Chapter 5

In sweep command there is

a) Two sketch profiles

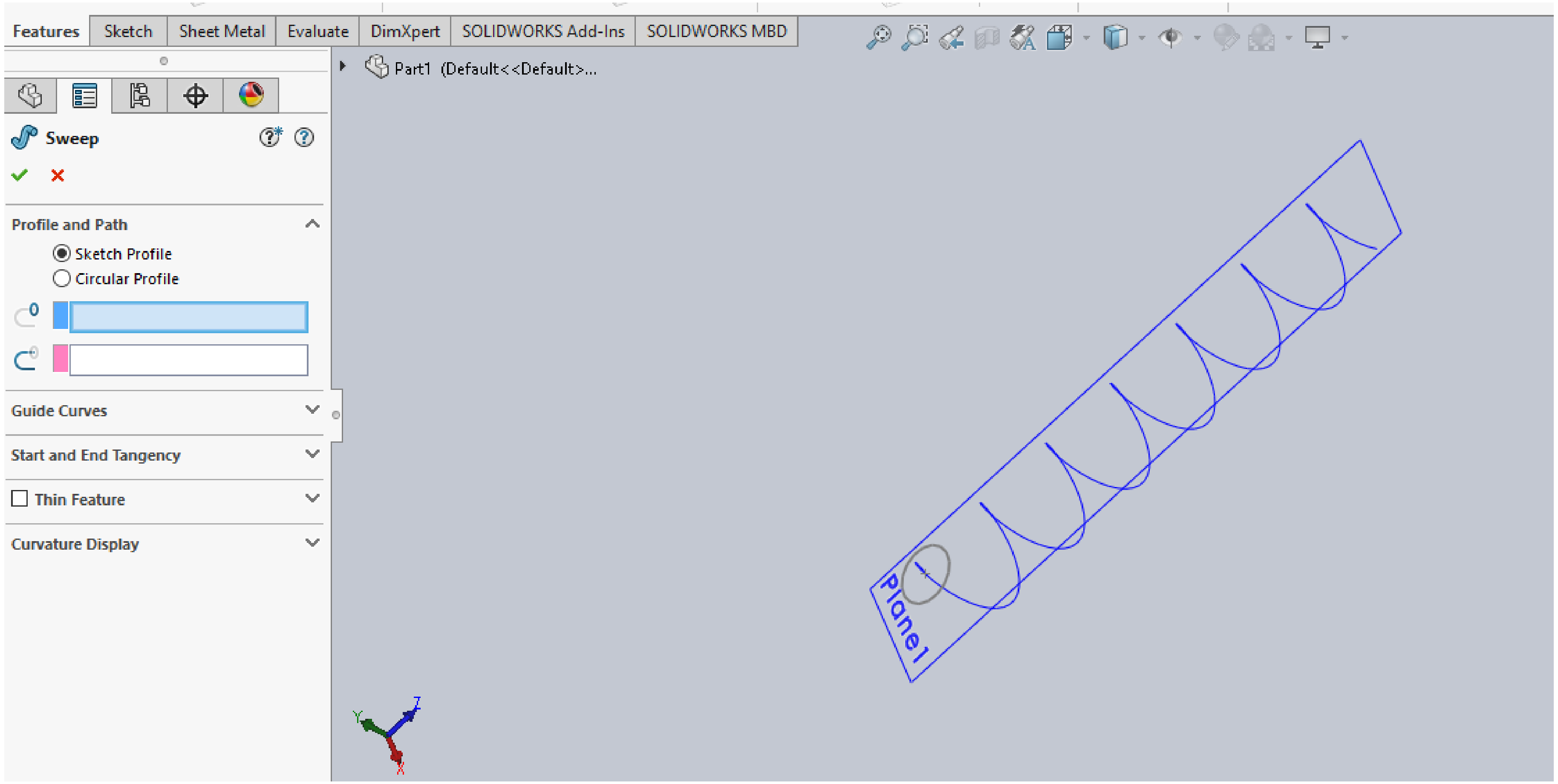
b) Two path

c) One sketch profile and one path

The sweep profile is used to create threads springs circular things and difficult geometry.

For sweep profile

- The sketch must be non intersecting that mean the two sketches(profile and path) are at different planes
- Its profile must be close


Features
Sketch
Sheet Metal
Evaluate
DimXpert
SOLIDWORKS Add-Ins
SOLIDWORKS MBD
Part1 (Default<<Default>...
Sweep
Profile and Path
Sketch Profile
Circular Profile
Guide Curves
Start and End Tangency
Thin Feature
Curvature Display
Plane1

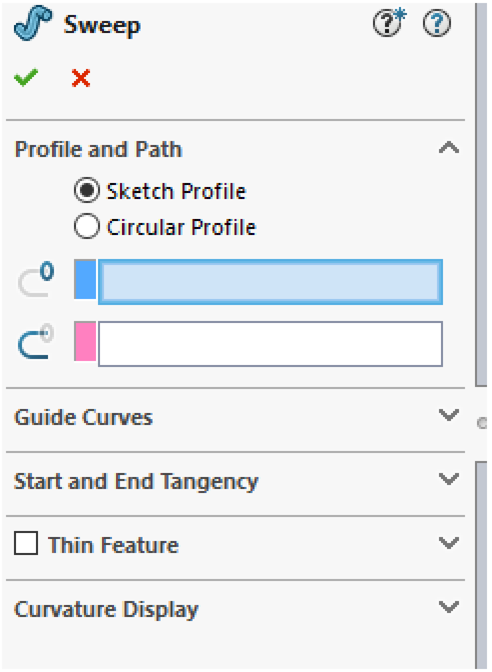
From the above fig.

After sketching the path and profile select the sweep command

First select the profile from property manager tree

And then select the path then 'ok'

The guide curve can also be used

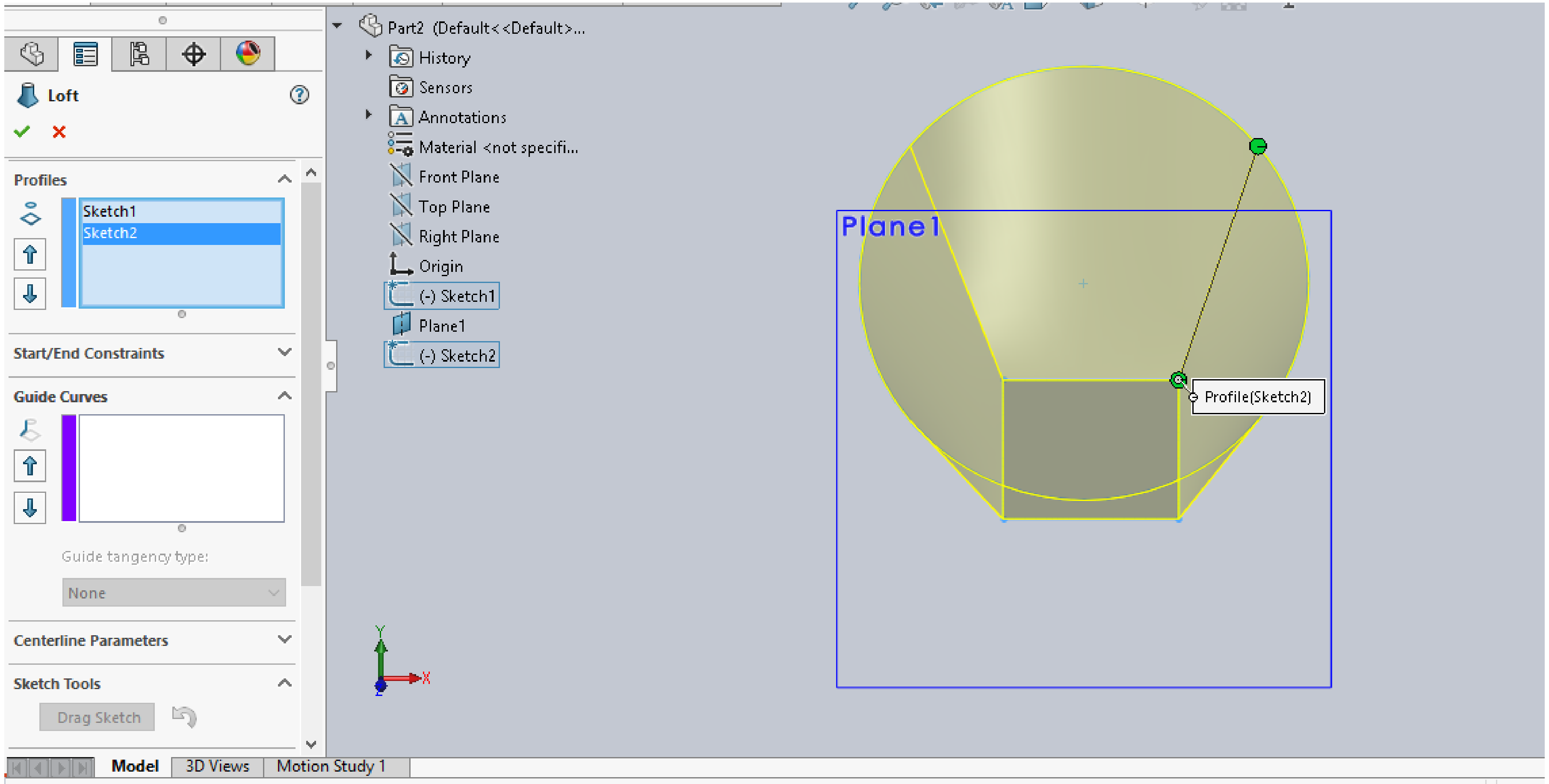


Loft:

The loft command is used between two sketches.(true,false)

- The loft feature is an important tool for surface modeling.
- The loft feature creates a shape by making transitions between multiple profiles and guide curves.
- The sketch profile can be either closed or open
- The guide curves are used to create complex shapes

- Draw first sketch
- Draw 2nd sketch
- Select the 'lofted boss/base command'
- Then in 'profile' option select first sketch then 2nd and use guide curves if need as shown in following fig.
- Then 'ok'

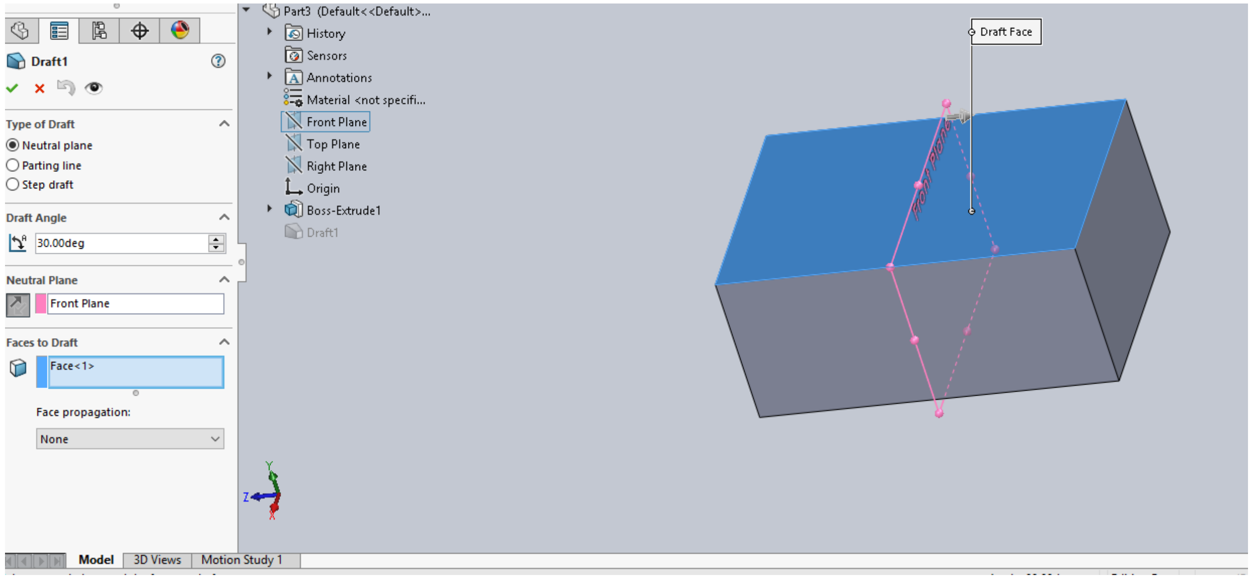

Loft
Profiles
Sketch1
Sketch2
Start/End Constraints
Guide Curves
Guide tangency type:
None
Centerline Parameters
Sketch Tools
Drag Sketch
Model
3D Views
Motion Study 1
Part2 (Default<<Default>...
History
Sensors
Annotations
Material <not specifi...
Front Plane
Top Plane
Right Plane
Origin
(-) Sketch1
Plane1
(-) Sketch2
Plane1
Profile(Sketch2)

Draft

In how many ways the draft command is used in SolidWorks?

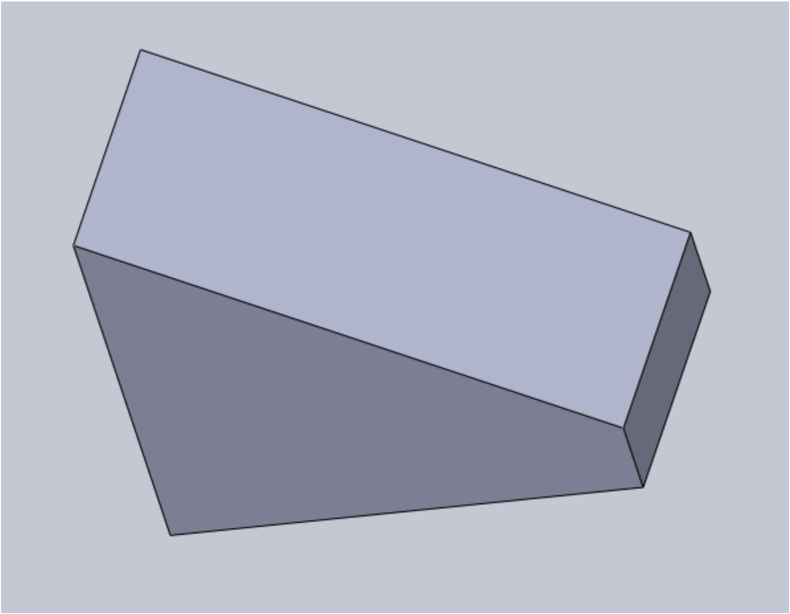
Draft command used to remove material from part

It used in different ways in SolidWorks



- The natural draft will be like this

This is at 30 degree



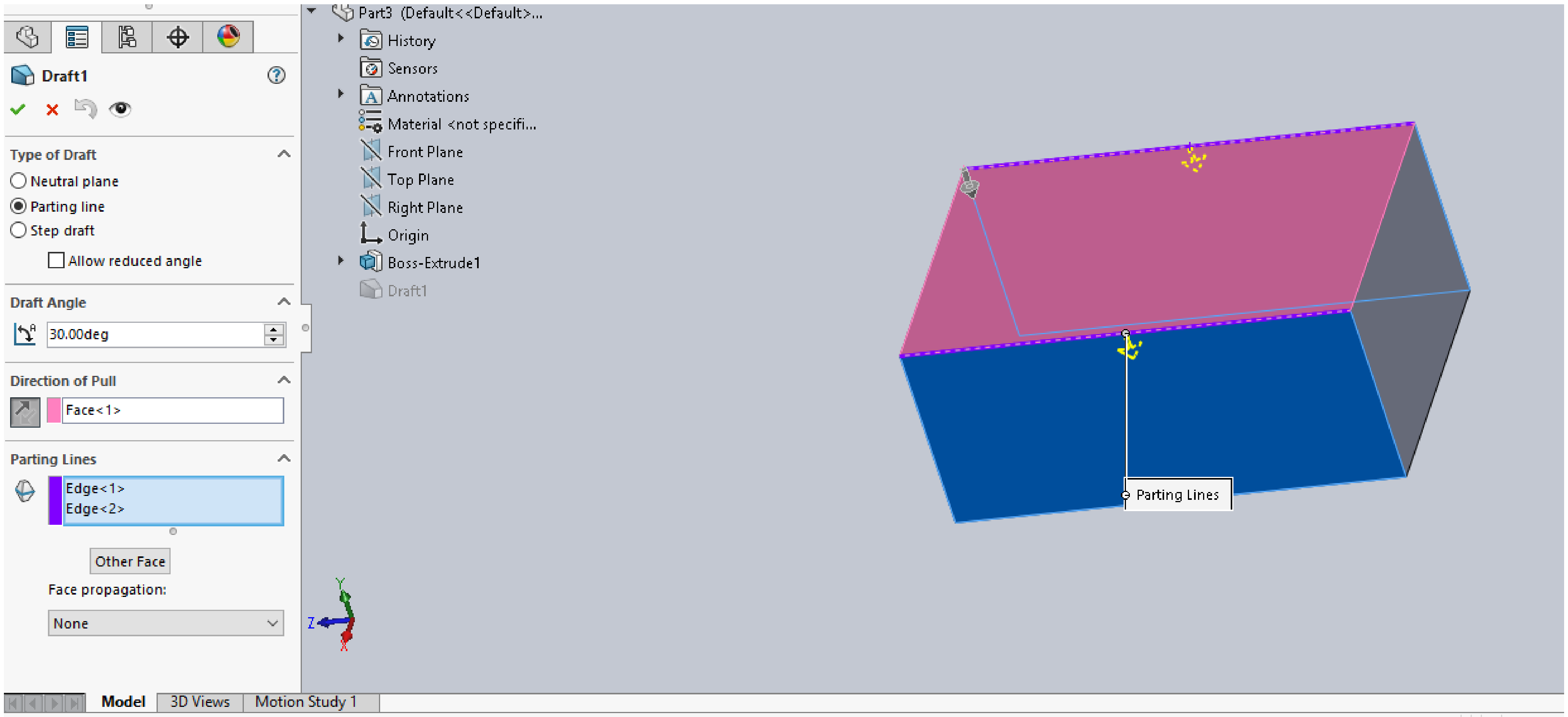
‘Parting line’ draft

First select the upper face

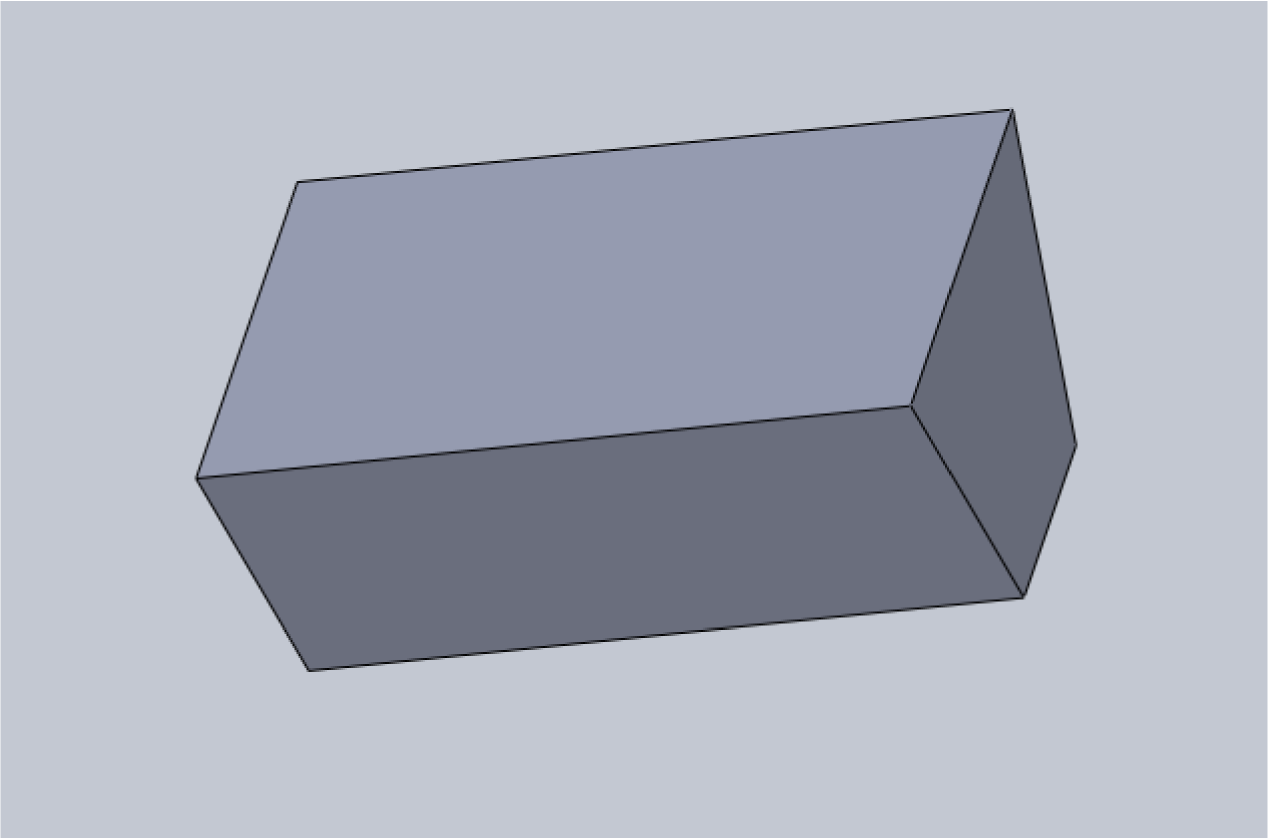
Then select the two edges which are parallel as shown then give angle

Then ‘ok’

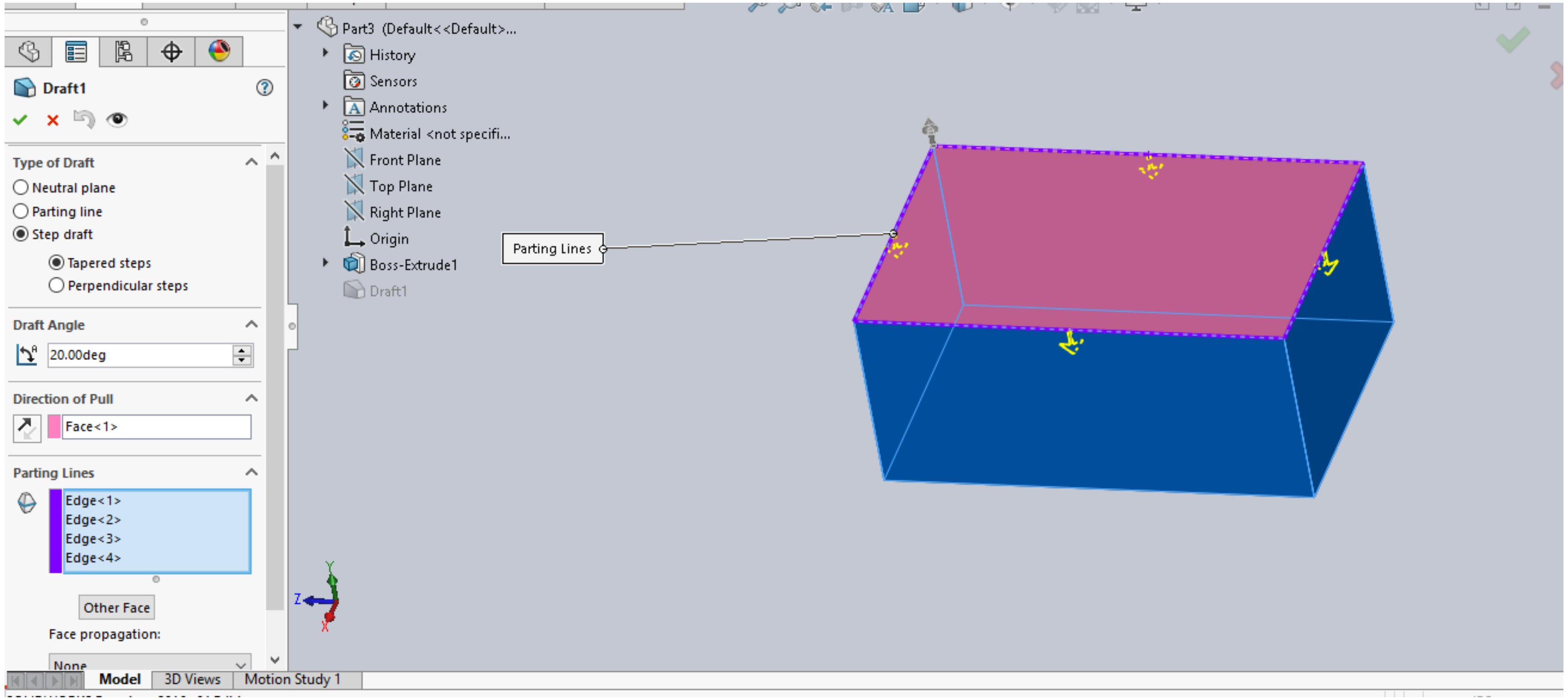
As shown in the following fig.


Draft1
Type of Draft
Neutral plane
Parting line
Step draft
Allow reduced angle
Draft Angle
30.00deg
Direction of Pull
Face<1>
Parting Lines
Edge<1>
Edge<2>
Other Face
Face propagation:
None
Part3 (Default<<Default>...
History
Sensors
Annotations
Material <not specifi...
Front Plane
Top Plane
Right Plane
Origin
Boss-Extrude1
Draft1
Parting Lines
Model
3D Views
Motion Study 1

And this is ‘parting line’ draft result



This is ‘step draft’



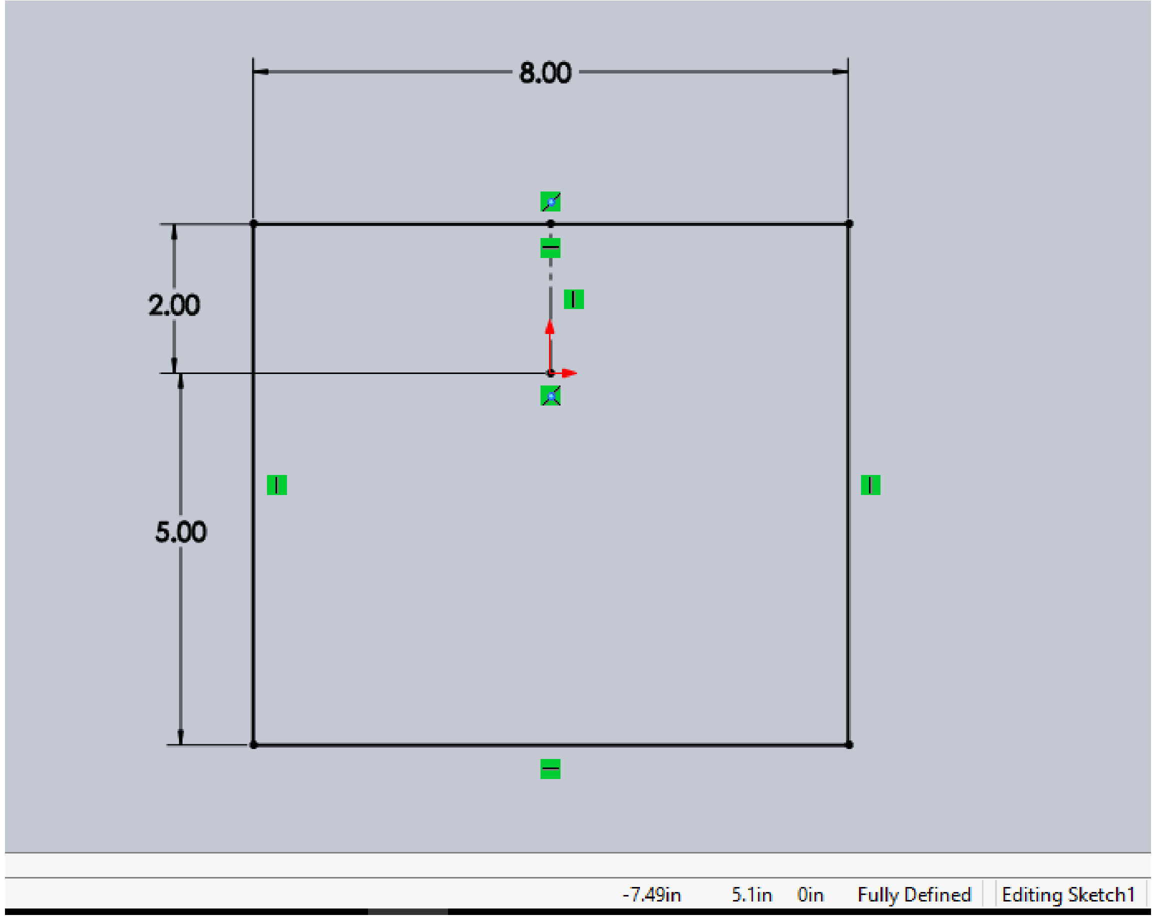
Opening a part document:

i)How the dimensions are fully defined in a sketch?

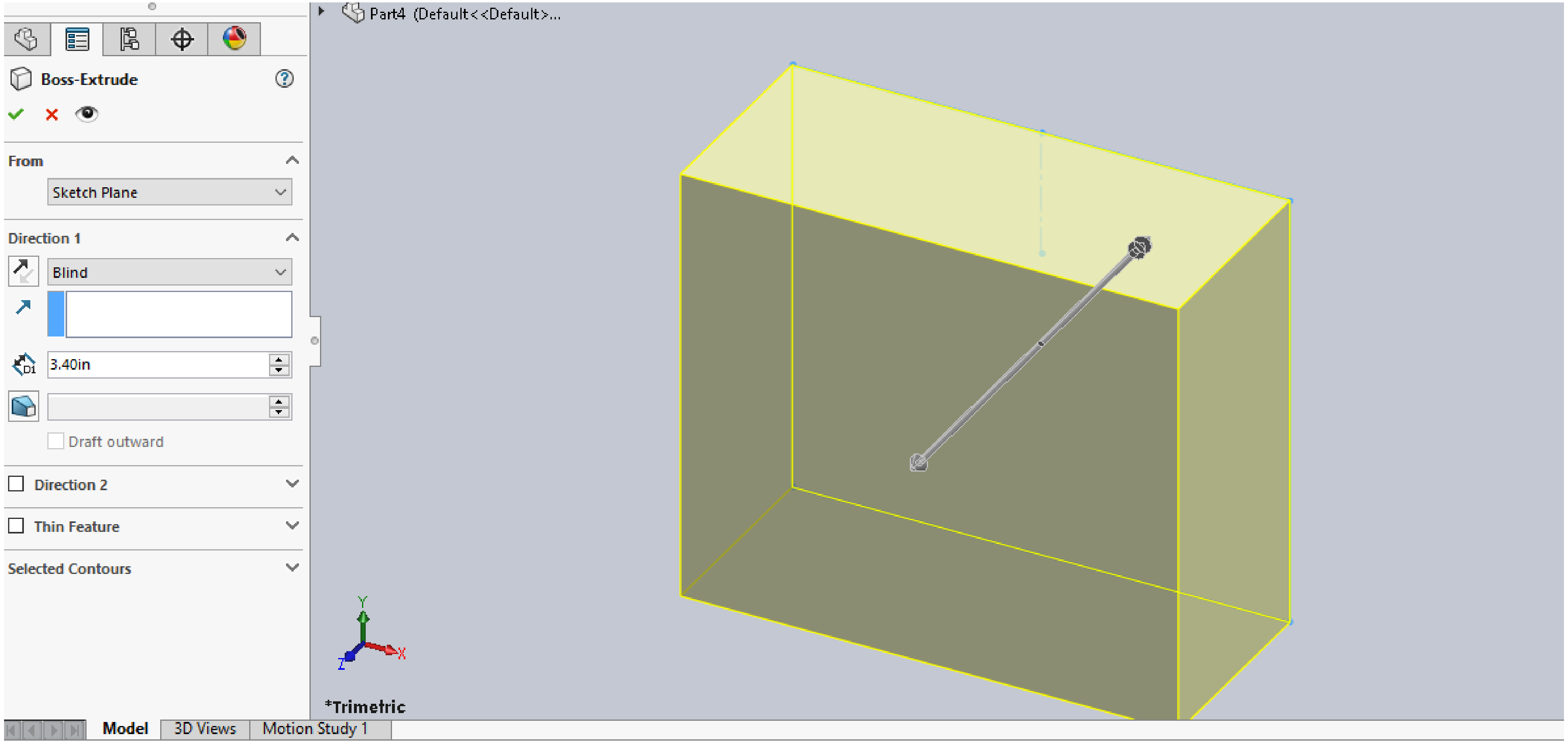
ii)How to make a sketch after exit the sketch?

- Open a part from a folder
- Edit the sketch and change it if you want and give all dimensions so that it becomes fully defined as shown in the following

Fully defined sketch

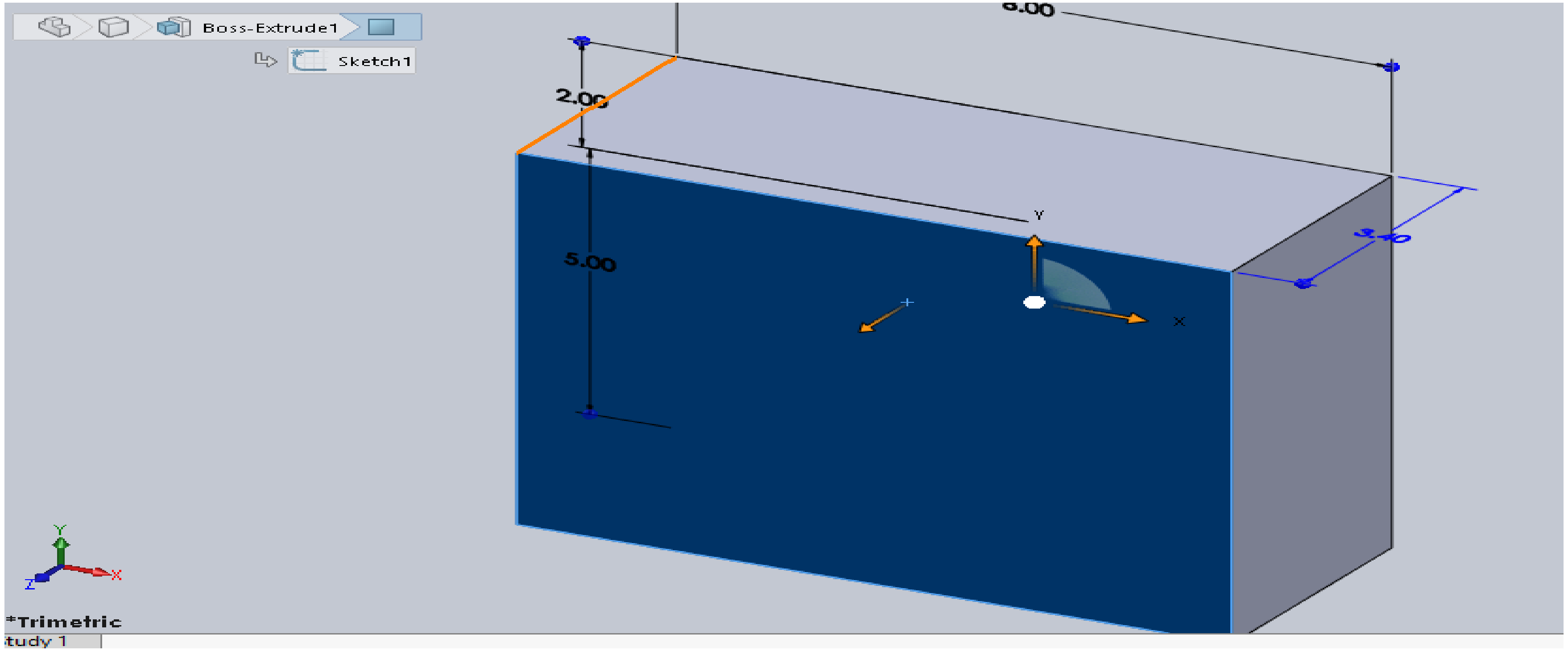


- Exit the sketch
- The feature manager tree will appear
- Click on ‘extruded boss/base’
- The property manager will be like this
- There are other options like draft and direction 2 which may be used

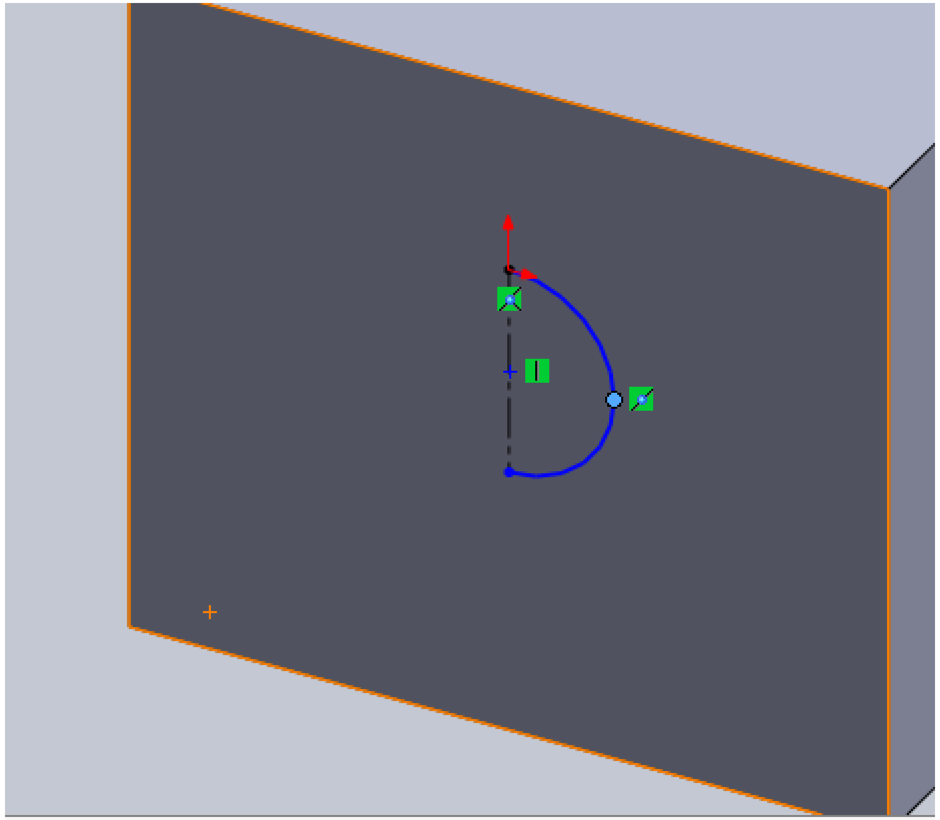

Part4 (Default<<Default>...
Boss-Extrude
From
Sketch Plane
Direction 1
Blind
3.40in
Draft outward
Direction 2
Thin Feature
Selected Contours
*Trimetric
Model
3D Views
Motion Study 1

Sketching the upper inlet port:

- Select the plane and click on the sketch to make new drawing on it
- Select the front plane in this case and sketch on it as shown



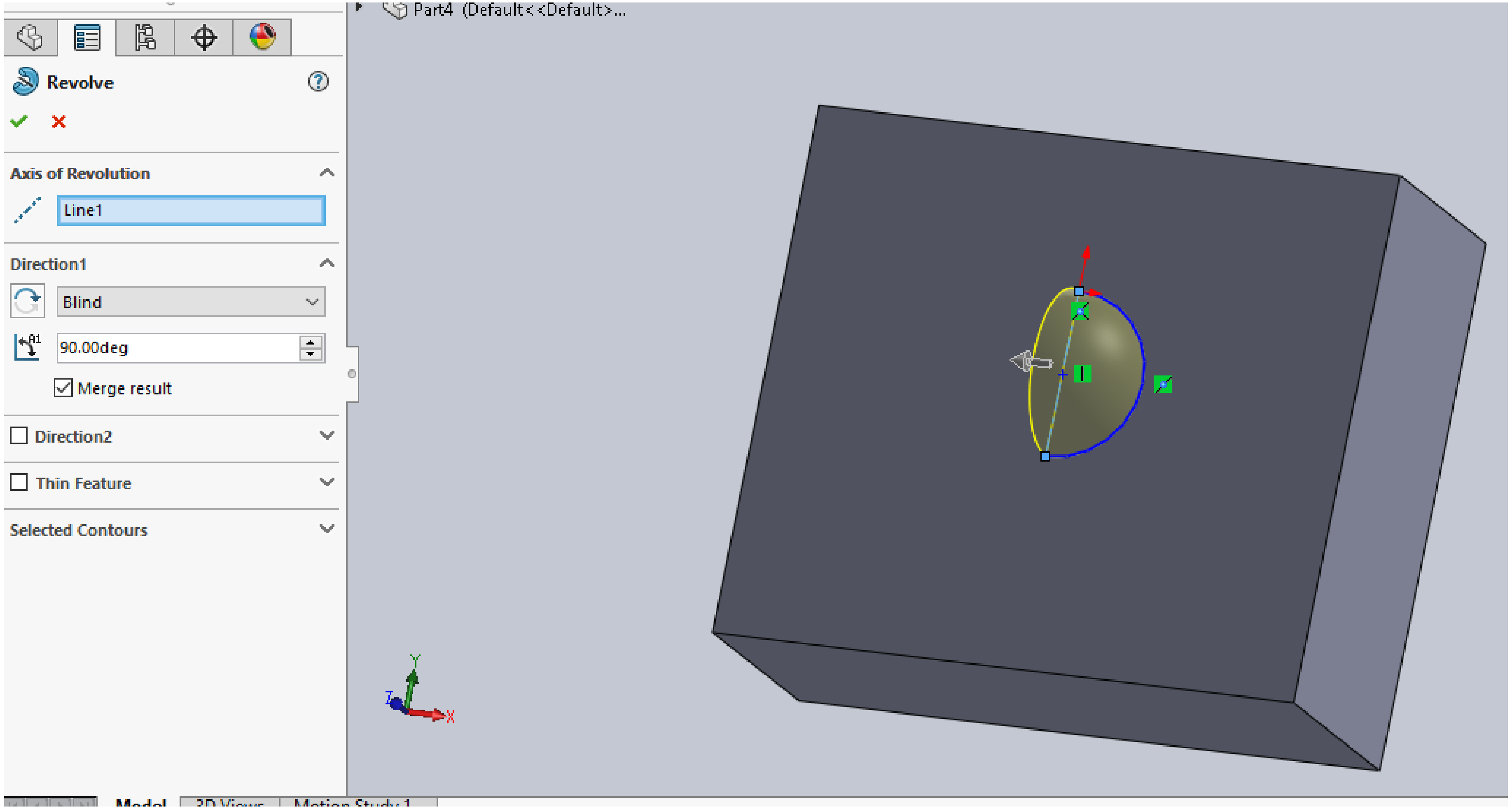
Sketch on it as shown



Revolve command

- the use of revolve command is easy and from this command we can draw complex geometry
- In revolve command first select the axis of rotation
- Then give the angle
- Then ‘ok’ as shown in fig.

Revolve command



Fillet:

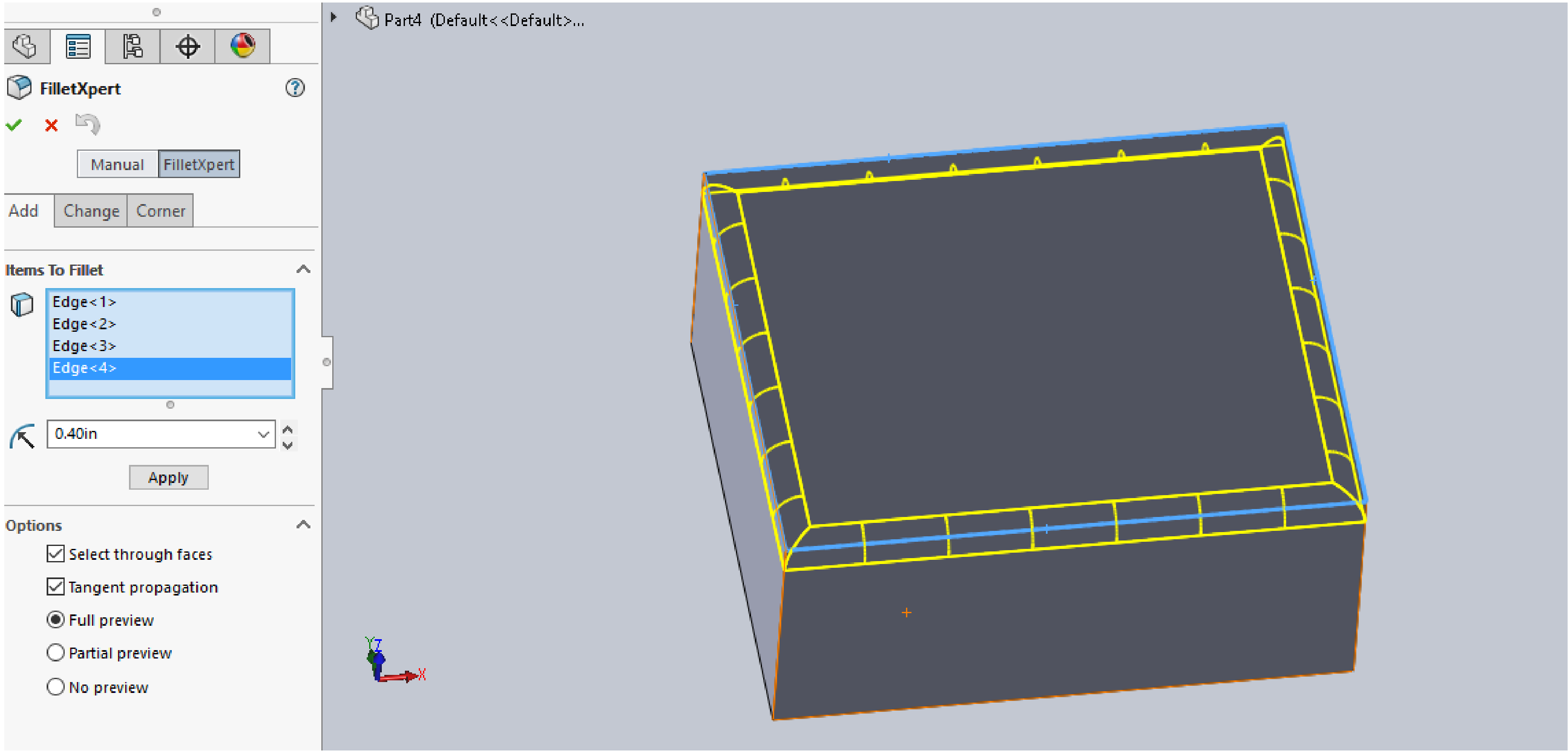
a) Why fillet are used?
b) What is difference between fillet and chamfer?

- ➢Select the edges you want to fillet
- ➢Then give radius value
- ➢The edges selected can be edited by right click on the item to fillet

And the delete and add other corner or edges

The fillet command is shown in following fig/

Fillet



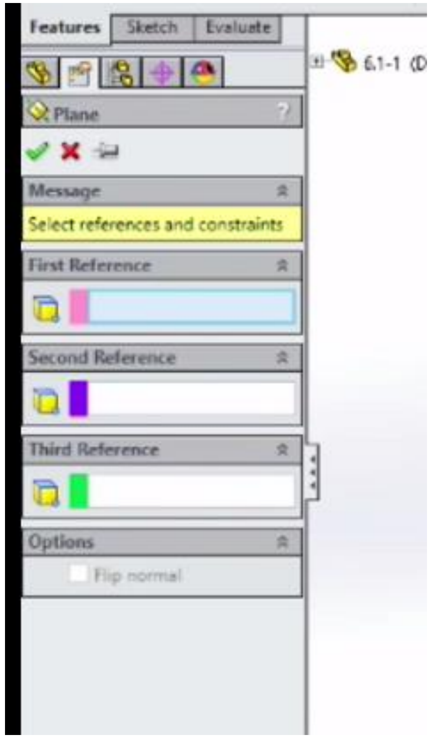
Part4 (Default<<Default>...
FilletXpert
Manual
FilletXpert
Add
Change
Corner
Items To Fillet
Edge<1>
Edge<2>
Edge<3>
Edge<4>
0.40in
Apply
Options
Select through faces
Tangent propagation
Full preview
Partial preview
No preview

Creating offset distance plane:

a) Why offset plane is used?

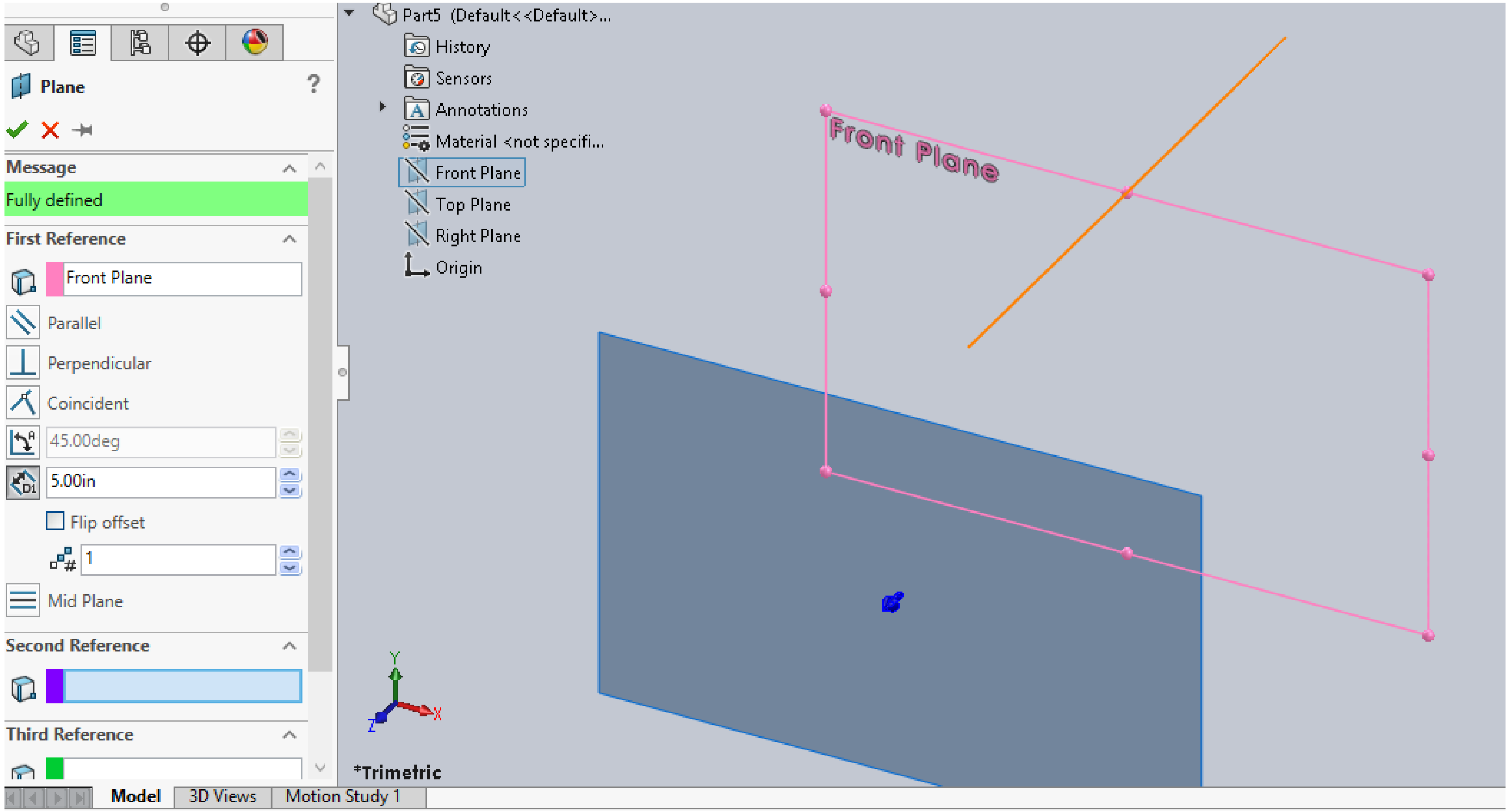
b) How many references are used to fully defined a new plane?

➢The new plane need 3 references

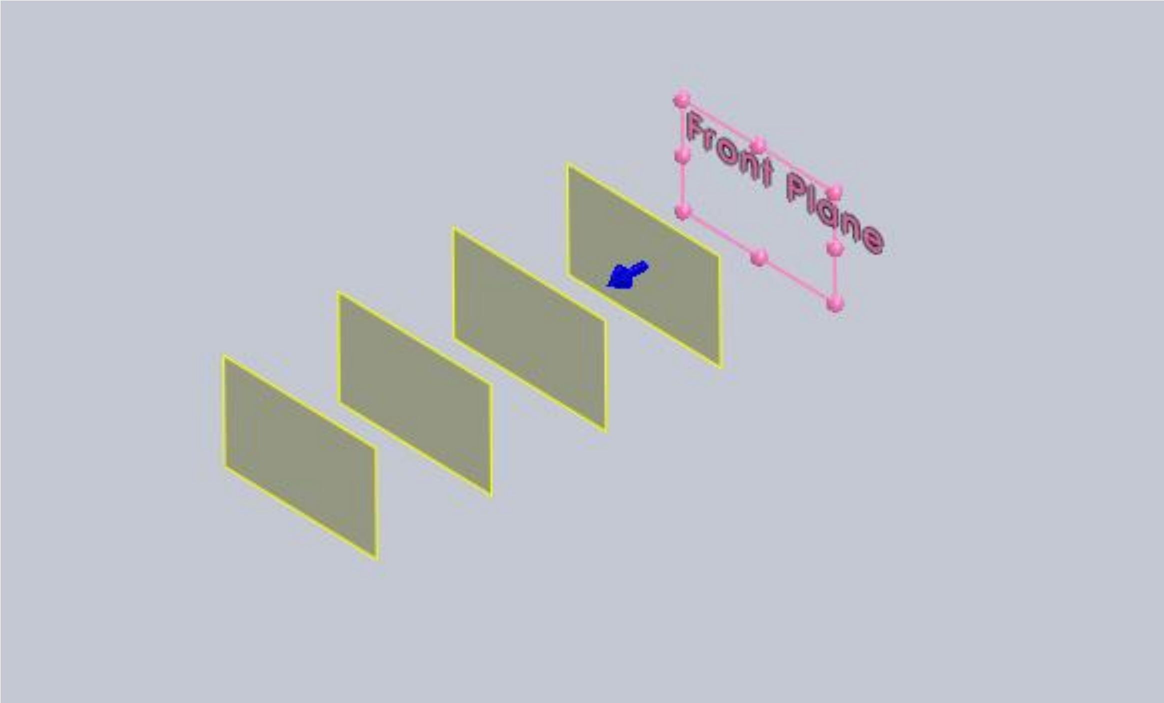


- From feature manager tree select the **front** plane to offset from
- Then enter any offset distance e.g. 5 inches
- Here we can set its direction either left or right from the front plane
- We can make number of plane

Offset plane

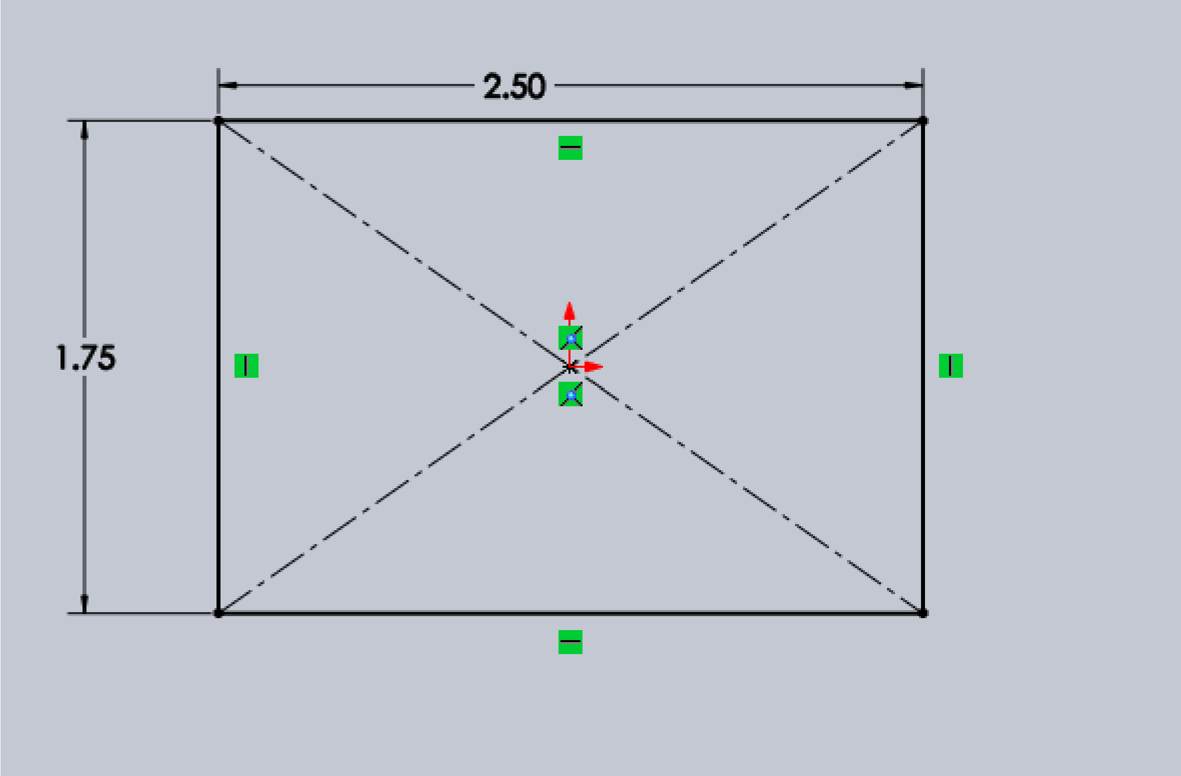


4 planes are created at equal distance as shown below:



Sketch the 1st loft profile

This is 1st lofted profile
And its fully defined
As all line are black
❖Select plane 1
And open a new sketch
Select a rectangle and
Draw sketch
Add the dimensions
And relations to fully
defined



The 2nd loft is sketch on the 2nd plane as shown

❖Select plane 2

And open a new sketch

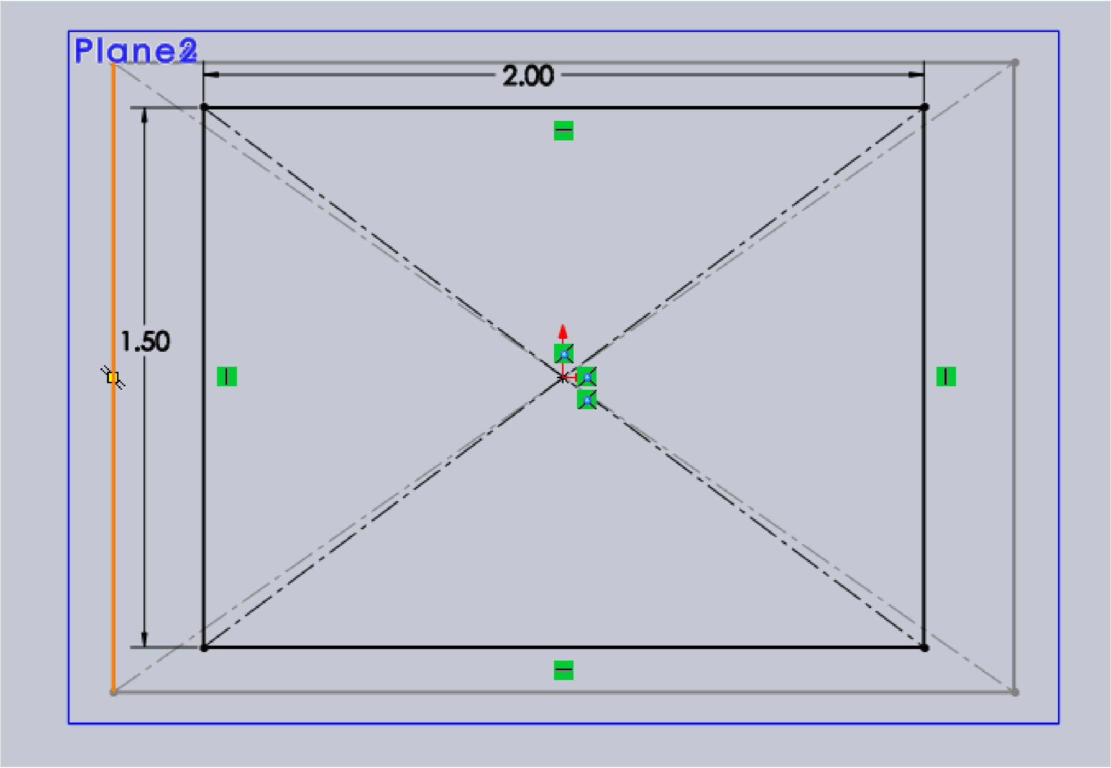
Select a rectangle and

Draw sketch

Add the dimensions

And relations to fully

defined



3rd sketch for loft on 3rd plane

❖Select plane 2

And open a new sketch

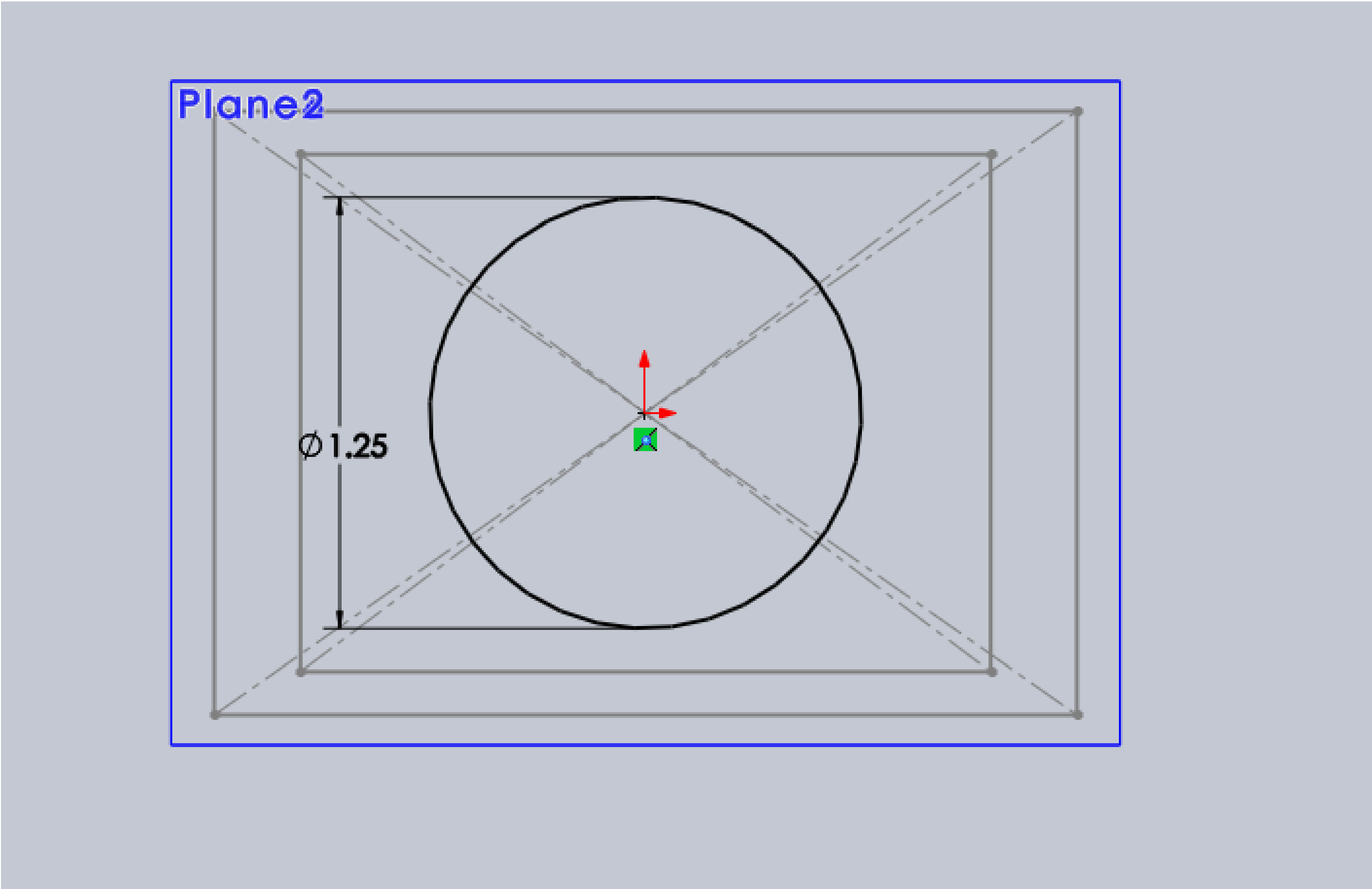
Select a circle and

Draw sketch

Add the dimensions

And relations to fully

defined



- Create 4th loft

❖Select plane 4

And open a new sketch

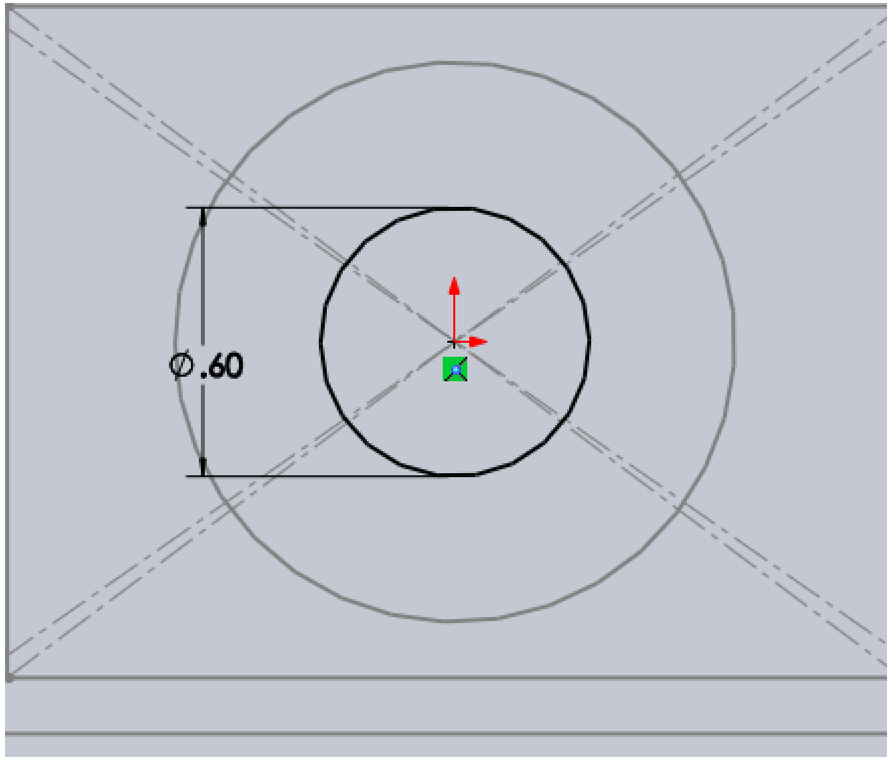
Select a circle and

Draw sketch

Add the dimensions

And relations to fully

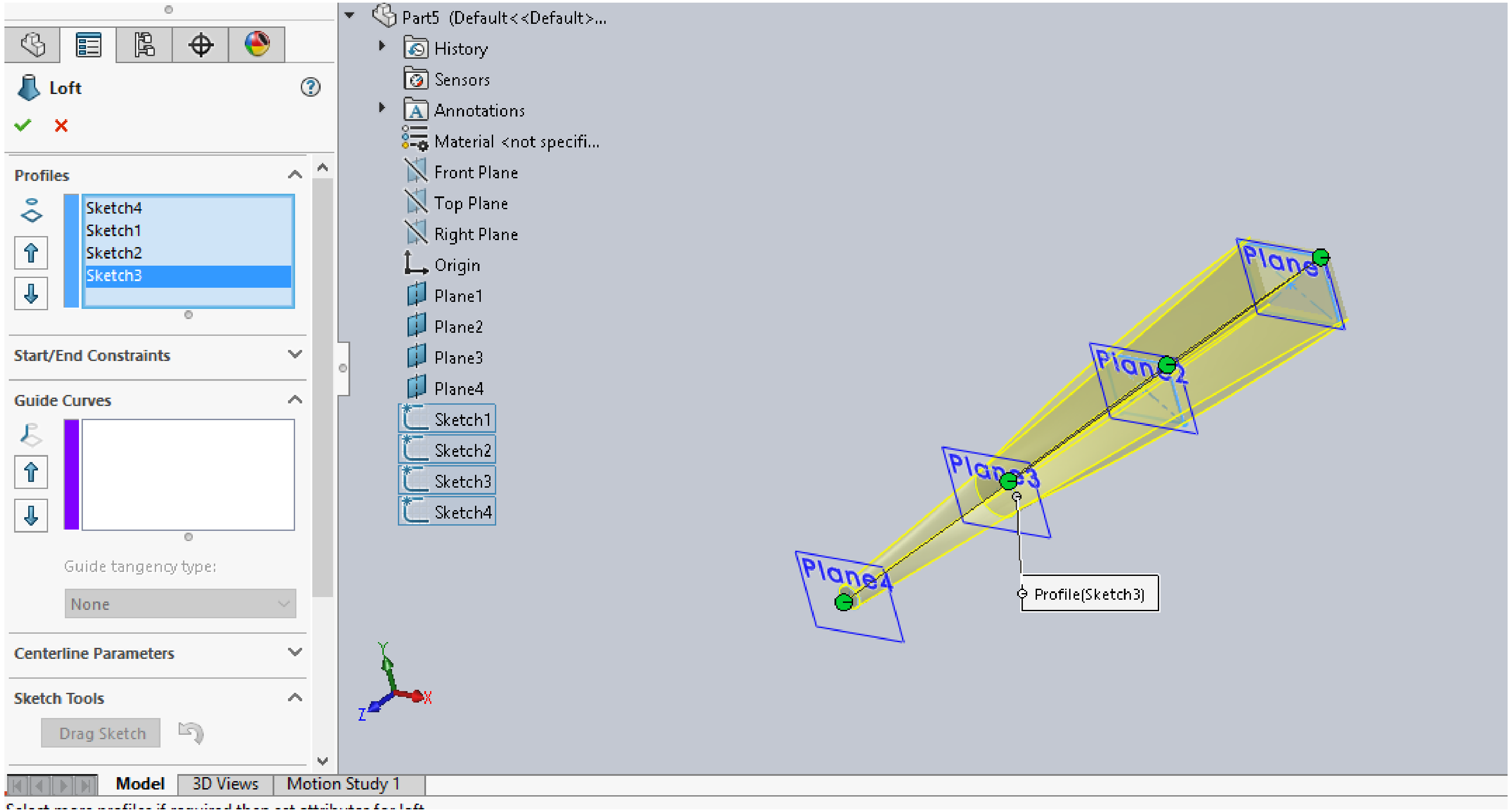
defined



Creating a loft feature:

Loft create when planes are out of phase angle(true,false)

- Exit the sketch and click on the **lofted boss/base** command
- Select all 4 sketches as shown in fig.in next slide
- Click ‘ok’


Loft
Profiles
Sketch4
Sketch1
Sketch2
Sketch3
Start/End Constraints
Guide Curves
Guide tangency type:
None
Centerline Parameters
Sketch Tools
Drag Sketch
Part5 (Default<<Default>...
History
Sensors
Annotations
Material <not specifi...
Front Plane
Top Plane
Right Plane
Origin
Plane1
Plane2
Plane3
Plane4
Sketch1
Sketch2
Sketch3
Sketch4
Plane1
Plane2
Plane3
Plane4
Profile(Sketch3)
Model
3D Views
Motion Study 1

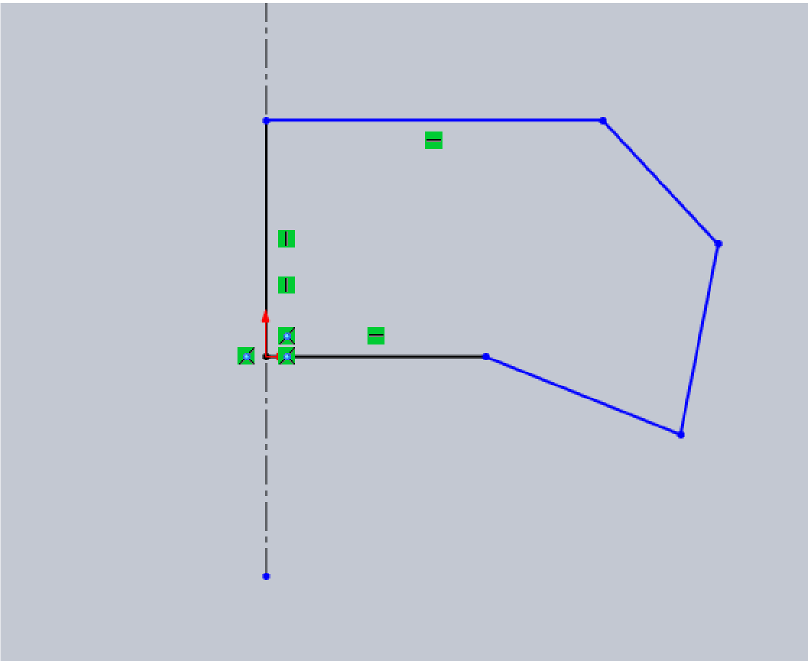
Mirror command

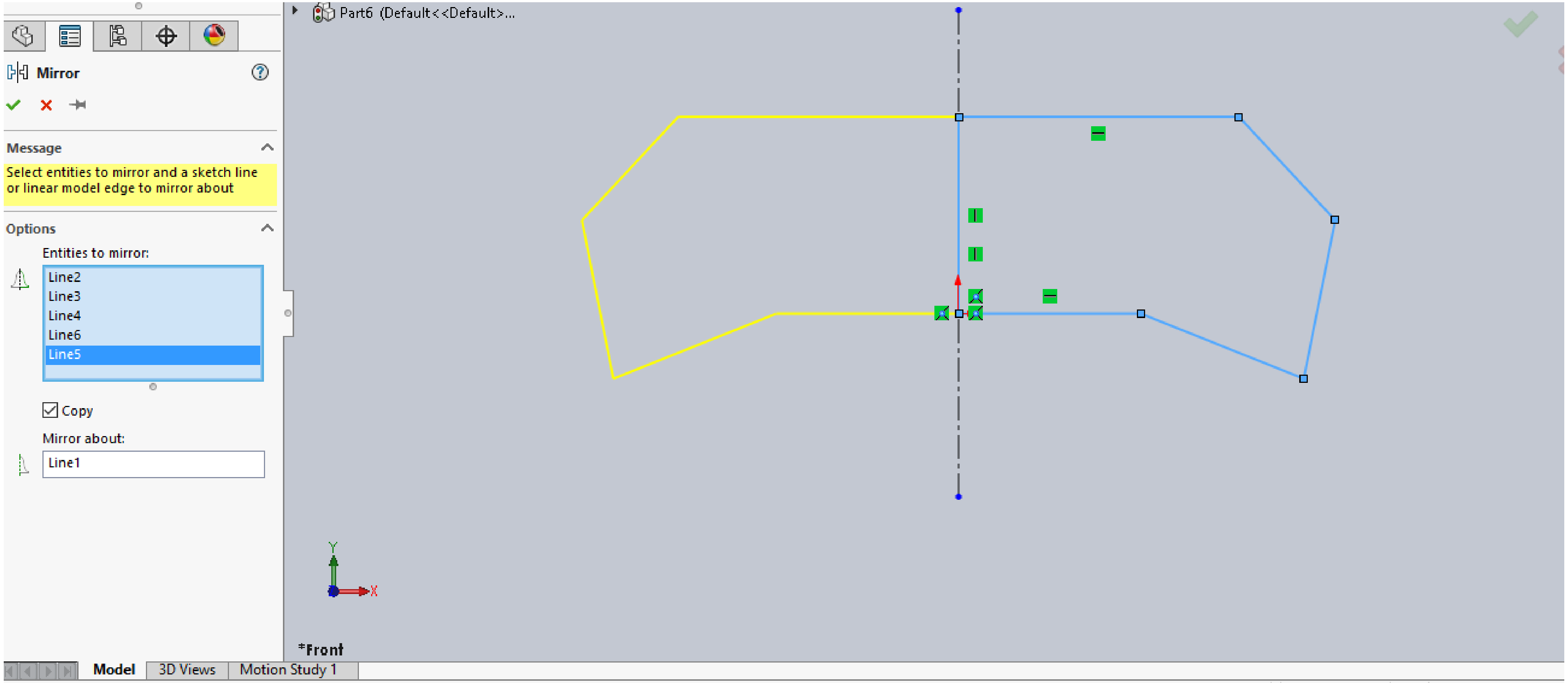
a) What is the difference between mirror and copy?

b) Mirror in sketch manager is same as in feature manager(true,false)

This is sketch I want to mirror about Centre line

- Select command **mirror** in sketch manager
- Select the **entities to mirror** then
- Selected the lines



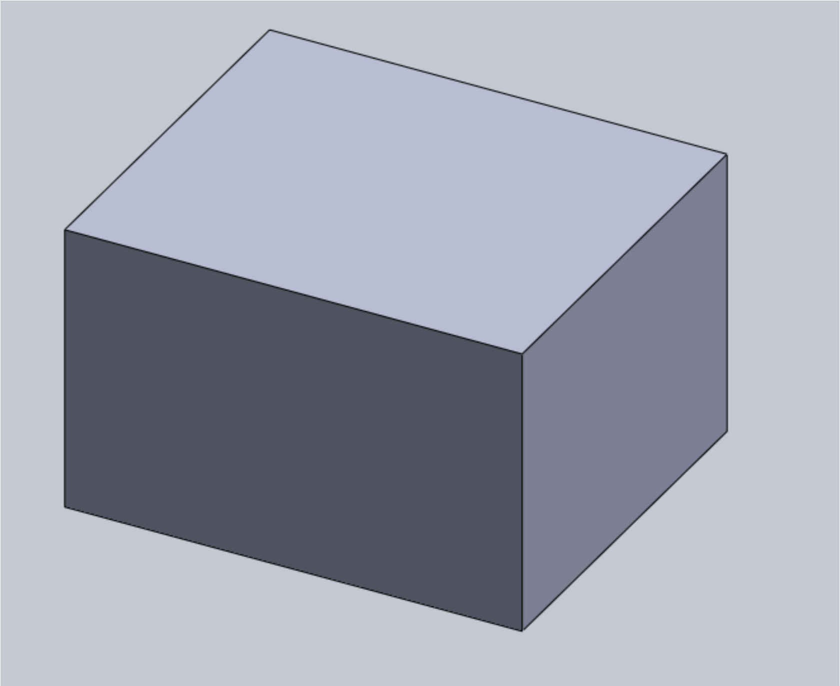

Part6 (Default<<Default>...
Mirror
Message
Select entities to mirror and a sketch line or linear model edge to mirror about
Options
Entities to mirror:
Line2
Line3
Line4
Line6
Line5
Copy
Mirror about:
Line1
*Front
Model
3D Views
Motion Study 1

- Then **mirror about** and select center line
- **‘Ok’**

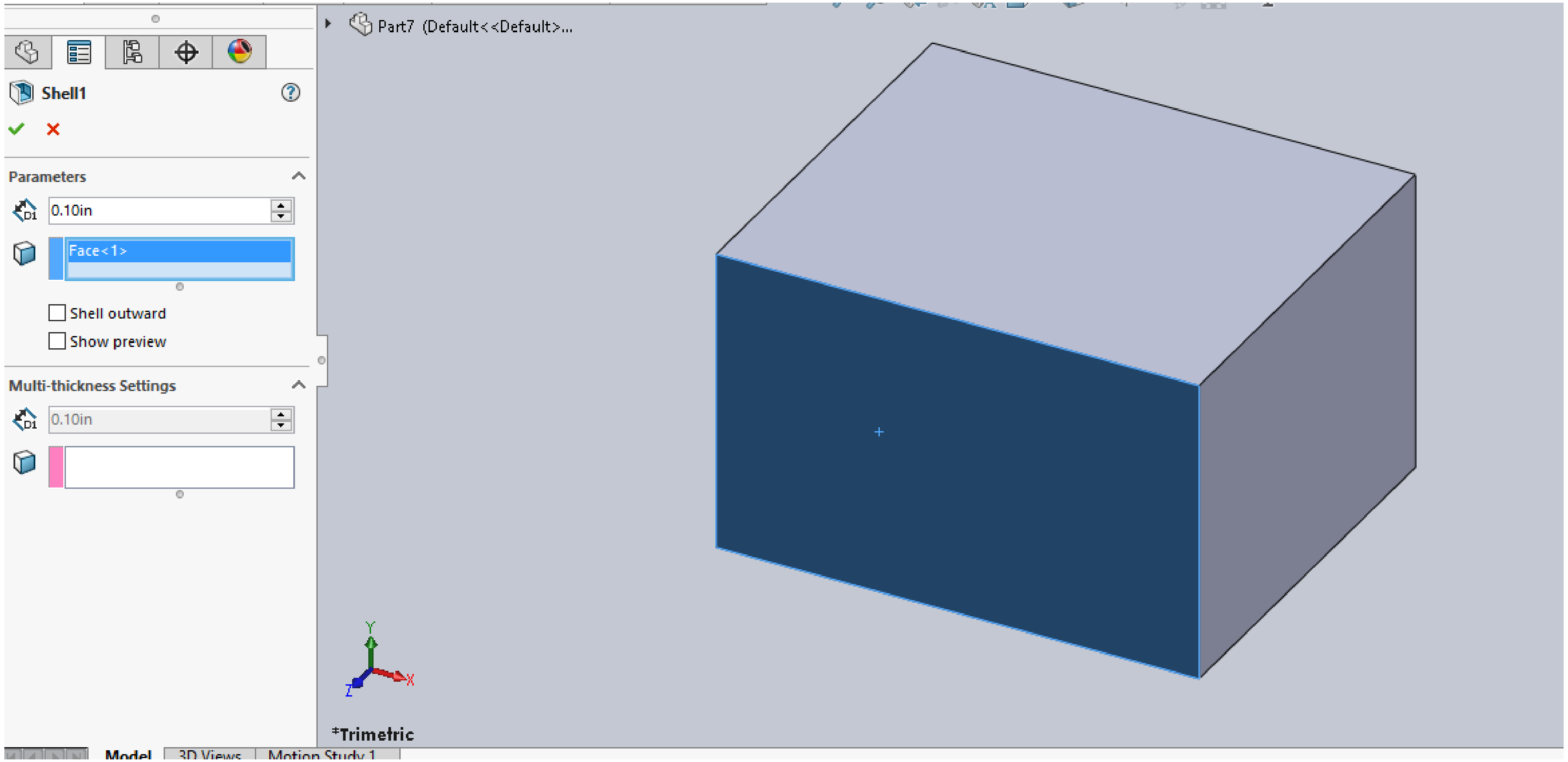
shell:

What is the purpose of shell command in solidworks?

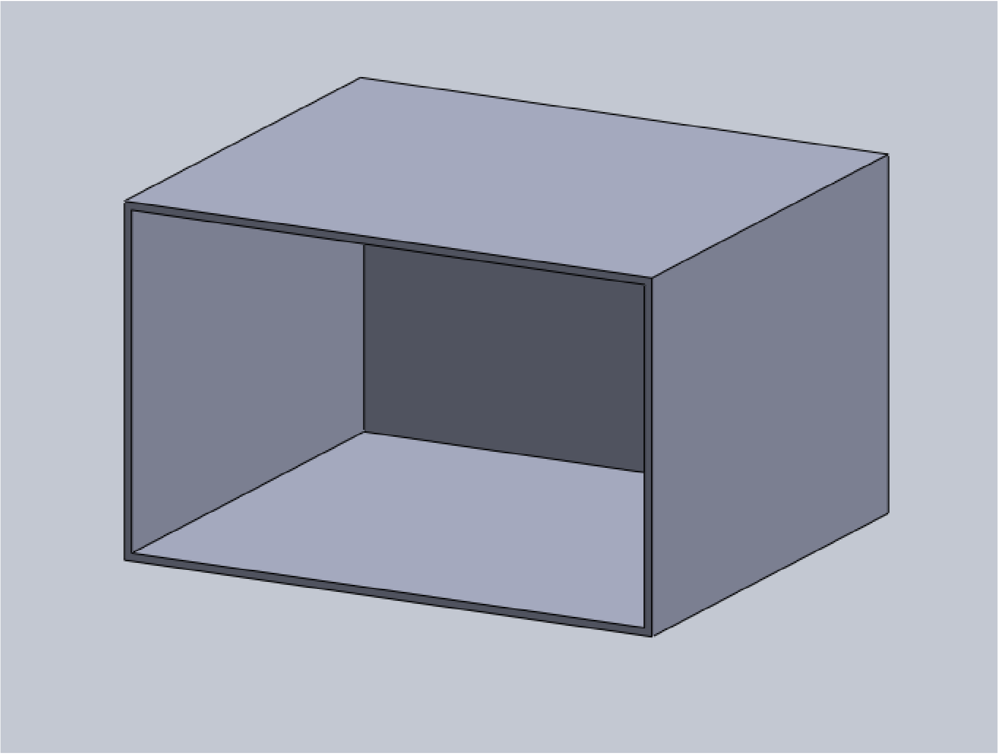
- Make the part as shown
- Click on command **shell**
- Then give thickness to the part
- Then faces which are to be removed



➢**Multi-thickness setting** we can set thickness of different faces



The final shape after shell command is shown beloow

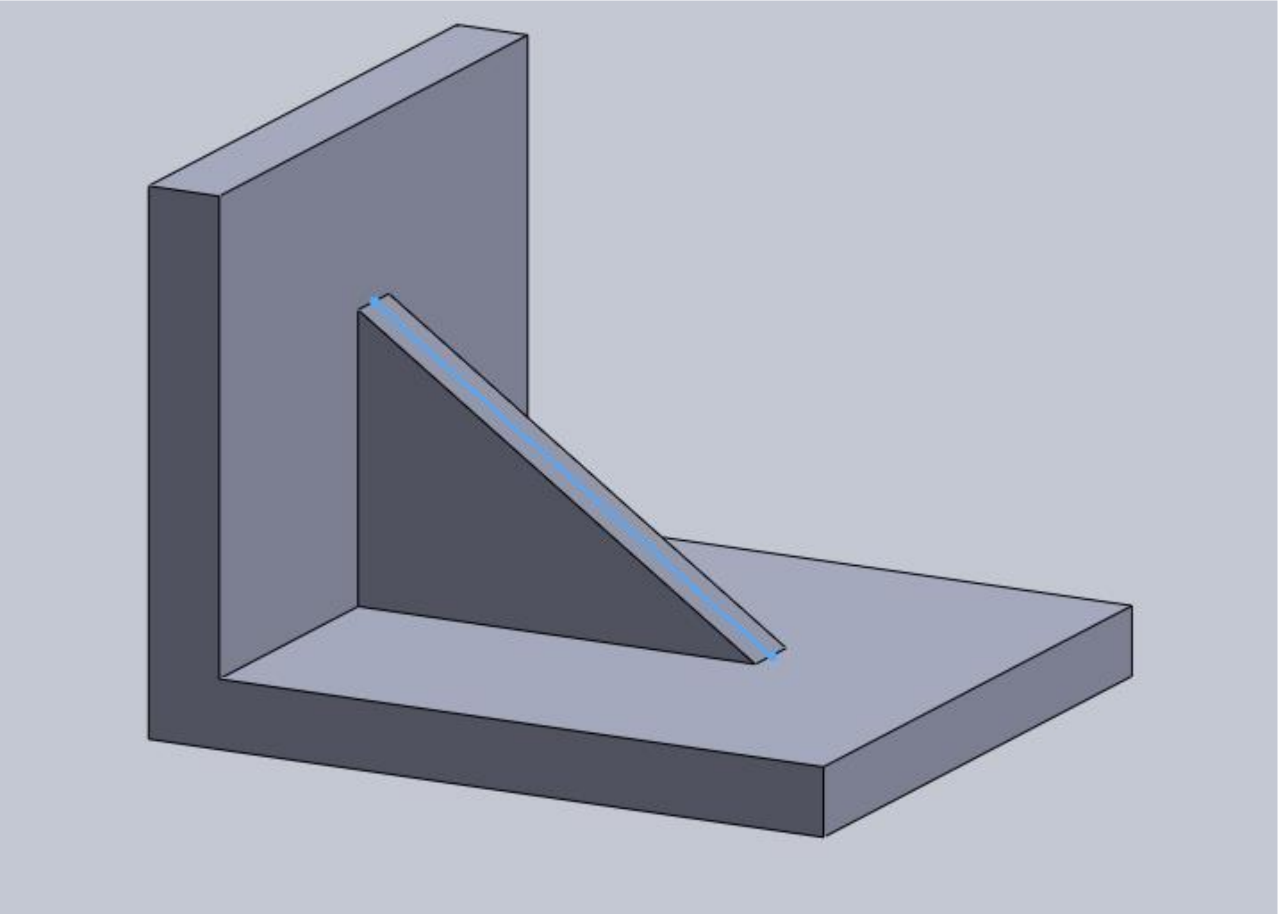


Rib:

a) If the direction of arrow is outside the rib then it will not make rib(true, false)
b) What are the application of rib?

- Make a sketch and extrude it and then select **rib** command
- Select a plane where you want to make a rib
- Sketch at this plane and give thickness to this rib and then ‘ok’

As shown in following fig.



You can save the work as **ctrl + s**