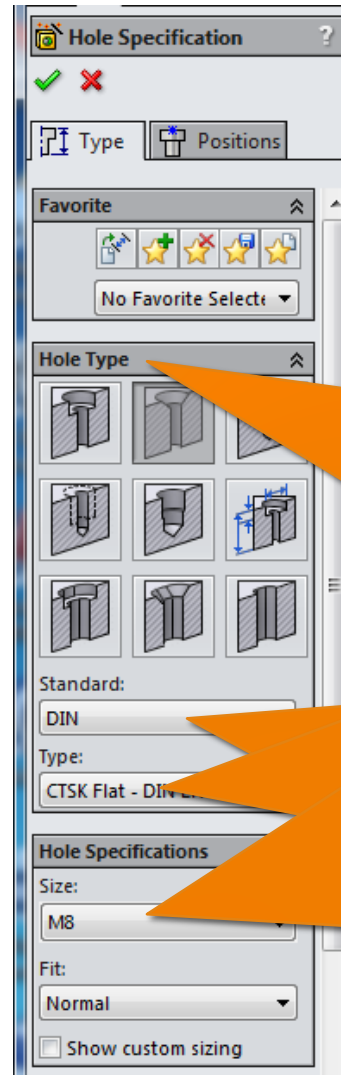
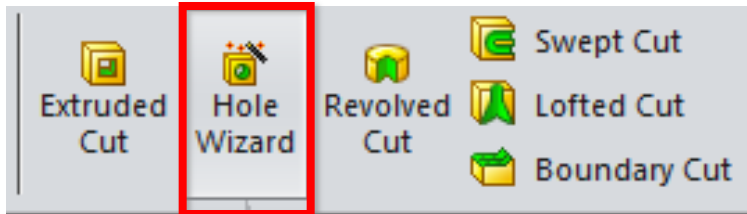
- Top Left:** A ribbon menu with the 'Display/Delete Relations' button highlighted by a red rounded rectangle. A dropdown menu is open, showing 'Display/Delete Relations', 'Add Relation', and 'Fully Define Sketch'.
- Center:** The 'Add Relations' dialog box. The 'Selected Entities' list contains 'Line1' and 'Line2'. The 'Existing Relations' list shows 'Collinear0' with a red dashed arrow pointing to it. Below this, an information icon and the text 'Under Defined' are visible. The 'Add Relations' section at the bottom of the dialog has a red rectangle around the 'Collinear' option, which is also being pointed to by a red hand cursor.
- Bottom:** Three panels showing the sketching process. The left panel shows two vertical lines with red hand cursors on their endpoints. The middle panel shows a mouse cursor clicking on the 'Collinear' option in the dialog. The right panel shows the two lines now aligned vertically, with green checkmarks indicating the relation is applied.

More Commands

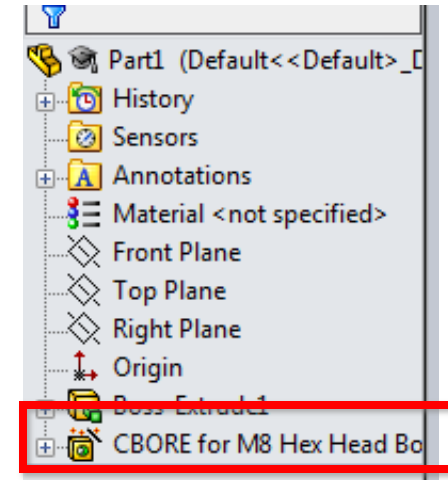
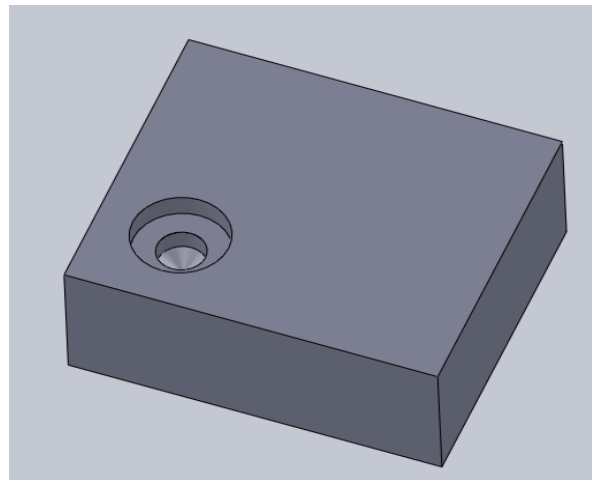
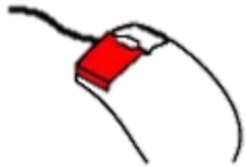
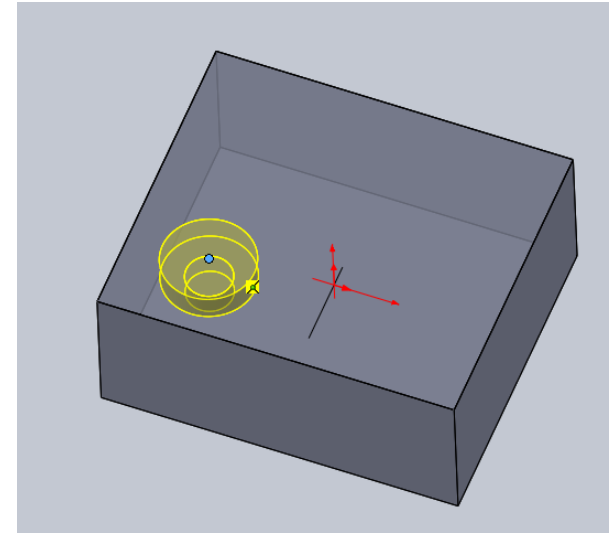
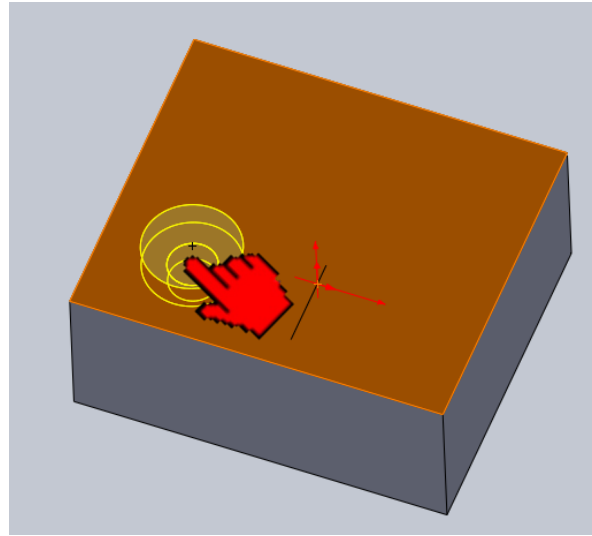
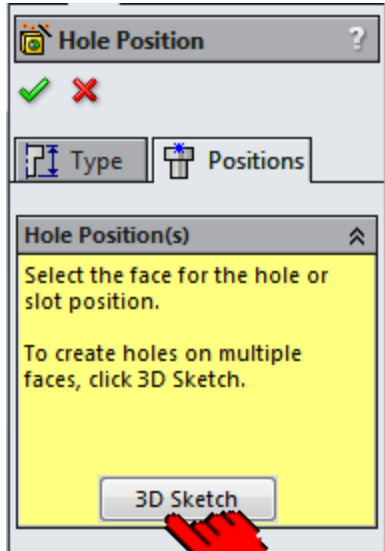
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Hole Wizard

To insert a hole using a pre-defined cross-section (like threaded, conterbore)

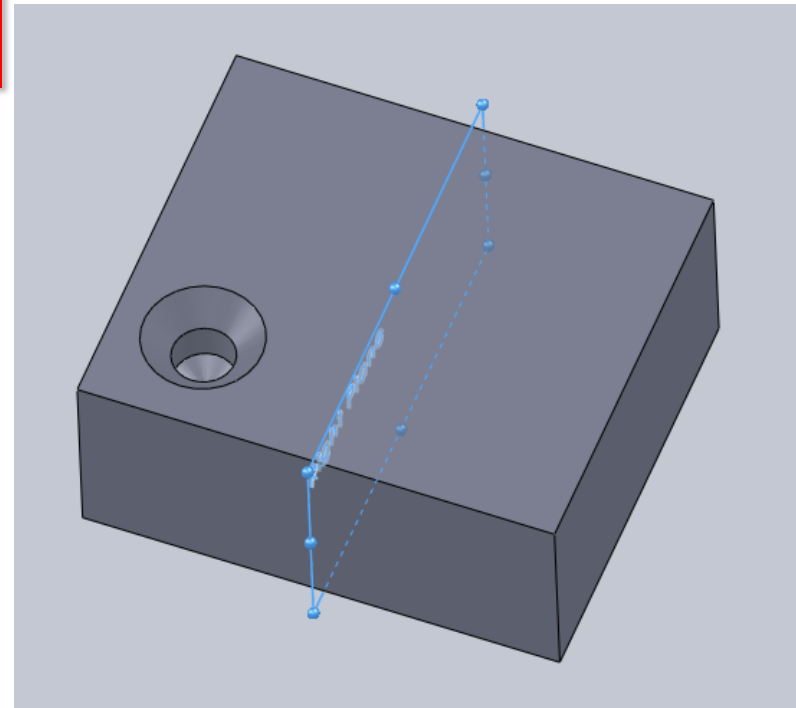
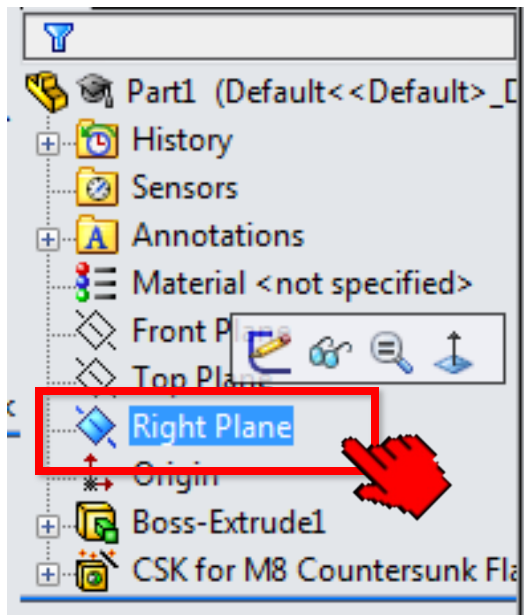
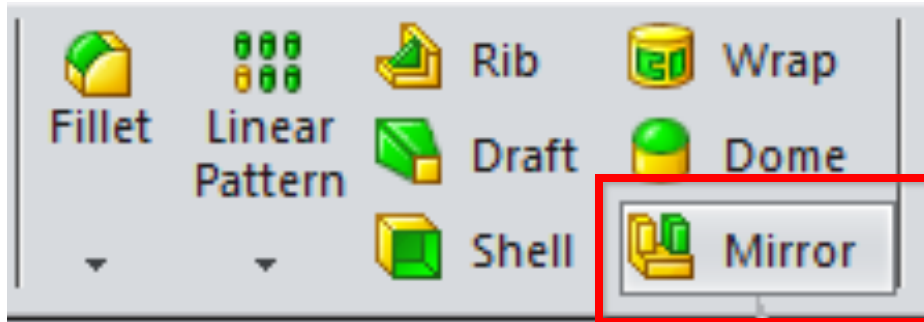


Hole Wizard

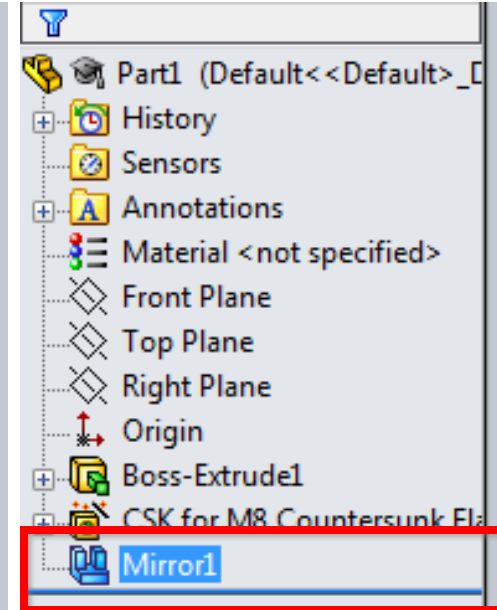
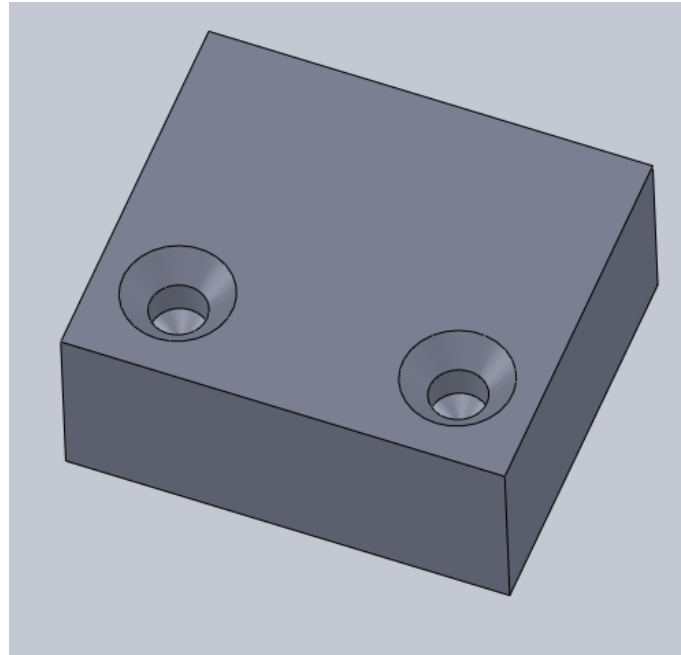
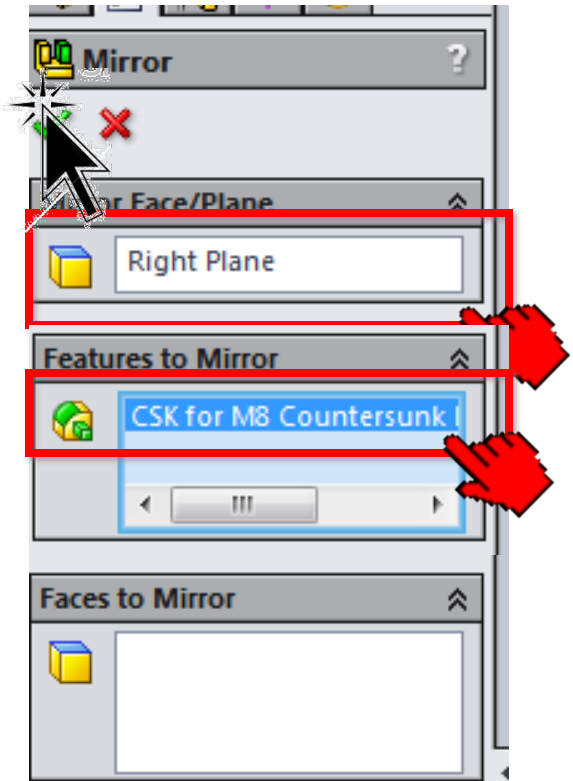


Mirror in 3D

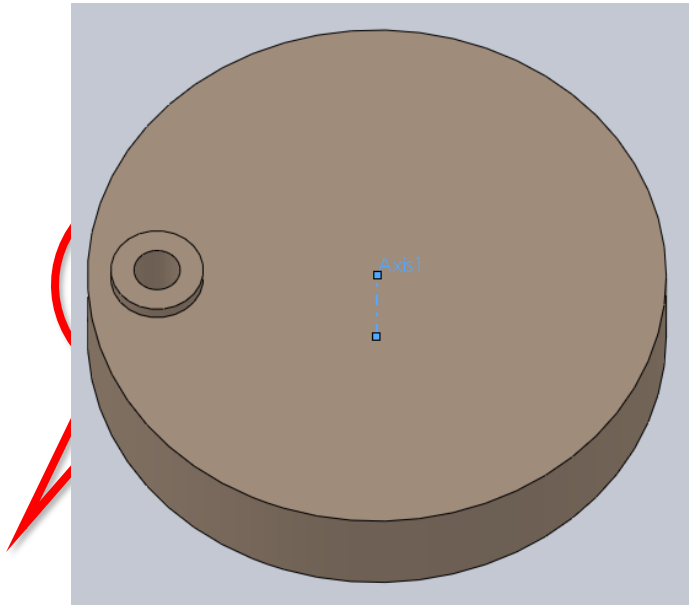
Mirrors features, faces and bodies about a face or a plane



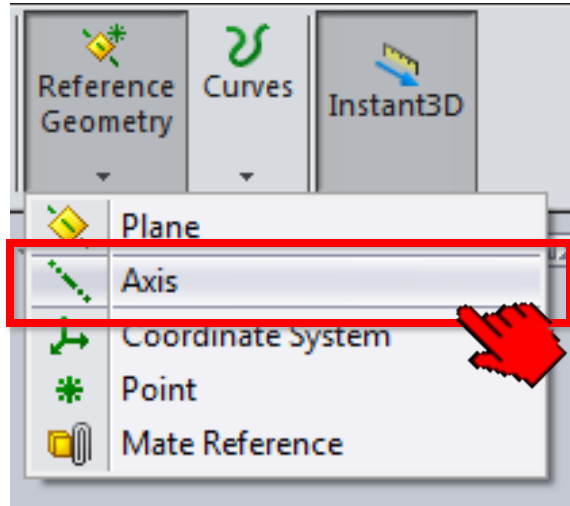
Mirror in 3D



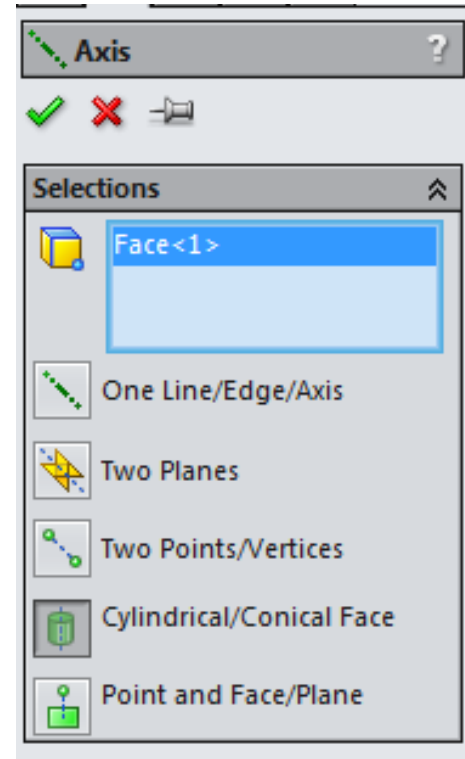
Pattern in 3D



To be copied feature
with PATTERN

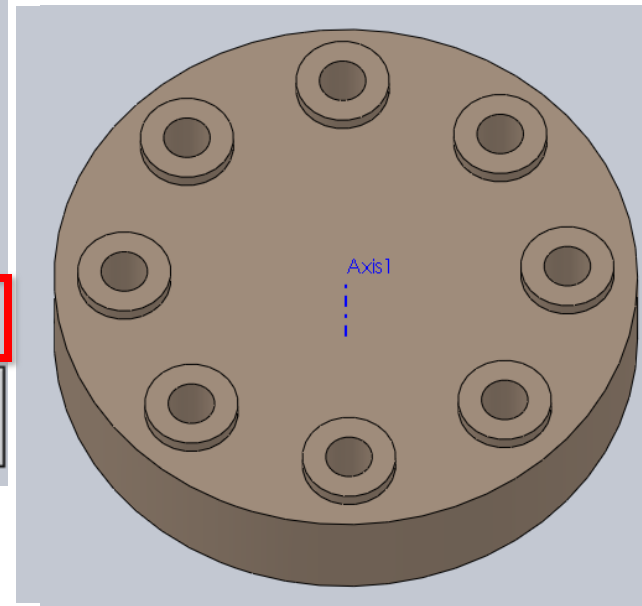
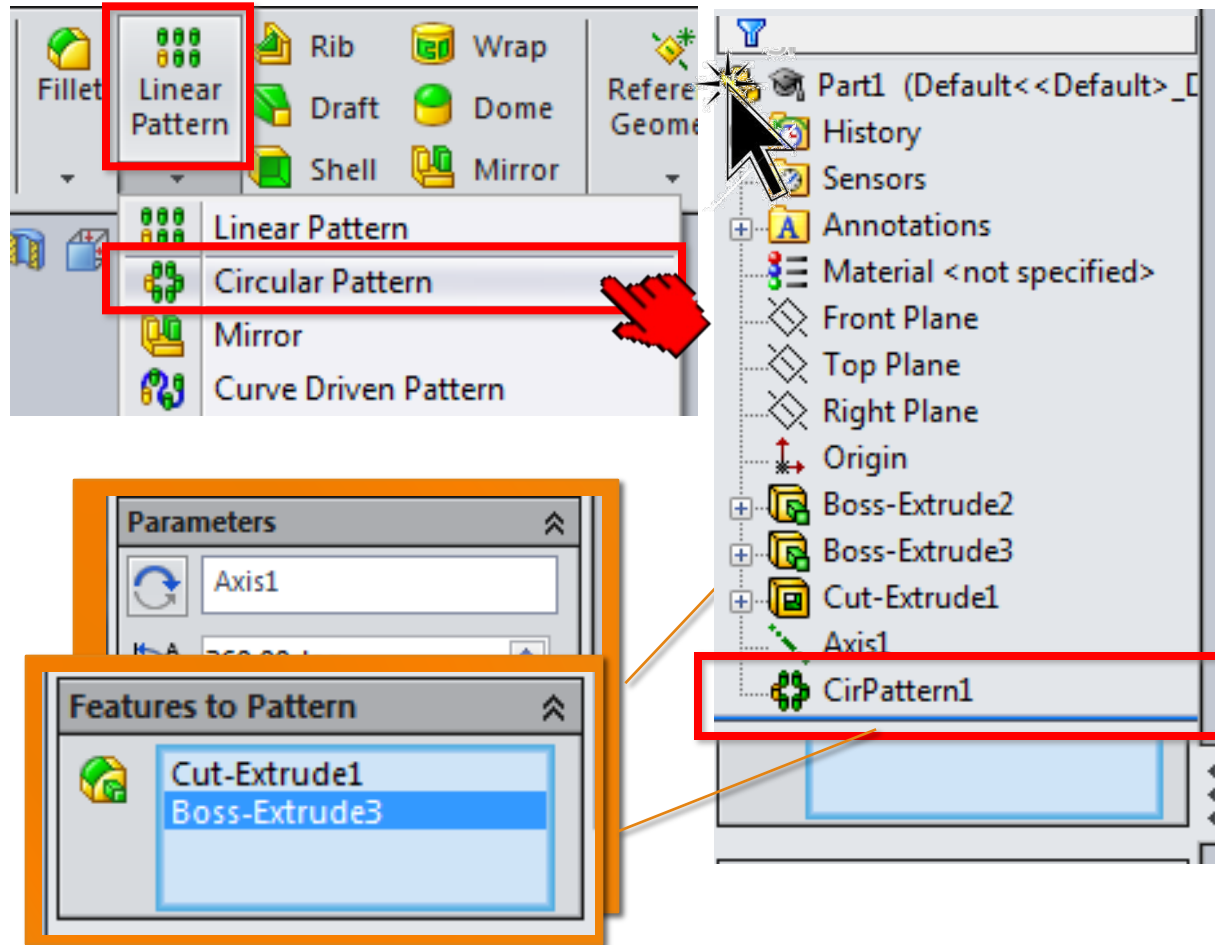


① Define an axis



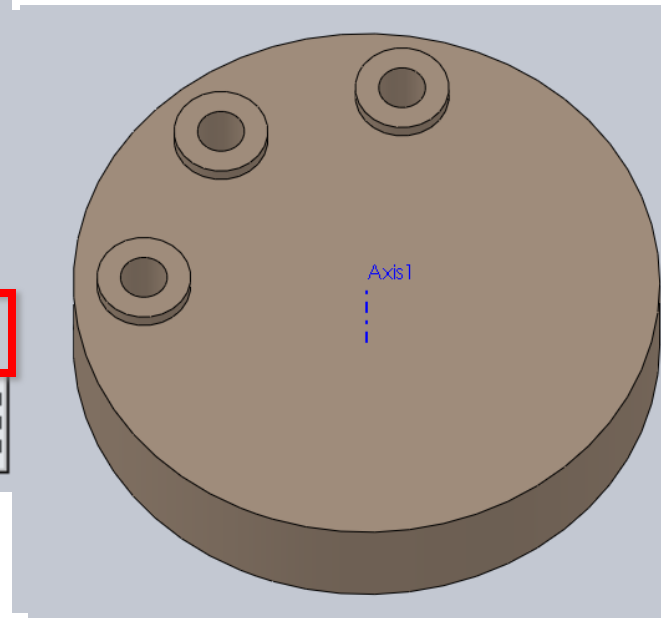
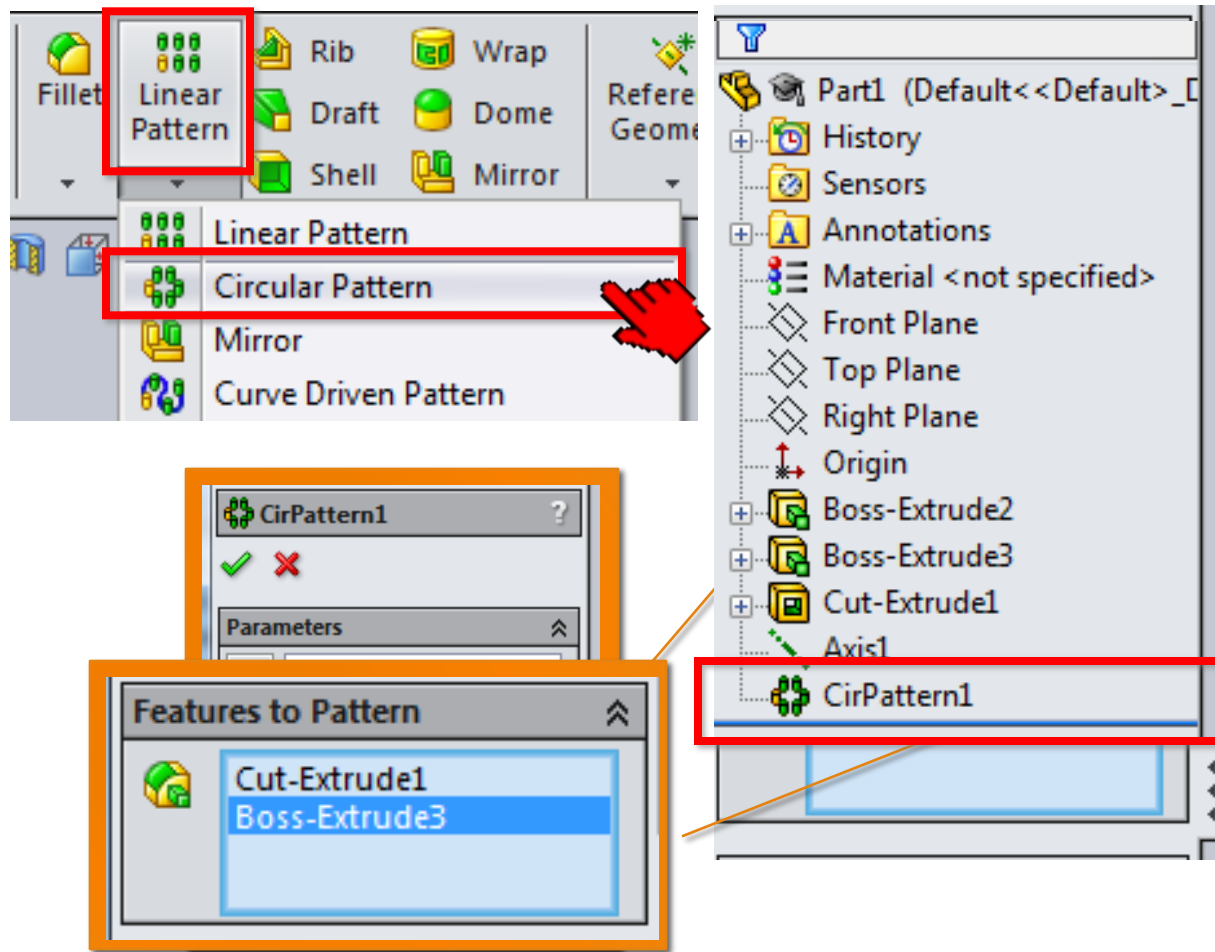
Pattern in 3D

② Select CIRCULAR PATTERN

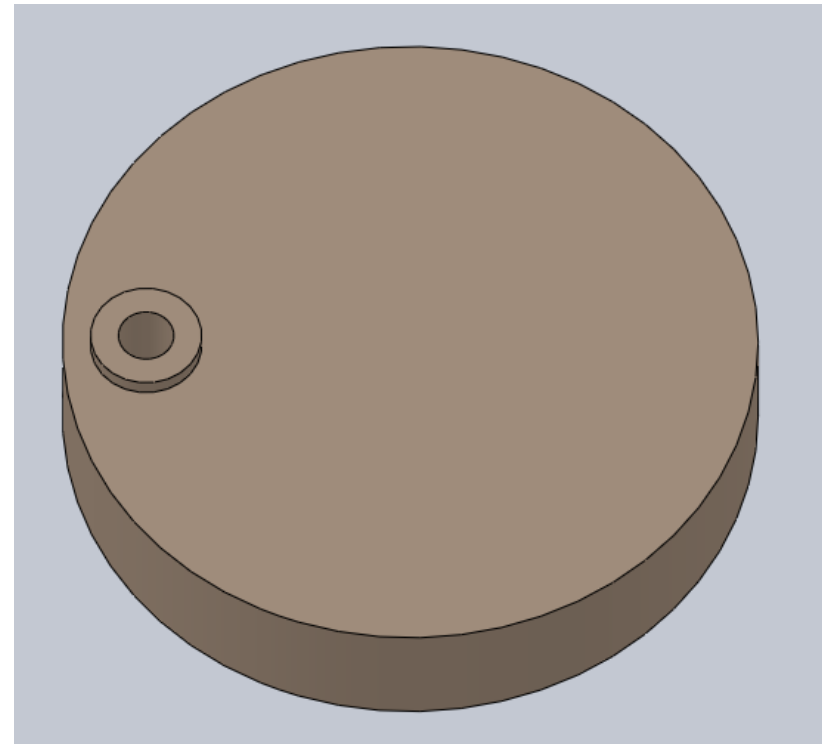
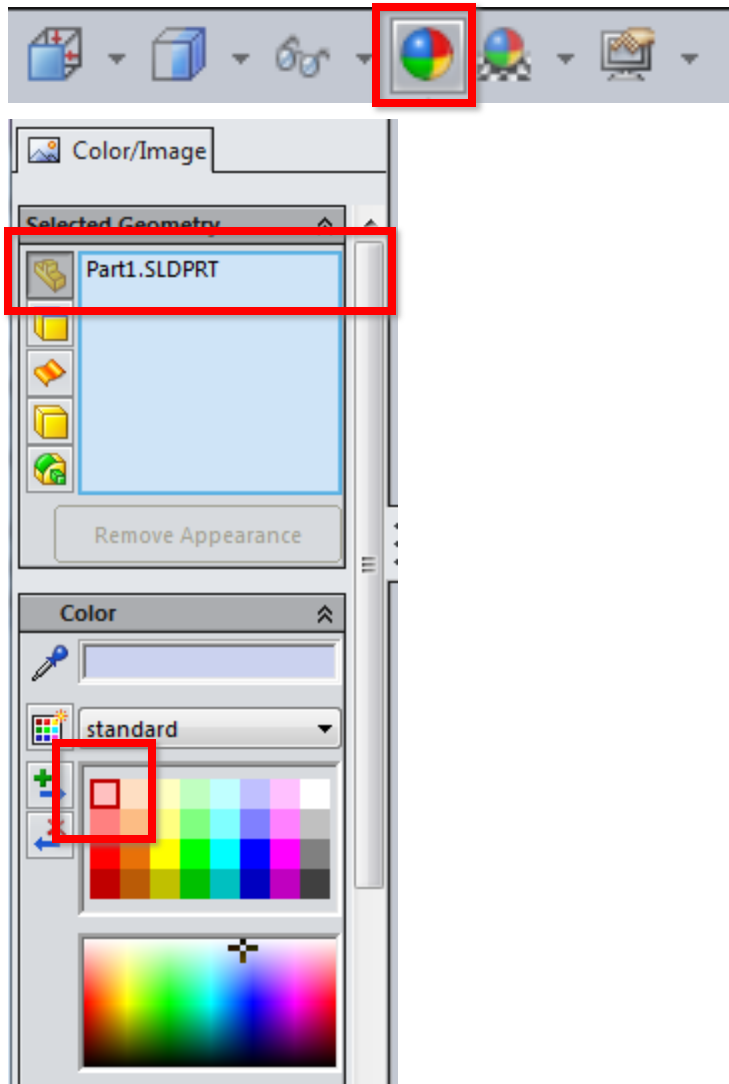


Pattern in 3D

② Select CIRCULAR PATTERN



Edit Appearance

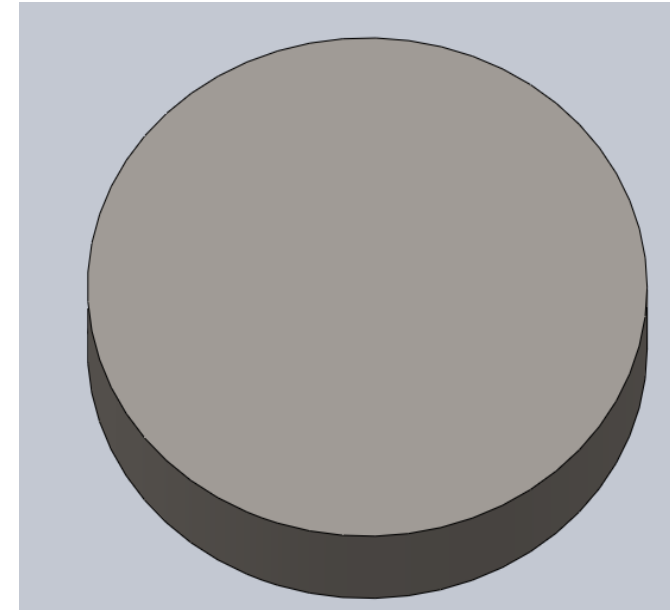


Material Specifying

The screenshot shows a CAD software interface with a tree view on the left and a material specification dialog box on the right. The tree view includes 'Part1 (Default<<Default>>_...', 'History', 'Sensors', 'Annotations', 'AISI 1020 Steel, Cold Rolled' (highlighted with a red box), 'Front Plane', 'Top Plane', 'Right Plane', 'Origin', and 'Boss-Extrude2'. The material specification dialog box has tabs for 'Custom', 'Application Data', and 'Favorites'. It contains a warning: 'not be edited. You must first copy the material'. Below this are several input fields, including a dropdown menu set to 'tropic' and a checkbox for 'Cold Rolled'. A table of material properties is displayed:









Value	Units
732736.22	psi
9	N/A
603019.01	psi
84322	lb/in^3
915.85	psi
	psi
763.21	psi
ie-006	/°F
10069415	Btu/(in·sec·°F)
Specific Heat	0.116079 Btu/(lb·°F)
Material Damping Ratio	N/A

At the bottom of the dialog box are buttons for 'Apply', 'Close', 'Save', 'Config...', and 'Help'. A red hand cursor is pointing at the 'Apply' button. Below the dialog box, a 'Customize Menu' dialog is partially visible with options like 'Collapse Items'.



For more [details](#) ...

SolidWorks

-  SolidWorks 2001 Version Tutorial
PDF, 873,86 KB
-  SolidWorks 2003 Version Tutorial
PDF, 4,40 MB
-  SolidWorks 2004 Version Tutorial
PDF, 3,87 MB
-  SolidWorks 2005 Version Tutorial
PDF, 182,60 KB
-  Solidworks 2014 Version Tutorial
PDF, 7,42 MB
-  Solidworks 2014 Part
PDF, 6,42 MB
-  Solidworks 2014 Drawing
PDF, 7,66 MB
-  Solidworks 2014 Assembly
PDF, 49,46 MB



SolidWorks 2014 Part I - Basic Tools

Parts, Assemblies and Drawings



Paul Tran CSWE, CSM

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The following week

You will learn how to align parts in an assembly. The steps to follow are:

- Introduction to Assembly
- Assembly Drawing
- Exploded Assemblies
- Bill of Materials (or Part List)
- **Assignment # 9**



The following week



MIDTERM EXAM #2

The following week ...

DON'T FORGET



To complete and submit
the
Progress Report #2

<http://transport.itu.edu.tr/dersler/lisansdersleri/mak110e/desingproject>

The following week ...



DON'T FORGET



Homework #3

Due to : 11th course week

Design one of the components for fabrication by welding, using layout A3 (adjusted)
draw in full-size

- make a two-view working drawing (draw cross-sectional view of the welding joints)
- indicate the dimensions using the welding symbols
- include on the drawing a bill of materials and identify each part on the assembly
- complete enlarged (detail) views of the welded joints to show the edge preparation for the drawing callouts
- use notes to explain any additional welding requirements

http://transport.itu.edu.tr/mak110e/mak110e_HW.htm

Assignment #8

Page

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Figure

7.61

You will generate the solid model

Submit the assignment on time

Upload file into NINOVA

