

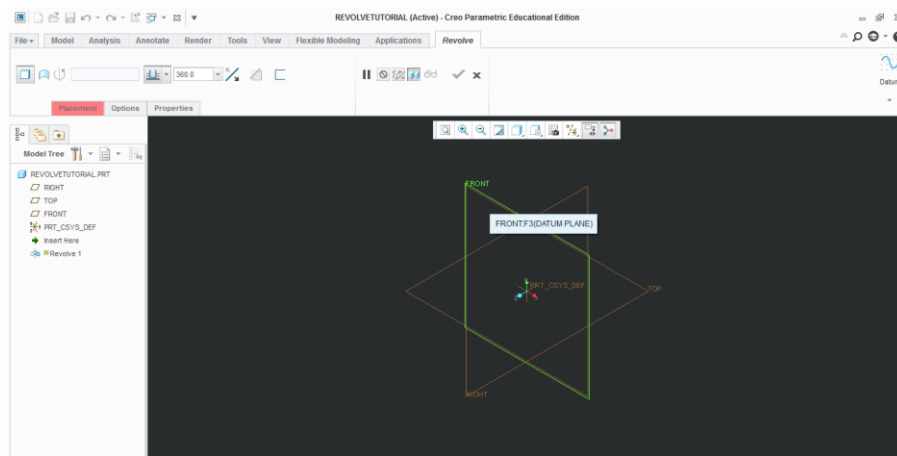
Revolve Modeling Tutorial

By: Matthew Jourden

Brighton High School

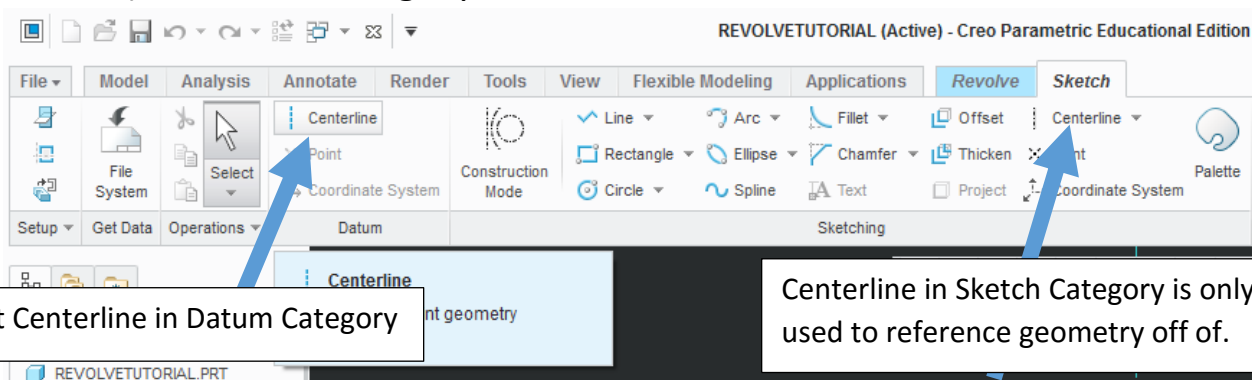
Revolve: Allows the designer to take a profile a spin around an axis line. Typically used for models that are symmetrical. Revolves can be used to add or subtract material. Each Revolve as two distinct features Axis Line (Line Profile will spin around) and Profile (Sketch Profile that is a cross-section of the revolved object)

1. Open Creo Parametric > Select New > Select Part > Name RevolveTutorial
2. Select Model Tab > Select Revolve Icon > Select Front Datum



3. Axis Line: In most instances drawing the axis line first helps to constrain the part.

Select Centerline (located datum category; not under the sketch icon (NOTE: Sketch > Centerline is only used for references; it cannot be used to revolve around) > From the origin place a Vertical Centerline



5. Green Check to exit the sketch > May modify total number of degrees the part is spun around in the Revolve Info Tab (keep it 360 degrees) > Green Check complete the object. Completed Image as shown below

