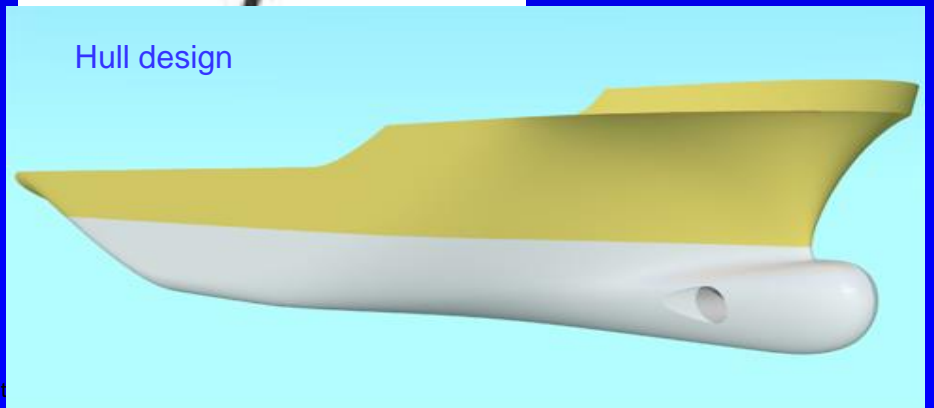


Spline



SolidWorks Sketch Commands

Drawing tools

Modifying tools

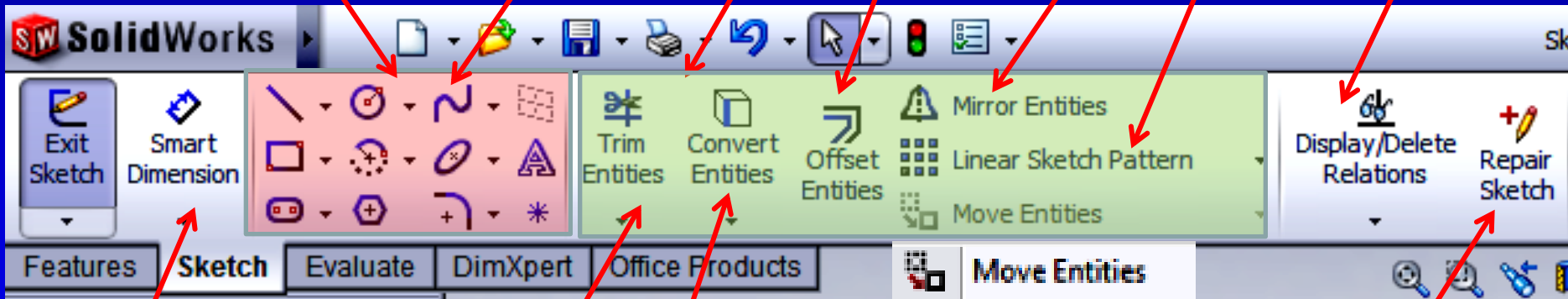
Rectangular and Polar (circular) array

Spline

Offset

Mirror

Constraints



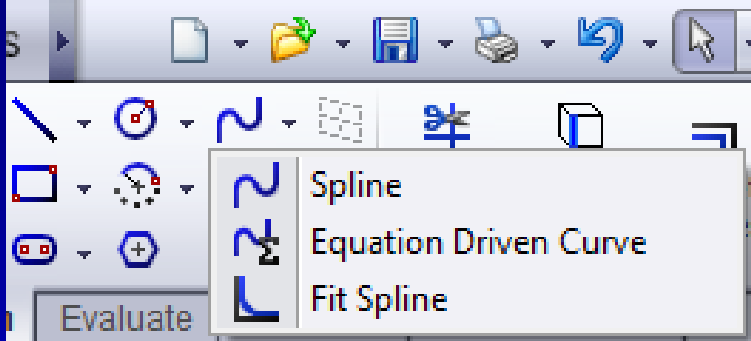
Dimensioning tool

Trim and Extend

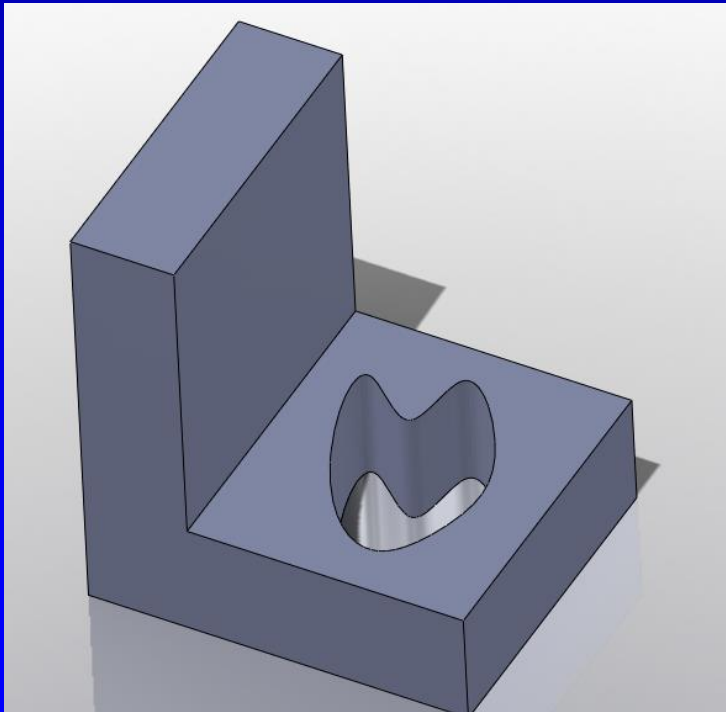
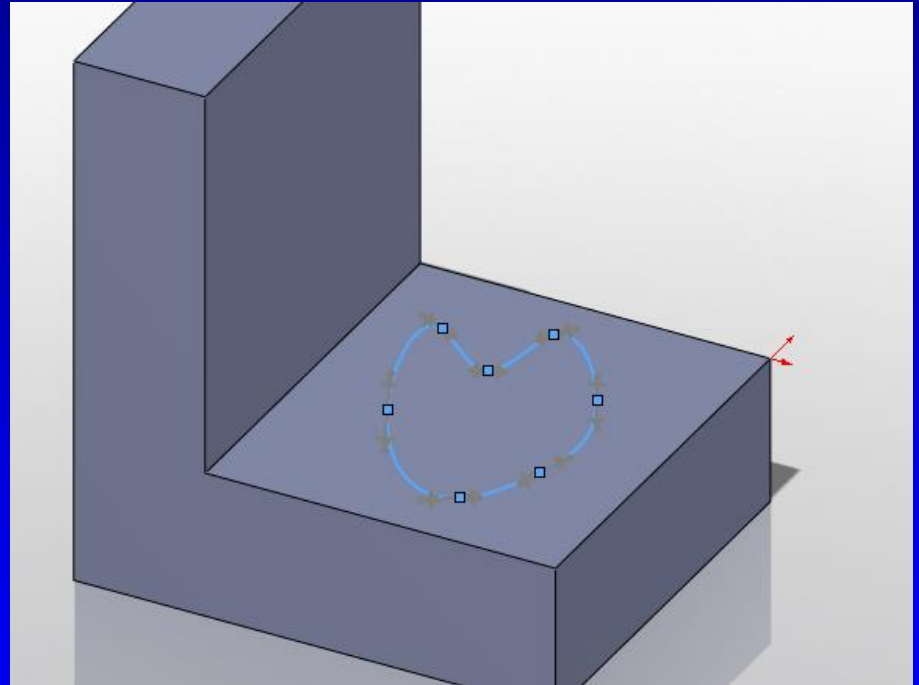
Project entities

Checks sketch for problems

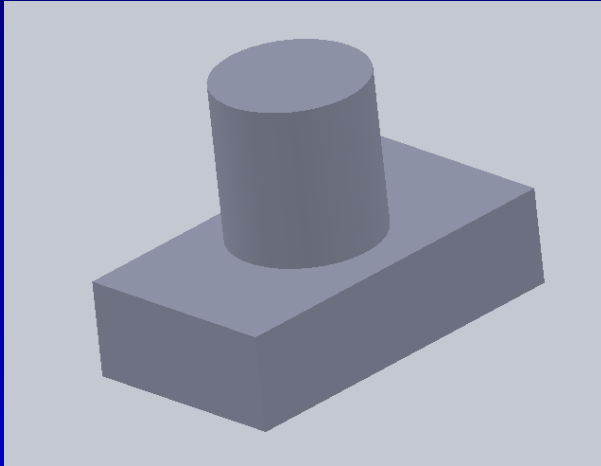
Spline



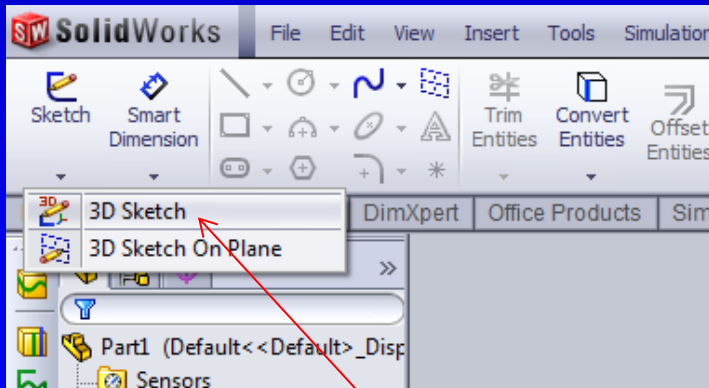
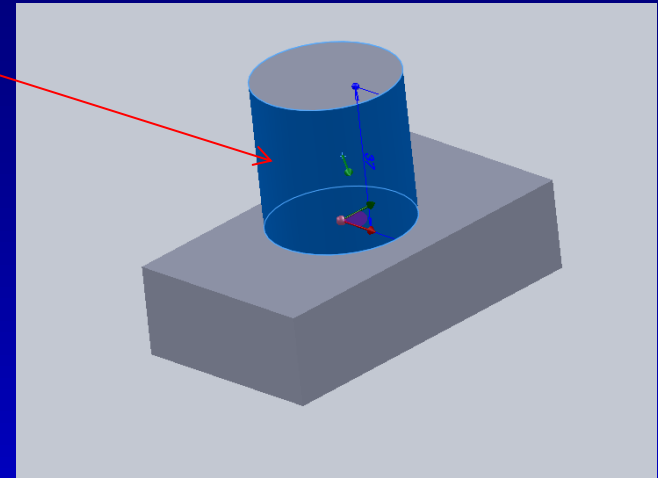
A smooth curve is fitted through selected points



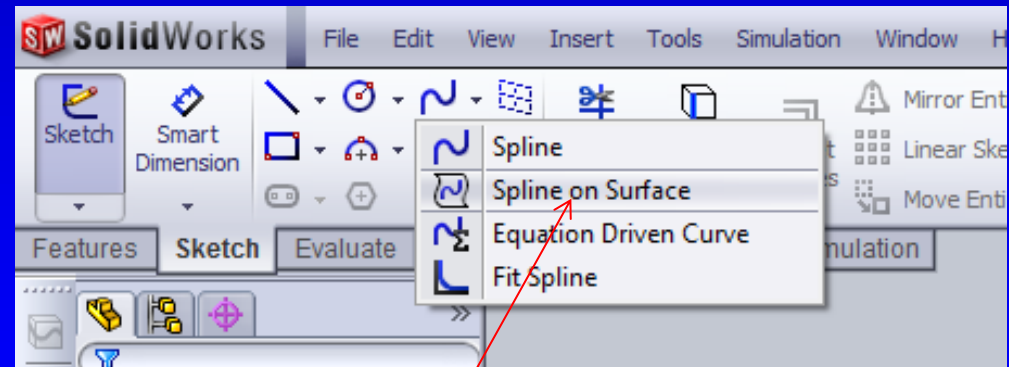
Spline on a curved surface



Select the surface to draw

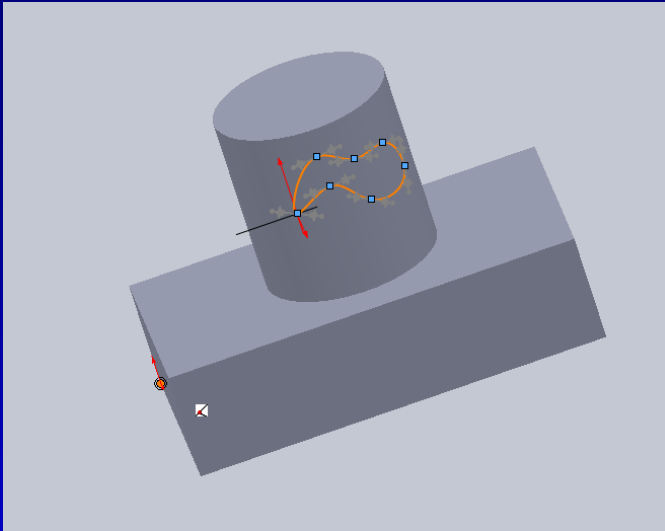


Select the **3D Sketch** option

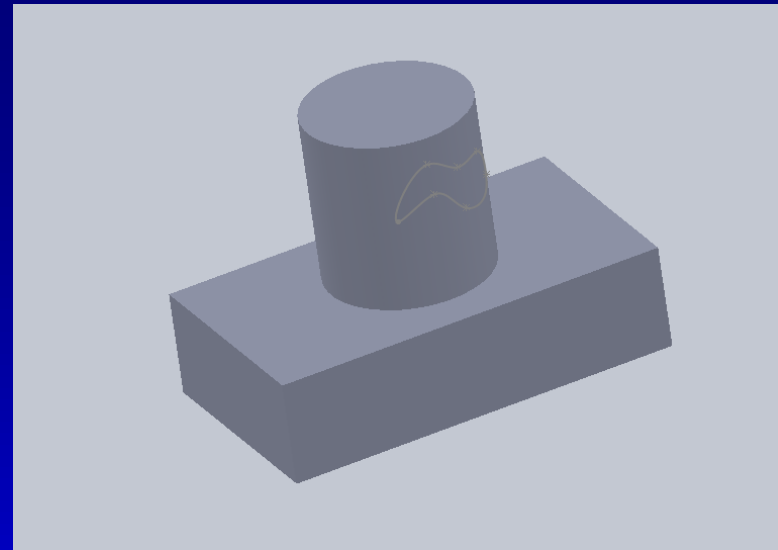


Select the **Spline on Surface** option

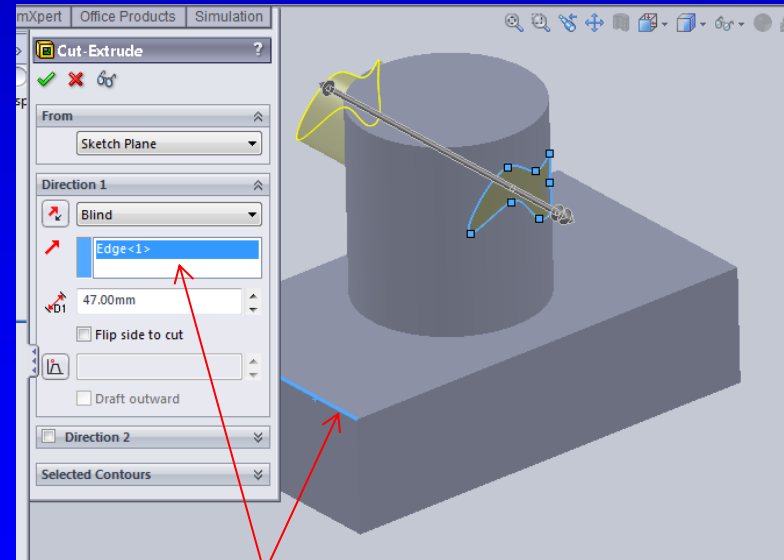
Draw the curve



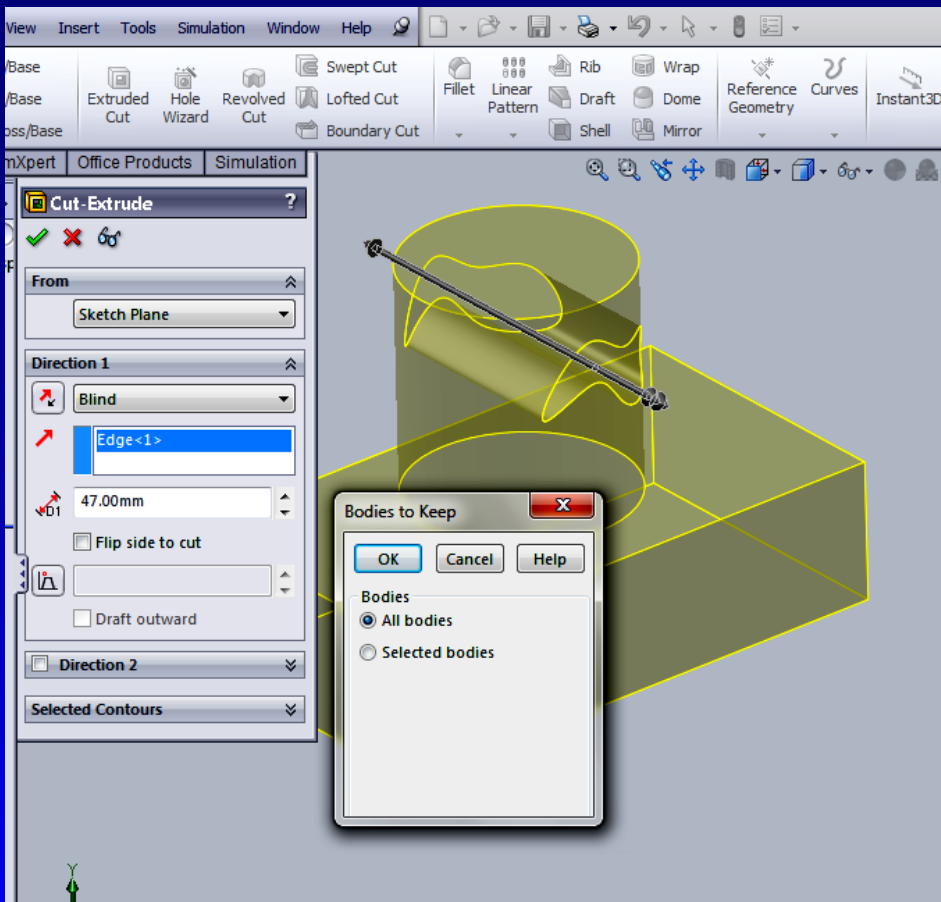
Exit the sketch



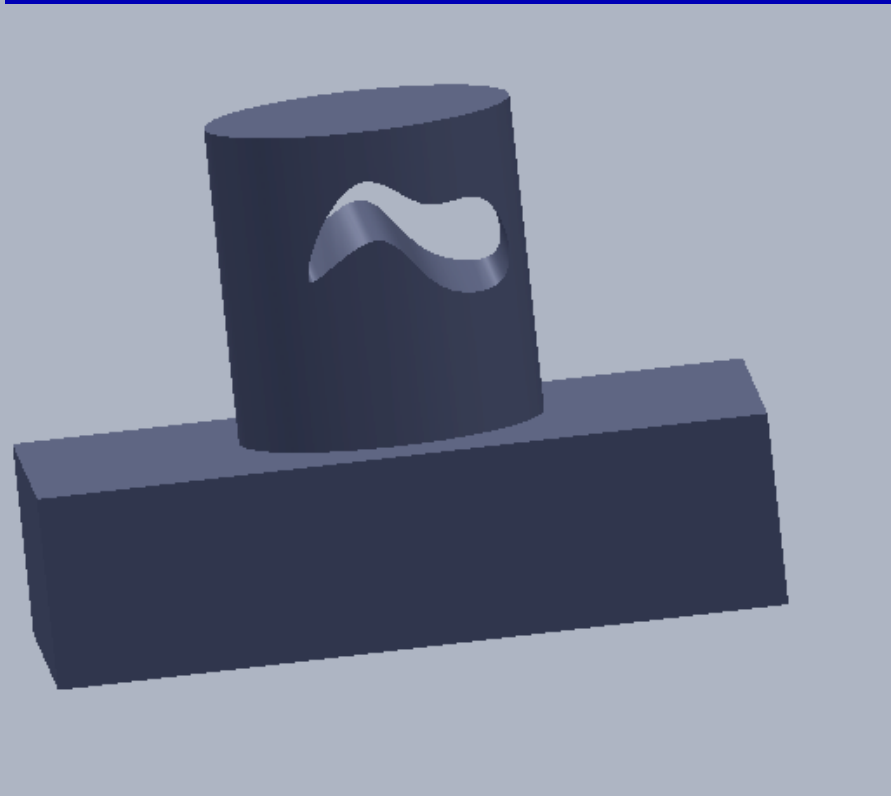
Select **Extrude Cut** option ,
choose the spline curve and
the direction for extrusion



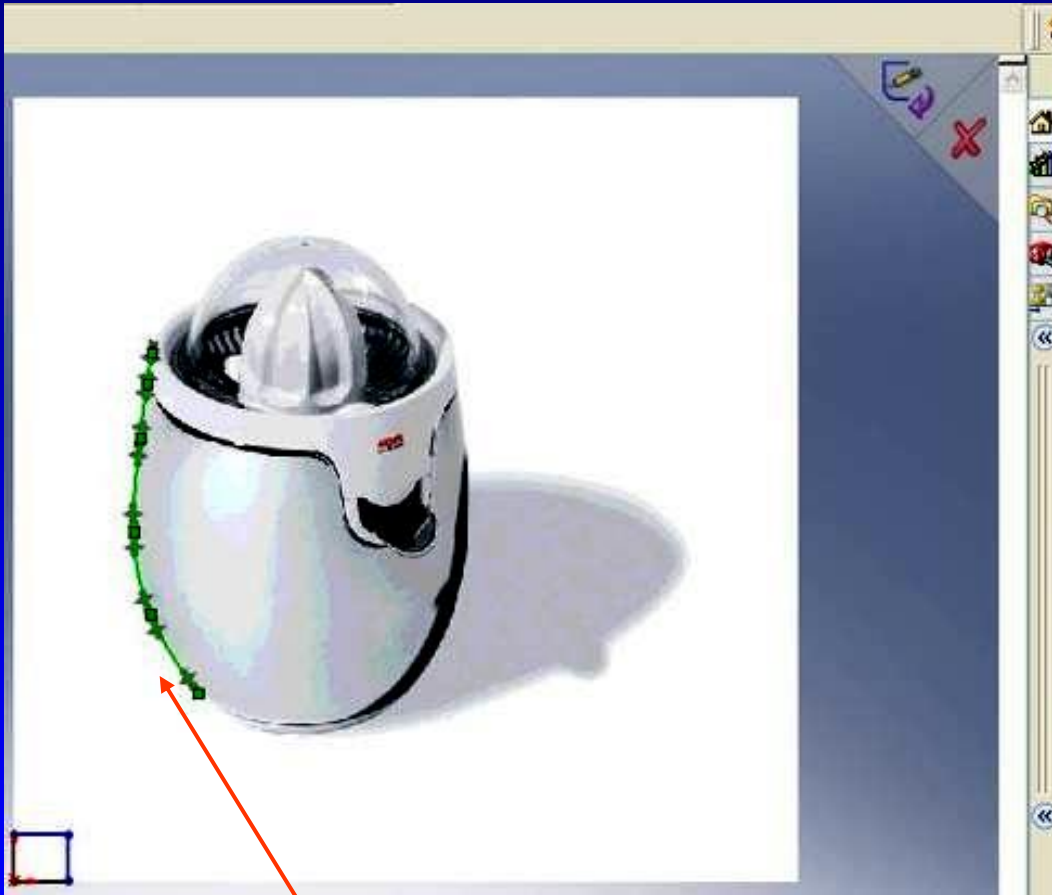
Direction



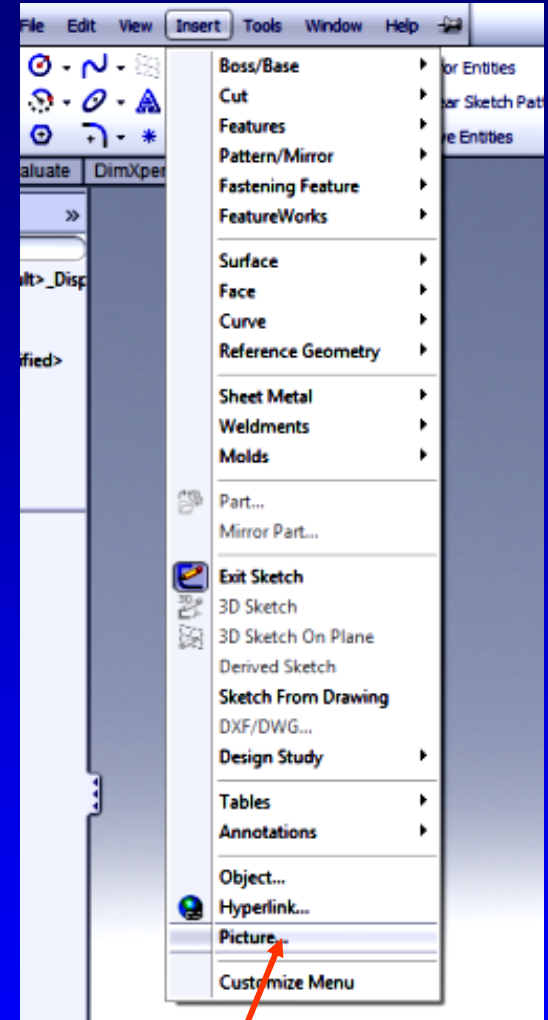
Select body to keep



SolidWorks Sketch Commands

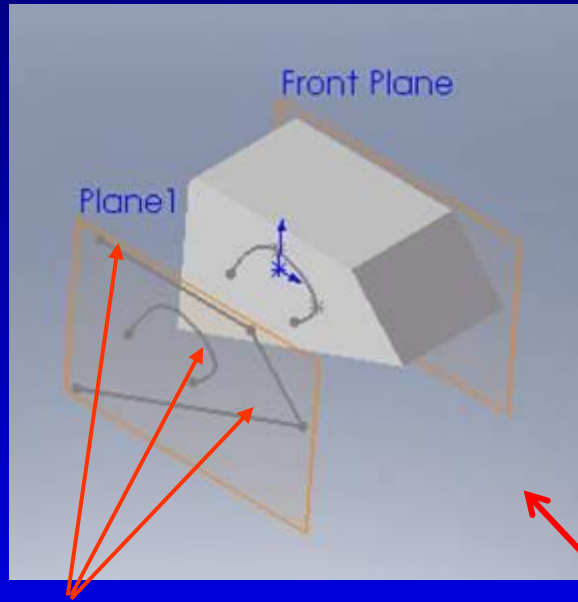


Use **Spline** to trace the curve

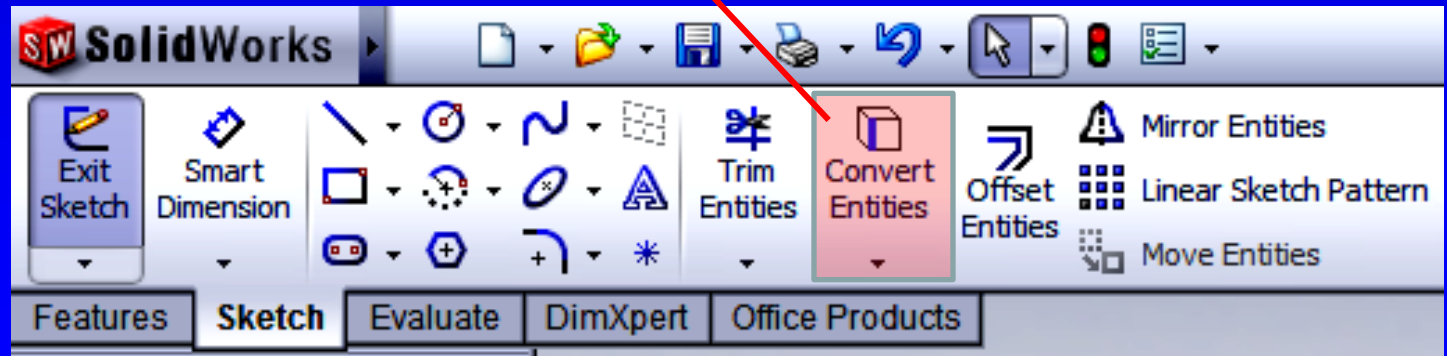


Importing an image into the sketch environment

Projecting Entities onto a Sketch plane



Projected entities



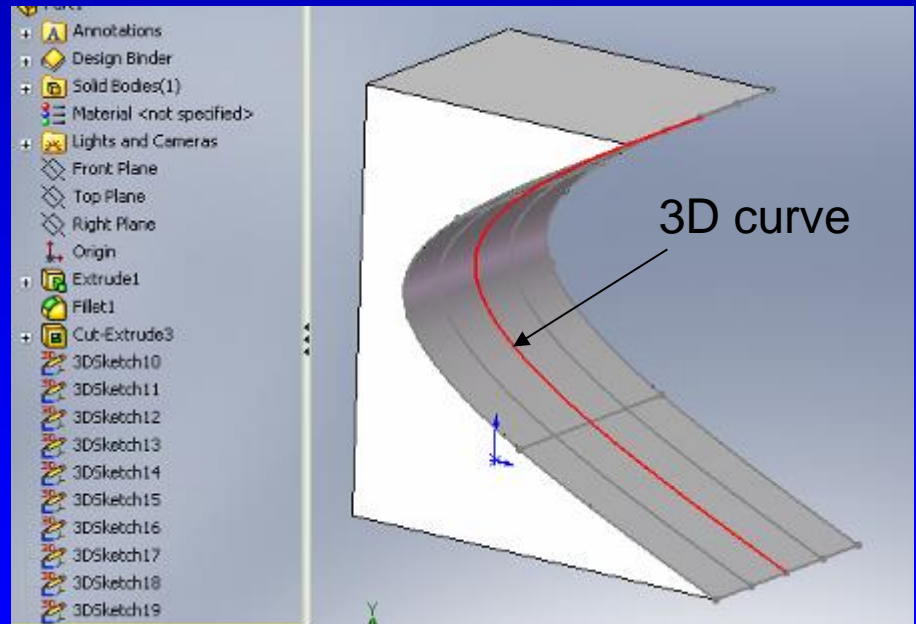
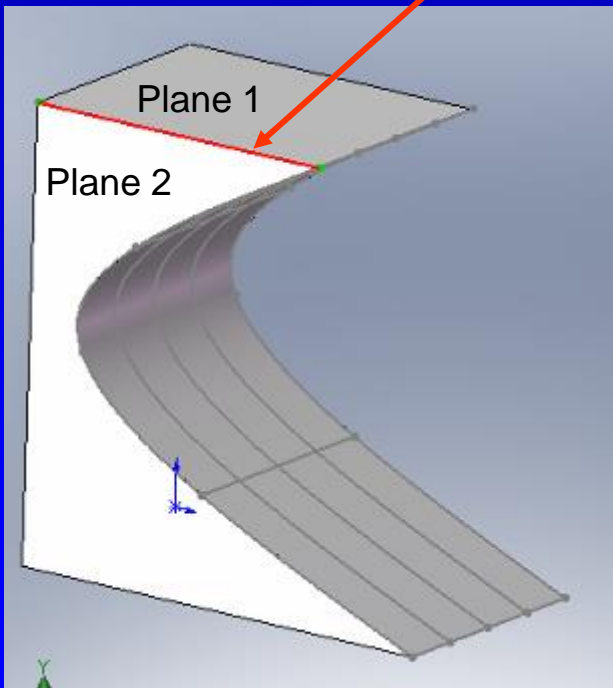
SolidWorks Sketch Commands



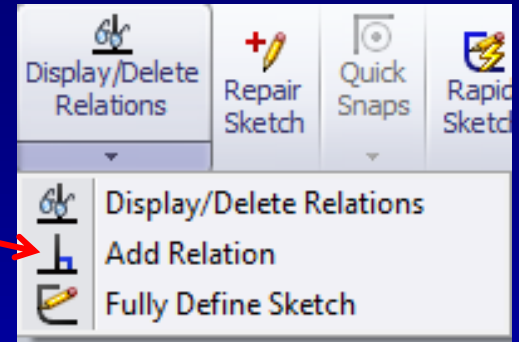
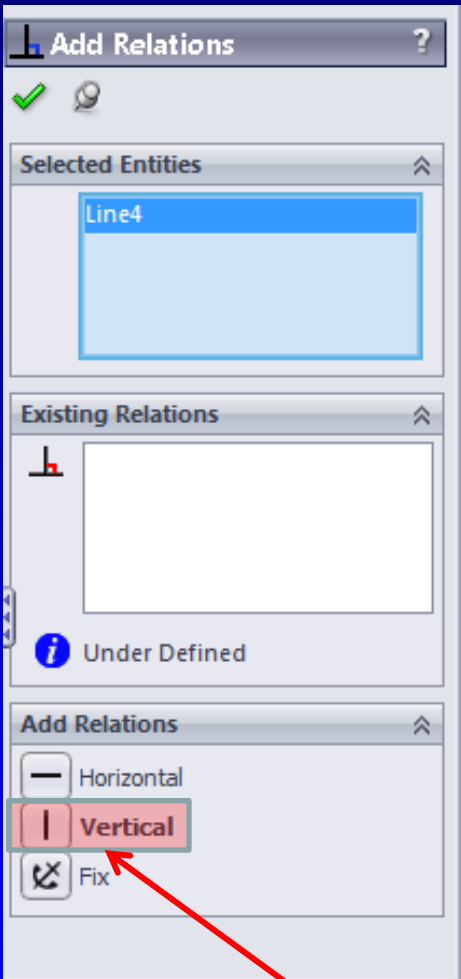
Locates the intersection of two planes and turns it into a sketch curve



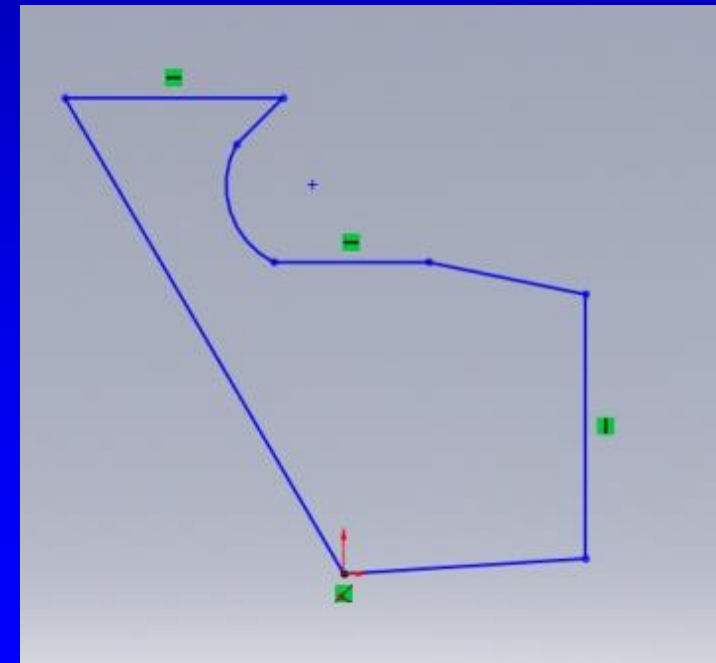
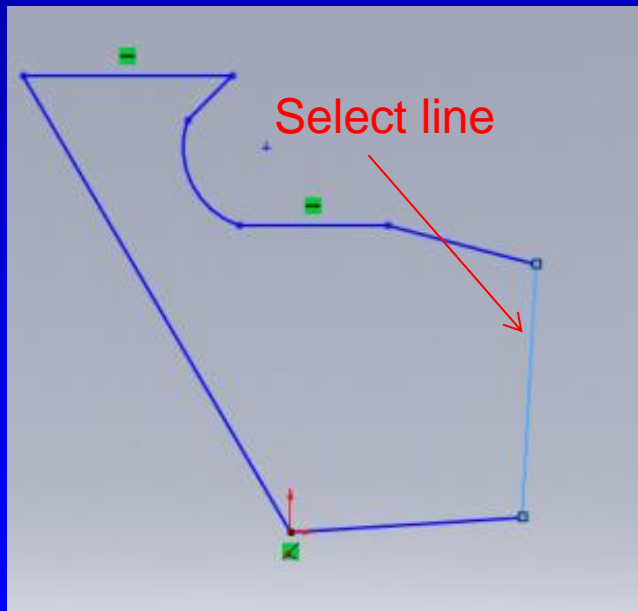
Allows you to extract a curve for other applications, **not active in sketch mode.**



Constraints



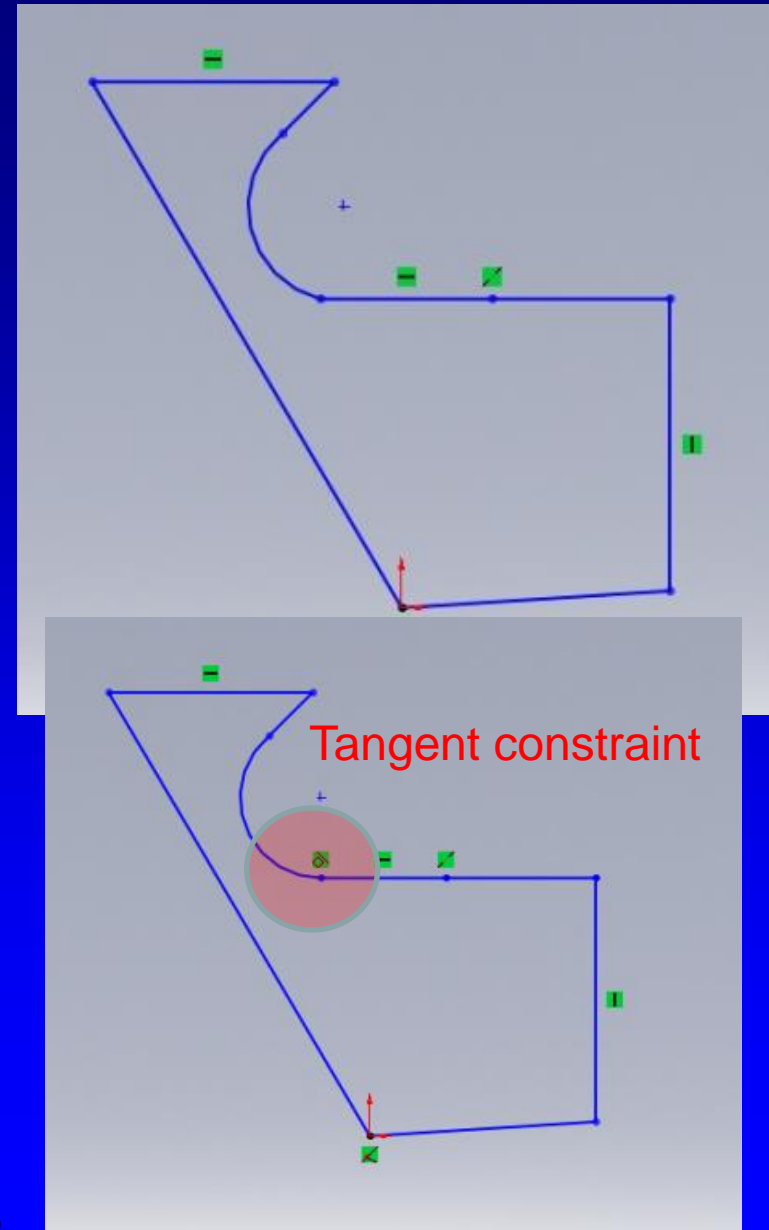
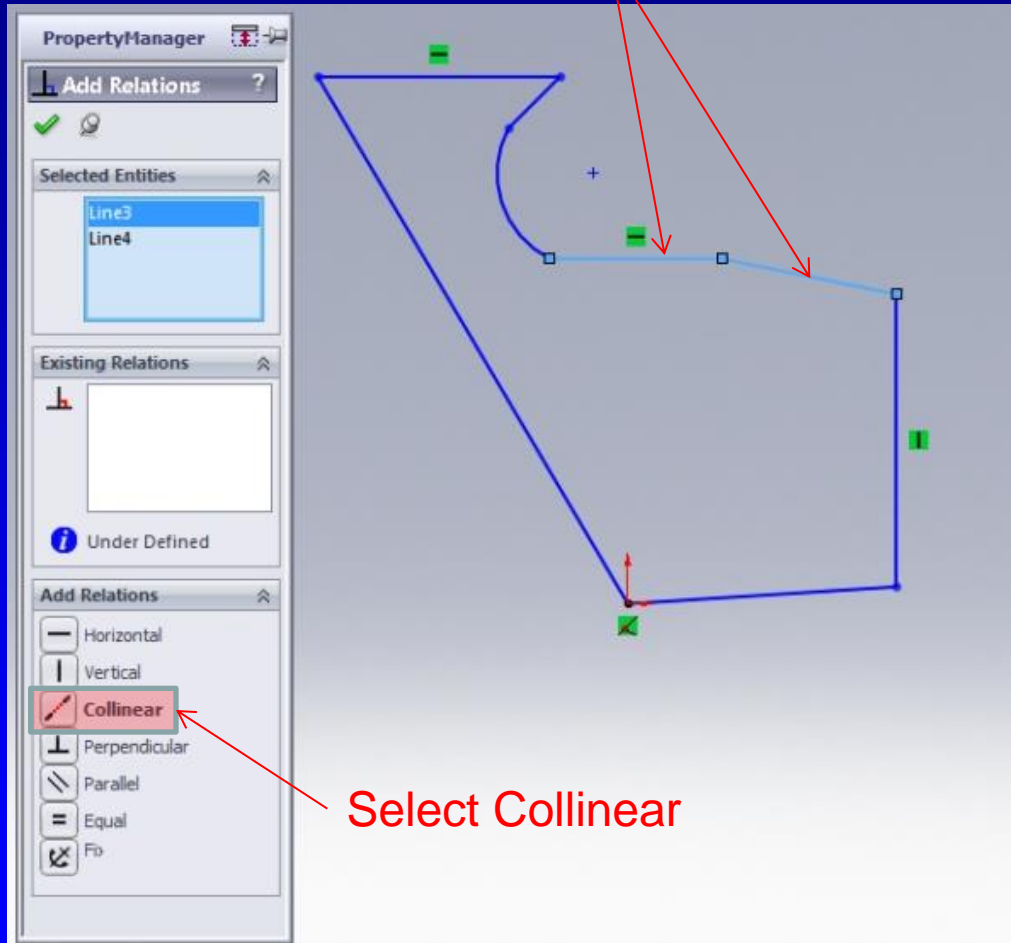
Select Add Relations



Select Vertical

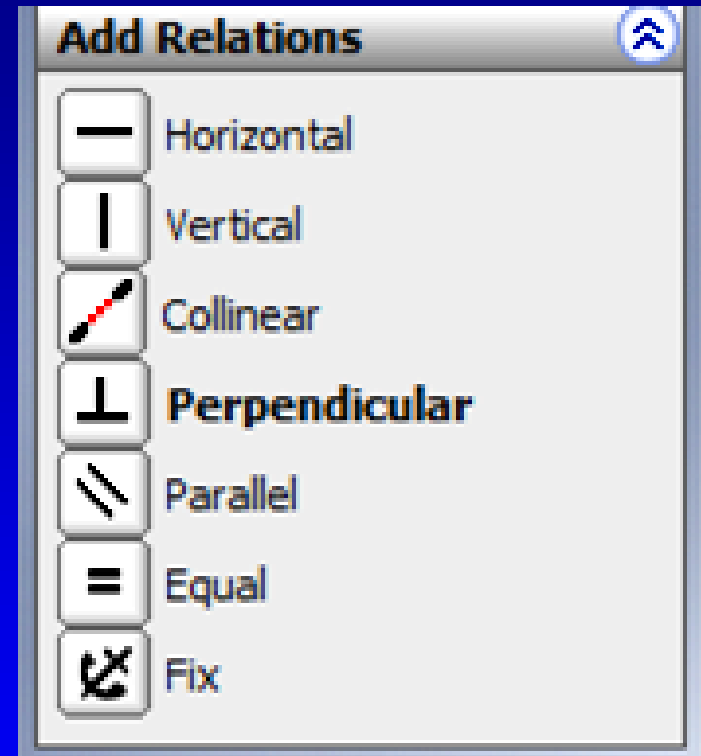
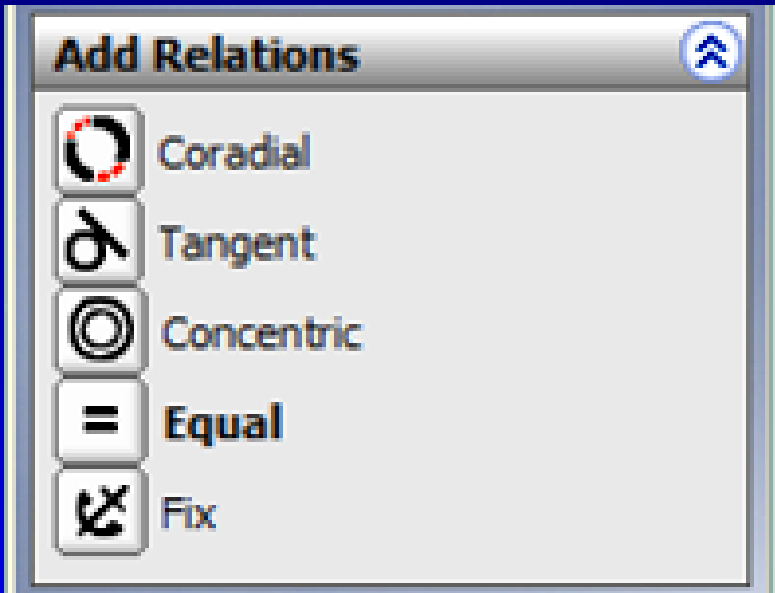
Constraints

Select two entities (lines)



Constraints

Two curved entities



Two straight entities



Repair Sketch Option

