# Lecture #8

## SOLID MODELING

# This week

You will learn 3D (or solid) modeling.

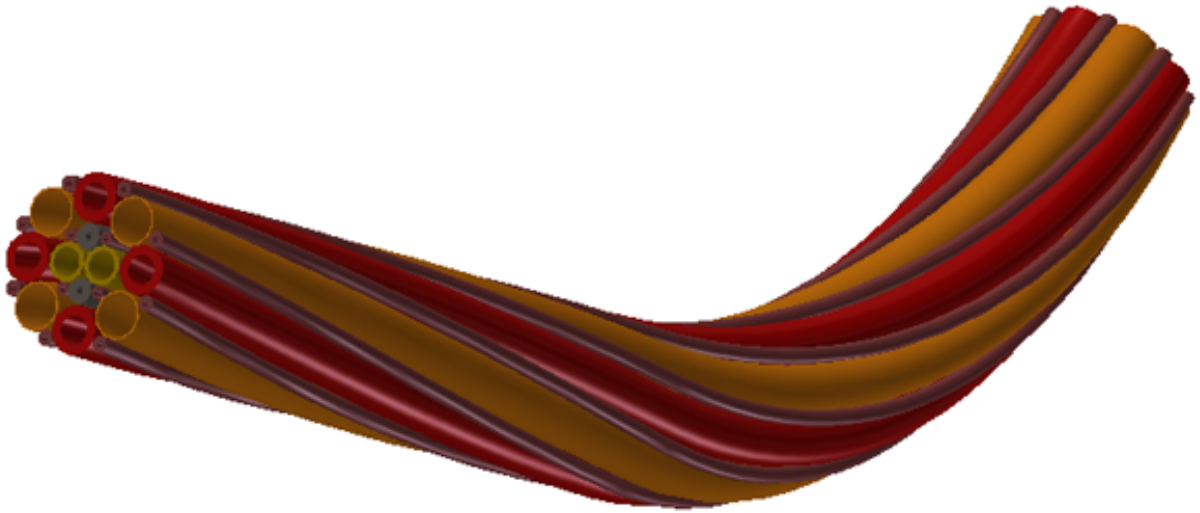
The steps to follow are:

- Swept features
- Helical sweep
- Blended features
- Lofting
- Assignment # 7

# Swept features

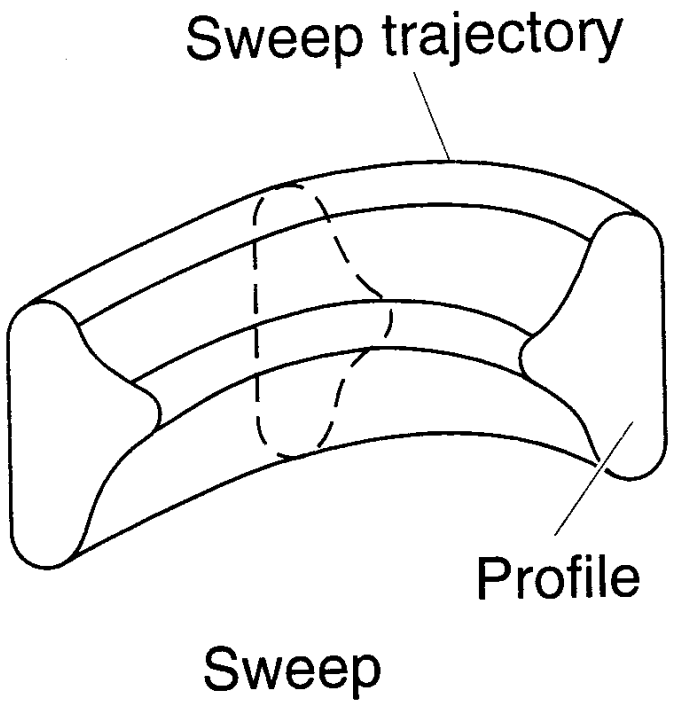
The sweep option can be compared to the extrude option.

The sweep option creates a section along a user-defined trajectory.

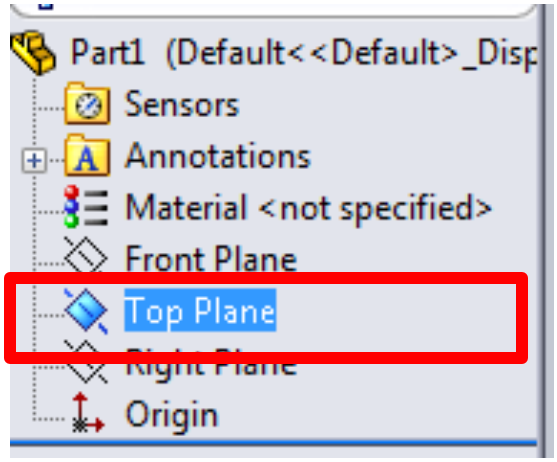


# Swept features

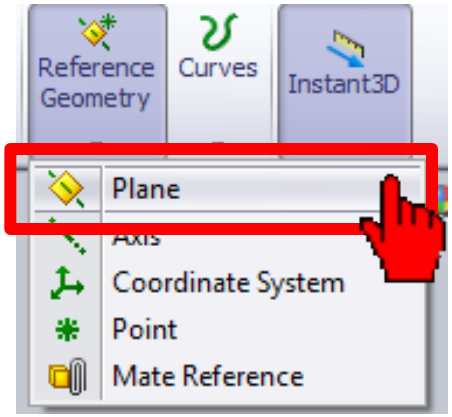
This trajectory can be either user-sketched or selected on the work screen.



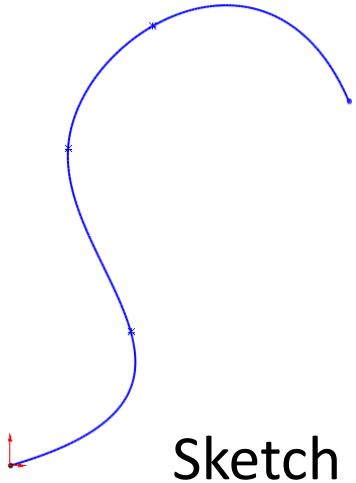
# Sweep




Working plane

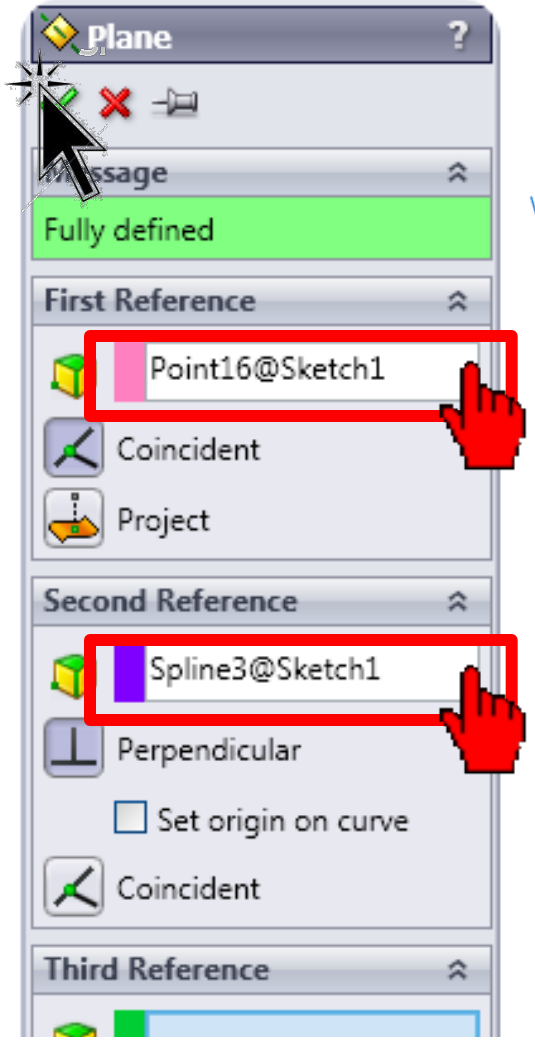


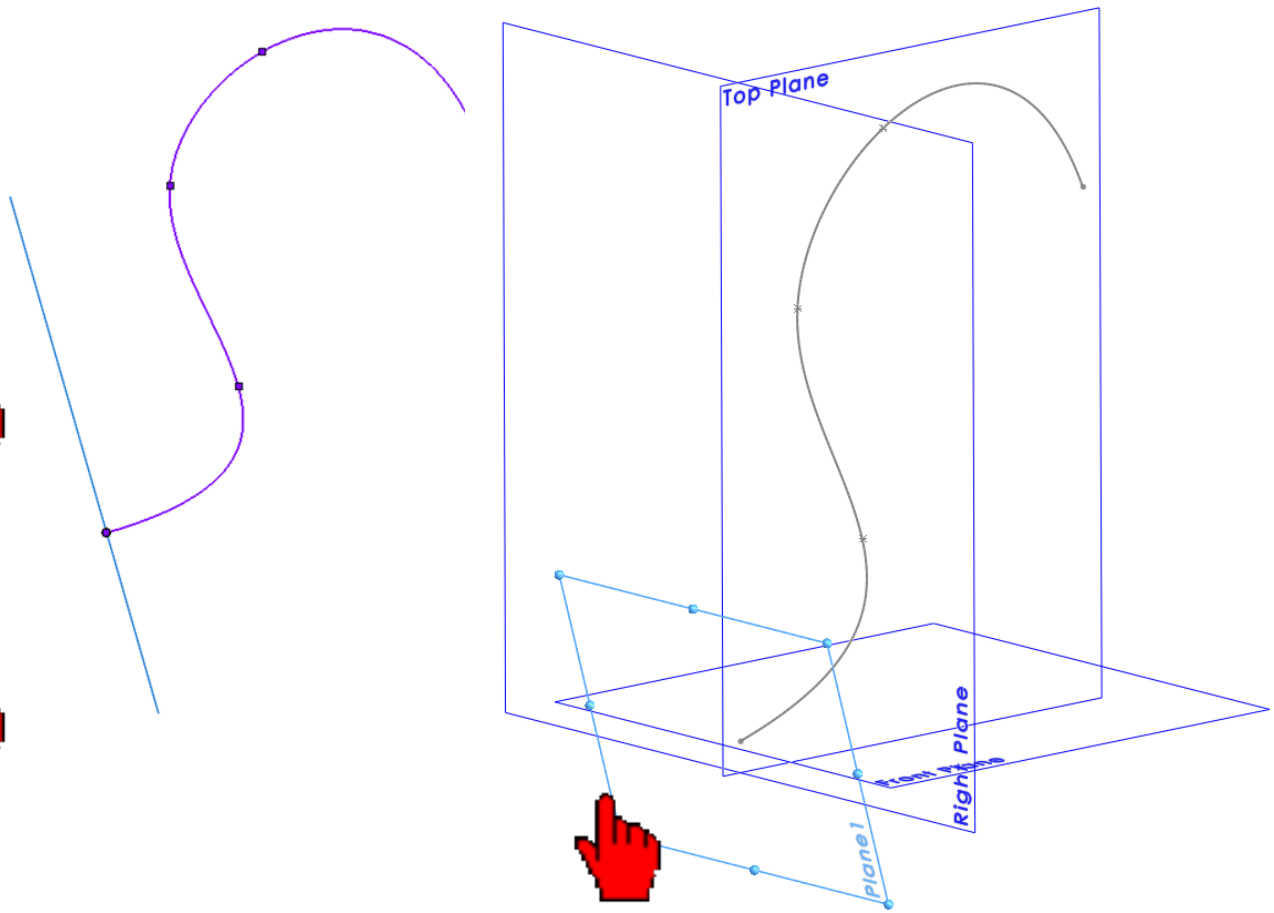

Make a Plane perpendicular to the endpoint of the spline.



Sketch a spline

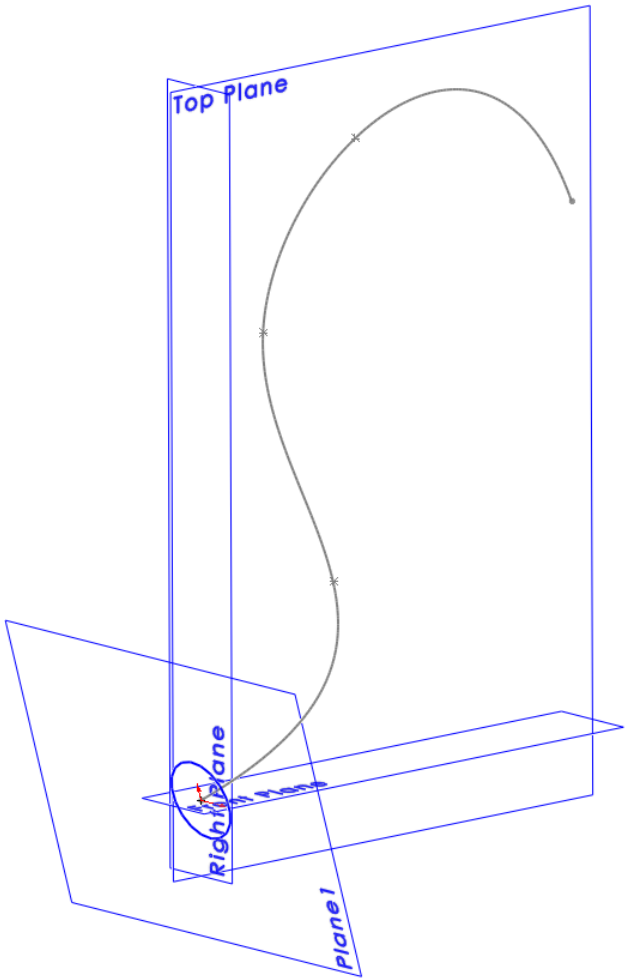
# Sweep



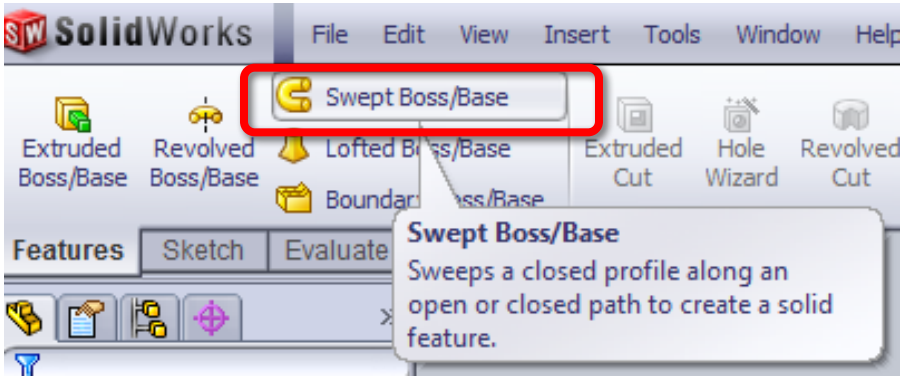



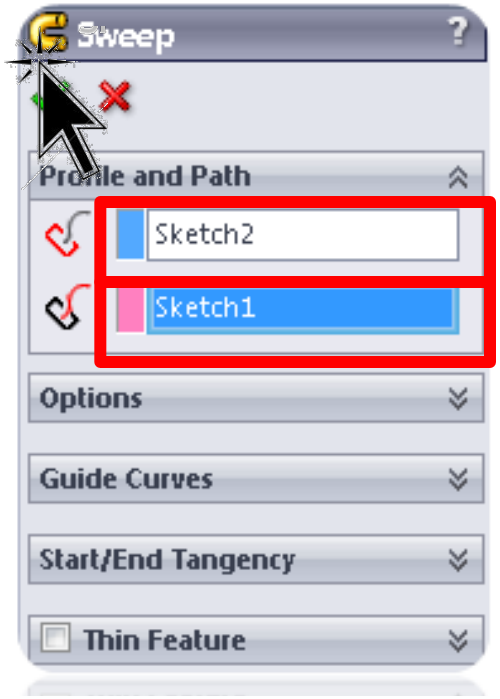

Select the new Plane1 in the feature tree and create a sketch by clicking on the 2D Sketch icon.

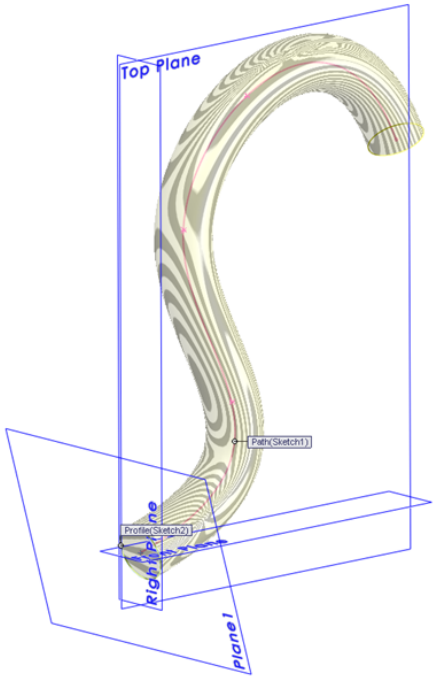
# Sweep



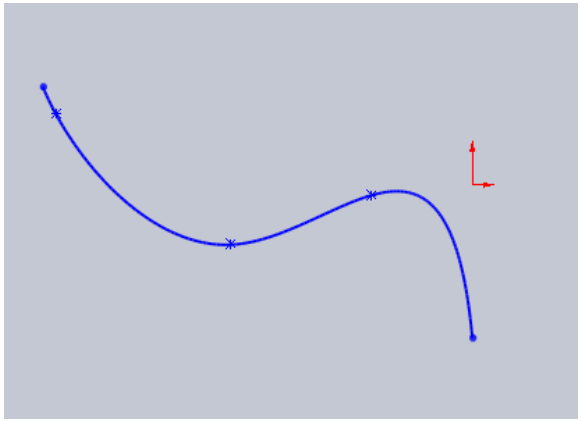

Sketch circle

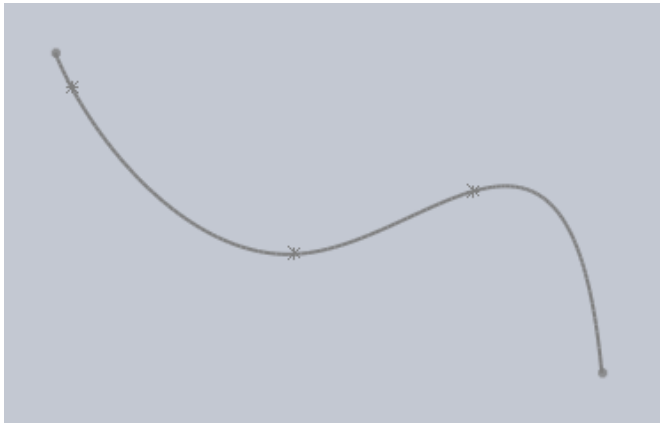


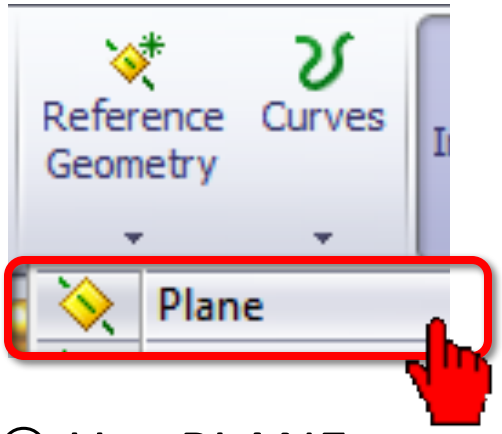



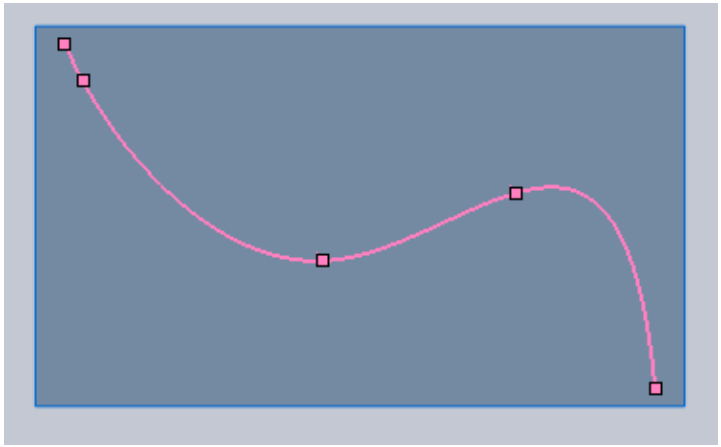
# Example : Swept Boss/Base

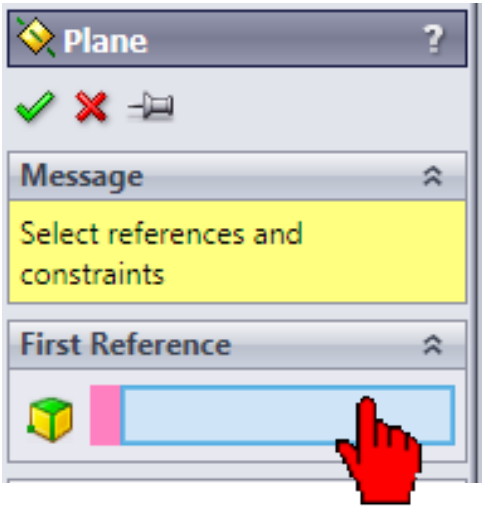




① sketch sweep patch ② Select REBUILD

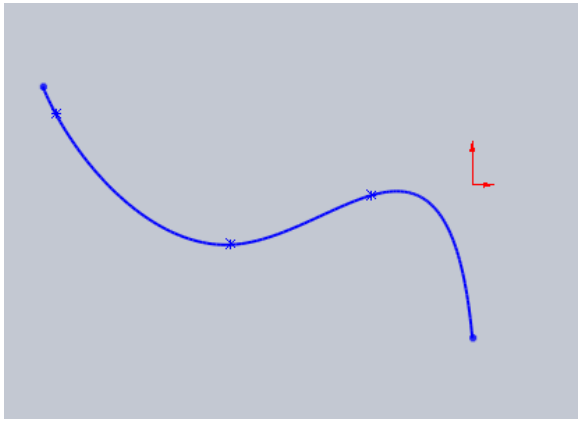


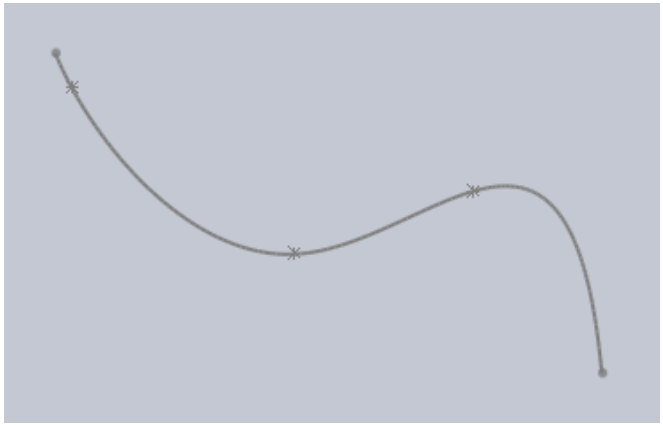





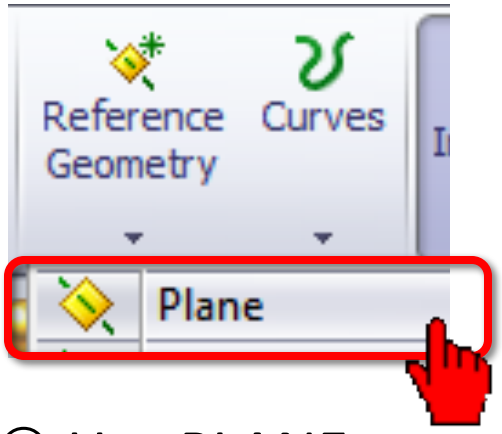

③ Use PLANE ④ Select any point on the path

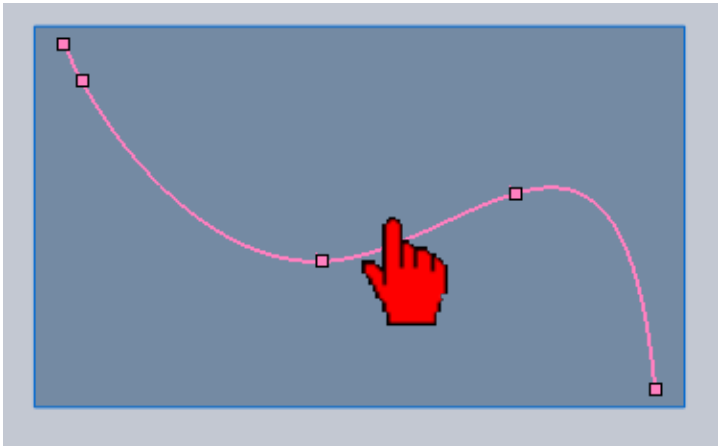
# Example : Swept Boss/Base

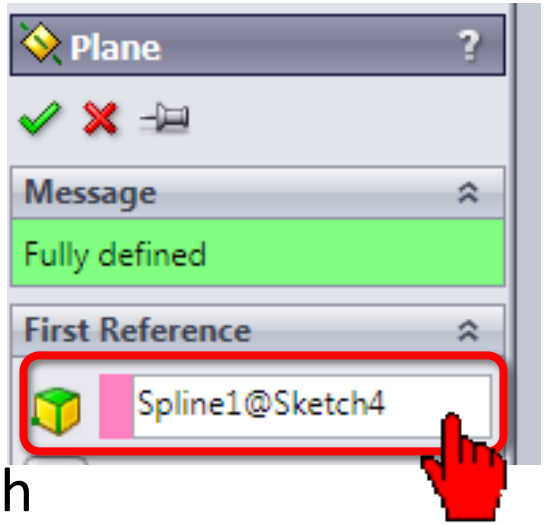




① sketch sweep patch ② Select REBUILD

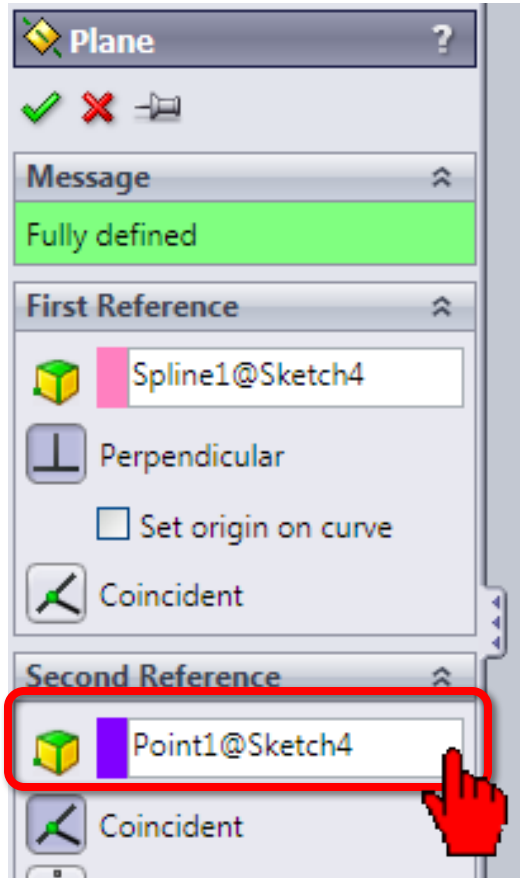


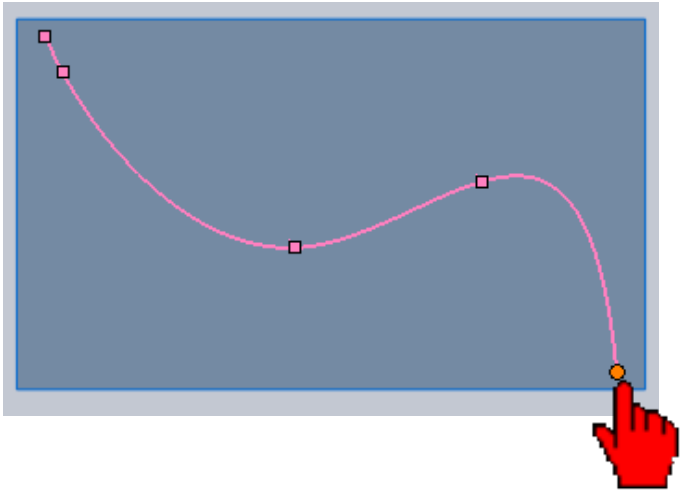






③ Use PLANE ④ Select any point on the path

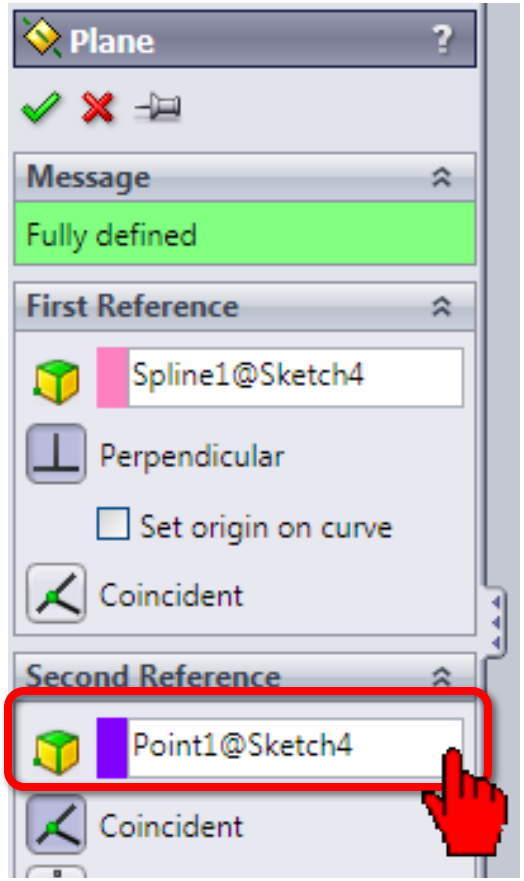
# Example : Swept Boss/Base

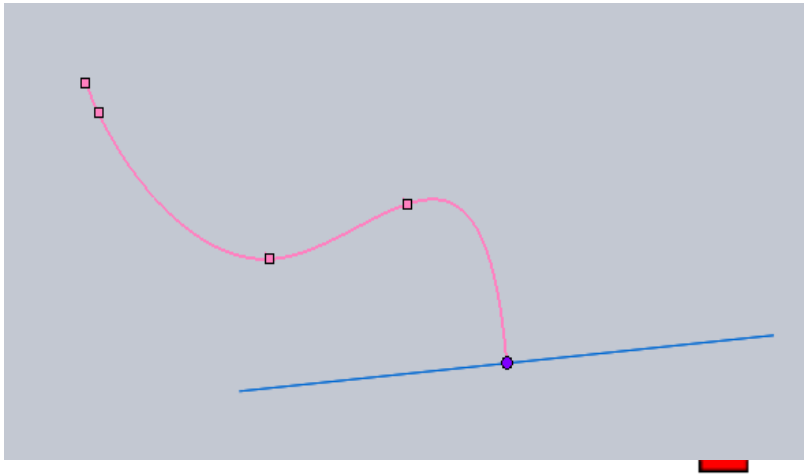


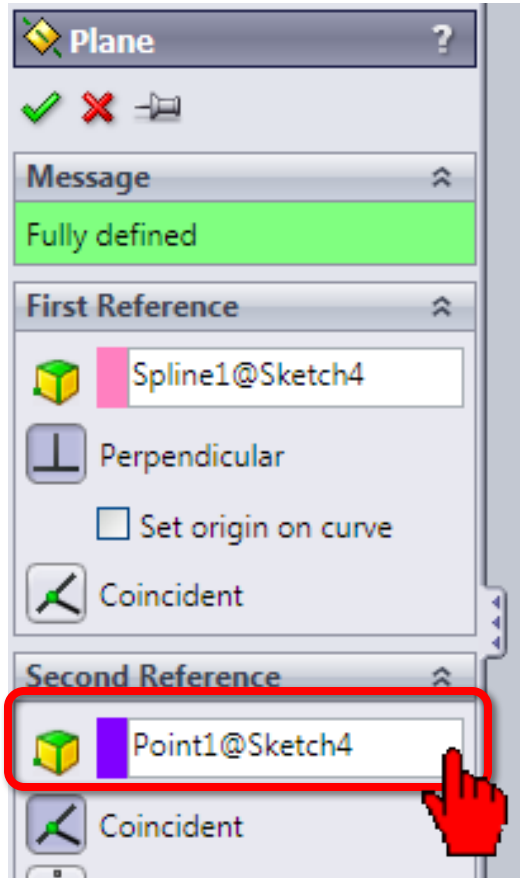
⑤ Select the end point

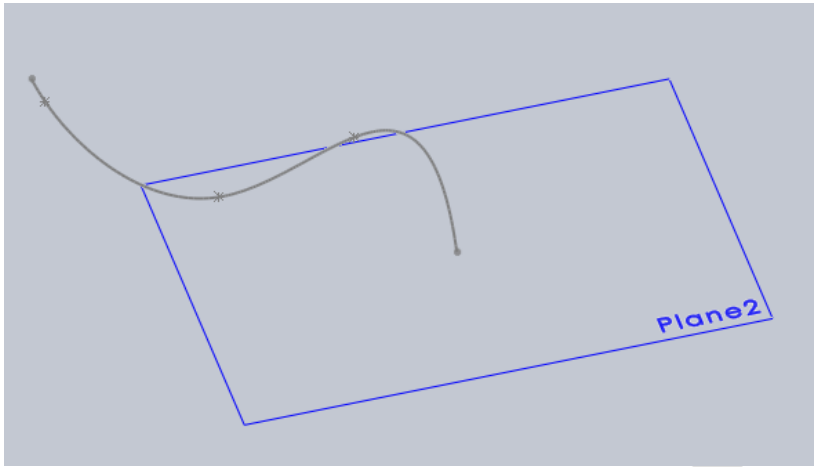
# Example : Swept Boss/Base



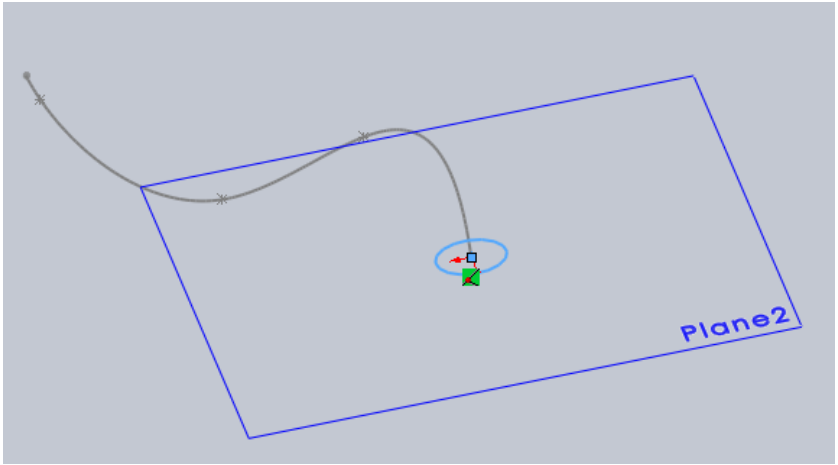



# Example : Swept Boss/Base



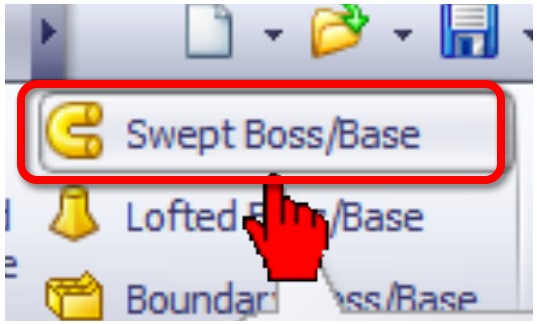

⑥ Draw a profile (CIRCLE)

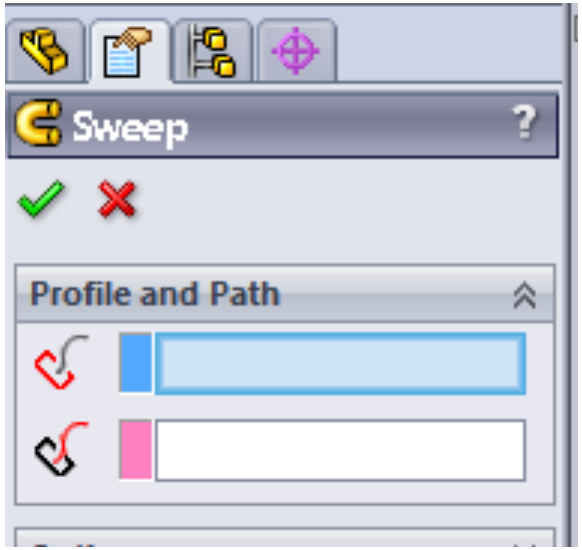



⑦ Select REBUILD

# Example : Swept Boss/Base

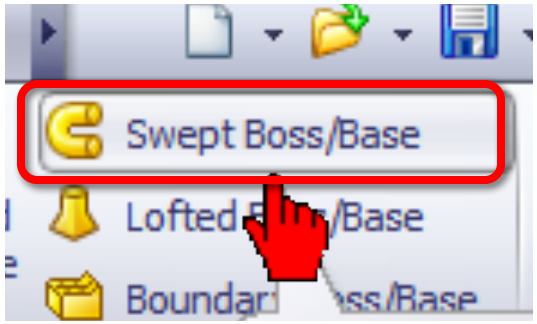
⑧ Select SWEPT BOSS/BASE

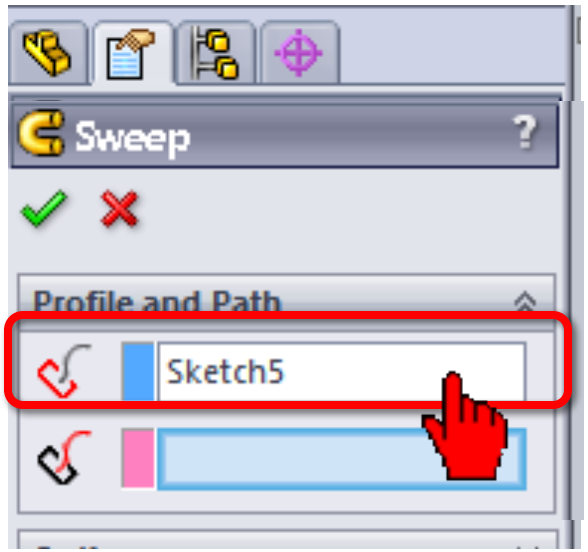


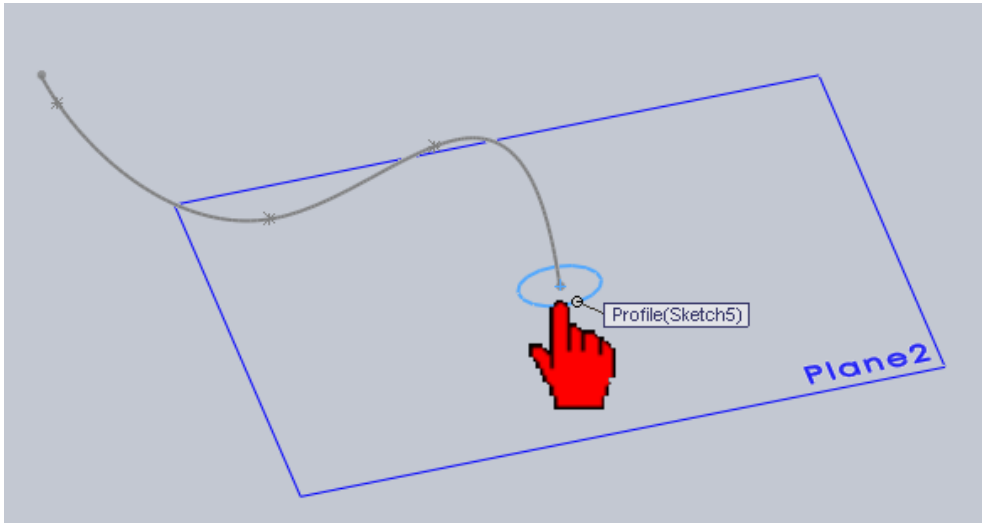
# Example : Swept Boss/Base

⑧ Select SWEPT BOSS/BASE



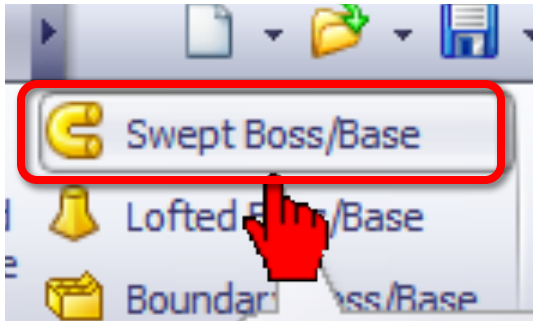



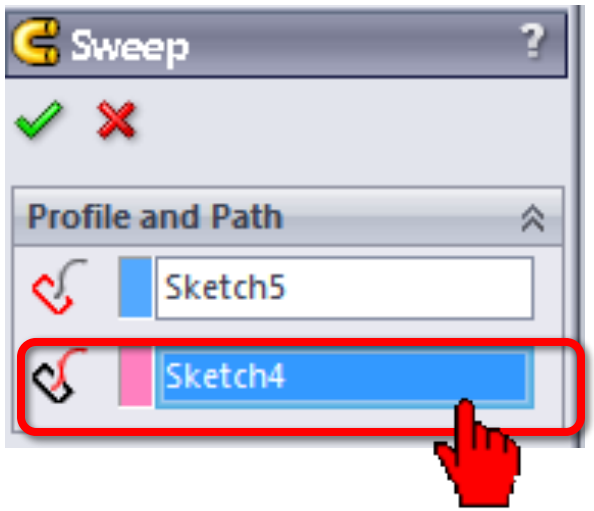

⑨ Select PROFILE



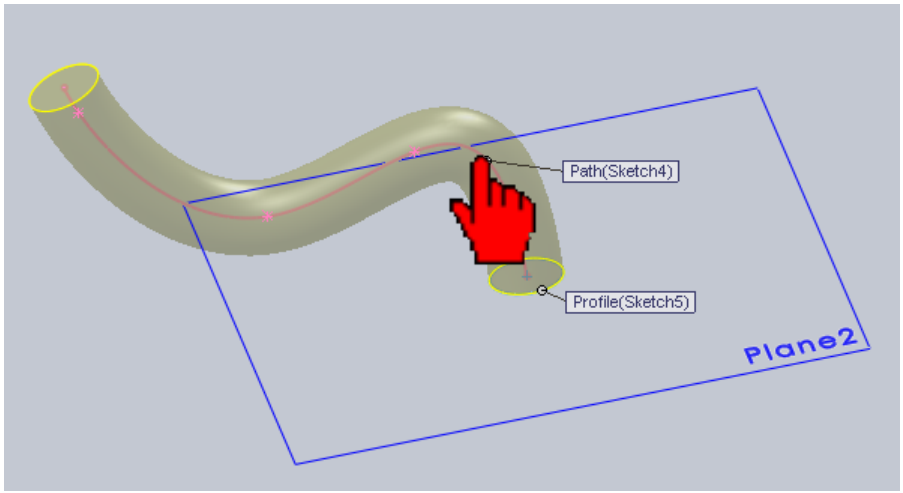
# Example : Swept Boss/Base

⑧ Select SWEPT BOSS/BASE



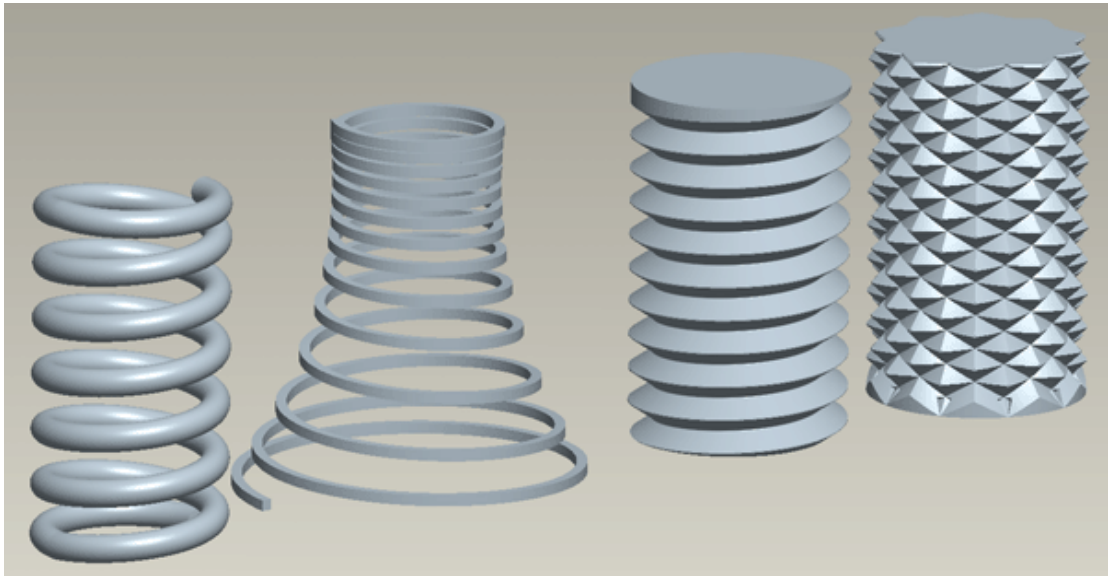

⑩ Select PATH



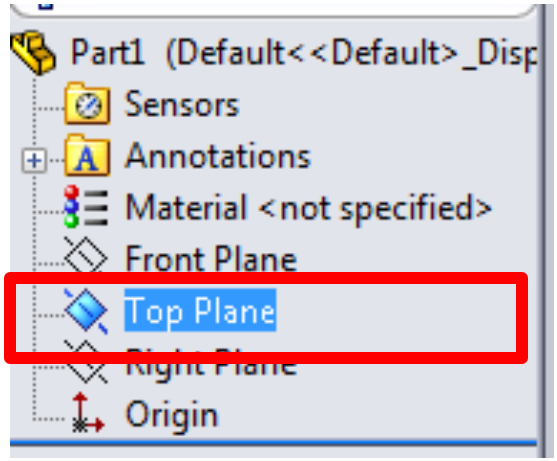
# Helical sweep

As its name implies, the Helical sweep option is useful for creating parts that consists of helical features.

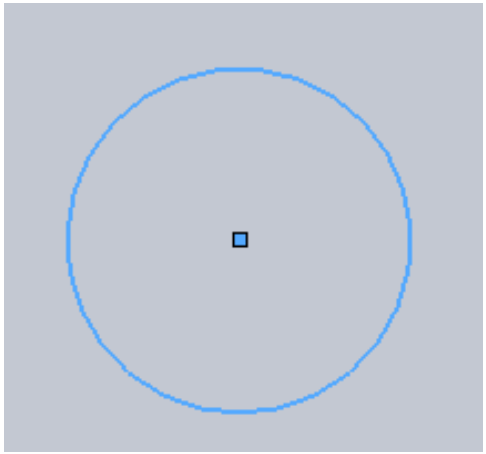
Two features often created with the helical sweep options are spring and threads.



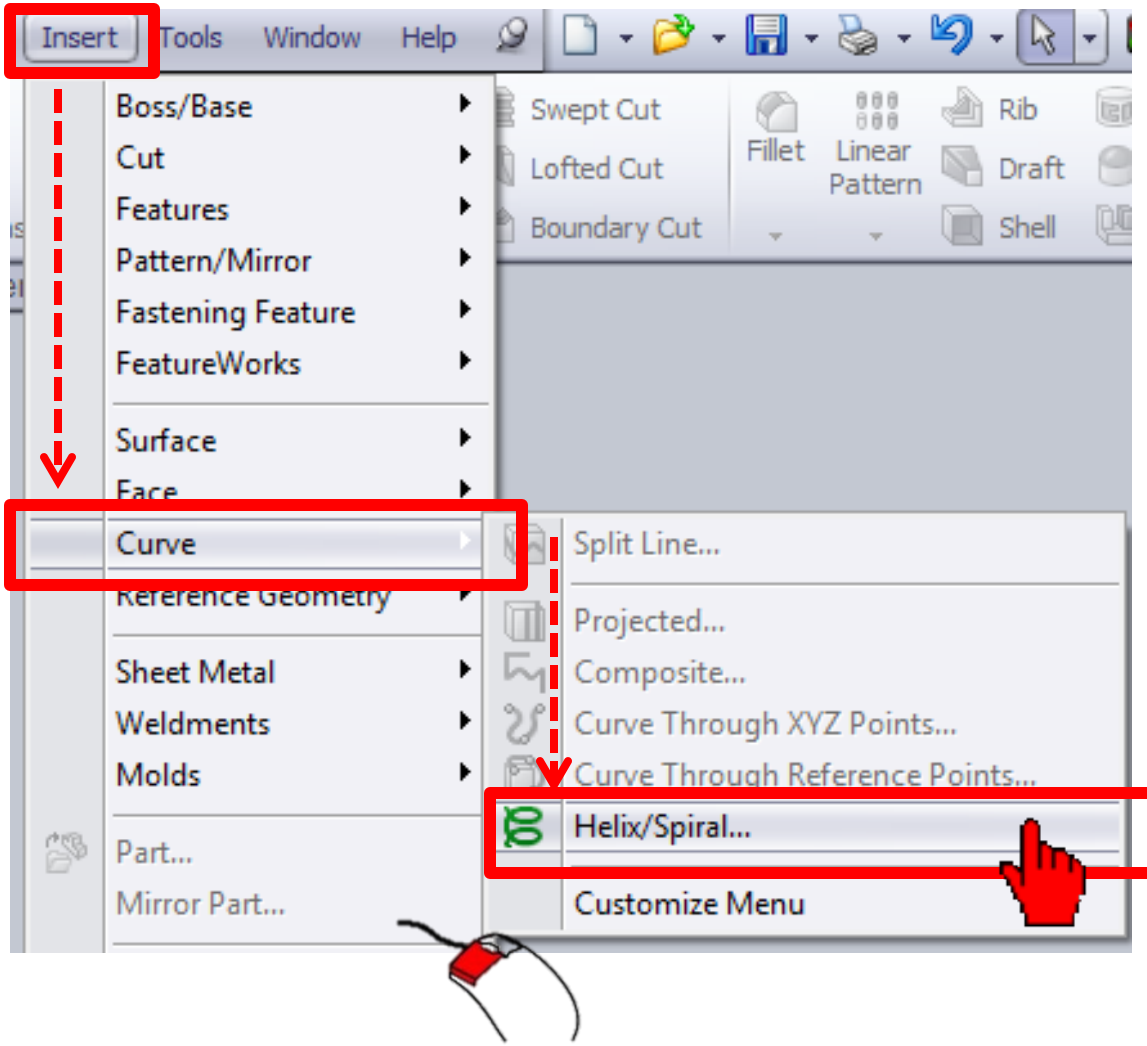
# Helical sweep



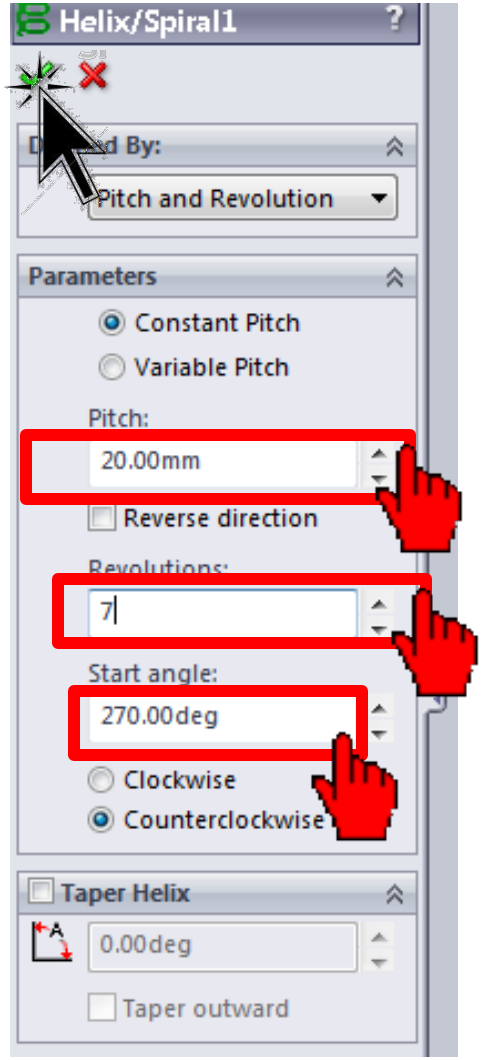

Working plane

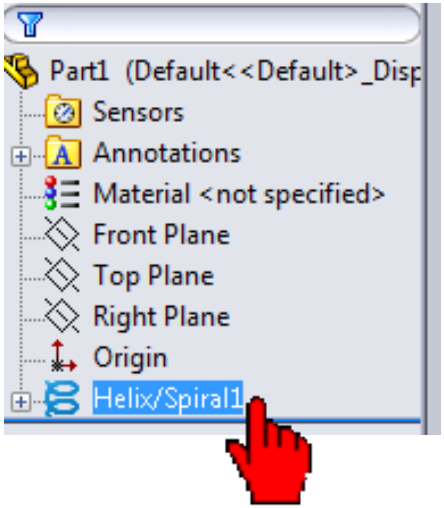


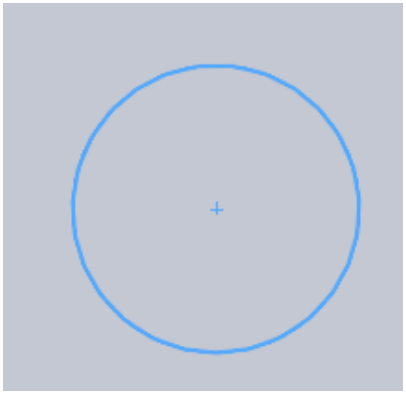
Sketch circle

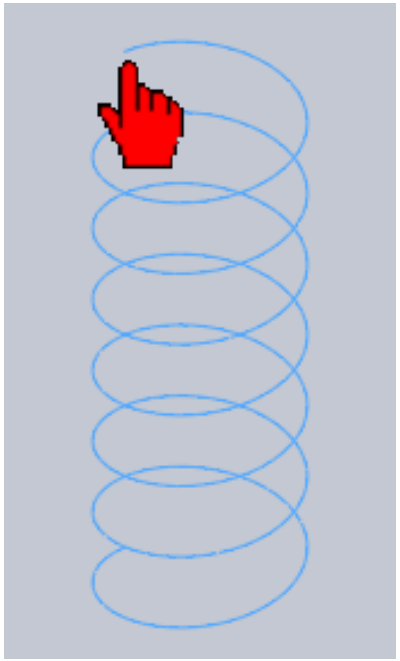


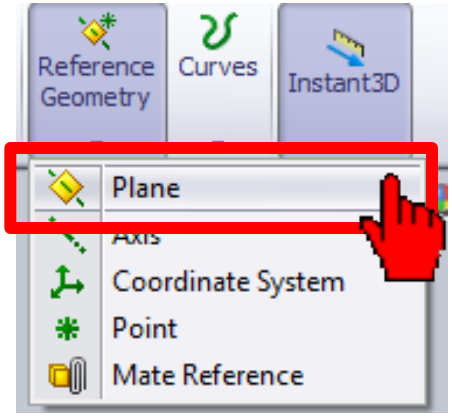
# Helical sweep



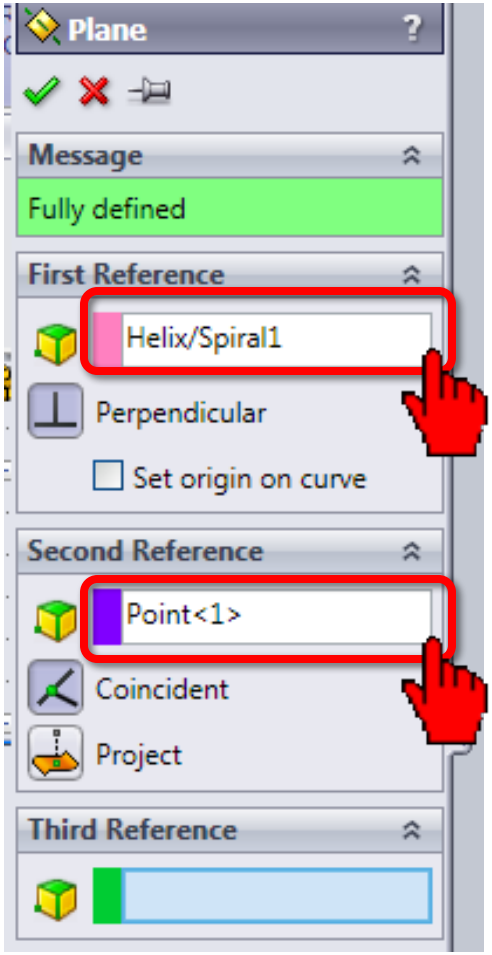



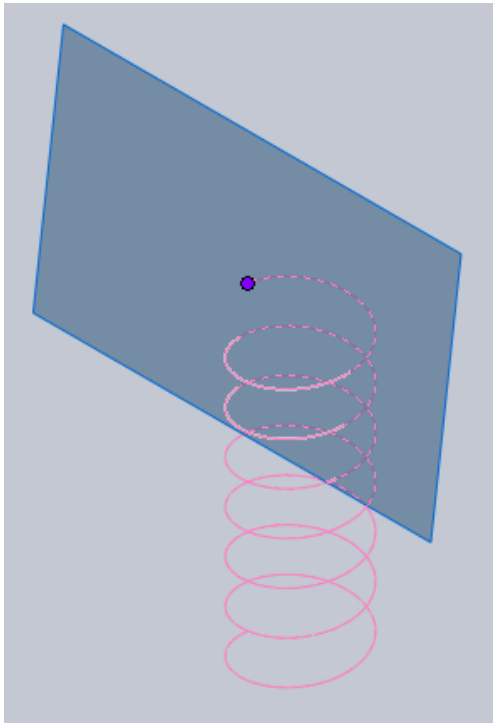


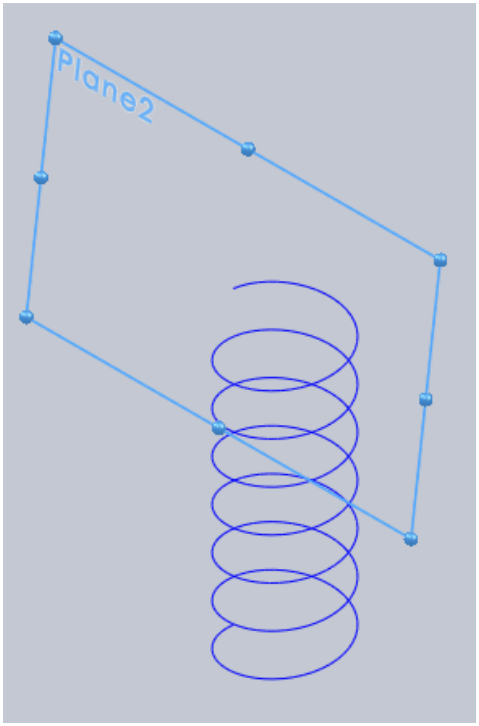



To sketch a circle new plane shall be located in this point

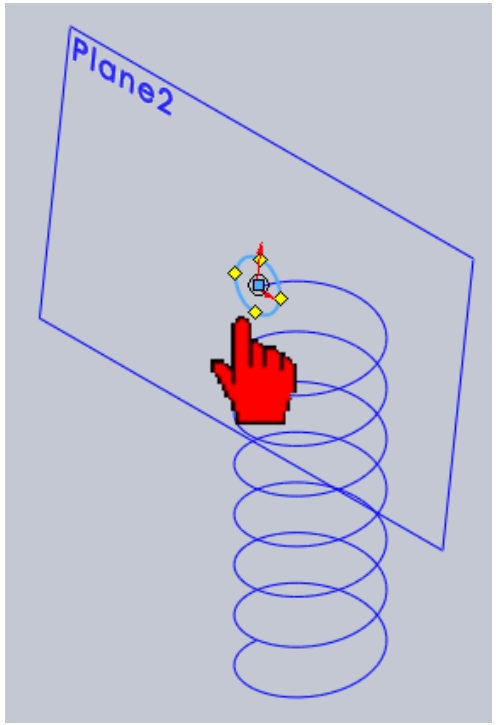
# Helical sweep

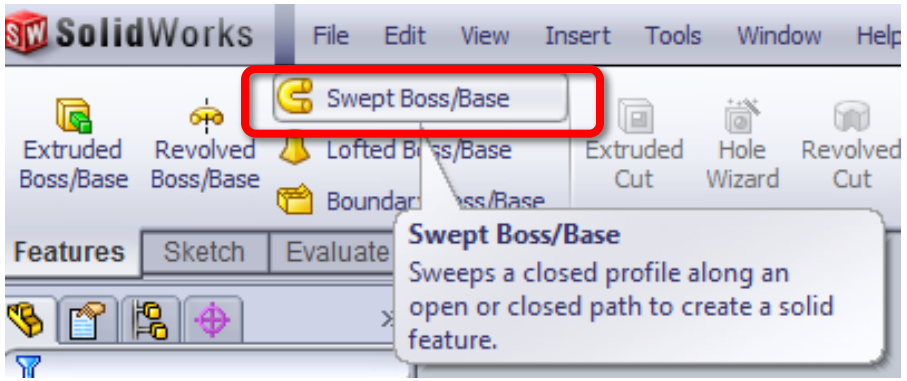


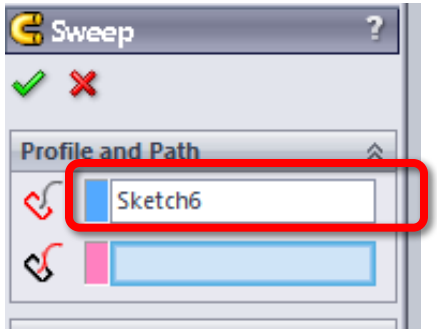



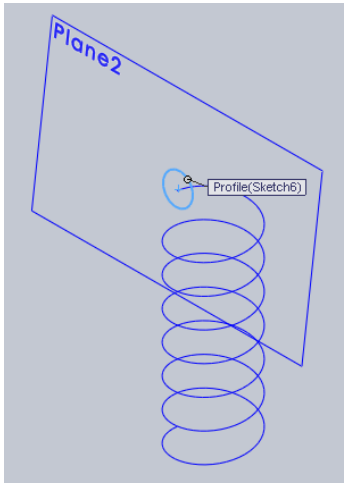



# Helical sweep



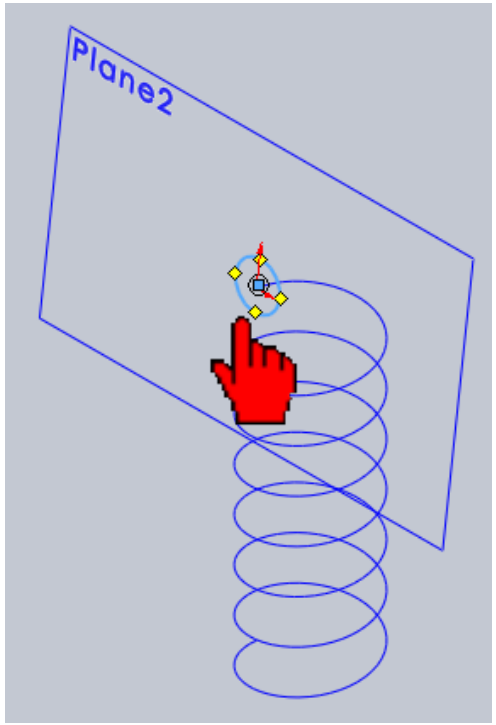



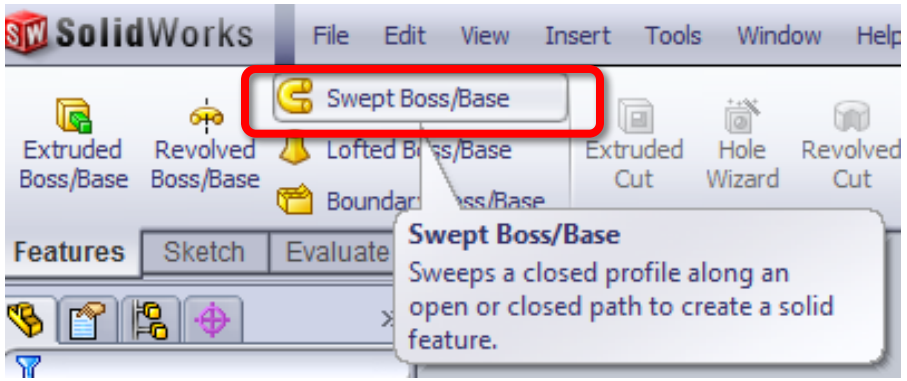



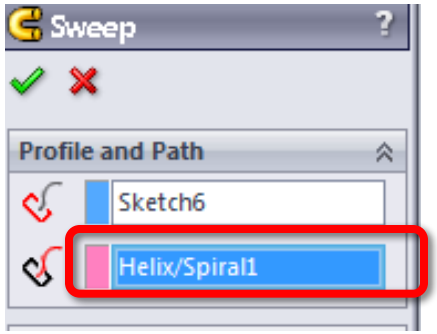

Sketch circle

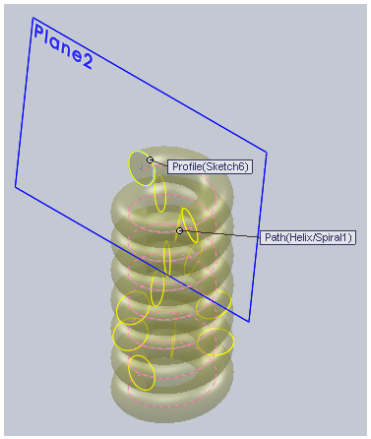
# Helical sweep




Sketch circle



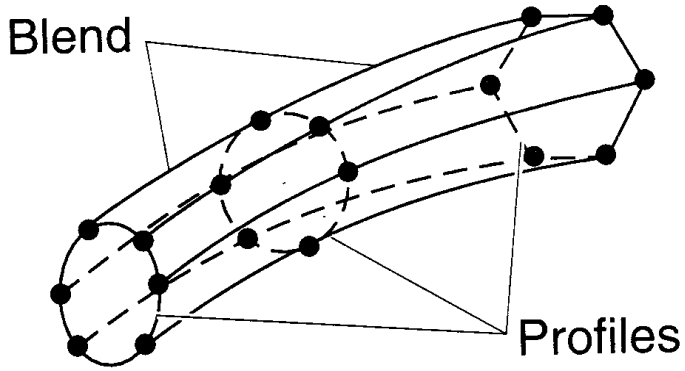



# Blended features

The blend option can also be compared to the extrude option.

Primarily, the blend option creates a feature by protruding along a straight trajectory between two or more user-defined sections.

A partially revolved blend can be created also.

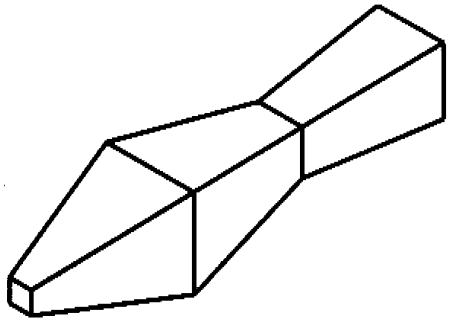



Blend

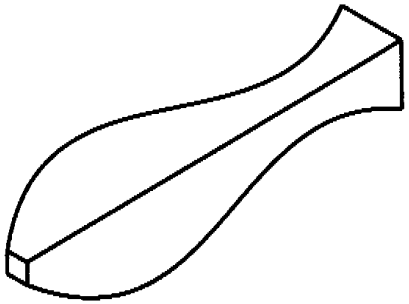
# Blended features

Three types of blends are available:

parallel
rotational
general



STRAIGHT BLEND



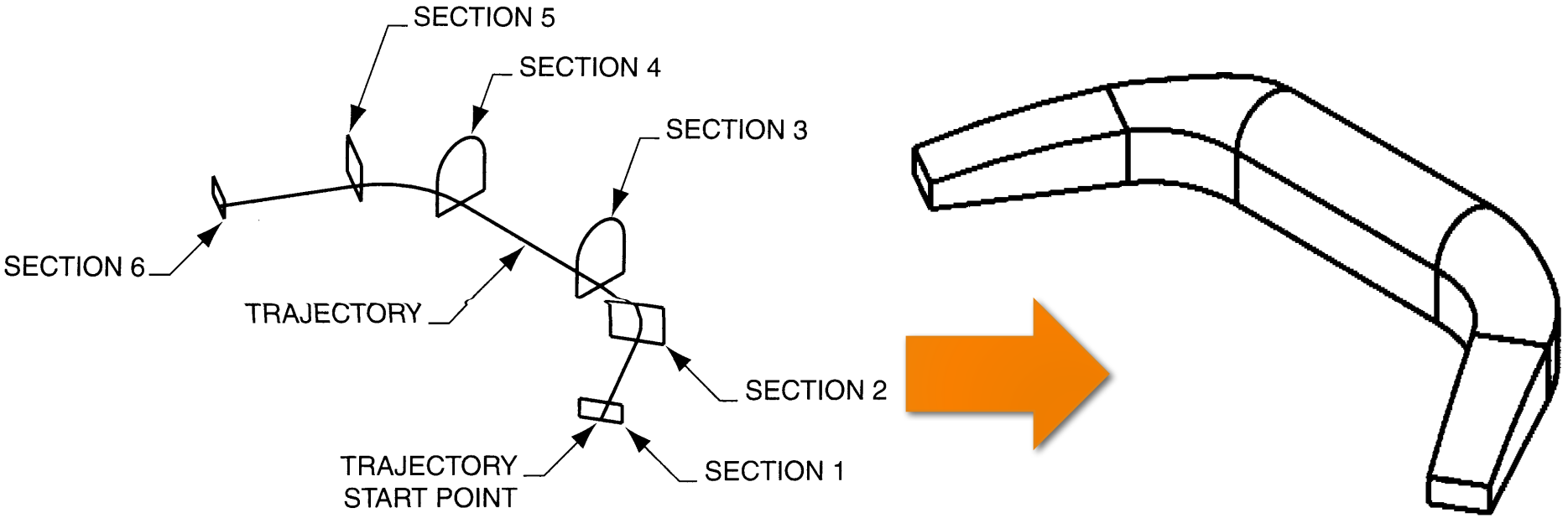
SMOOTH BLEND

# Swept Blend

A swept blend is a combination of a sweep and a blend.

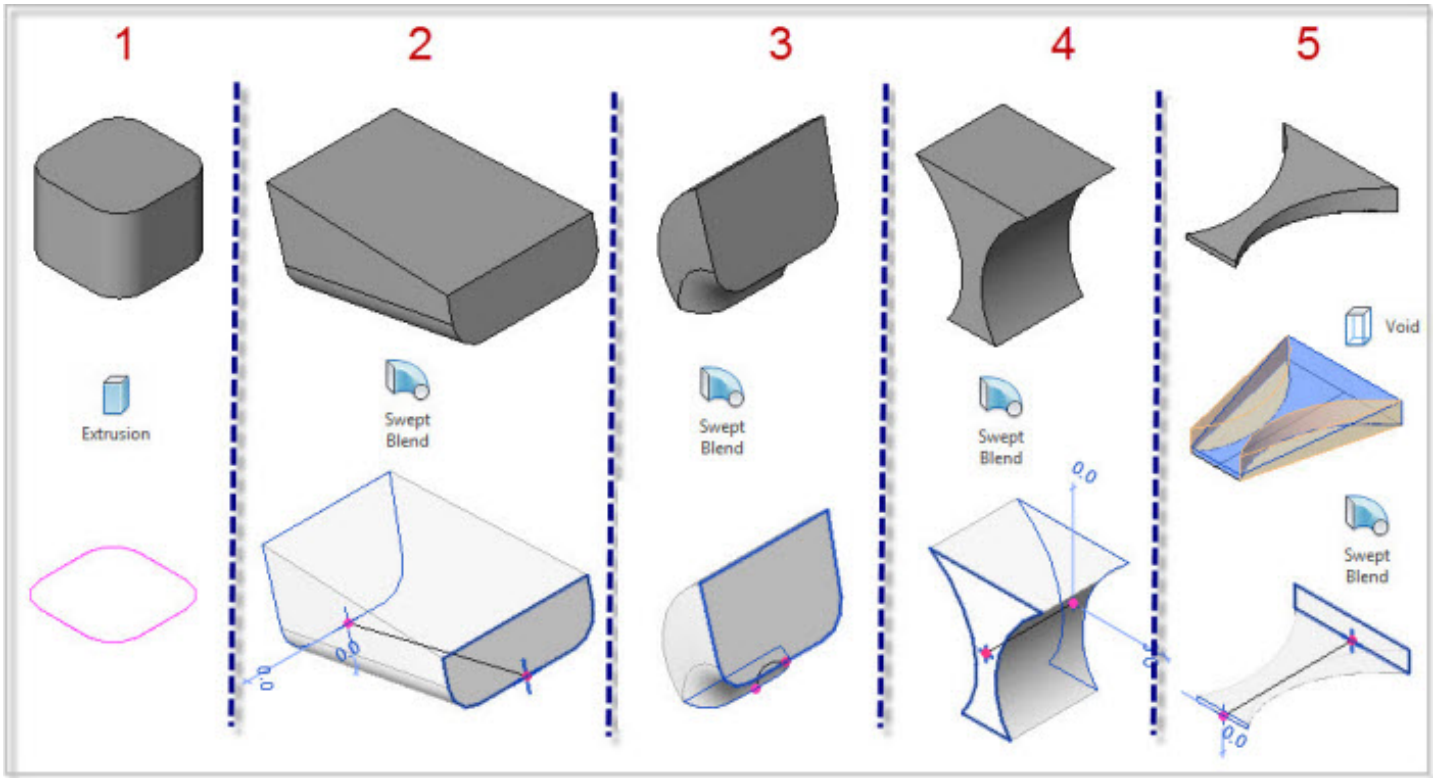
A swept feature is a section protruded along a defined trajectory.

This trajectory can be either sketched or selected.



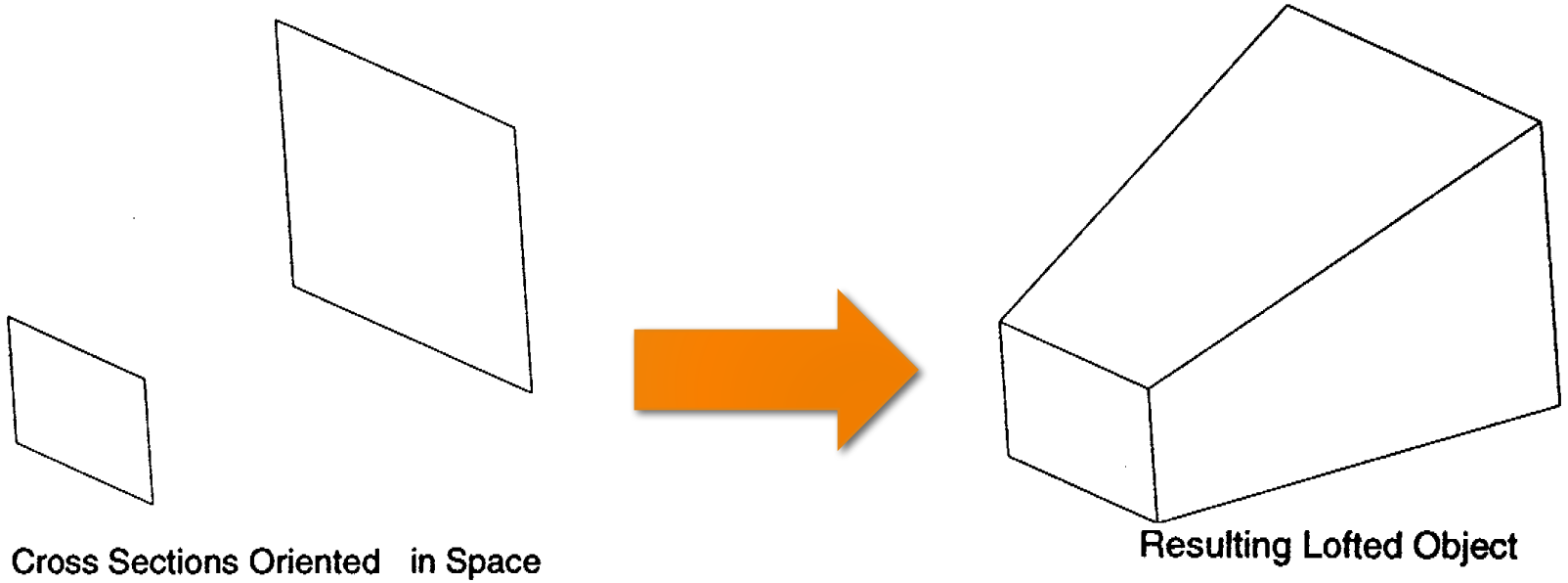
# Swept Blend

A parallel blended feature is a feature protruded along a straight trajectory between two or more user-defined sections.

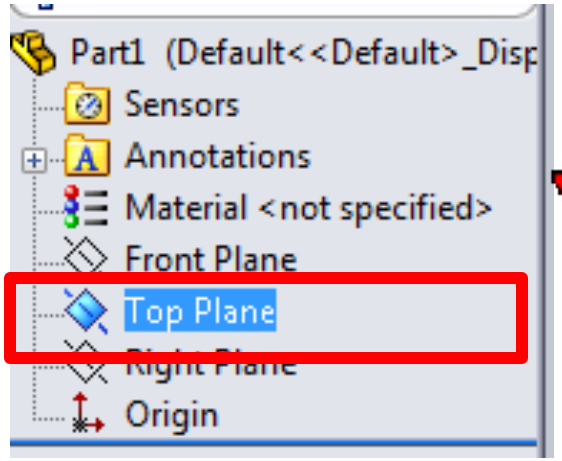


# Lofting

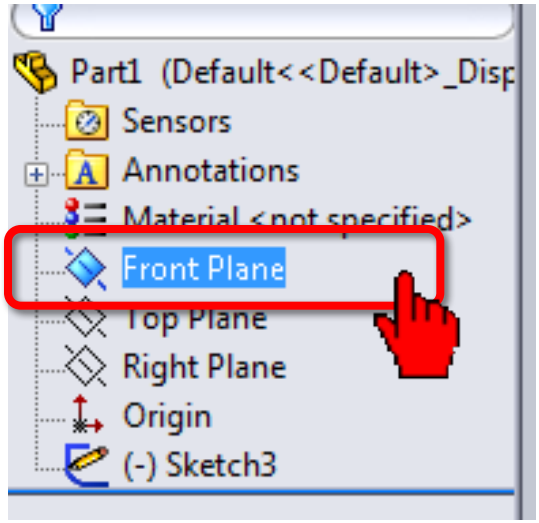
Lofting is typically used to create objects of varying cross section.
Lofting could also be used to create objects of constant cross section in which the cross section is copied and oriented in space along something other than a straight-line path.

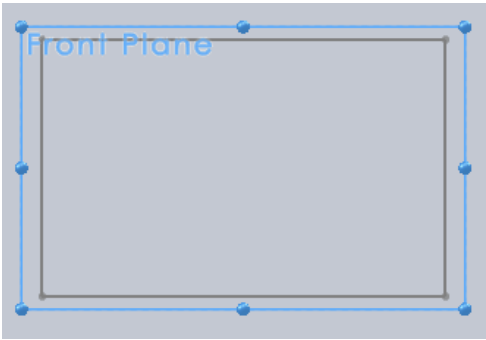


# Lofting

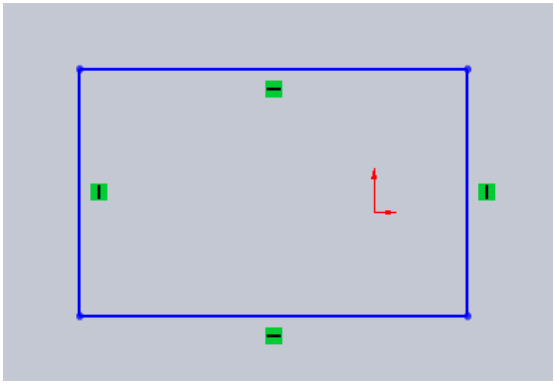



Working plane



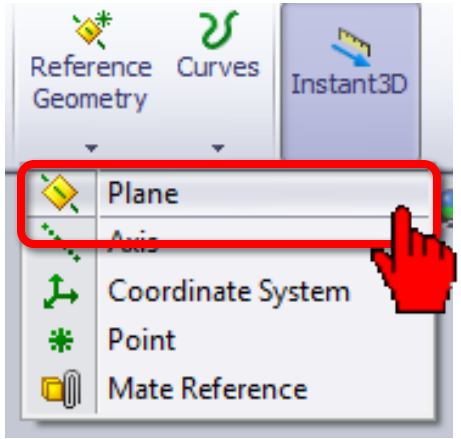




Select plane

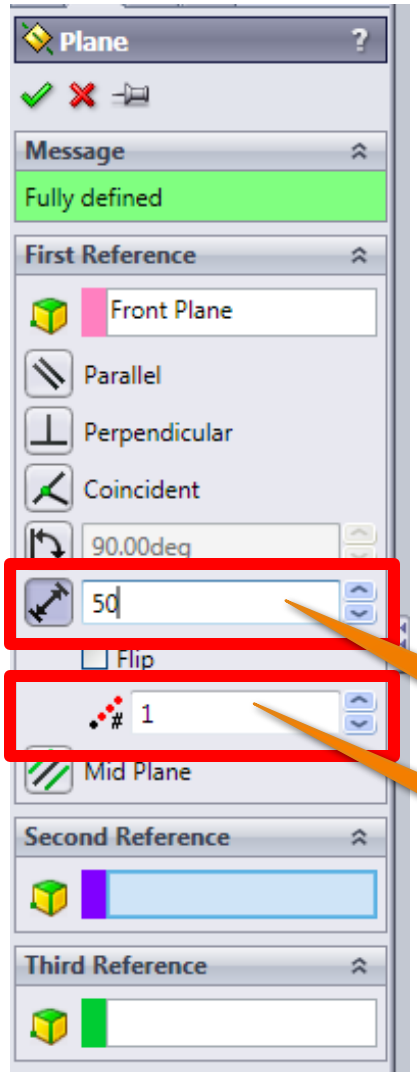


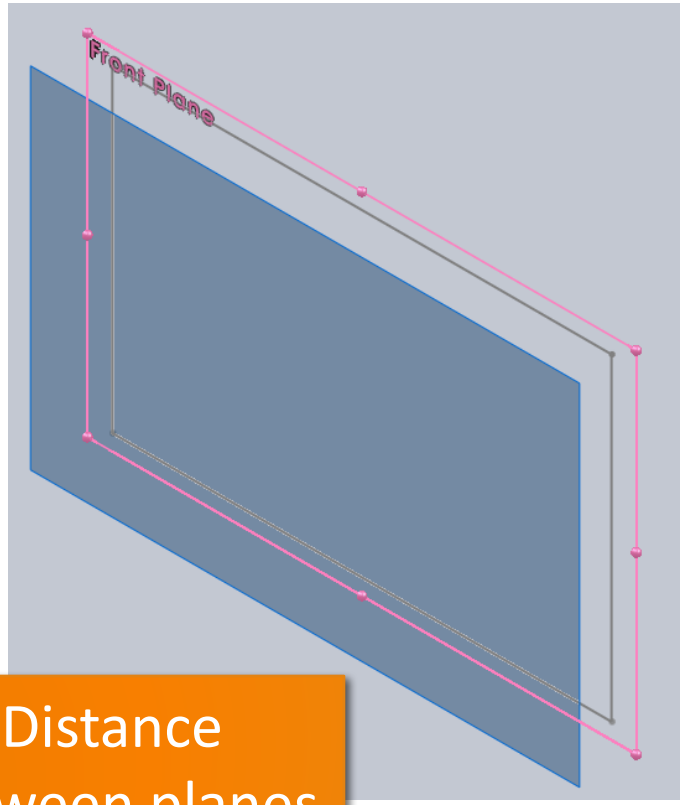
Sketch rectangle

# Lofting




To build a new working plane

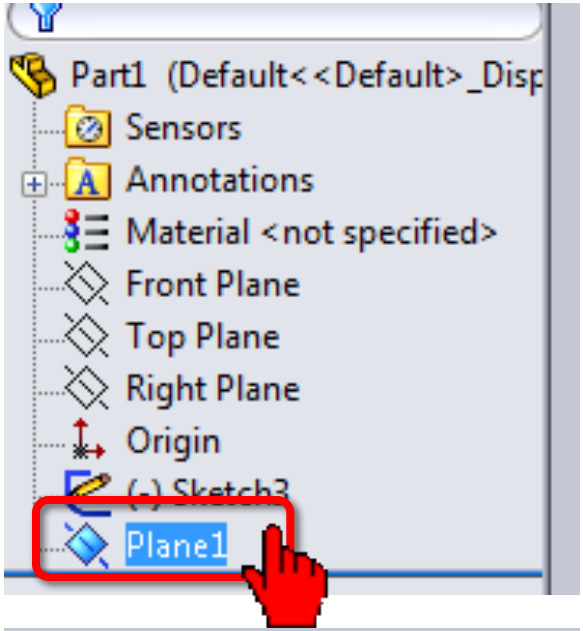


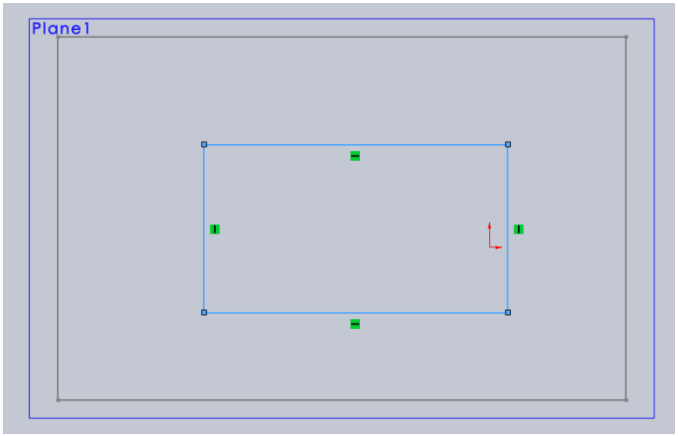




Distance between planes

Number of new plane

# Lofting

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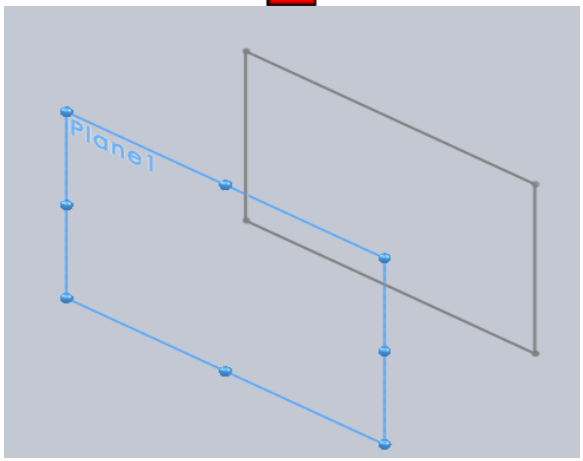
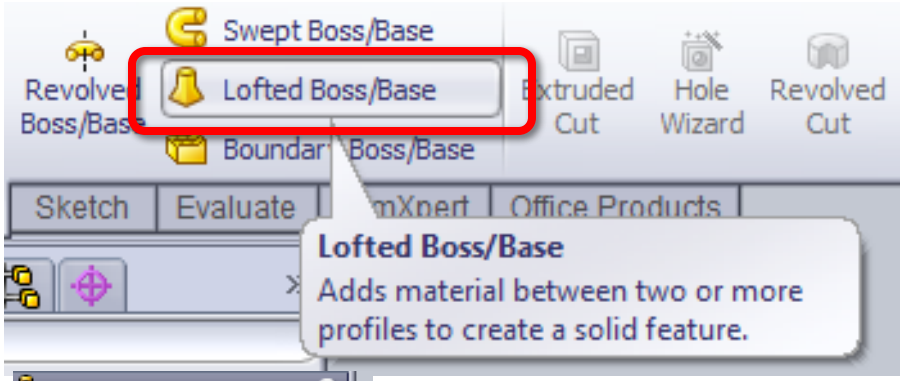
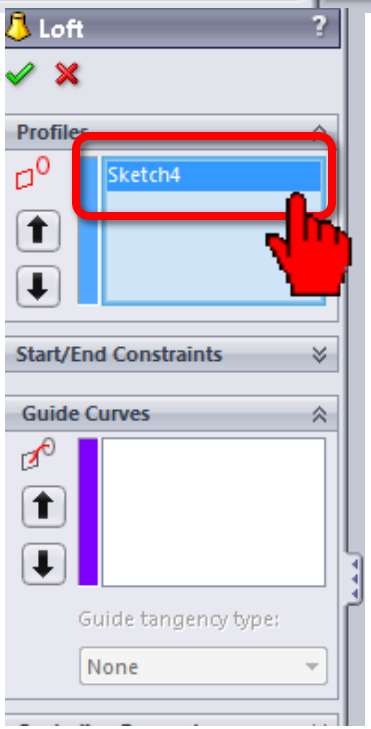
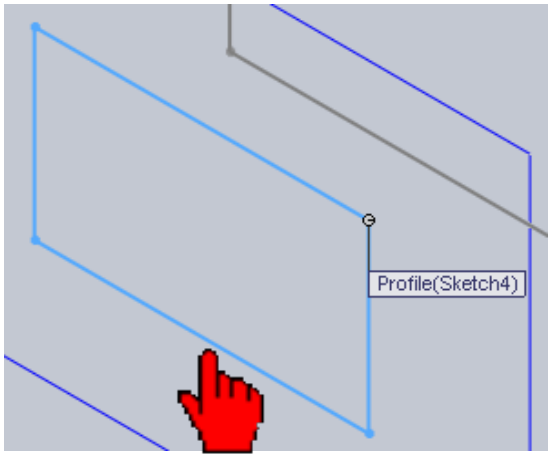
Sketch second profile



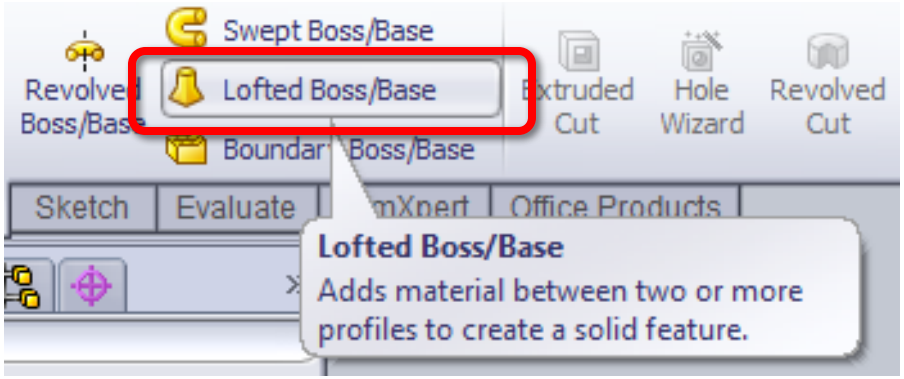
# Lofting

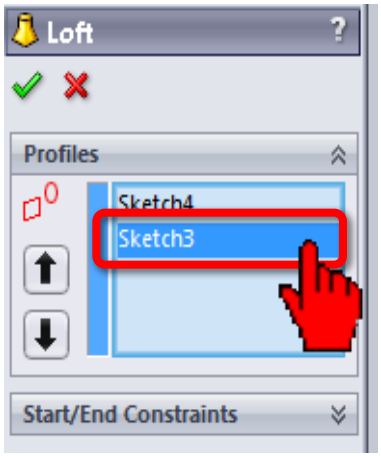

Swept Boss/Base
Revolved Boss/Base
Lofted Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Boundary Boss/Base
Sketch
Evaluate
Office Products
Lofted Boss/Base
Adds material between two or more profiles to create a solid feature.

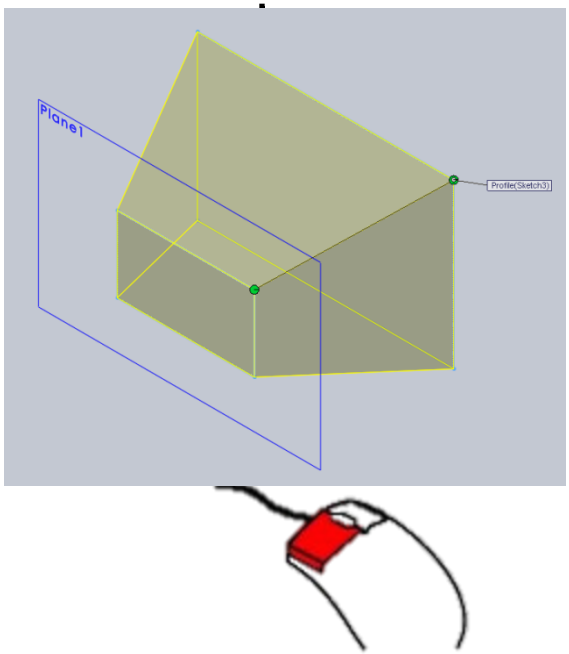

Loft
Profiles
Sketch4
Start/End Constraints
Guide Curves
Guide tangency type:
None


Profile(Sketch4)

# Lofting


Swept Boss/Base
Revolved Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Sketch
Evaluate
Office Products
Lofted Boss/Base
Adds material between two or more profiles to create a solid feature.


Loft
Profiles
Sketch4
Sketch3
Start/End Constraints


Plane1
Profile(Sketch3)

# The following 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

# Assignment #7

Page

Figure

You will generate the solid model

Submit the assignment on time

Upload file into NINOVA