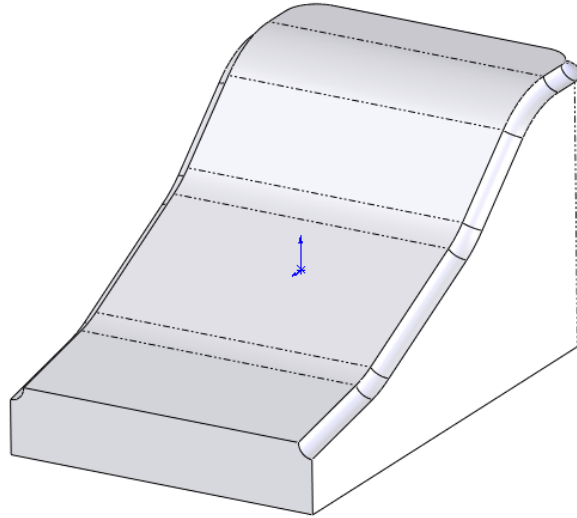


Composite Curve Tutorial

Complement to

[Beginner's Guide to SolidWorks Books](#)



Disclaimer: These models and drawings are not meant for manufacturing but as a teaching aid to practice SolidWorks modeling.

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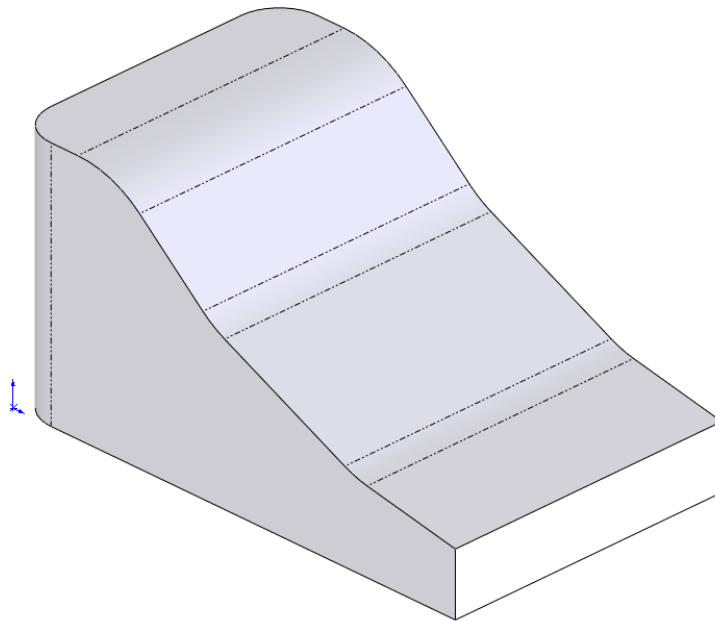
alejandro@mechanicad.com

Composite Curve:

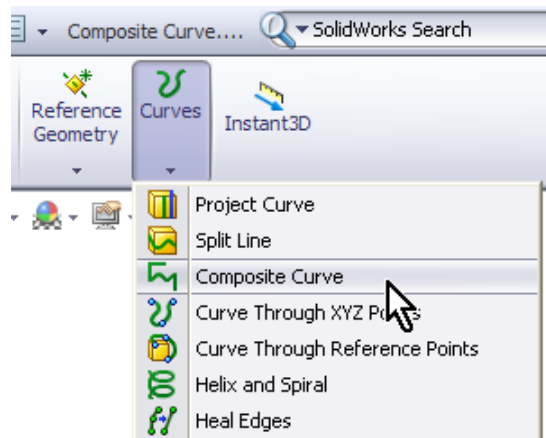
A composite curve is used to merge multiple model edges and/or sketch entities to make a single continuous curve to be used for sweeps and lofts as a path or guide curve.

Open the part located in our server or make a similar shape for this example.

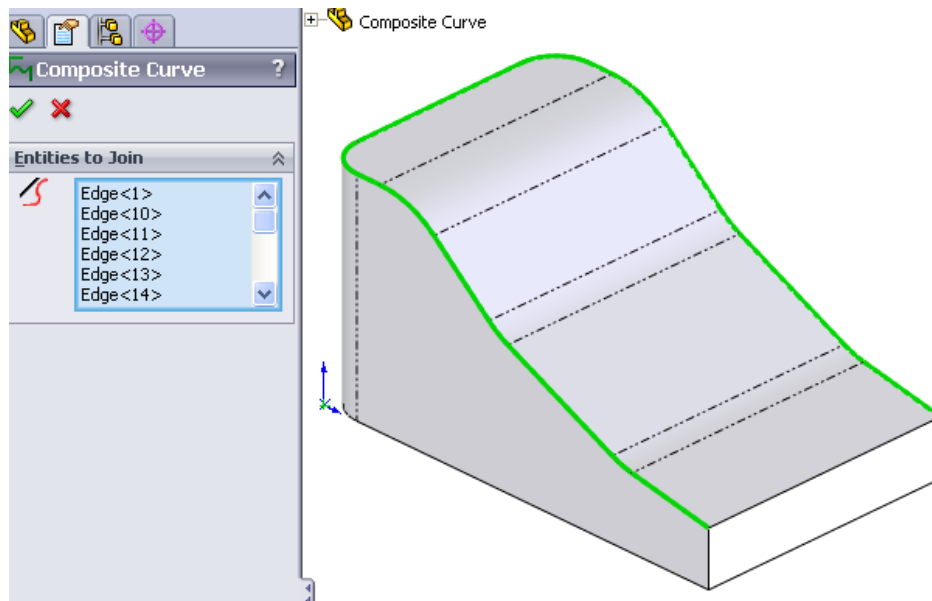
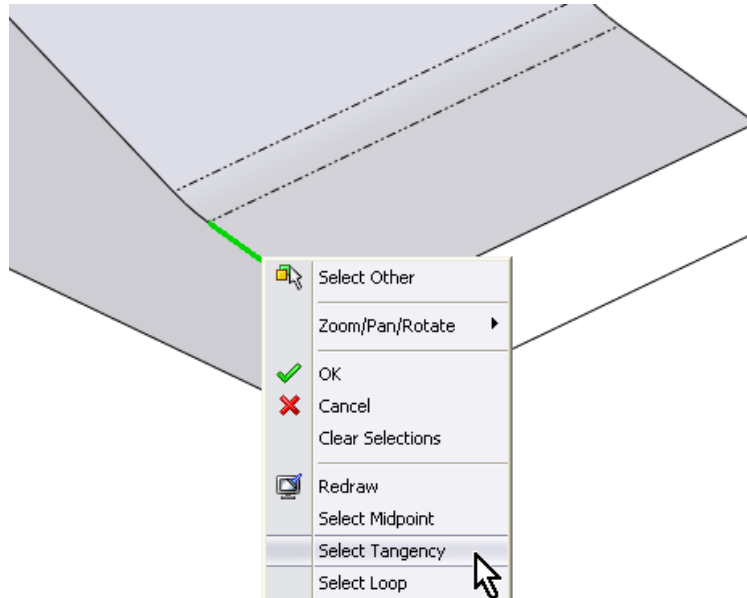
www.mechanicad.com/download/composite-curve.sldprt



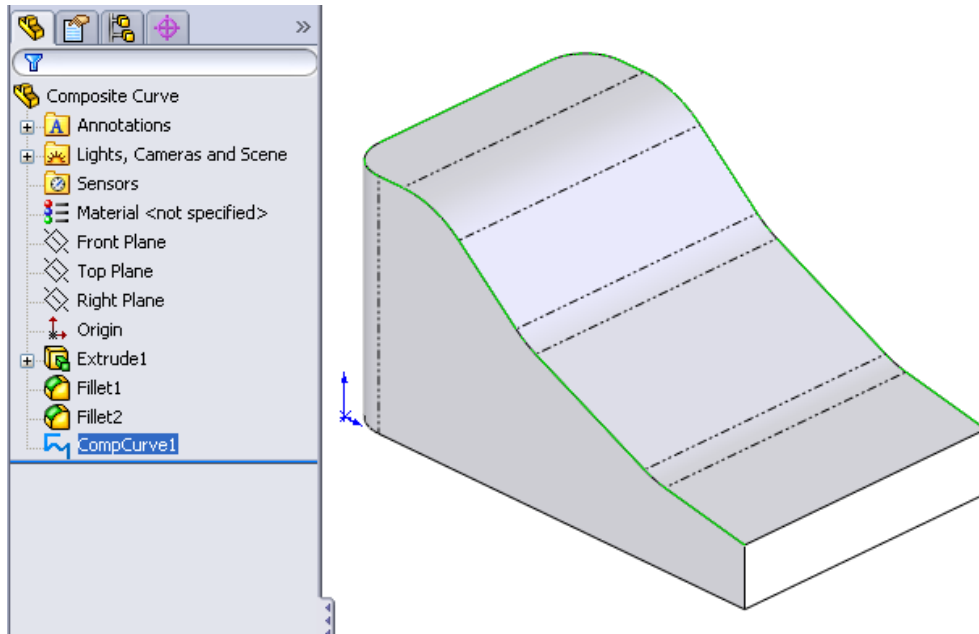
To make a composite curve, select the “Curves, Composite Curve” icon from the features tab in the Command Manager, or select the menu “Insert, Curve, Composite...”



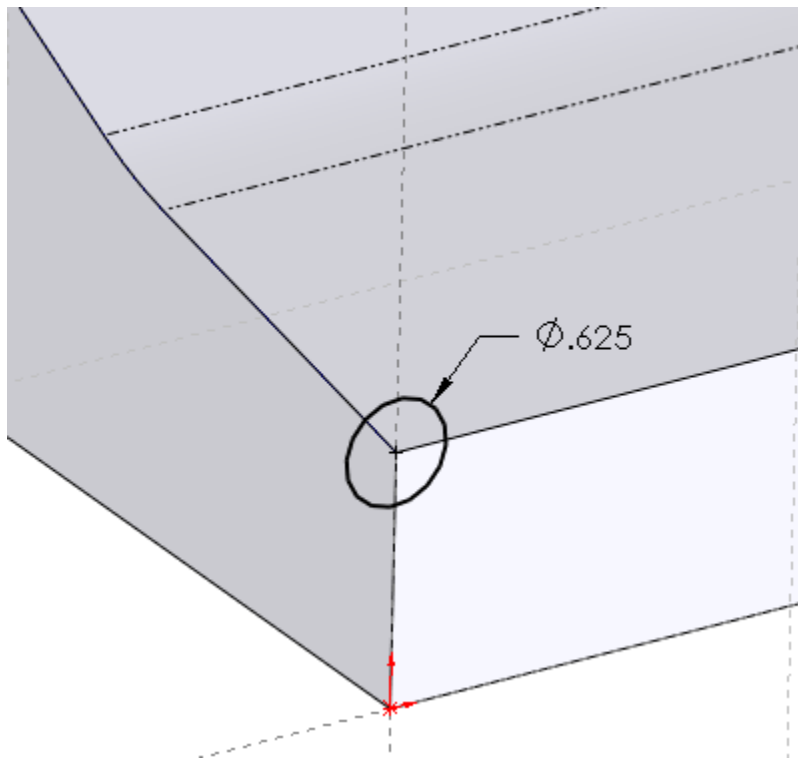
Select all the edges that will make the curve for the path. To select all the edges, we can either select them one by one, window-select them, or, since all of the edges are tangent, we can Right-Mouse-Click on one edge and pick “**Select Tangency**” from the pop-up menu. All tangent edges are selected automatically.



Click on “OK” to finish the command. The Composite Curve is created and added to the Feature Manager.

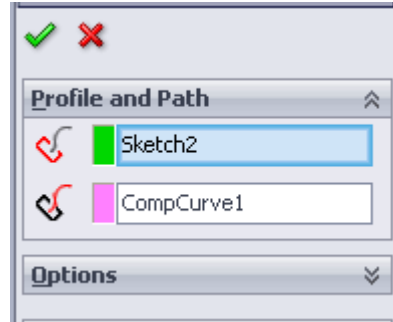


Now we can make a sketch to be used as a profile, and make a sweep using the Composite Curve just created as a path.

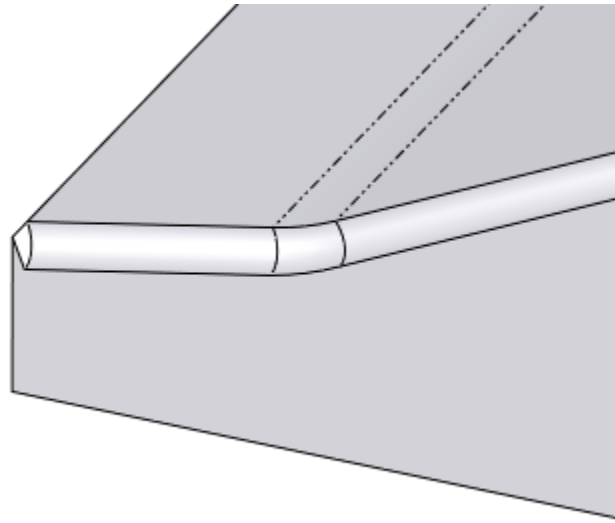


Exit the sketch.

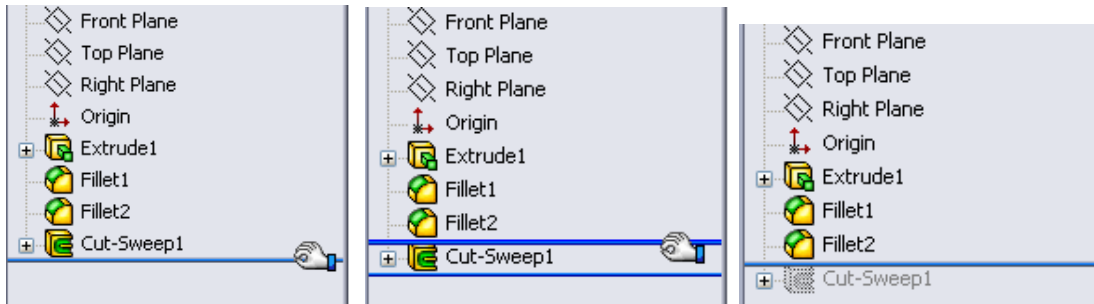
Select the menu “Insert, Cut, Sweep” and use this sketch as profile and the composite curve as a path.



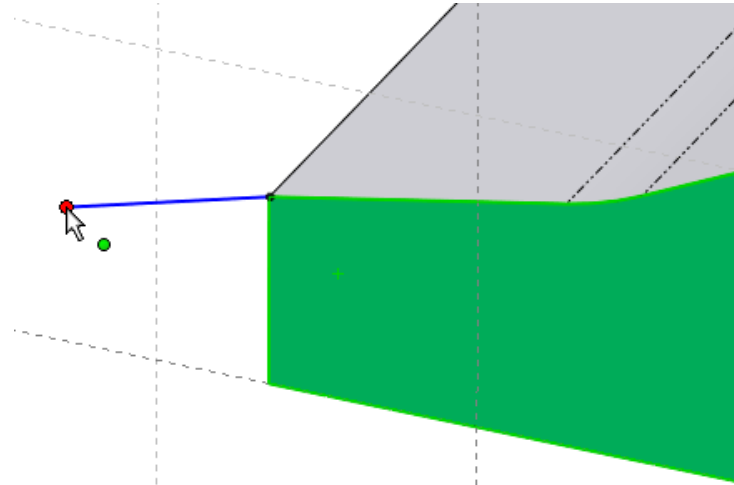
When completing the cut sweep, we realize that the resulting feature is not giving us the desired result at the end of the path.



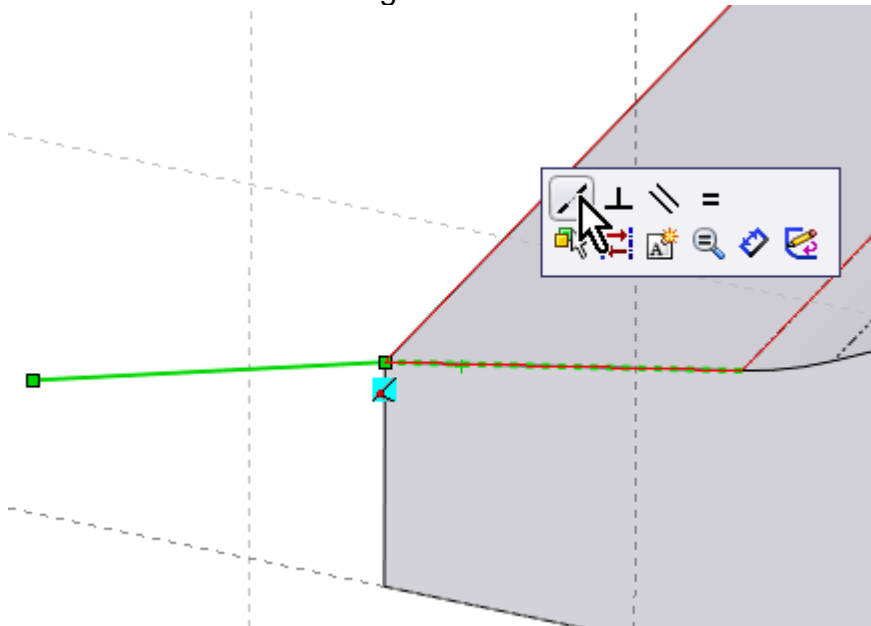
To get the result we want, move the rollback bar just above the Cut Sweep feature, and make a sketch as indicated next:



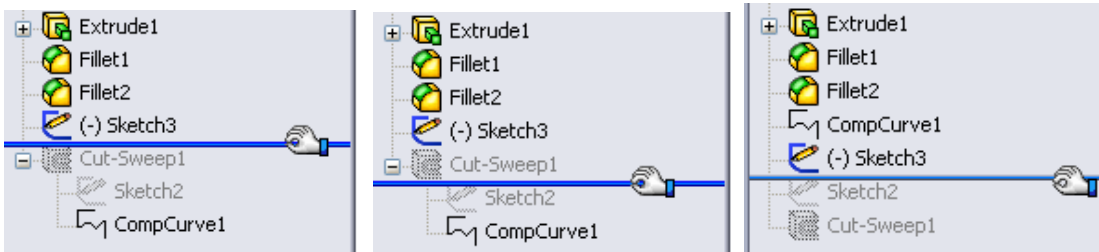
Now select the face indicated, and make a sketch as a single line. This line can be dimensioned if necessary.



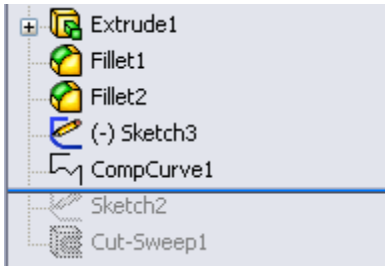
Make the line collinear to the last edge of the model.



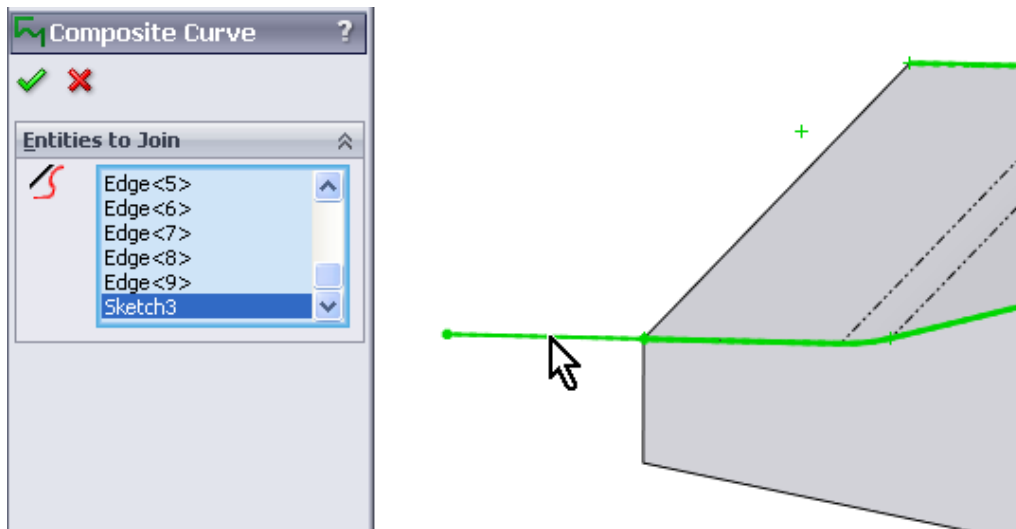
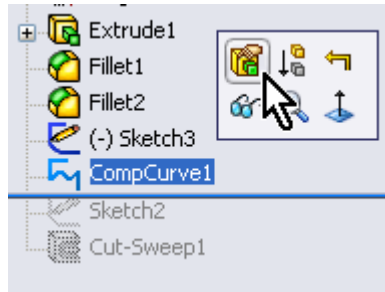
And exit the sketch. Expand the *Cut-Sweep1* feature, and move the rollback bar forward just before the *Cut-Sweep1*



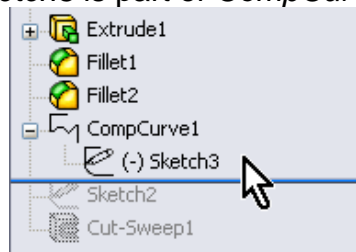
We can reorder features in the Feature Manager; in order for us to make *Sketch3* part of *CompCurve1*, *Sketch3* must be located above it. Click and drag *Sketch3* above as indicated.



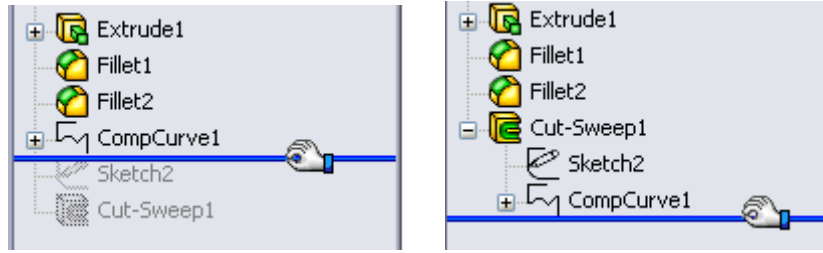
Now we can edit the *CompCurve1*, to add the *Sketch3* to it.



Click OK to finish. Now *Sketch3* is part of *CompCurve1*.



Now drag the rollback bar all the way to the bottom, and finish the part.



Now the *Cut-Sweep1* feature cuts through the part as desired.

