

Computer-aided Drafting (SolidWorks)

Tutorial (Basics in SolidWorks)

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Today's Outline

- Modeling

- 2D

- Sketching
 - Dimensioning
 - Mirror/Pattern
 - Trim/Extend
 - Fillet/Chamfer

- 3D

- Extruded/ Extruded Cut
 - Revolved
 - Fillet/Chamfer

Getting Started

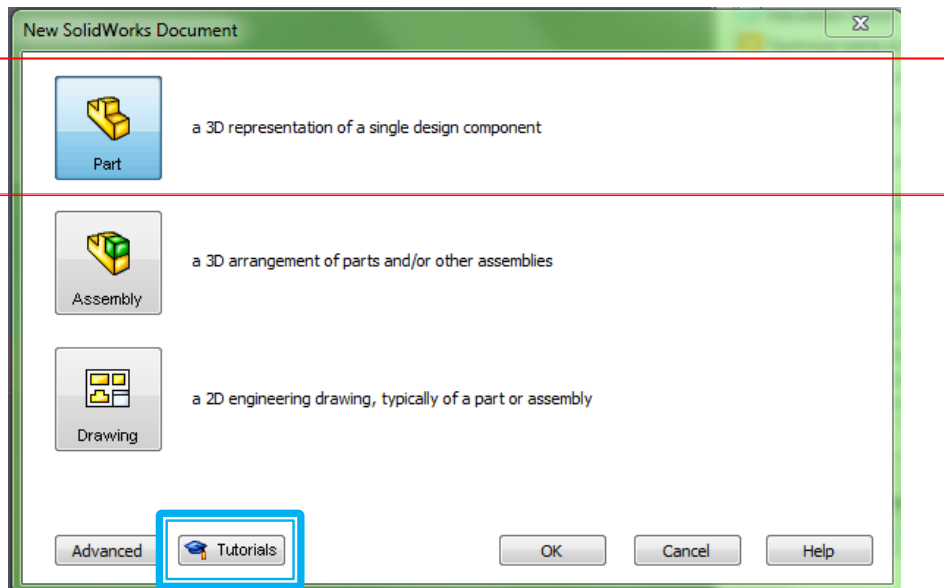
- Open Software

- Start > All programs > SolidWorks 2011 > SolidWorks 2011

- New Document



- Choose "Part" and then "OK" / "Enter"

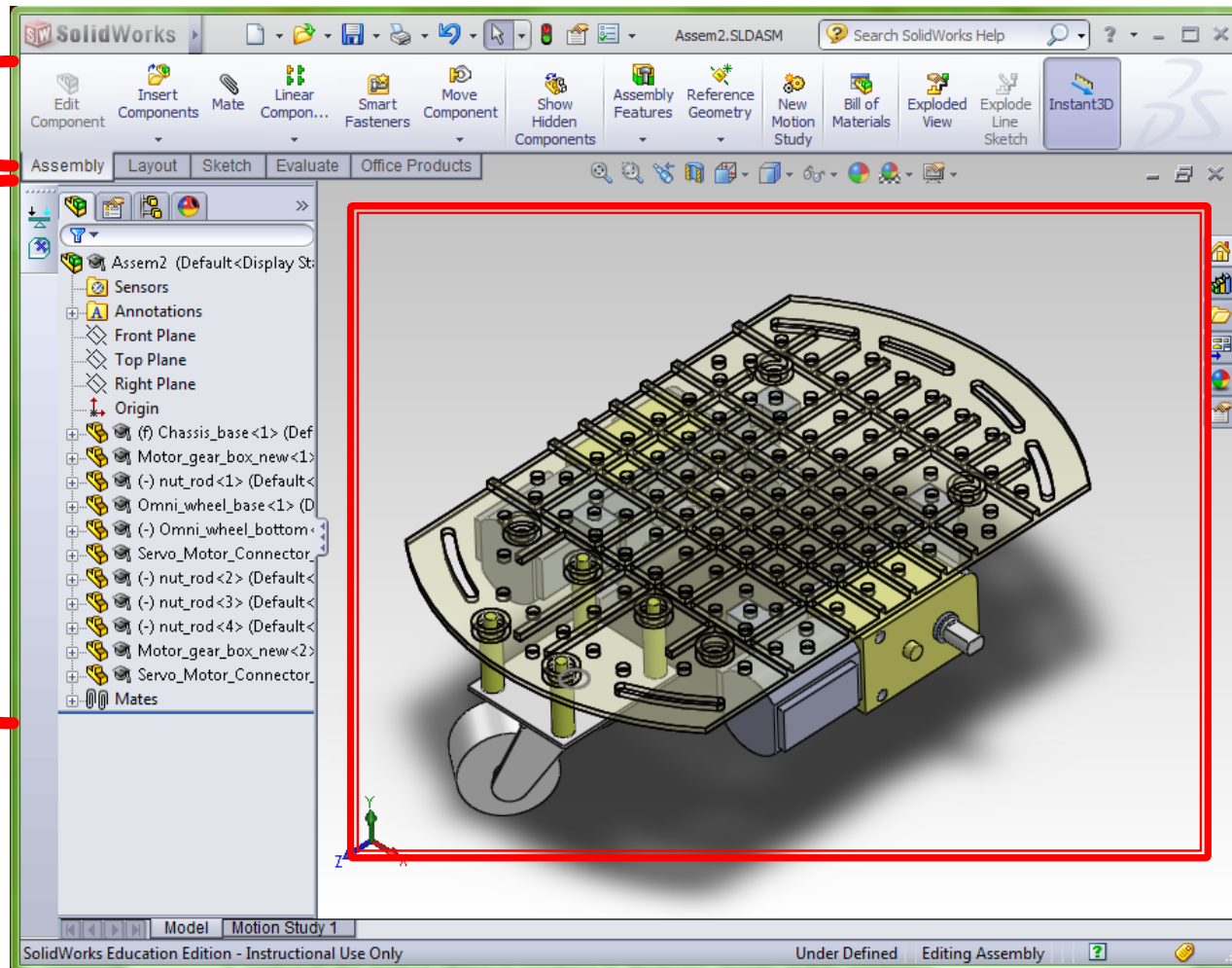


Create parts (File format -.sldprt)
Assembly parts (File format -.sldasm)
2D Engineering Draws (File format -.slddrw)

Basic Interface

Drawing tools

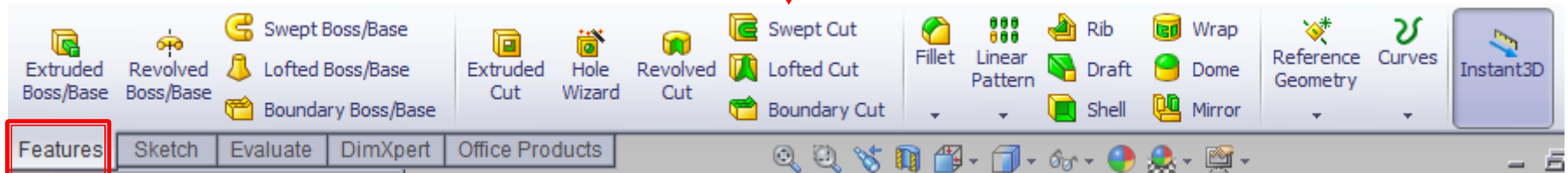
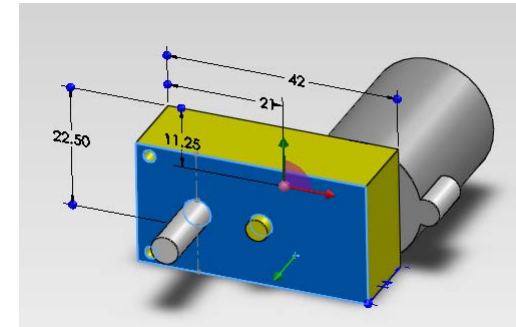
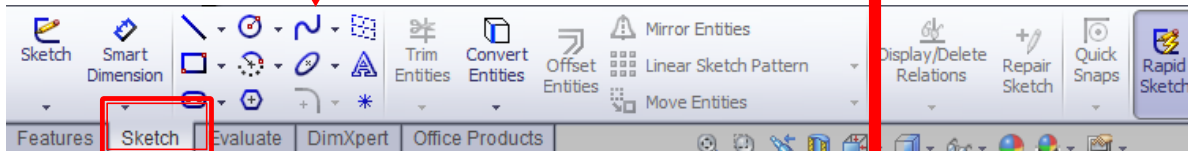
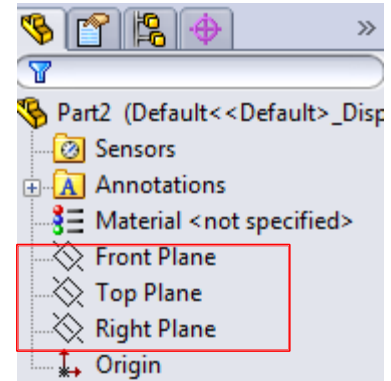
Design tree



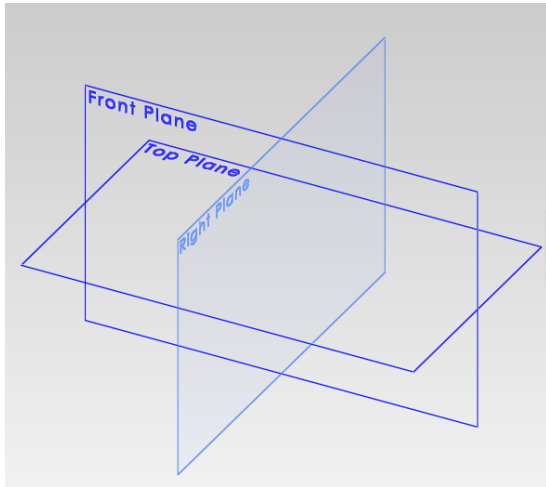
Drawing region

Basic procedures on sketching

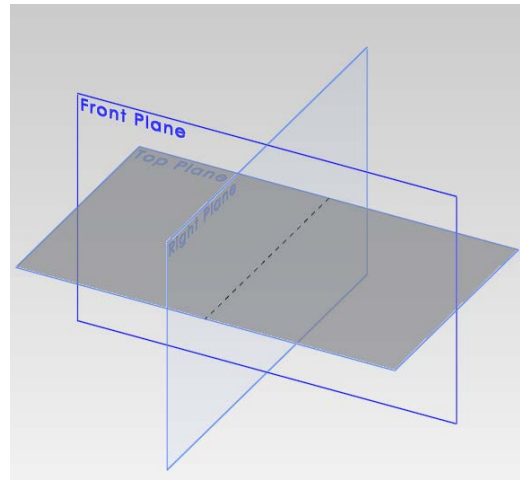
- Select plane for sketch
 - Invisible planes, planer surface on parts
- Create sketch(2-D)
- Draw features on sketch(3-D)



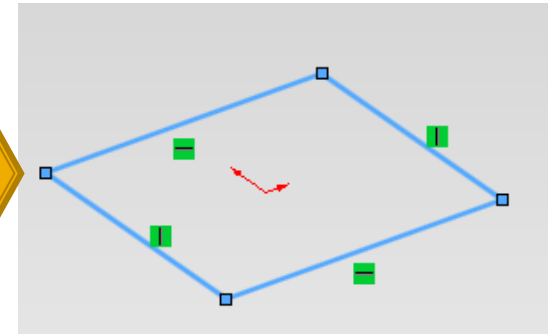
Basic steps



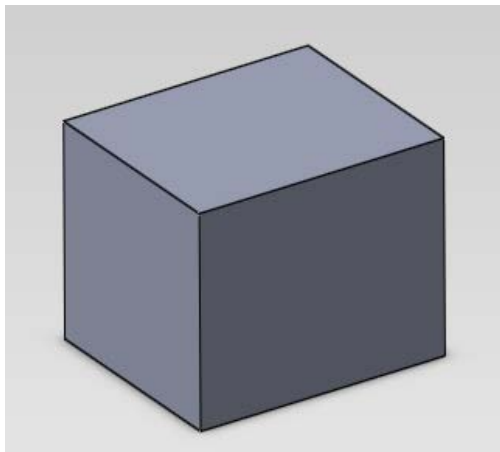
Nothing



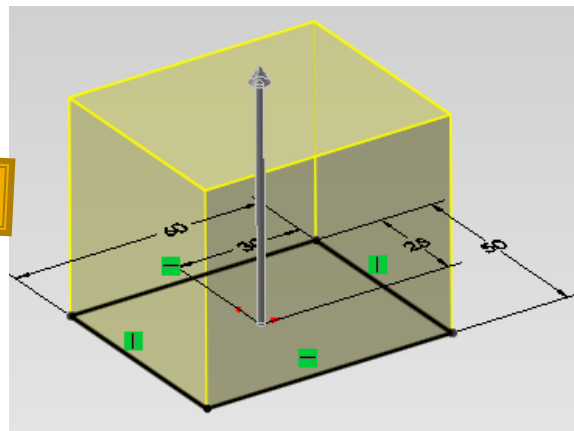
Select a plane



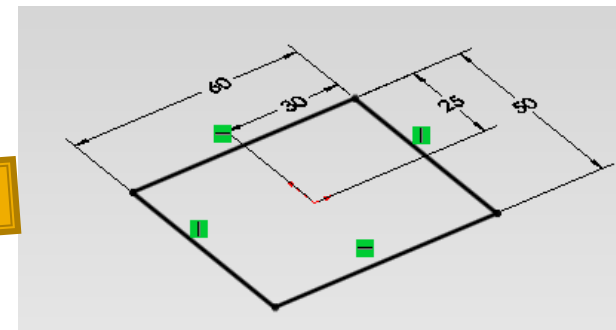
Sketch on the plane



Finish a feature

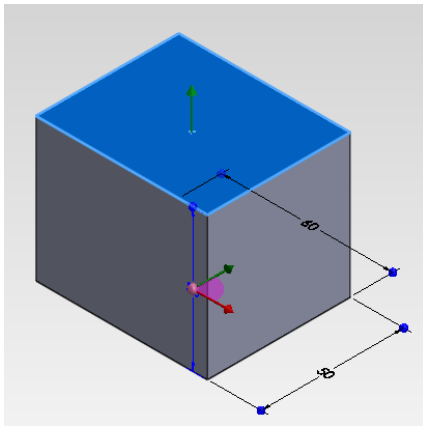


Create features (extrude)

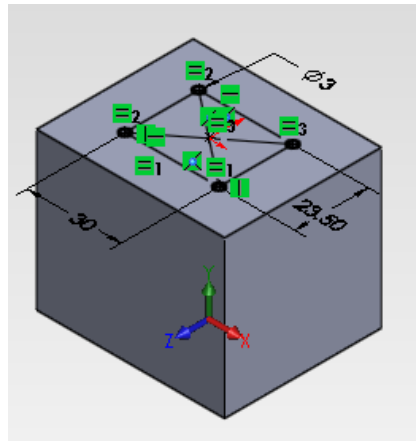


Dimensioning

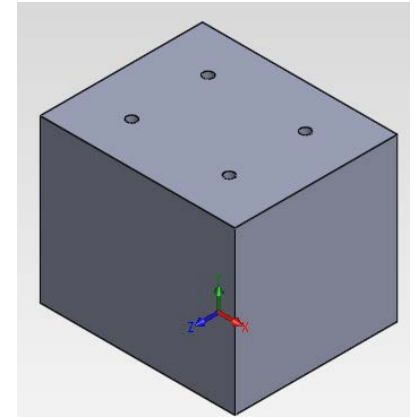
Basic steps [Con't]



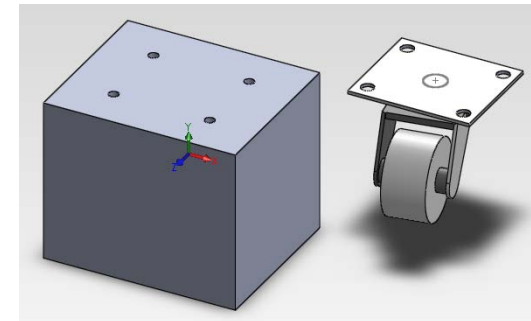
Select other planes



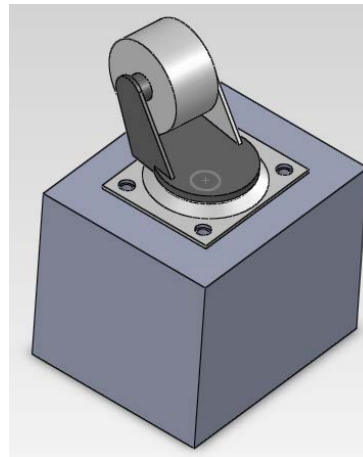
Create more sketches



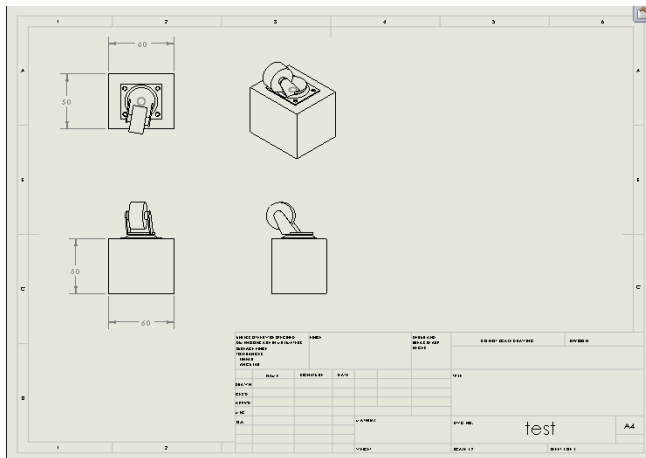
Create more features



Assemble with other parts



Assembly done,
or modify if necessary



Create engineering drawing

Control

- Standard View -Right click and hold

- (View>Toolbars>Standard View)



- Shading and background




- Others

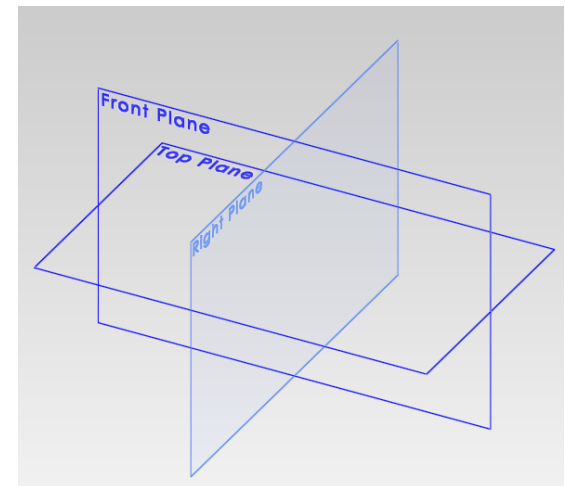
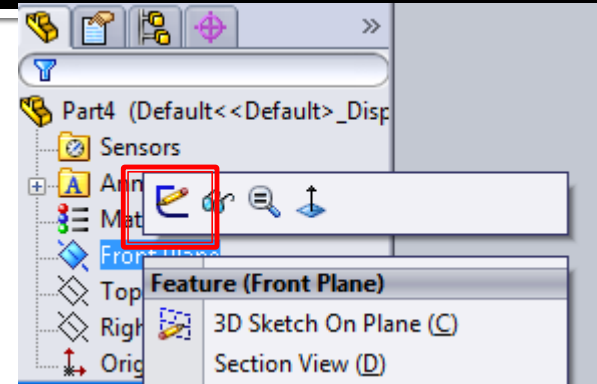
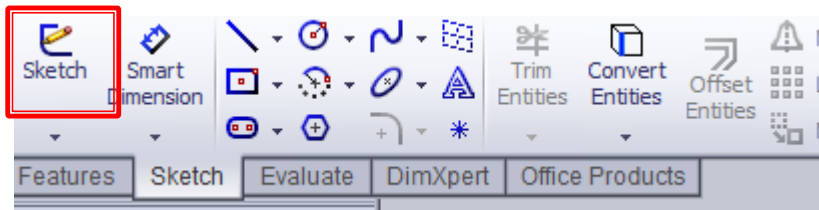
- Rotating –Middle click and hold
 - Panning –Ctrl + Middle click and hold
 - Zooming –Scroll Wheel
 - Fast views –Ctrl + 1,2,3,4,5,6,7



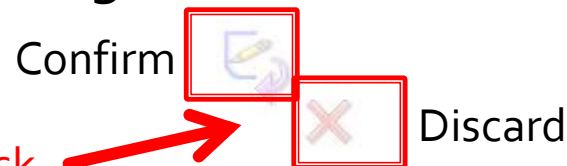
- *Many others.....(this tutorial covers only the basics)*

2-D sketch

- Select a plane and right click.
- In dialog box, click “sketch” 
- Or click “sketch” and select a plane



- Exit the sketch before you move on.
(at the top of the right side)



Debug: if you stuck
somewhere, check this!!

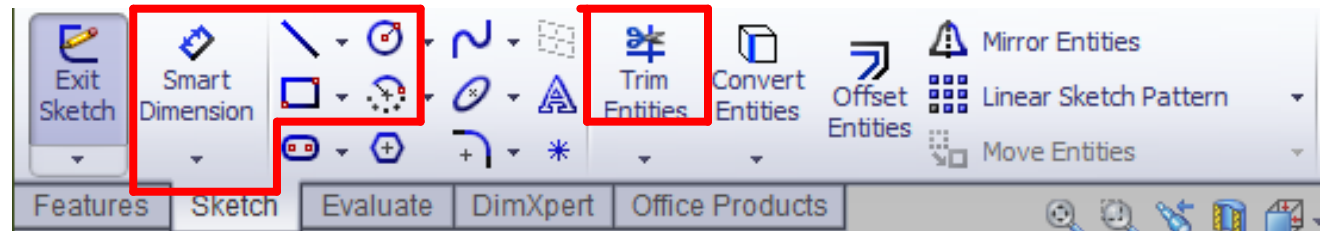
2-D

- Basic sketching

- Lines
- Circles
- Rectangles
- Arcs

- Modifications

- Dimensions (many types)
- Fillets/Chamfers
- Mirrors
- Patterns
 - Linear
 - Circular
- Trims



2-D lines



- Enter / Create Sketch
- Click line 

button

As sketch: click to define arbitrary first point, second point,...etc

Horizontal: click to draw horizontal line only

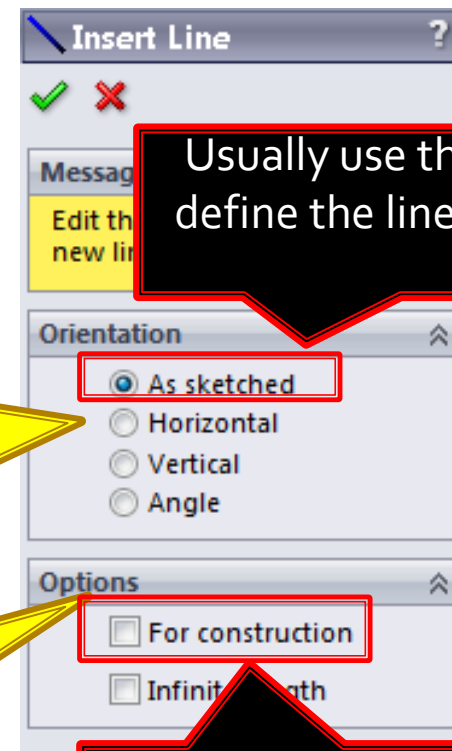
Vertical: click to draw vertical line only

Angle: click to draw line by defining angle with respect to horizontal line

For construction: can change the elements for reference

Infinite length: can draw line with infinite length

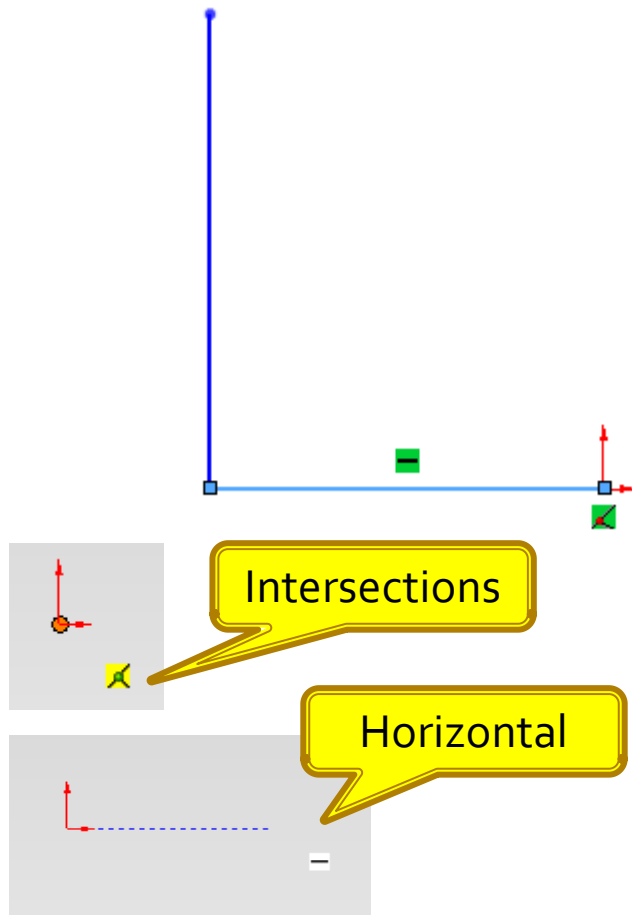
- ESC : leave the current command
- Delete elements: select the elements and click delete button



Usually use this and further define the line by dimension

Sometimes useful for later drawings

2-D lines



Line Properties ?

✓

Existing Relations ^

Horizontal13

Fully Defined

Add Relations ^

Horizontal

Vertical

Fix

Options ^

☐ For construction

☐ Infinite length

Parameters ^

59.32704311

180.00°

Additional Parameters v

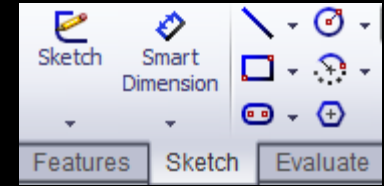
Tell you its existing relation/the relation with other elements

You can add relations here

Modify options (normally you can set "for construction" after all necessary lines are drawn)

Define parameter of current element. E.g. length of line

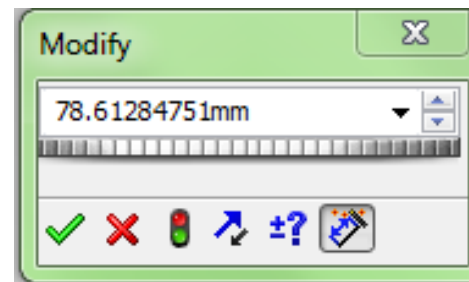
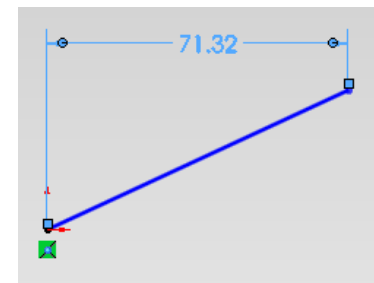
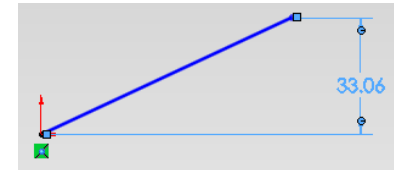
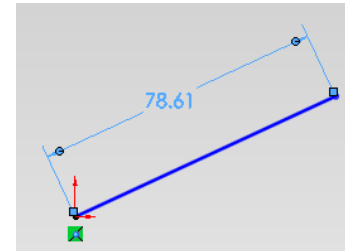
2-D lines (dimensions)



■ Click Smart Dimension

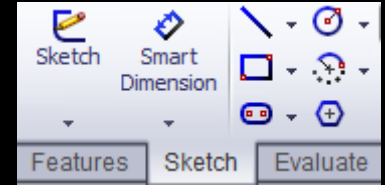



- Click on the line: define the line by length only
 - Pull the dimension to different directions
 - Along line
 - Vertical
 - horizontal
 - Then type in the value to modify the line
 - Click 'tick' when done

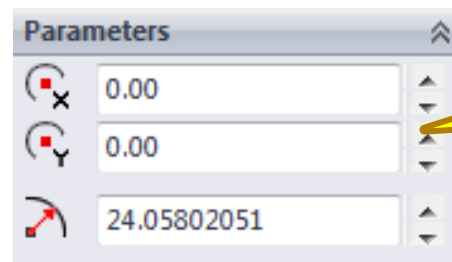
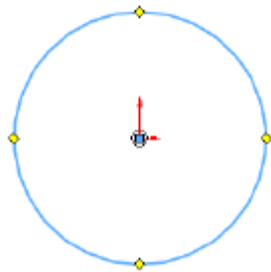


- Click on 2 points: define the line by 2 end points (similar)


2-D circles

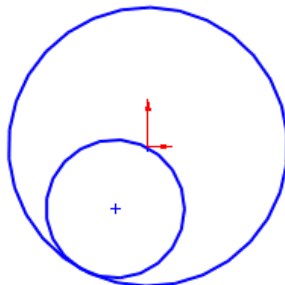


- Select circle 
 - Click a point(center), then click another point(radius)



Radius and
Center

- Select Perimeter circle 
 - Select three points to define a circle

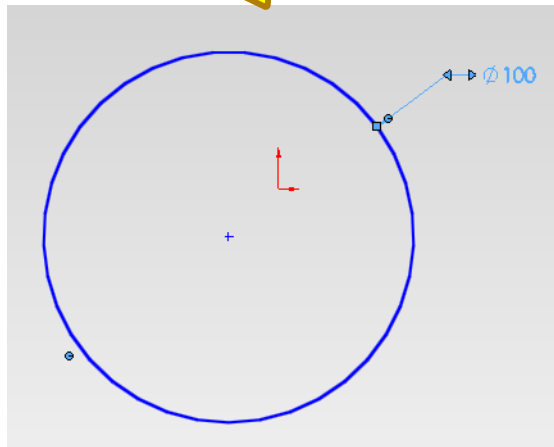


Useful when you want
to draw tangent circle

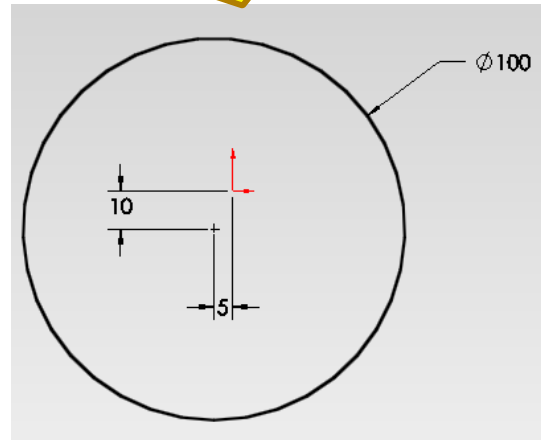
2-D circles (dimensions)

- Usually it is defined by its center and radius

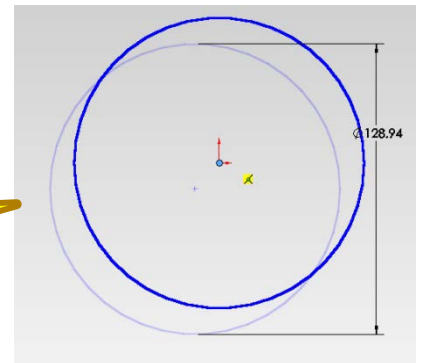
Not yet fully defined (**blue**)



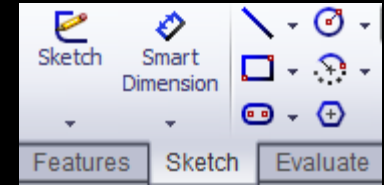
Fully defined (**Black**)




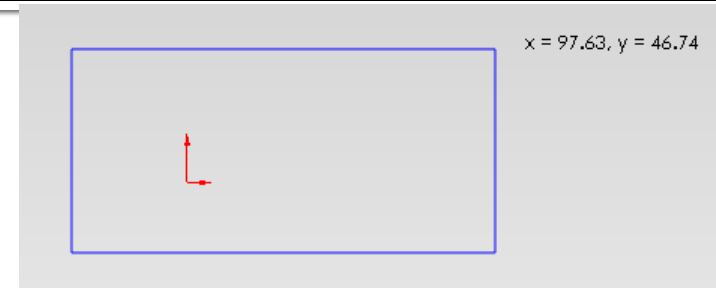
Or use the mouse to force set the relations of the center with some reference (e.g. the origin)



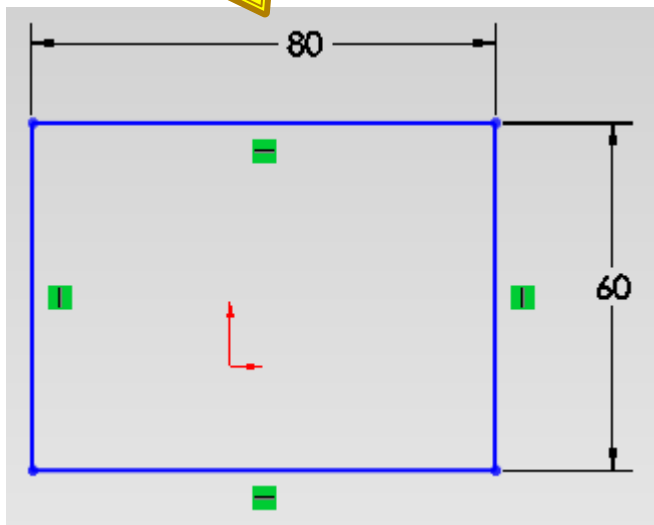
2-D rectangles



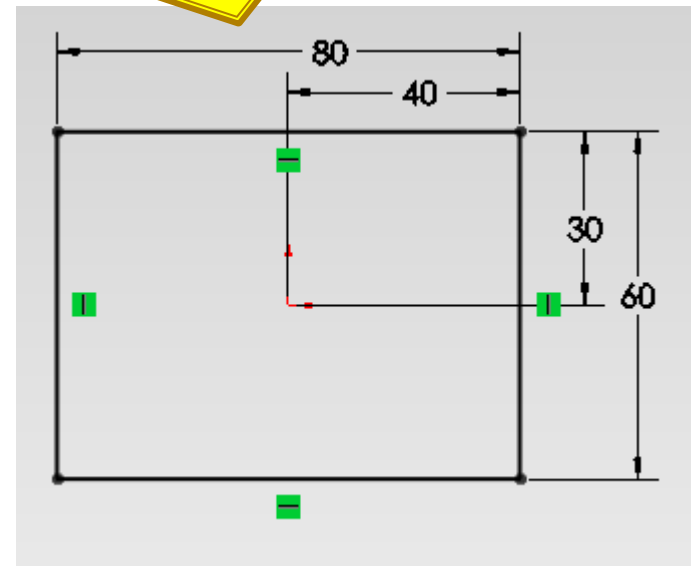
- Select rectangle 
- Click 2 more points to define it
- Usually it is defined by 2 sides



Not yet fully defined (blue)

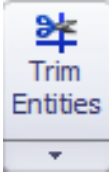


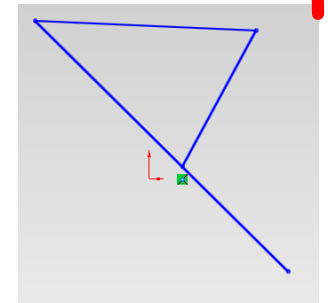
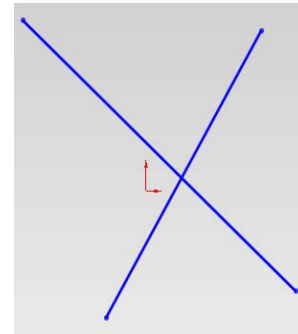
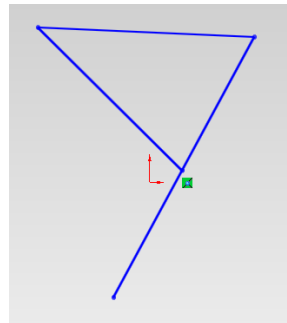
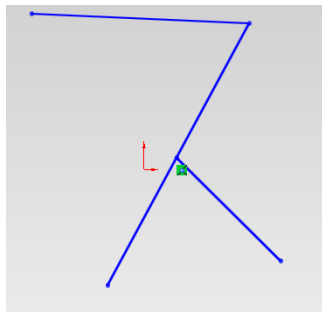
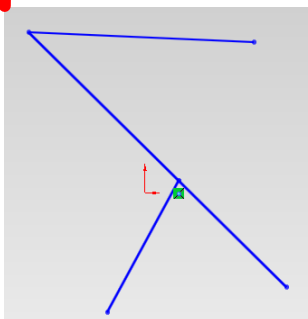
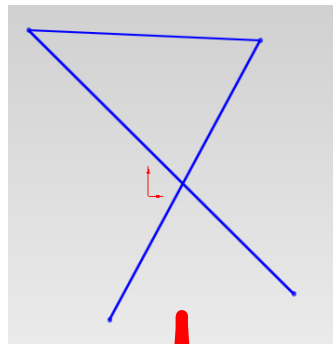
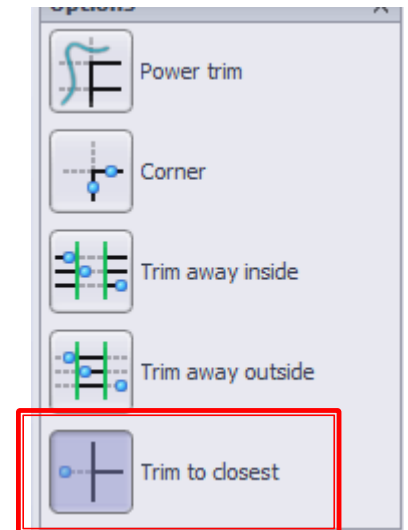
Fully defined (Black)



2-D Trim (cutting lines)



- Click trim  and select "trim to closest"
- Trim the line between 2 (intersecting) points

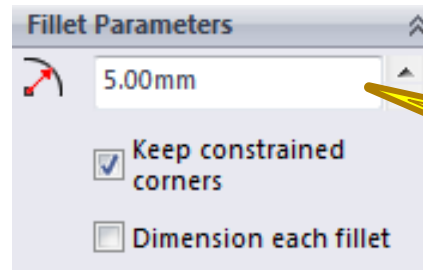
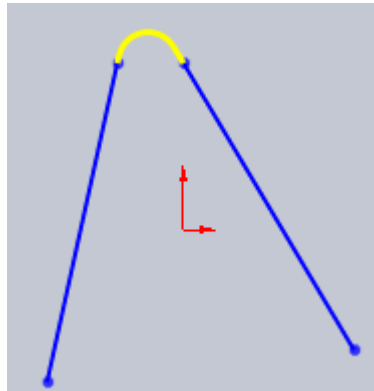


2-D fillets / chamfers

- Click fillet

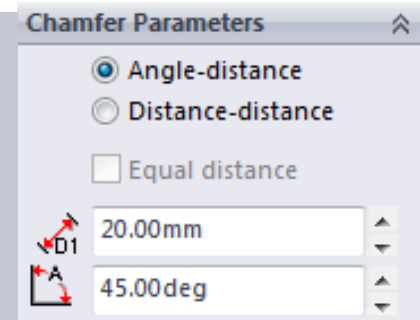
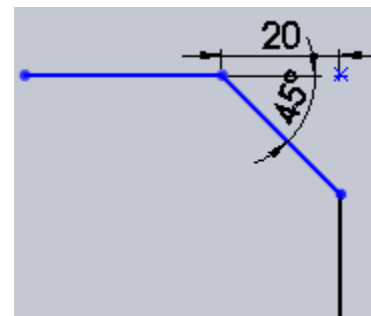
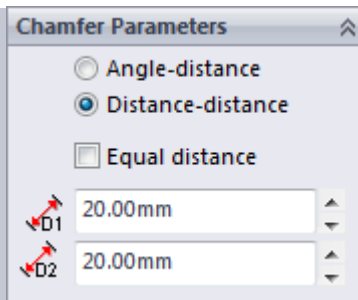
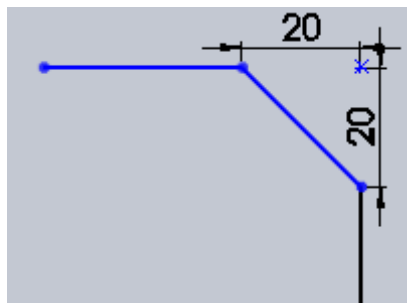


- Fillet the corner at the intersection of two sketch (choose a corner)



Set corner radius

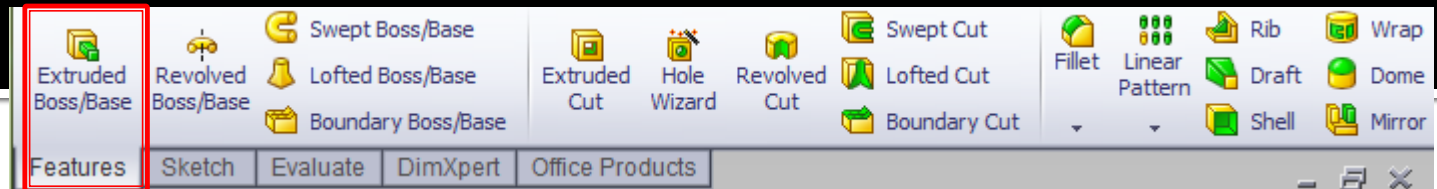
- Click chamfer



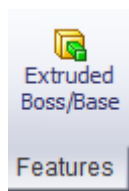
3-D

- Extrude / extrude cut
- Revolve / revolve cut
- Fillets / chamfers
- Mirror
- Patterns
 - Linear
 - Circular
- Many others
 - Loft base
 - Sweep base
 - Etc.

3-D extrudes

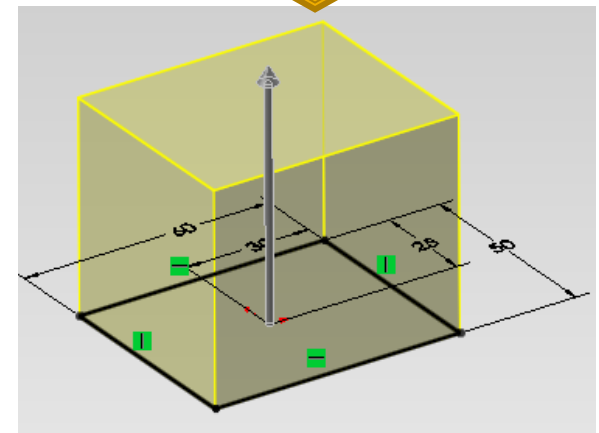
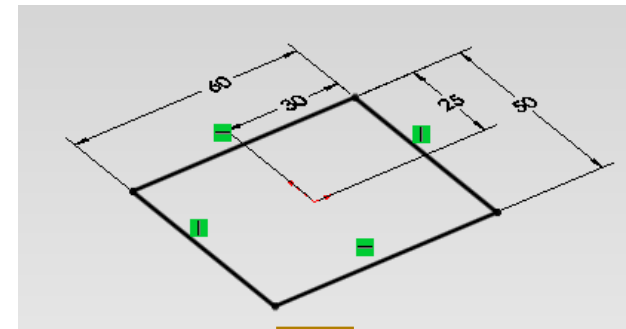
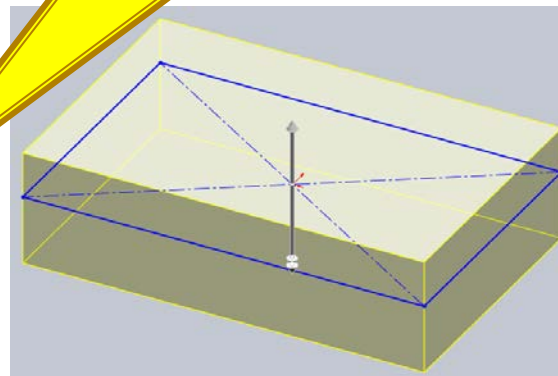
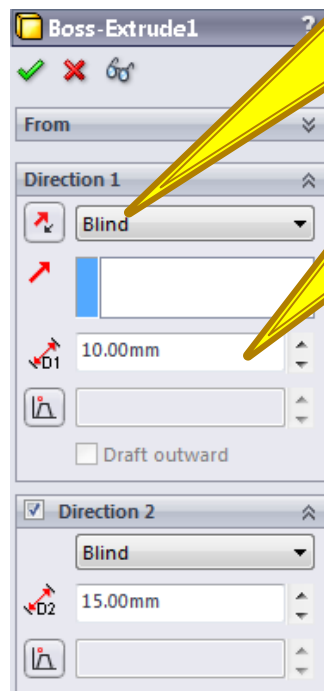


- Select a sketch (**must be close-loop**)
- Click



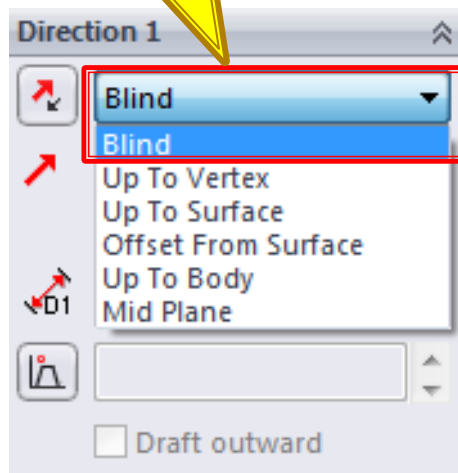
Extrude type

Extrude thickness



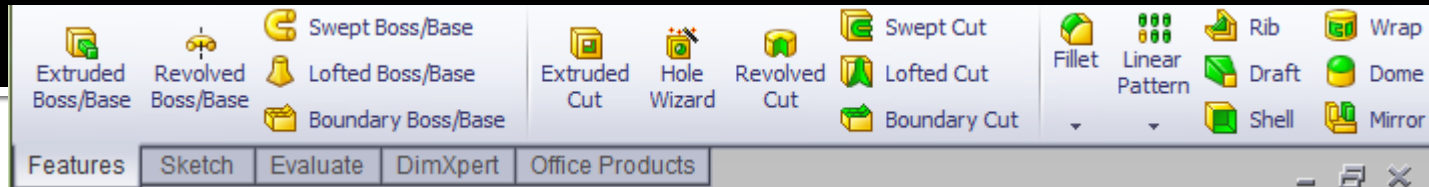
3-D extrudes

Frequently used

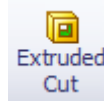


Blind	Set the Depth
Through all	Extends the feature from the sketch plane through all existing geometry.
Up to Vertex	Select a vertex in the graphics area for Vertex
Up to Surface	Select a face or plane to extend to in the graphics area for Face/Plane . Double-click a surface to change the End Condition to Up to Surface, with the selected surface as the termination surface. If the sketch that you extrude extends outside of the selected face or surface body, Up To Surface can do some automatic extension of one analytic face to terminate the extrusion.
Offset from surface	Select a face or plane in the graphics area for Face/Plane , and enter the Offset Distance . Select Translate surface to make the end of the extrusion a translation of the reference surface, rather than a true offset. If necessary, select Reverse offset to offset in the opposite direction.
Up to body	Select the body to extrude to in the graphics area for Solid/Surface Body . You can use Up To Body when making extrusions in an assembly to extend the sketch up to the selected body. Up To Body is also useful with mold parts, if the body you extrude to has an uneven surface.
Mid plane	Set the Depth

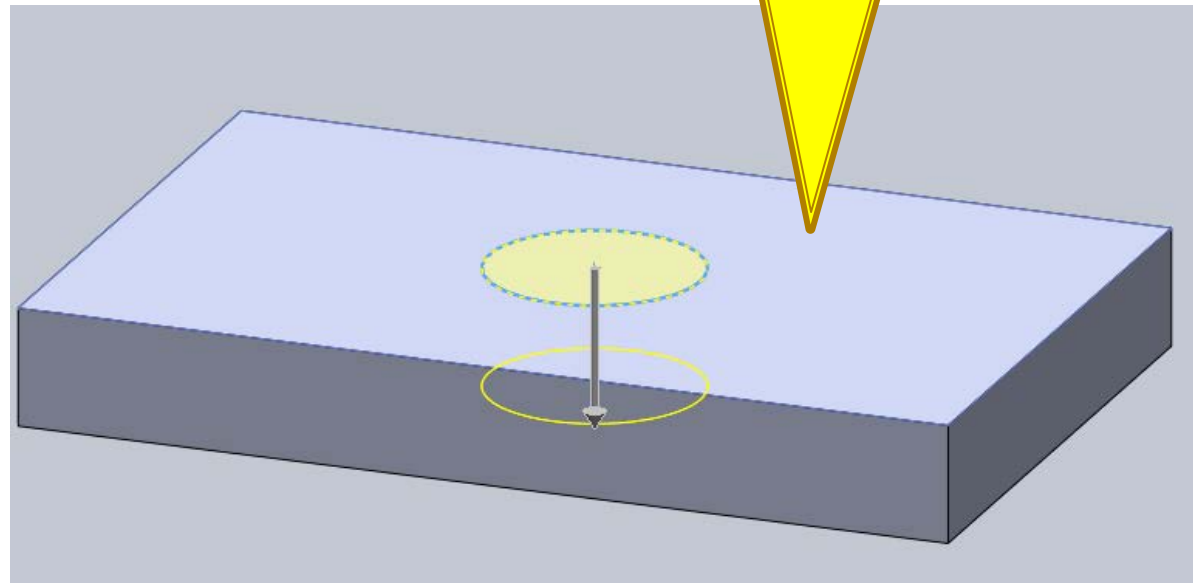
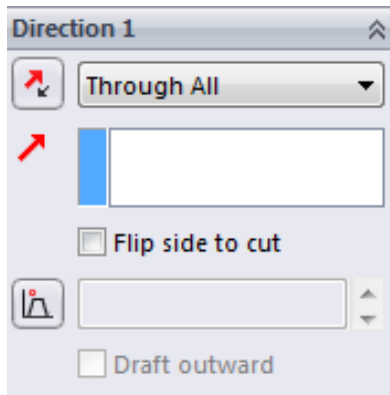
3-D extrude-cut



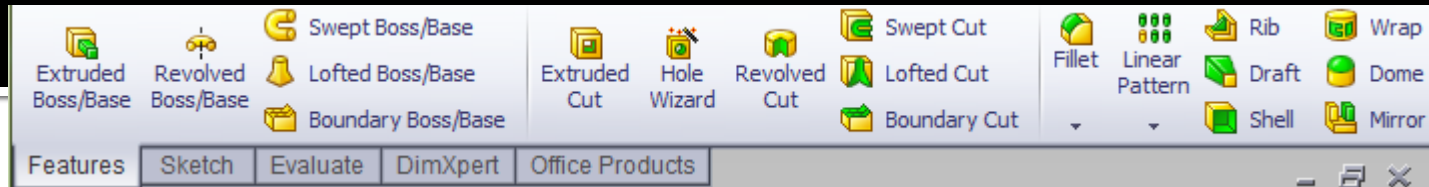
- Select a plane on feature
- Sketch on that plane
- Click Features > Extruded Cut



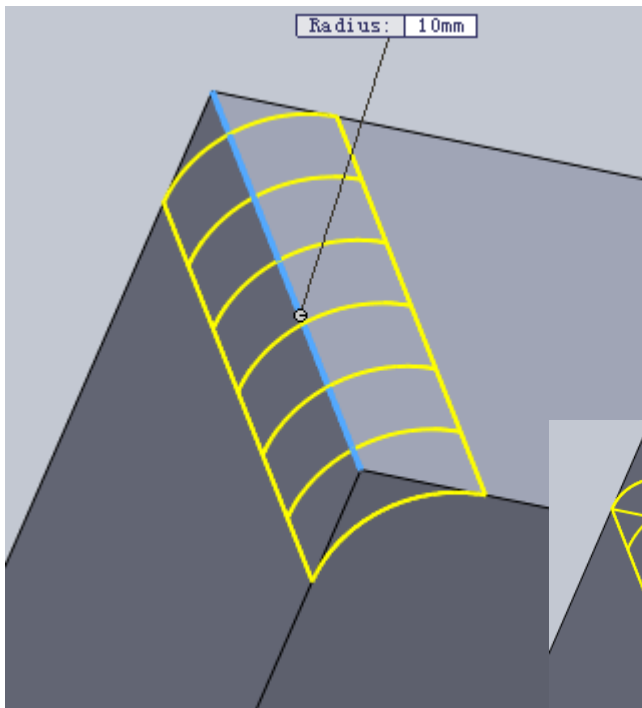
Select this plane and sketch first!!!



3-D fillets

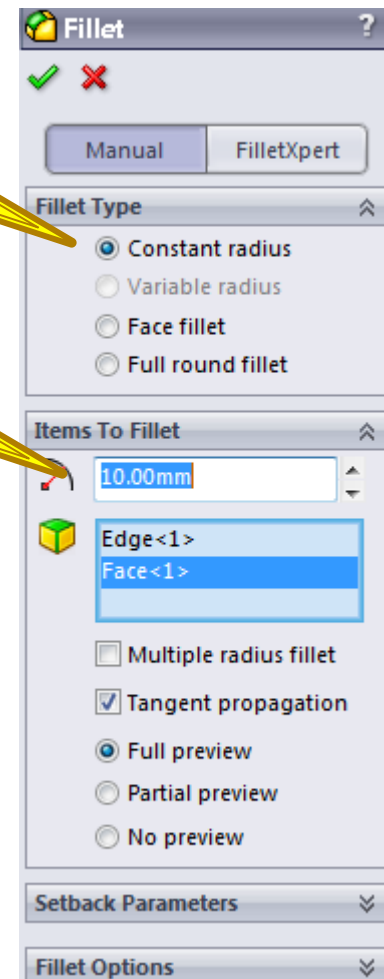
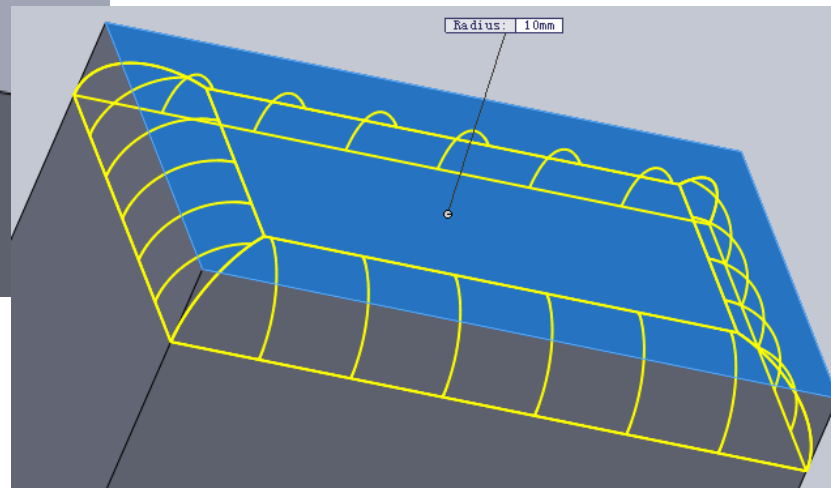


- Select edge/face of feature and click

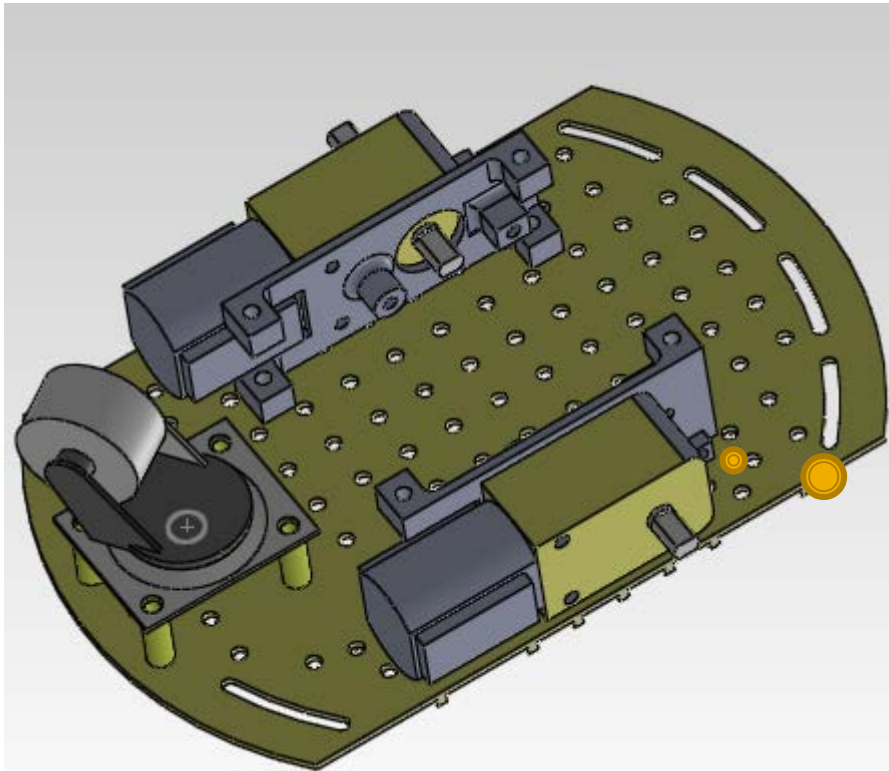


Constant radius

Radius



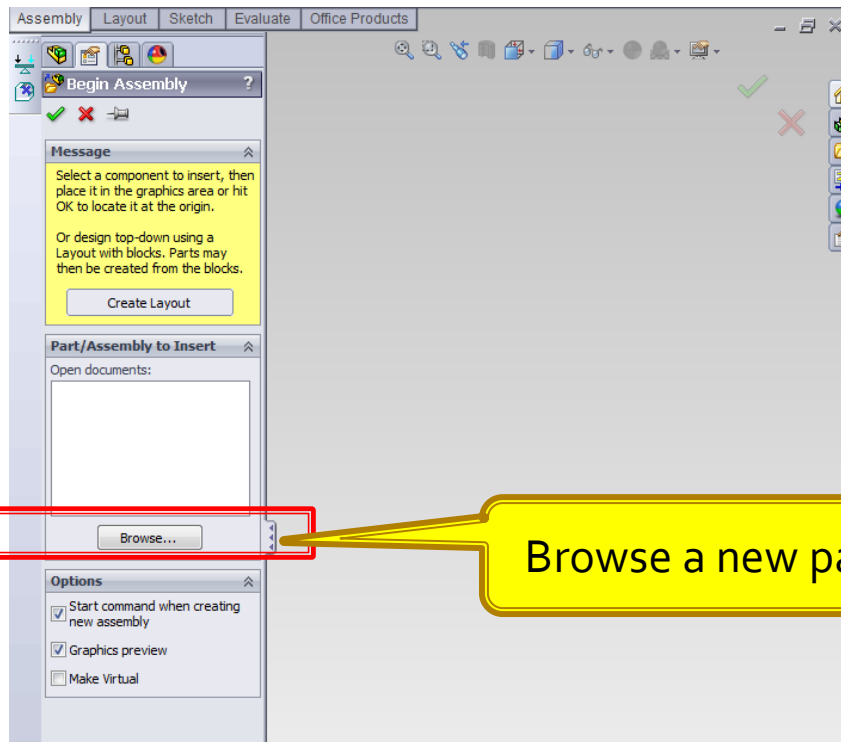
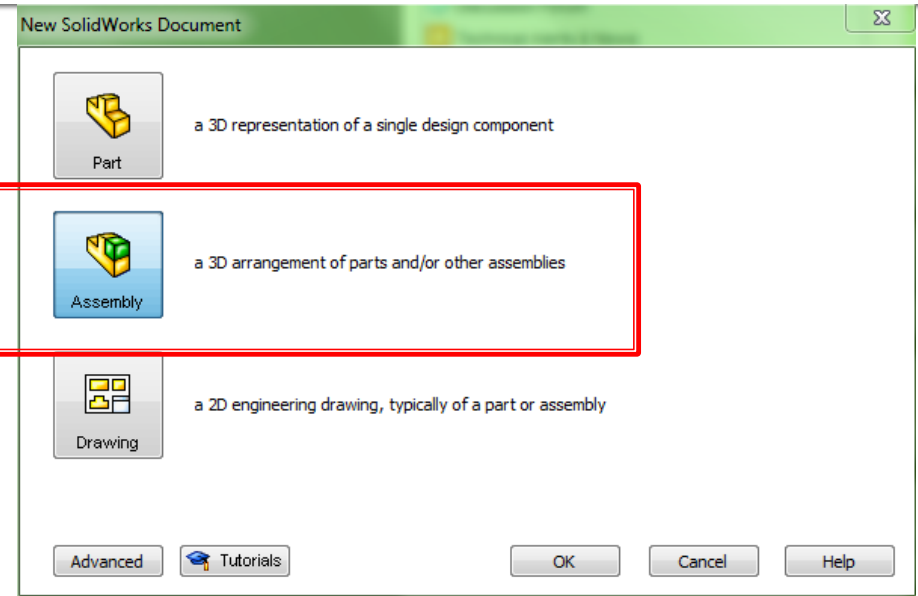
Assembly



Assemble virtually
before
manufacturing!

Assembly

- Open new assembly
- Insert new component

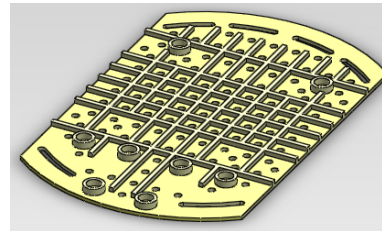


Assembly

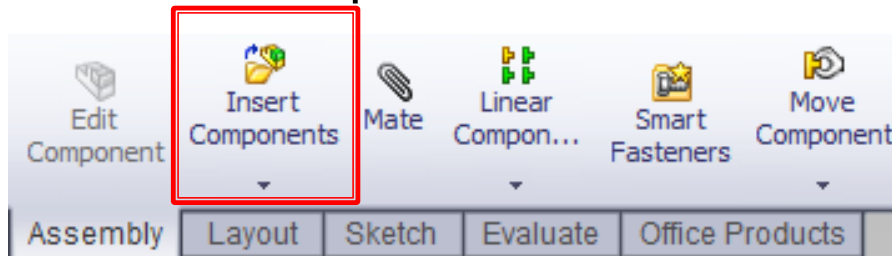
- Open the first (fixed) component

Chassis_base.SLDPRT	20/6/2012 16:24	SolidWorks Part Document	1,157
Motor_gear_box_new.SLDPRT	10/6/2012 17:58	SolidWorks Part Document	341
nut_rod.SLDPRT	4/7/2012 15:09	SolidWorks Part Document	211
Omni_wheel_base.SLDPRT	10/5/2012 11:13	SolidWorks Part Document	210
Servo_Motor_Connector_new.SLDPRT	21/6/2012 10:30	SolidWorks Part Document	390

- Click to place the component

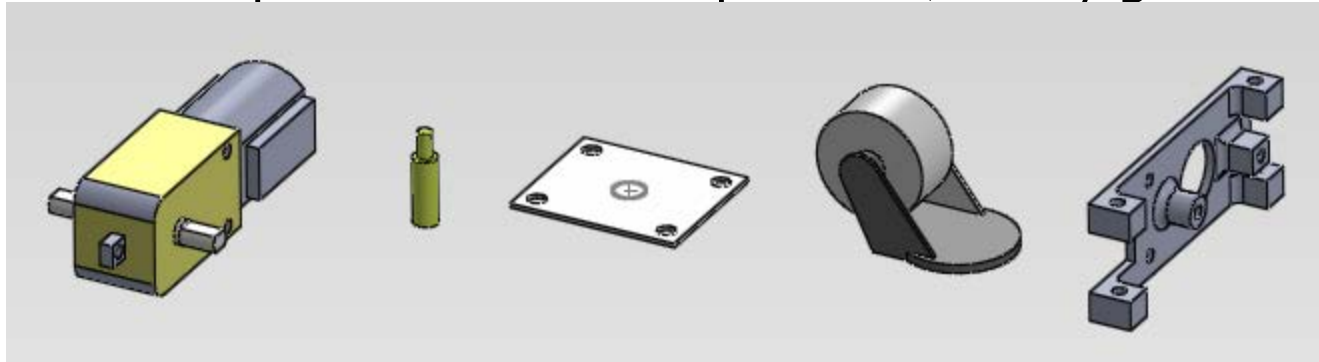


- In the “**Assembly**” tab, click “**Insert Components**” and browse more components

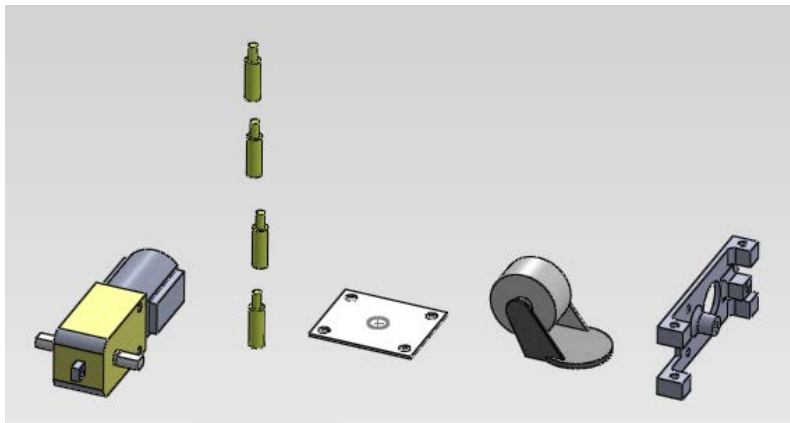


Assembly

- Take one part for each component (totally 5 more parts)

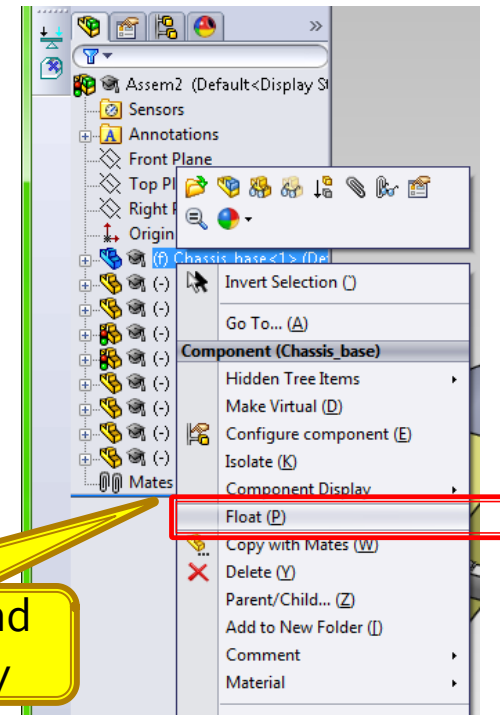
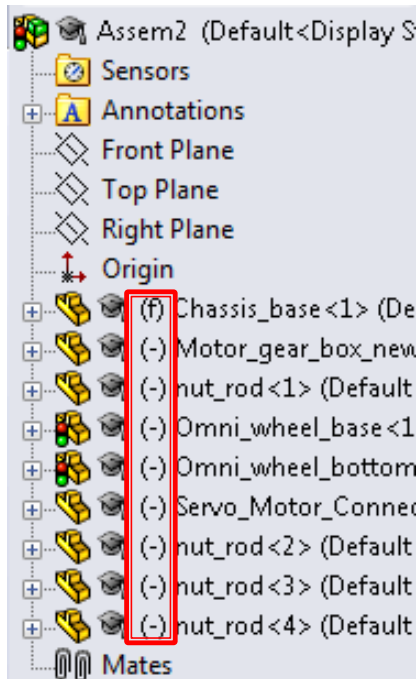


- Duplicate parts: Press Ctrl + mouse (left) key on the motor (yellow part), pull away from it.



Assembly

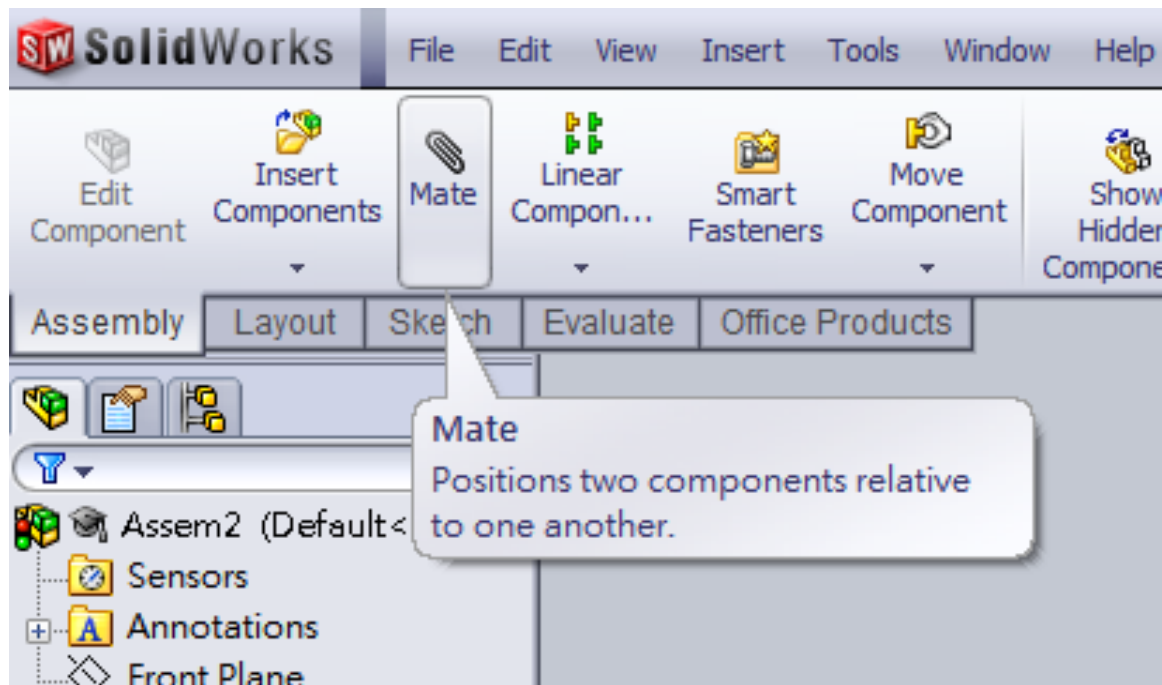
- **The prefixes:**
- **(f) Fixed:** position of this component is fixed, i.e., it CANNOT be moved / rotated.
- **(-) Float:** position of this component has NOT been defined, i.e., it can be moved / rotated freely.



You can right-click it, and set to float if necessary

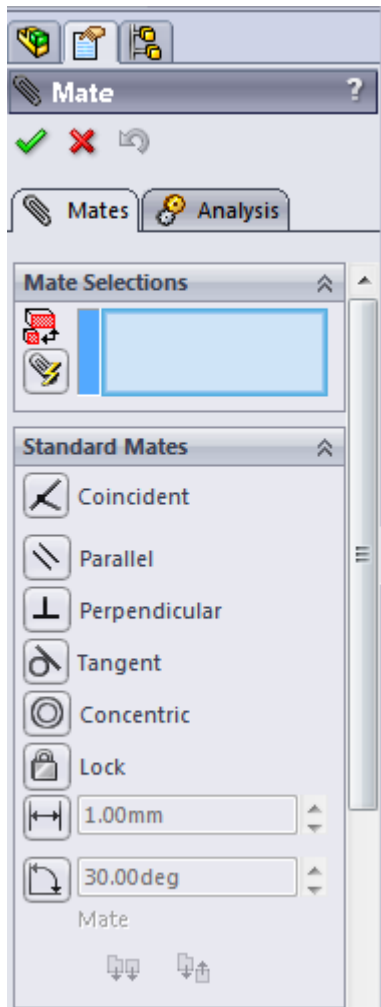
Assembly

- Adding relations, click Mate



Assembly

Most commonly used!



- **Standard Mates**

- **Coincident**

- configure two items(faces, edges or planes) so that they touch each other.

- **Parallel**

- configure two items so that they remain a specified distance from one another.

- **Perpendicular**

- configure two items so that they're 90 degrees from one another.

- **Tangent:**

- configure a curve and another item so that they meet at a single point. One of the selected items must be a cylindrical, conical or spherical face.

- **Concentric:**

- configure two items so that they share the same center axis.

- **Distance:**

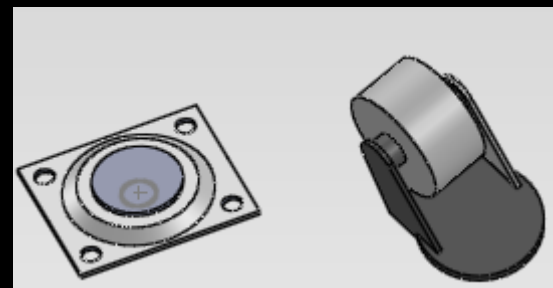
- configure two items at a specified distance from one another.

- **Angle:**

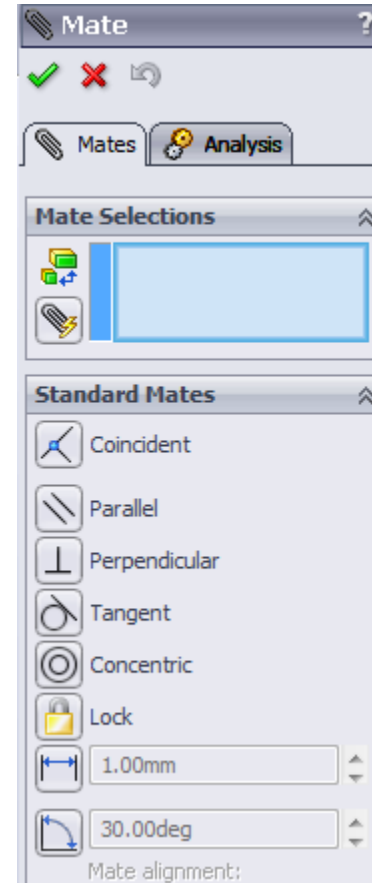
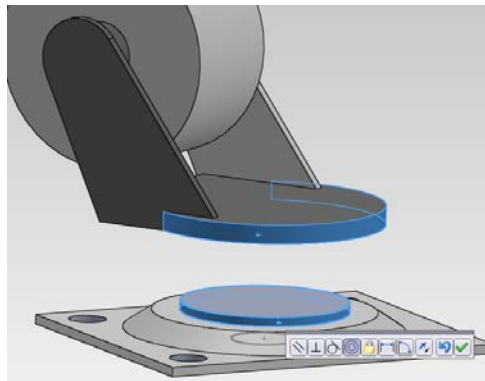
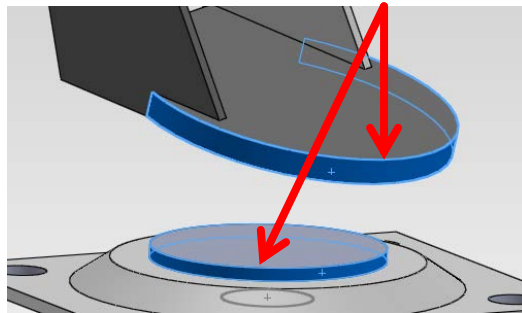
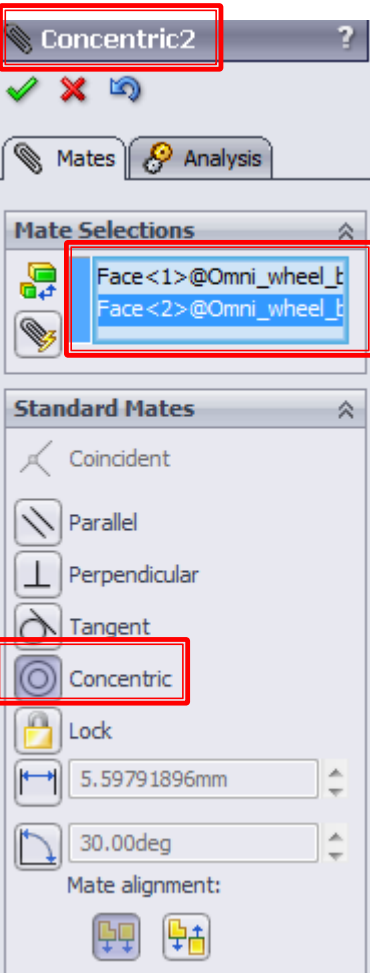
- configure two items at a specified angle to one another.

Mate can only handle 2 features each time!!!

Assembly (omni-wheel)

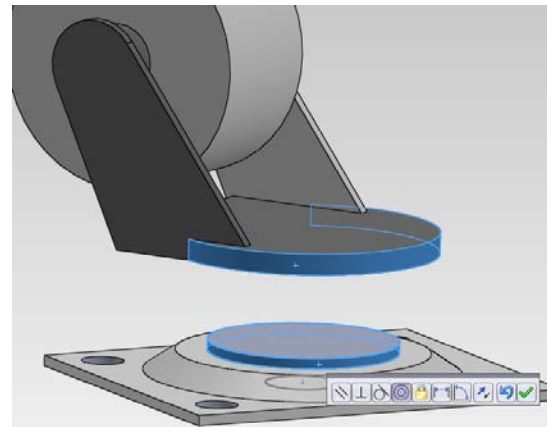
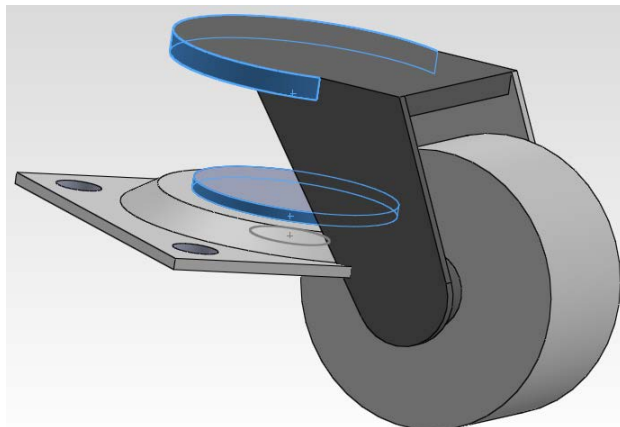
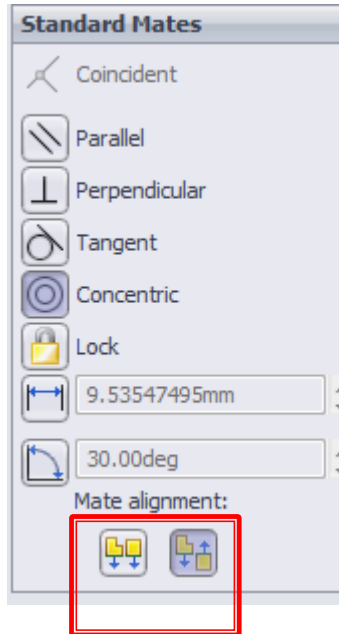


- Click "Mate", select 2 faces



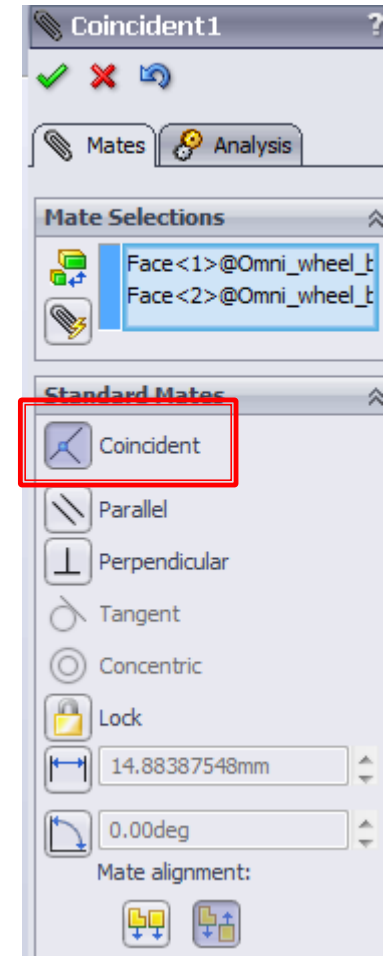
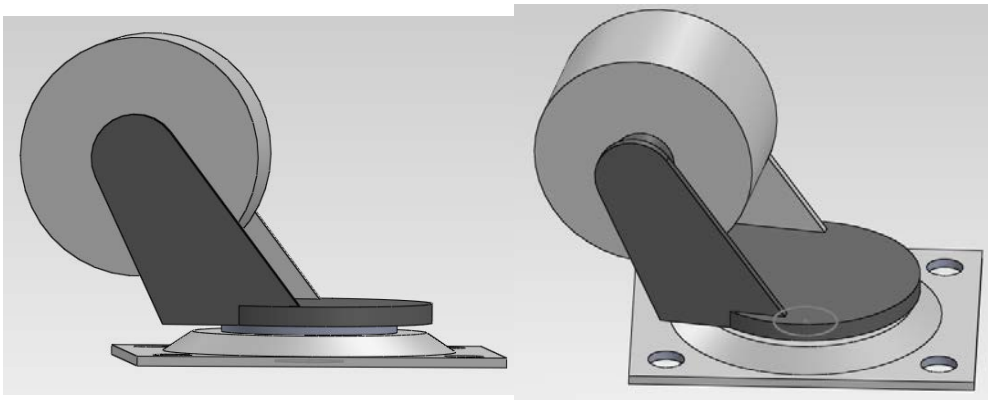
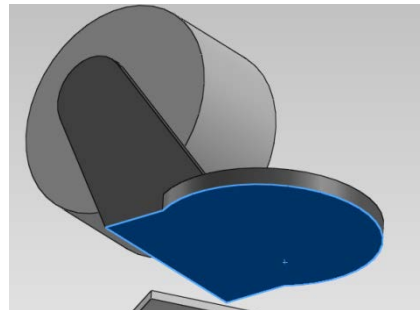
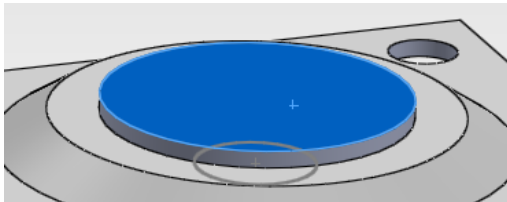
Assembly (omni-wheel)

- Reverse mating directions

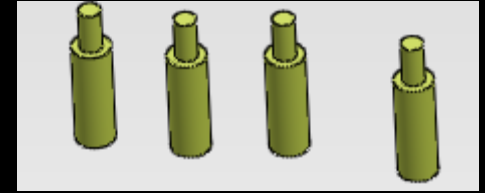


Assembly (omni-wheel)

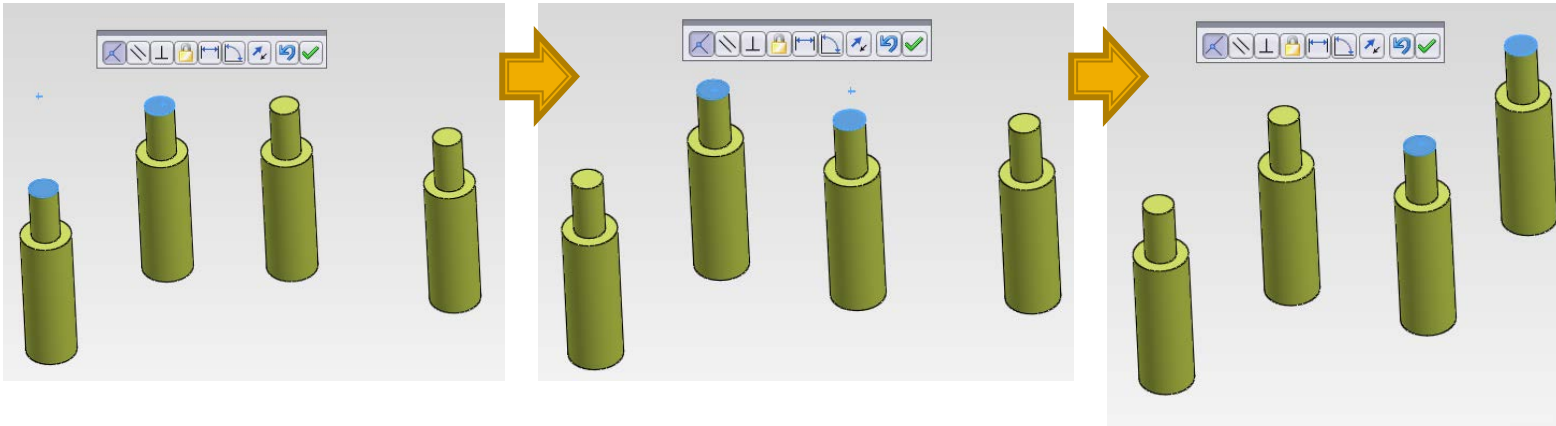
- Mate (coincident), click 2 surfaces, then "Mate", finally click "tick"



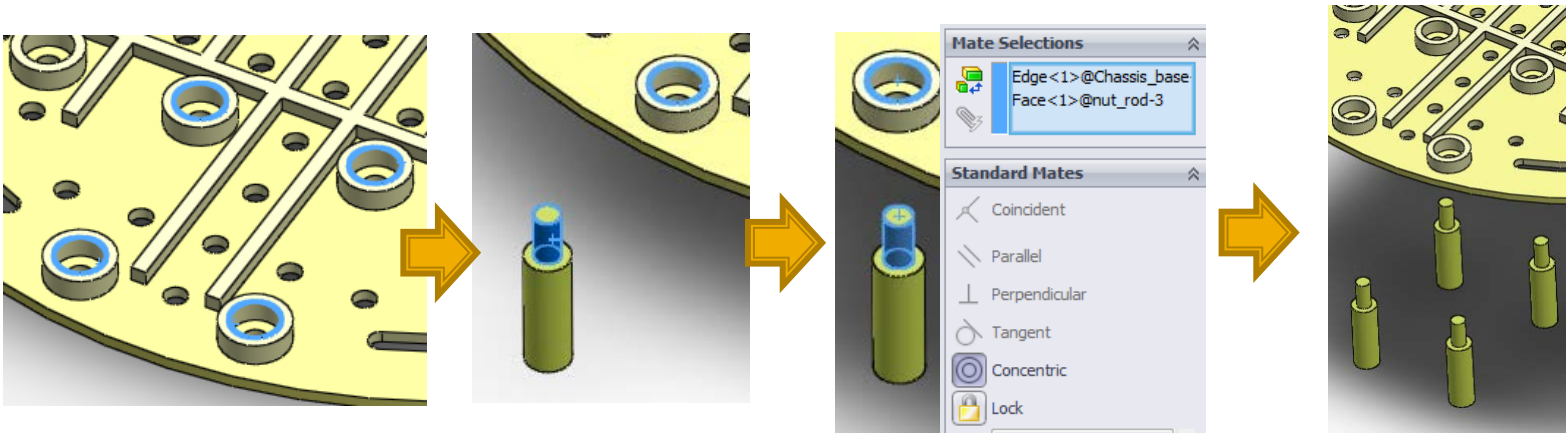
Assembly (screw rods)



- Mate (**coincident**) with all 4 surfaces,

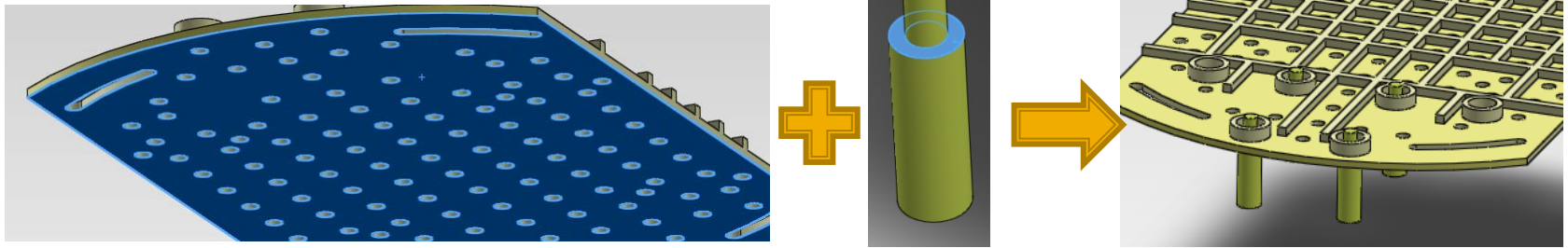


- Mate (**concentric**) the rods and the holes on the chassis,

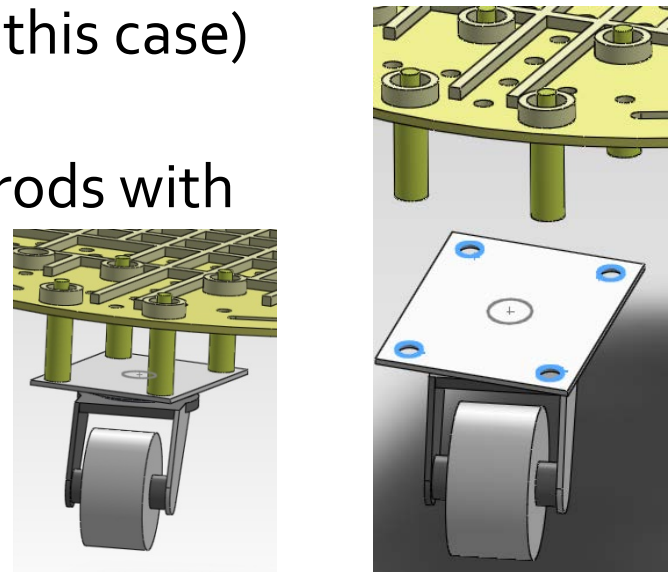


Assembly (screw rods)

- Mate (**coincident**) with 2 surfaces,

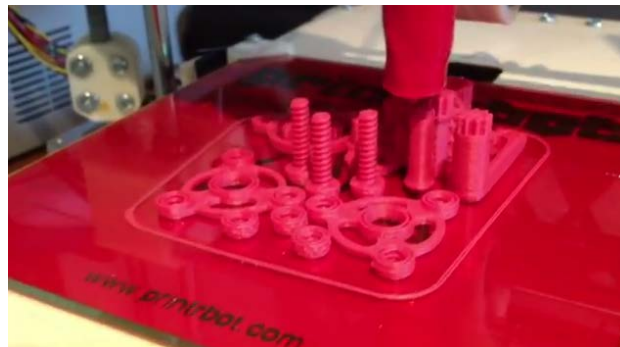



- Mate (**concentric**) the holes on the “omni-wheel” with the 4 rods, (either 2 holes are enough in this case)
- Mate (**coincident**) the head of the rods with the flat surface on the omni-wheel



Rapid Prototyping (RP)

- Flows of RP:
 - 1. Design some parts
 - 2. Save as “**part_name.stl**” format
 - 3. Send to RP machine for fabrication
 - 4. Post-treatment such as supporting materials removal
- Demonstration of the rapid prototyping process:
 - http://www.youtube.com/watch?feature=player_embedded&v=bpcwBQKUqK4





**Thank you!
Good luck with
your study!**