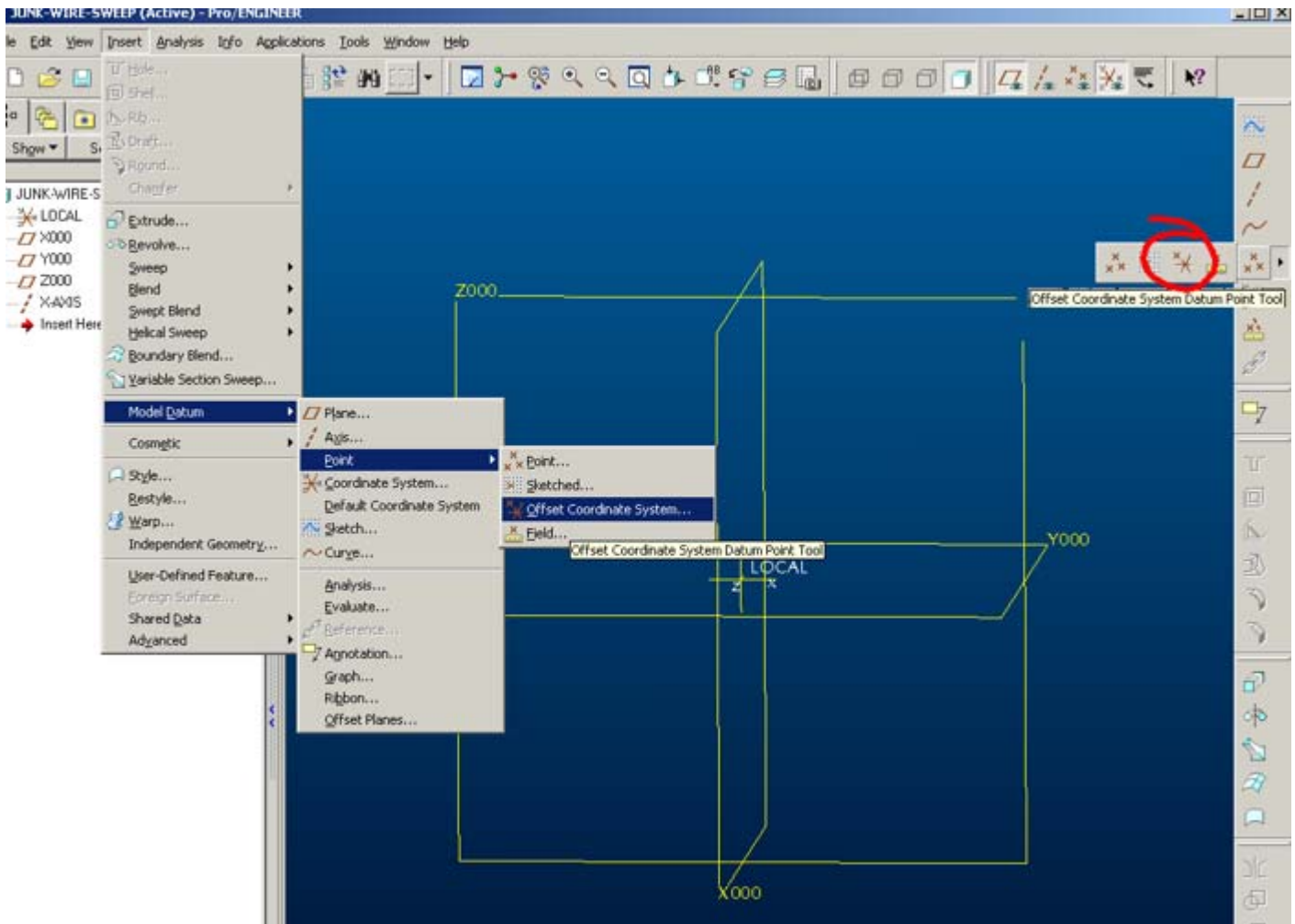


### 3D Sweep Curves

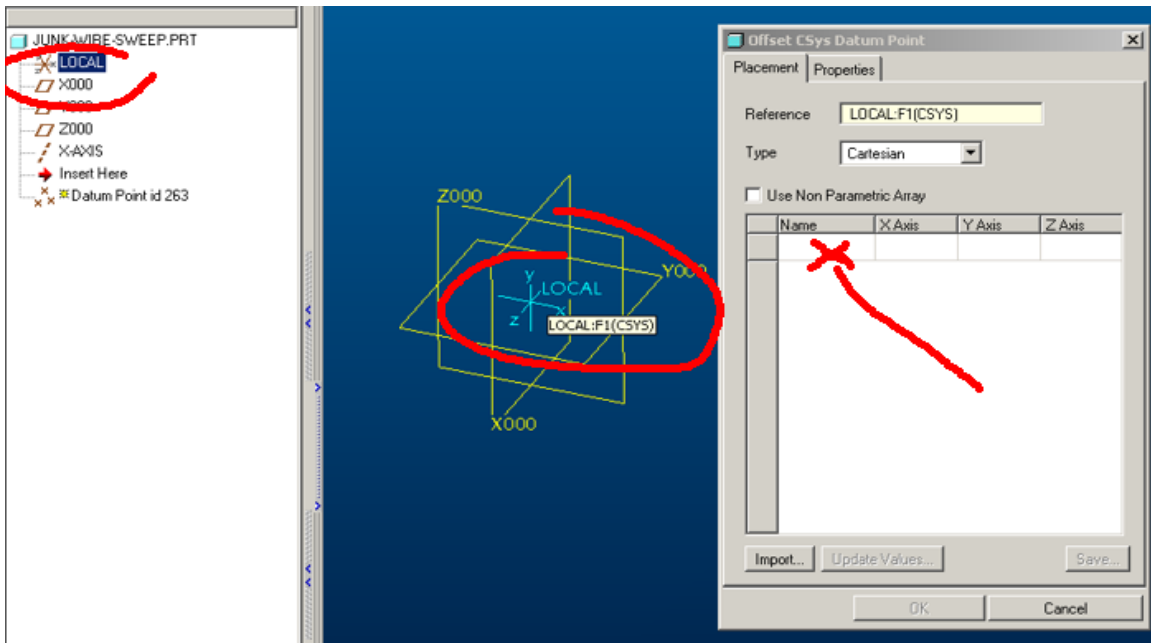
Using a 3D sweep is one way to create wires, cables and flex cables for assemblies in ProE. (Sheet metal is also good for flex cables and provides a flat pattern - See flex cable tutorial)

Process Steps:

- 1 - New Part
- 2 - Insert 3D Datum Points
- 3 - Create Datum Curve
- 4 - Generate Sweep

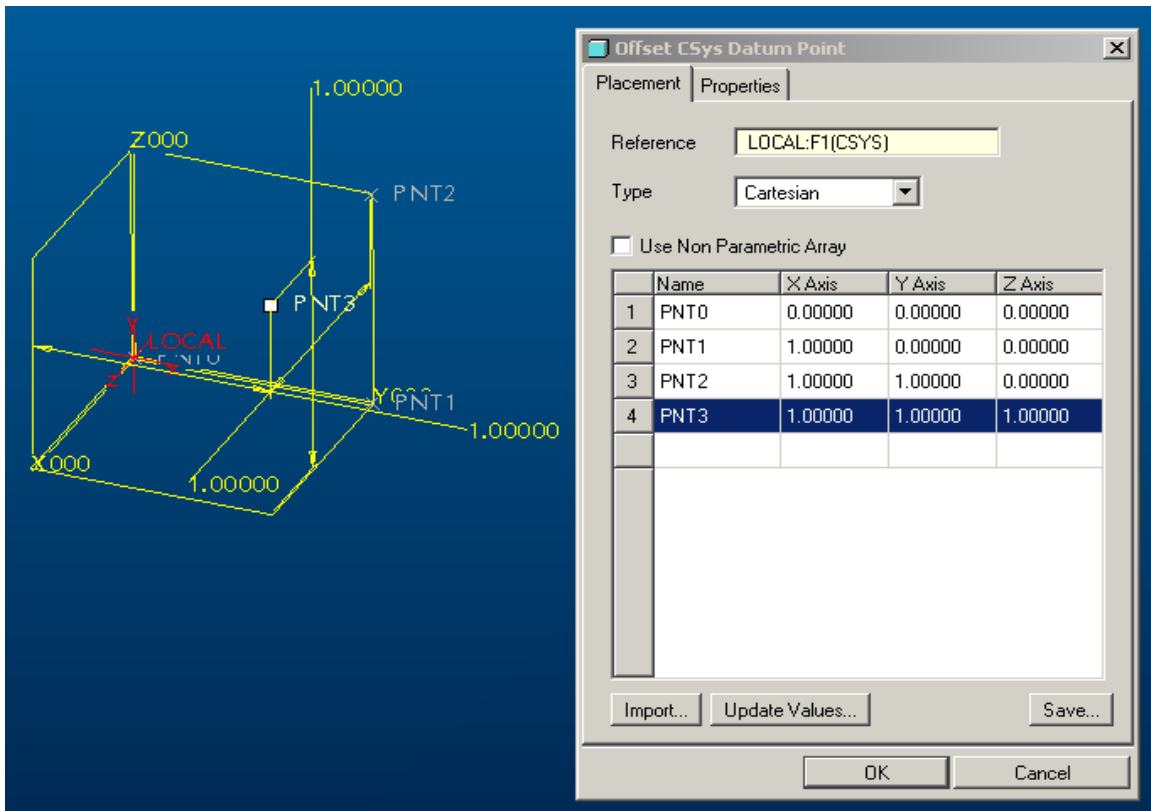


Starting with a new part, create datum points using the Offset Coordinate System Point Tool. Menu Picks: Insert --> Model Datum --> Point --> Offset Coordinate System... or right tool bar as shown.



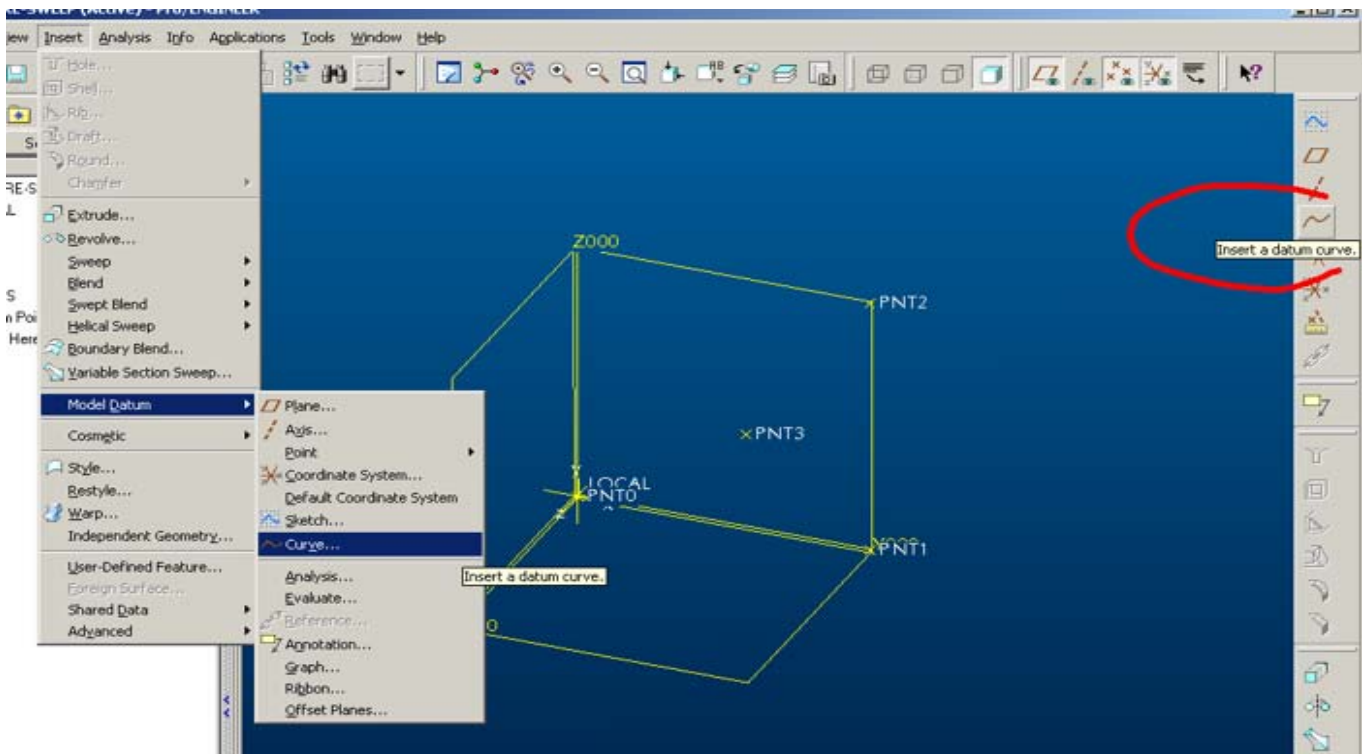
Select the Coordinate System to reference the points from.

Note: You can select the coordinate system from the Tree Column or from the Working Click in the empty box under NAME to begin defining points.

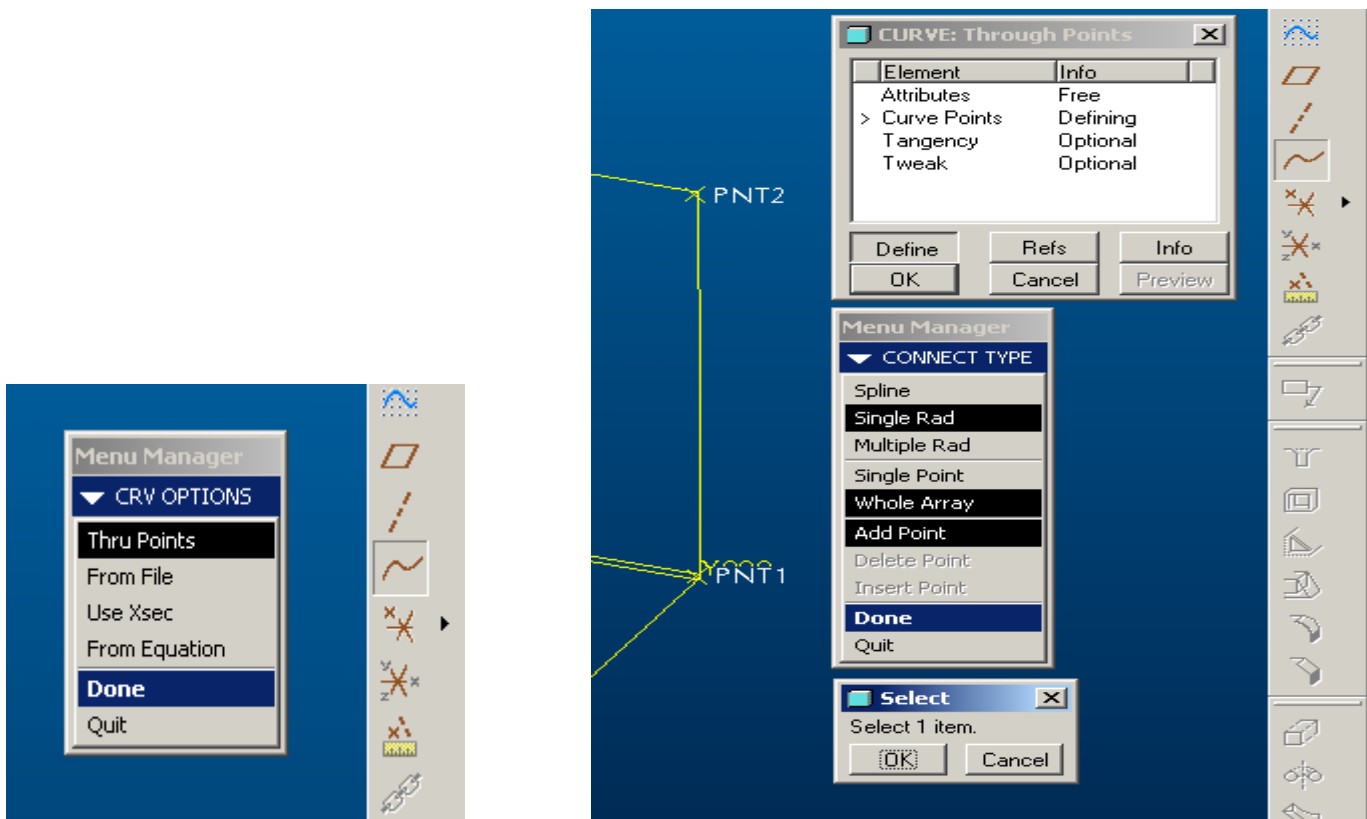


Enter the X, Y and Z values for the point locations.  
[OK] when complete

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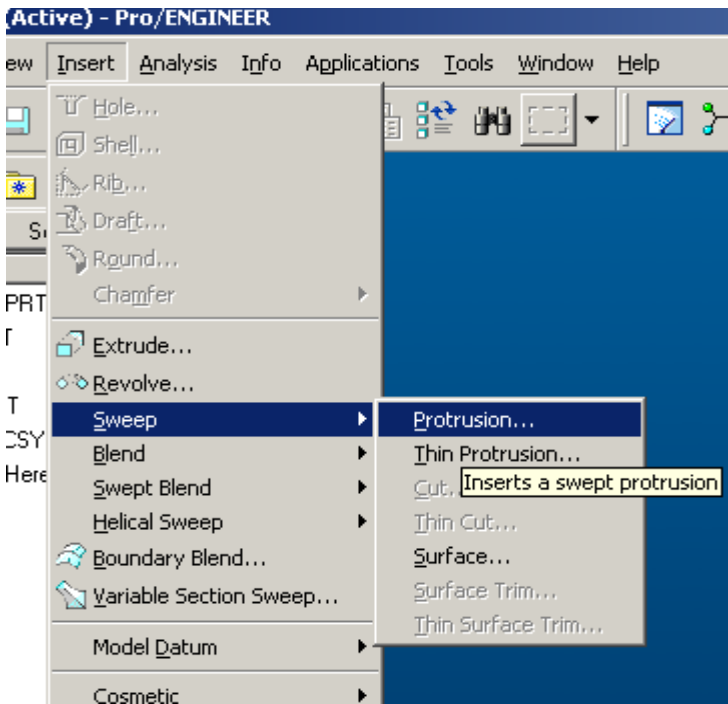


Create a datum curve through the point array:  
 Insert --> Model Datum --> Curve

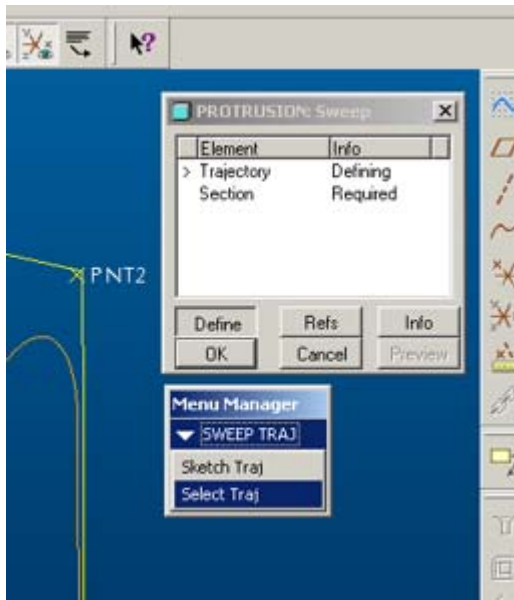


Select THRU POINTS  
 [Done]

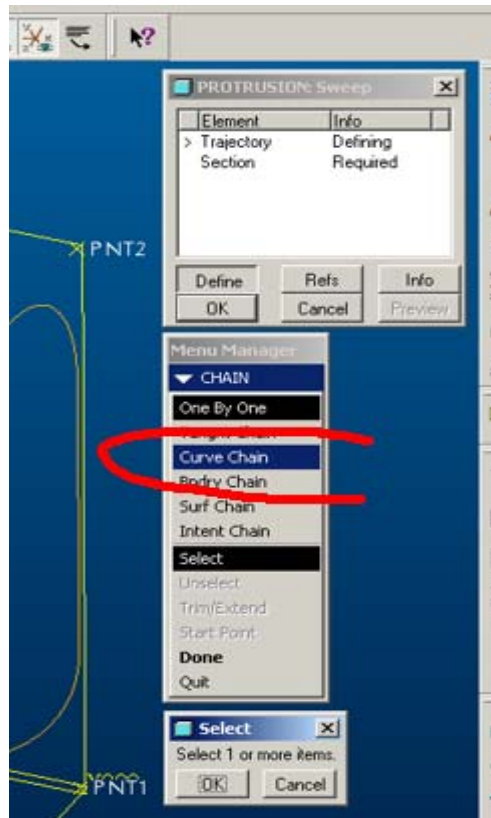
Single RADIUS  
 Select PNT0  
 Enter the Radius (.25 used in this example)  
 [Done] [OK]



Create a sweep protrusion: Insert --> Sweep --> Protrusion



[Select Traj] to define the trajectory

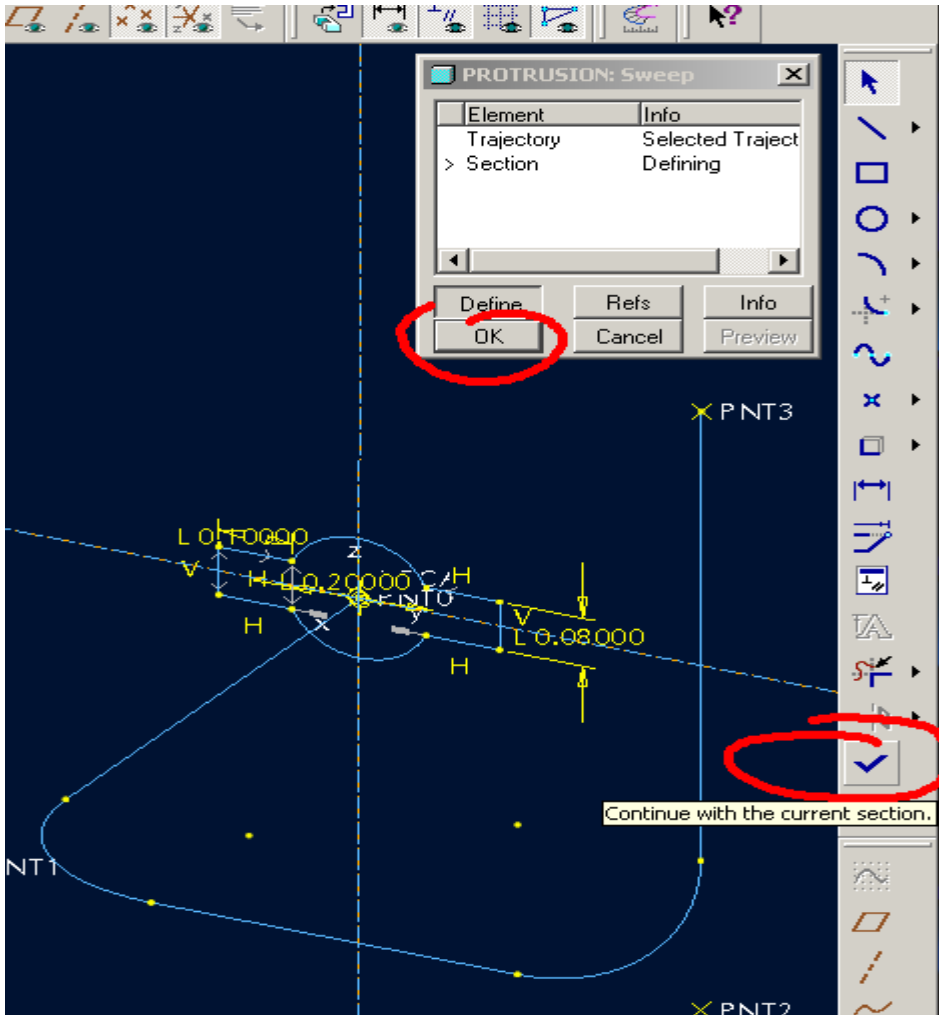


[Curve Chain]

Select the curve  
CHAIN OPT  
**Select All**

[Done]  
[Okay]

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The screen will now re-orientate for sketching the curve to be extruded.

Sketch the curve and Click the Continue **Check Mark** [OK] to complete the Protrusion Sweep box.

