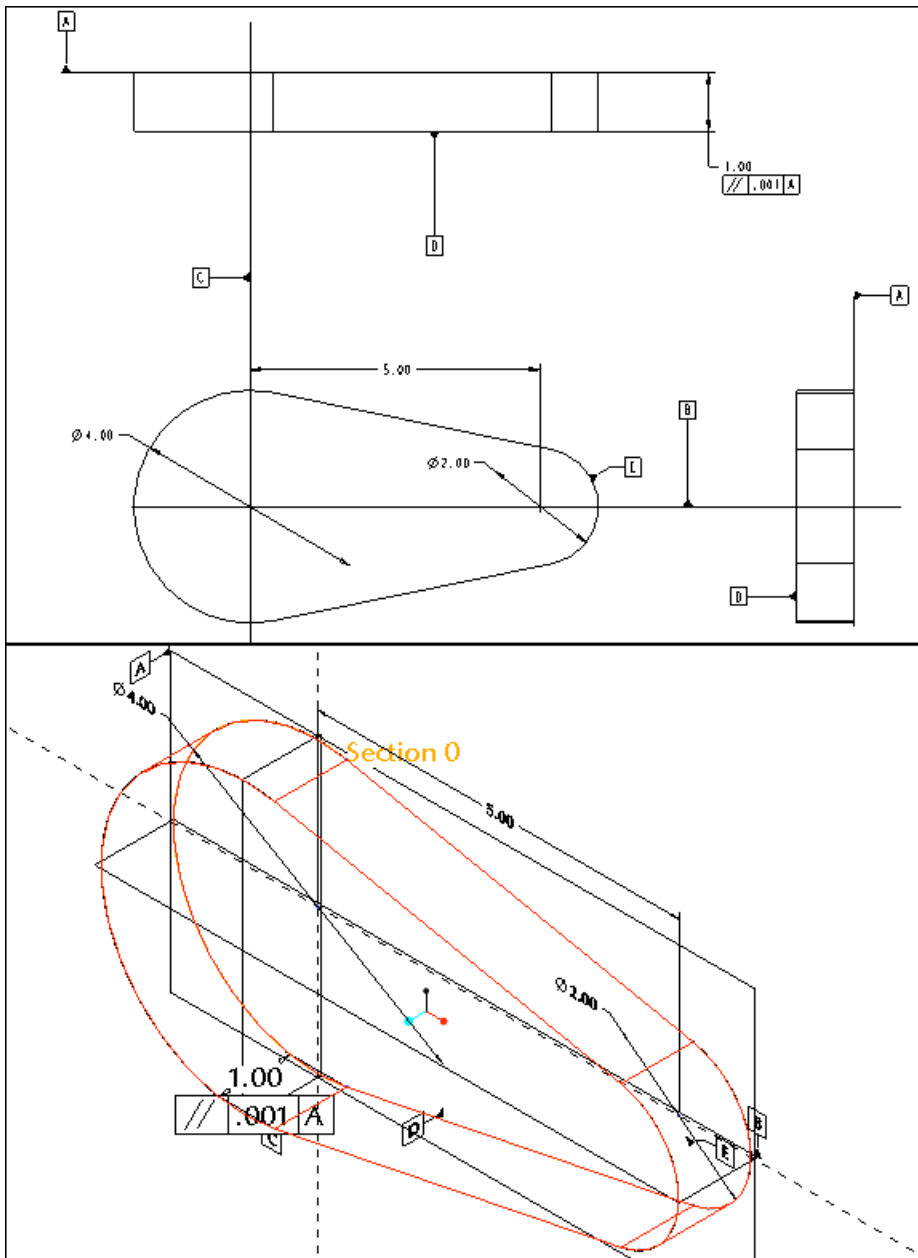


Tutorial 4 3D Annotations



3D Annotated Model and Traditional Drawing

(Tutorial does not correspond to *Moving from 2D to 3D CAD for Engineering Design*)

Introduction

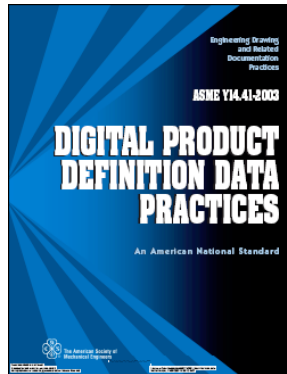
3D Annotations are quickly replacing many of the functions previously provided by 2D drawings. This tutorial is an introduction to inserting and placing datum features and driving dimensions on Pro/ENGINEER models using 3D annotations. A drawing will also be created to compare the two.

Annotation Features

3D Notes can also be added to an object using **Annotation Features**. Annotation features are data features that you can use to manage the model annotation and propagate model information to other models, or to manufacturing processes. The Annotation Feature Tool options correspond to the new ASME Y14.41 Digital Product Definition Data Practices.

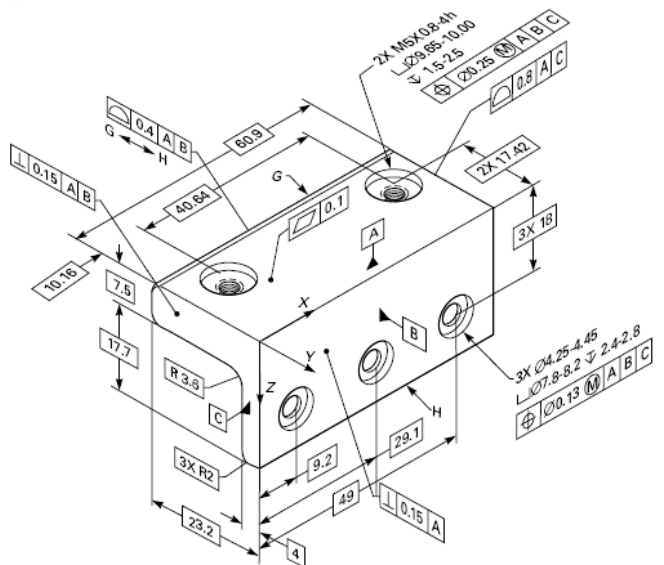
An Annotation feature consists of one or more Annotation Elements. Each Annotation Element can contain one annotation item, along with associated references and parameters. You can include the following types of annotations in an Annotation Element:

- Note
- Symbol
- Surface finish
- Geometric tolerance
- Set Datum Tag
- Ordinate baseline dimension
- Driven dimension
- Ordinate driven dimension
- Chamfer dimension
- Reference dimension
- Ordinate reference dimension



Digital Product Definition Data Practices

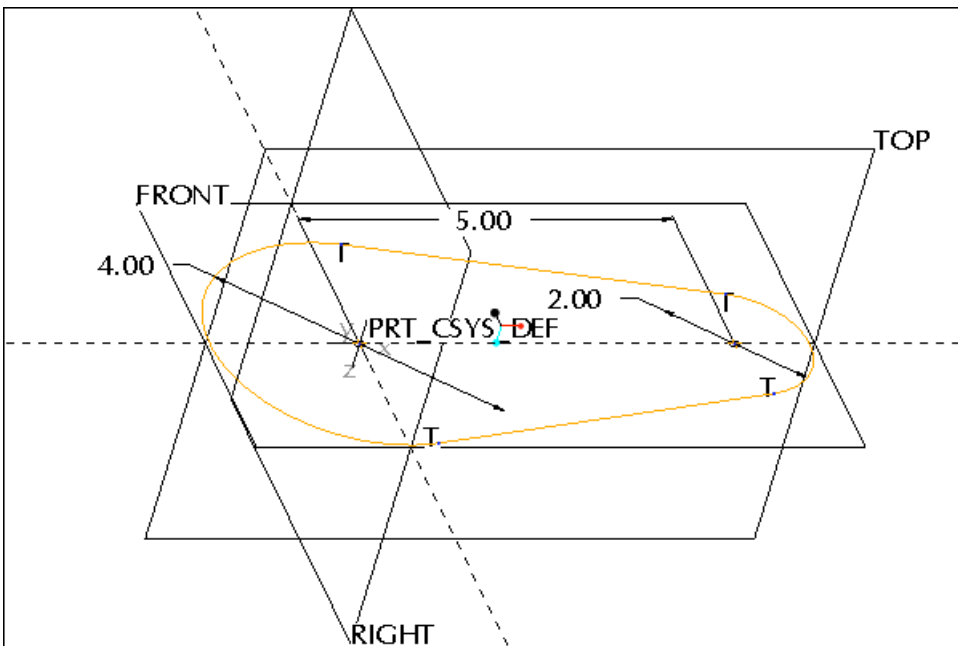
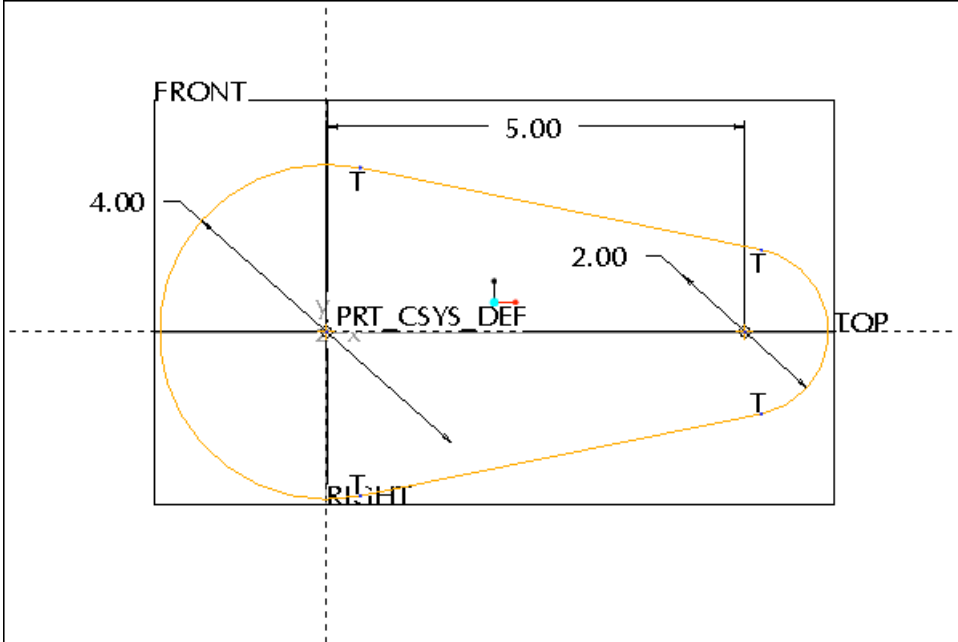
ASME Y14.41-2003 establishes requirements for preparing, organizing and interpreting 3-dimensional digital product images. Digital Product Definition Data Practices, which represents an extension of the popular Y14.5 standard for 2-dimensional drawings, reflects the growing need for a uniform method of documenting the data created in today's computer-aided design (CAD) environments. The standard provides a guide for CAD software developers working on improved modeling and annotation practices for the engineering community. ASME Y14.41 sets forth the requirements for tolerances, dimensional data, and other annotations. ASME Y14.41 advances the capabilities of Y14.5, Dimensioning and Tolerancing, the standard pertaining to 2-D engineering drawings”.

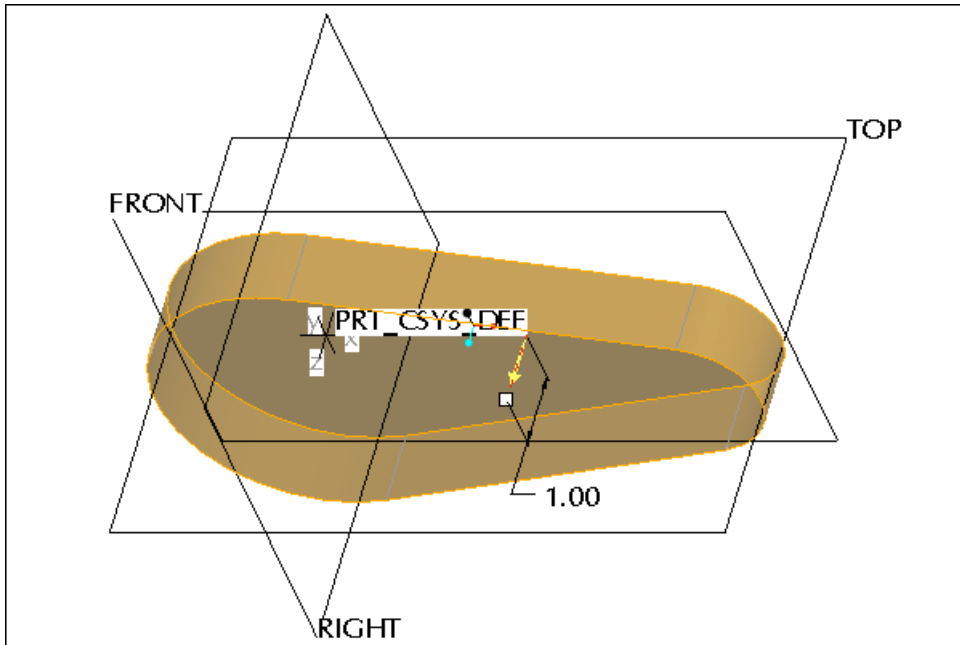


Modeling Commands

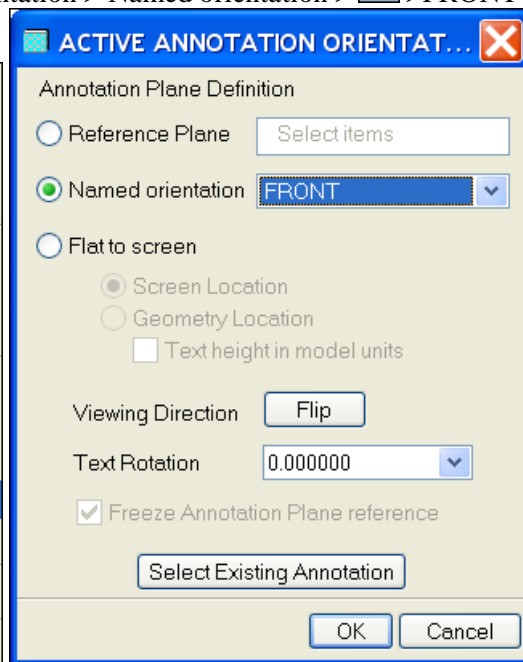
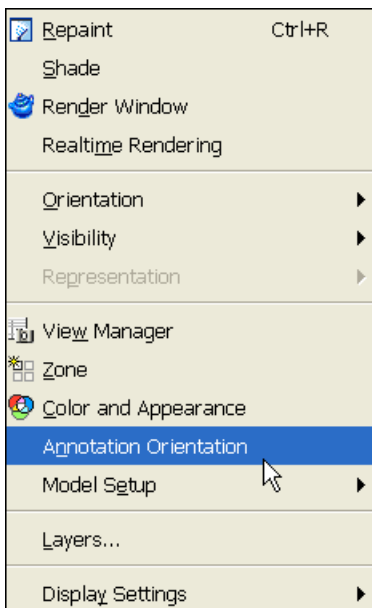
Commands are simplified in this tutorial. You will be called upon to create and edit items using skills mastered from the previous tutorials.

Launch Pro/ENGINEER WILDFIRE 4.0 > File > Set Working Directory > OK > File > New > name your file (annotations) > create an extrusion using the provided values

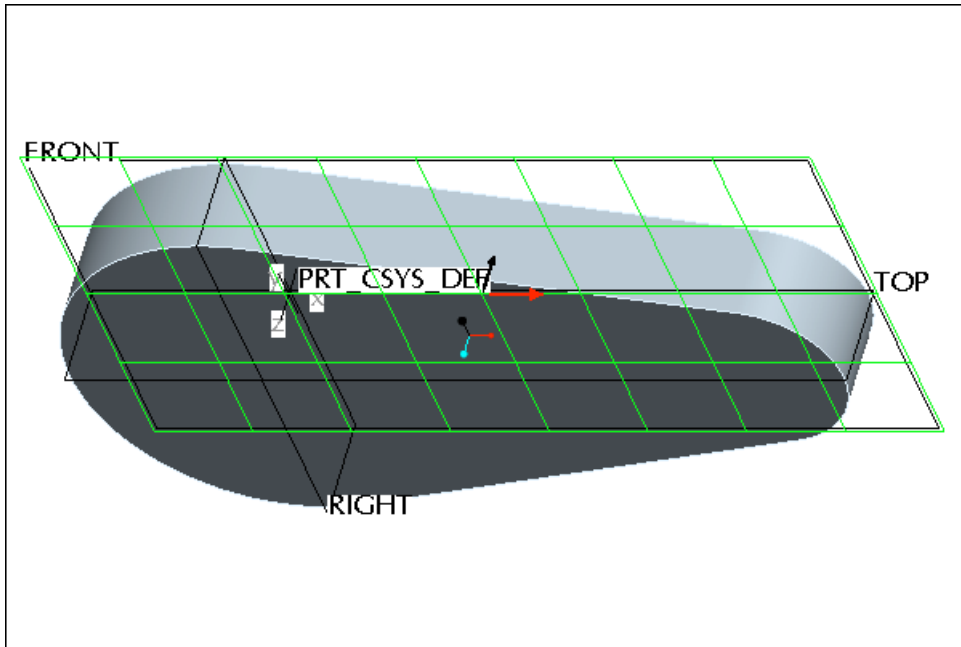




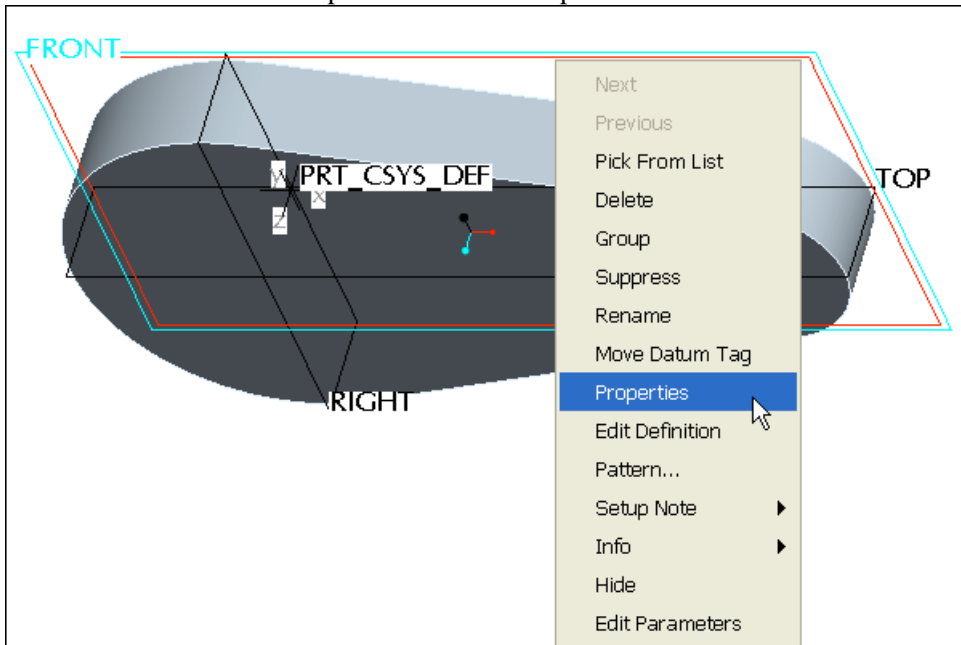
Click: View > Annotation Orientation > Named orientation > [dropdown] > FRONT



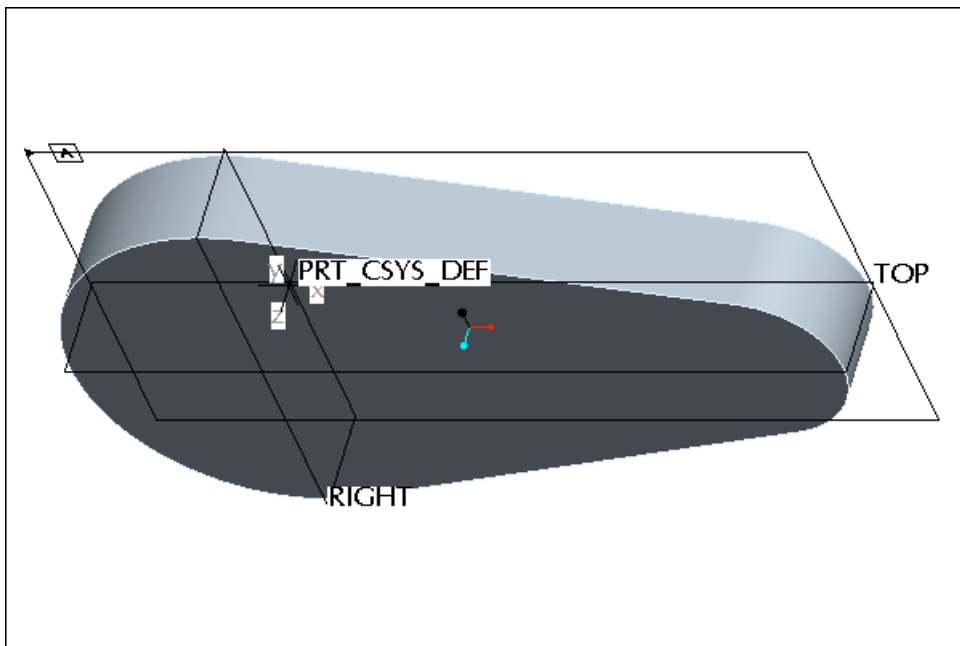
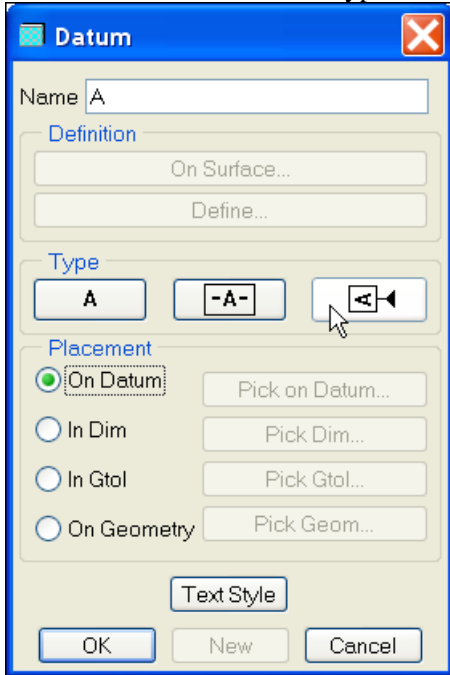
Click: OK



Pick on the FRONT datum plane > RMB > Properties

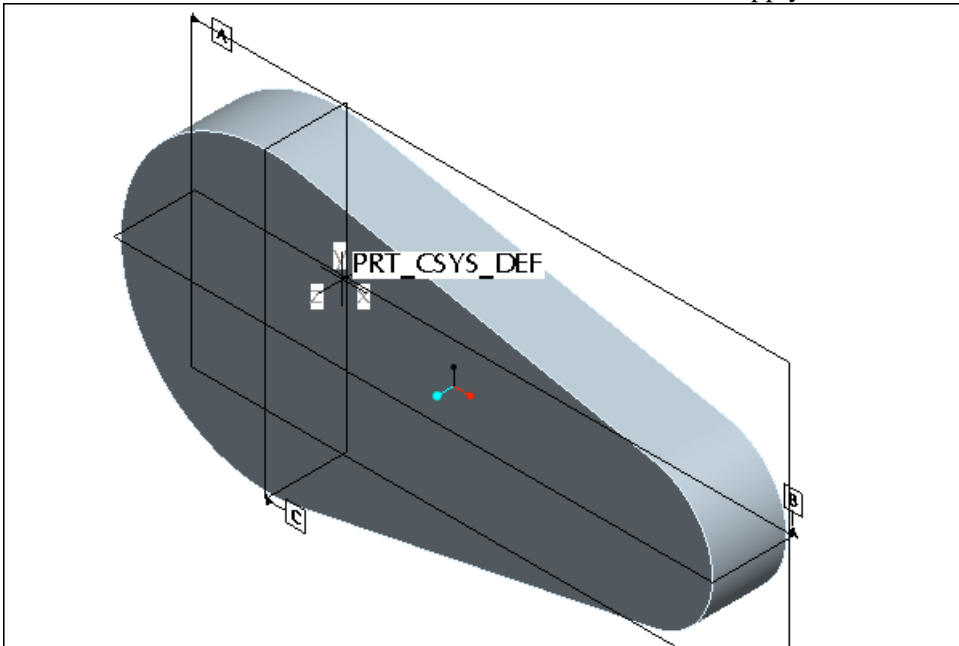


Click in the Name field and type: A > Enter > Type  > On Datum > OK

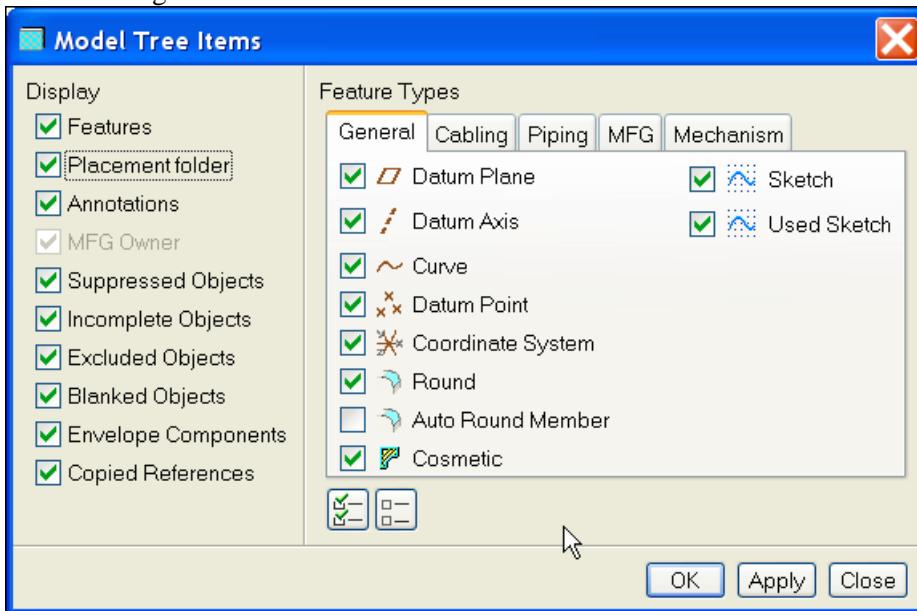


Repeat the commands for Datums B (TOP) and C (RIGHT)

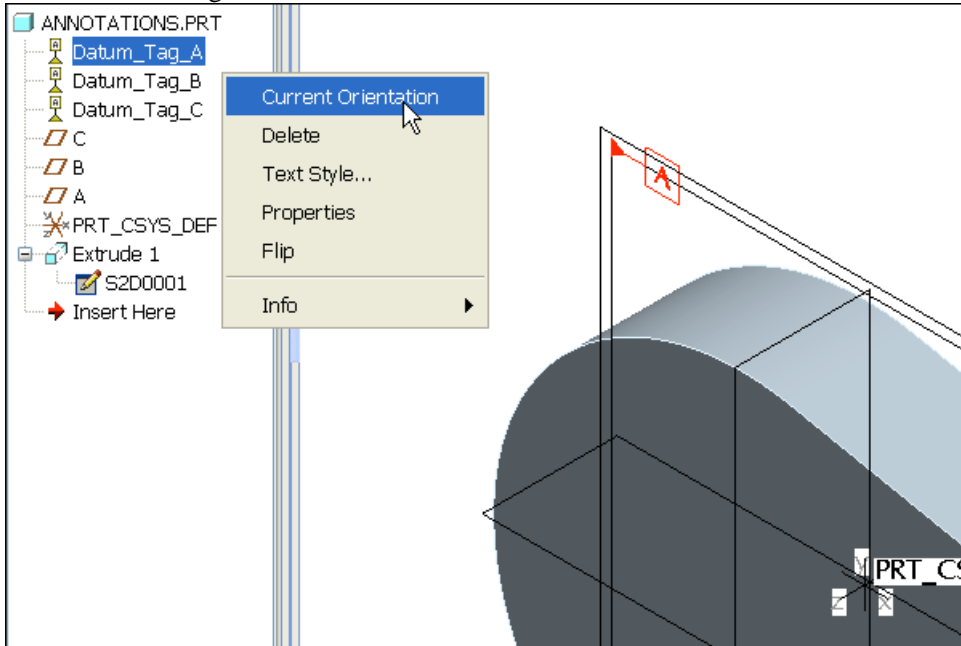
Click: Tools > Environment > Standard Orient to Isometric > Apply > OK



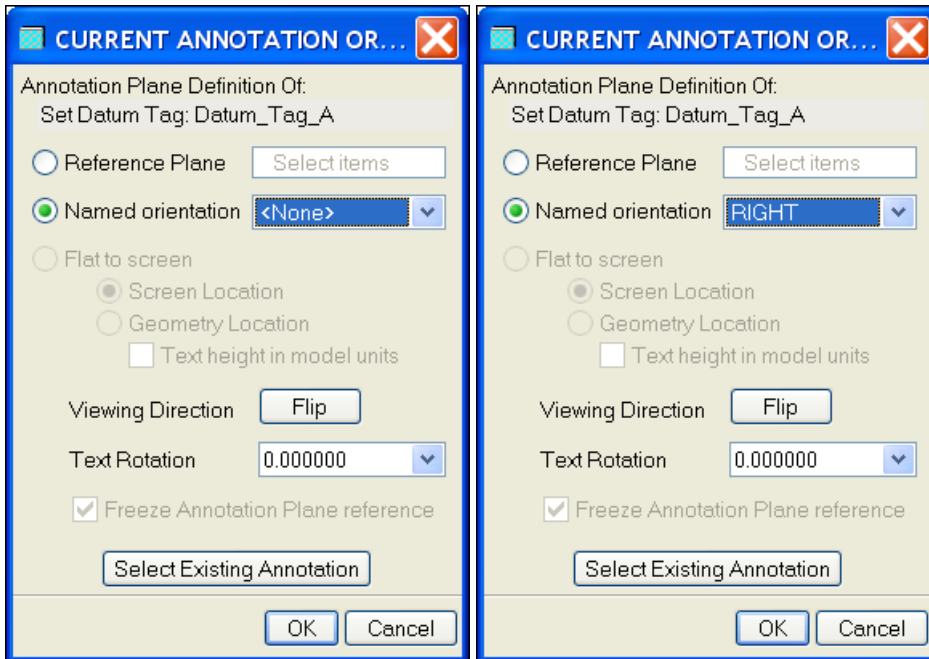
Click: Settings > Tree Filters > all on > OK



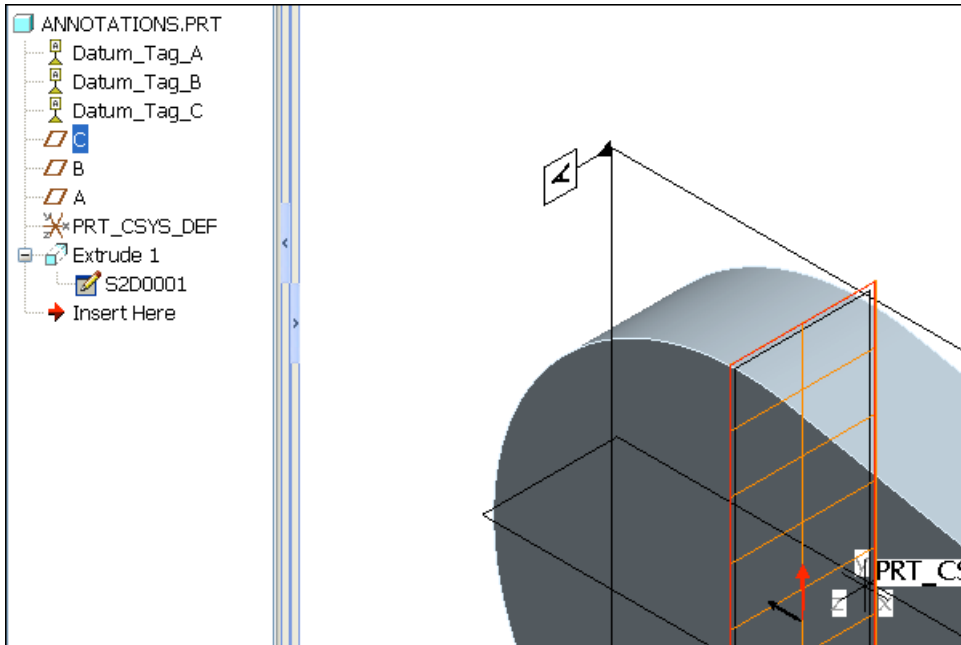
Select Datum_Tag_A in the Model Tree > RMB > Current Orientation



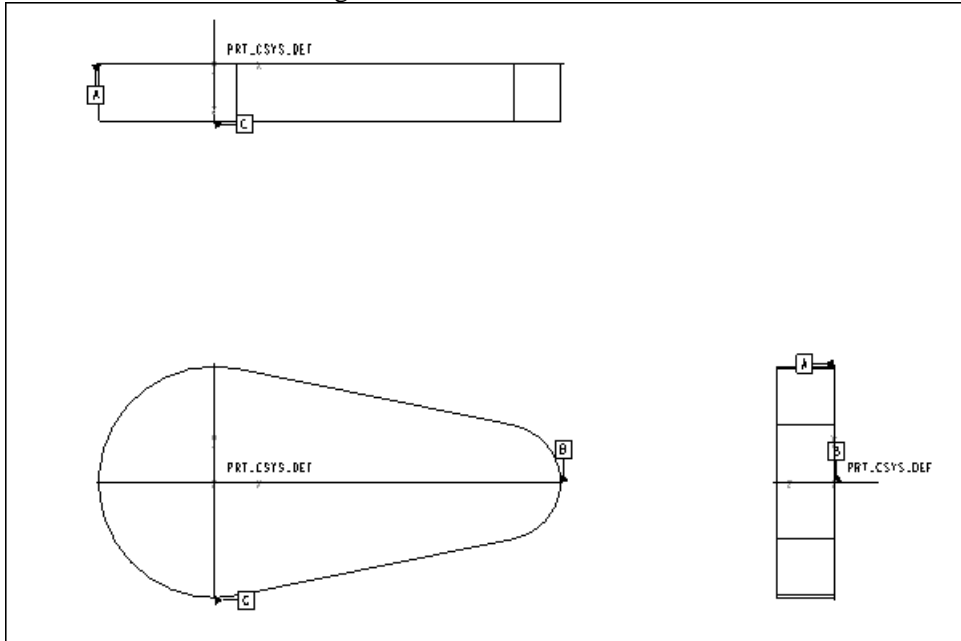
Click: Named orientation > [dropdown] > RIGHT > OK



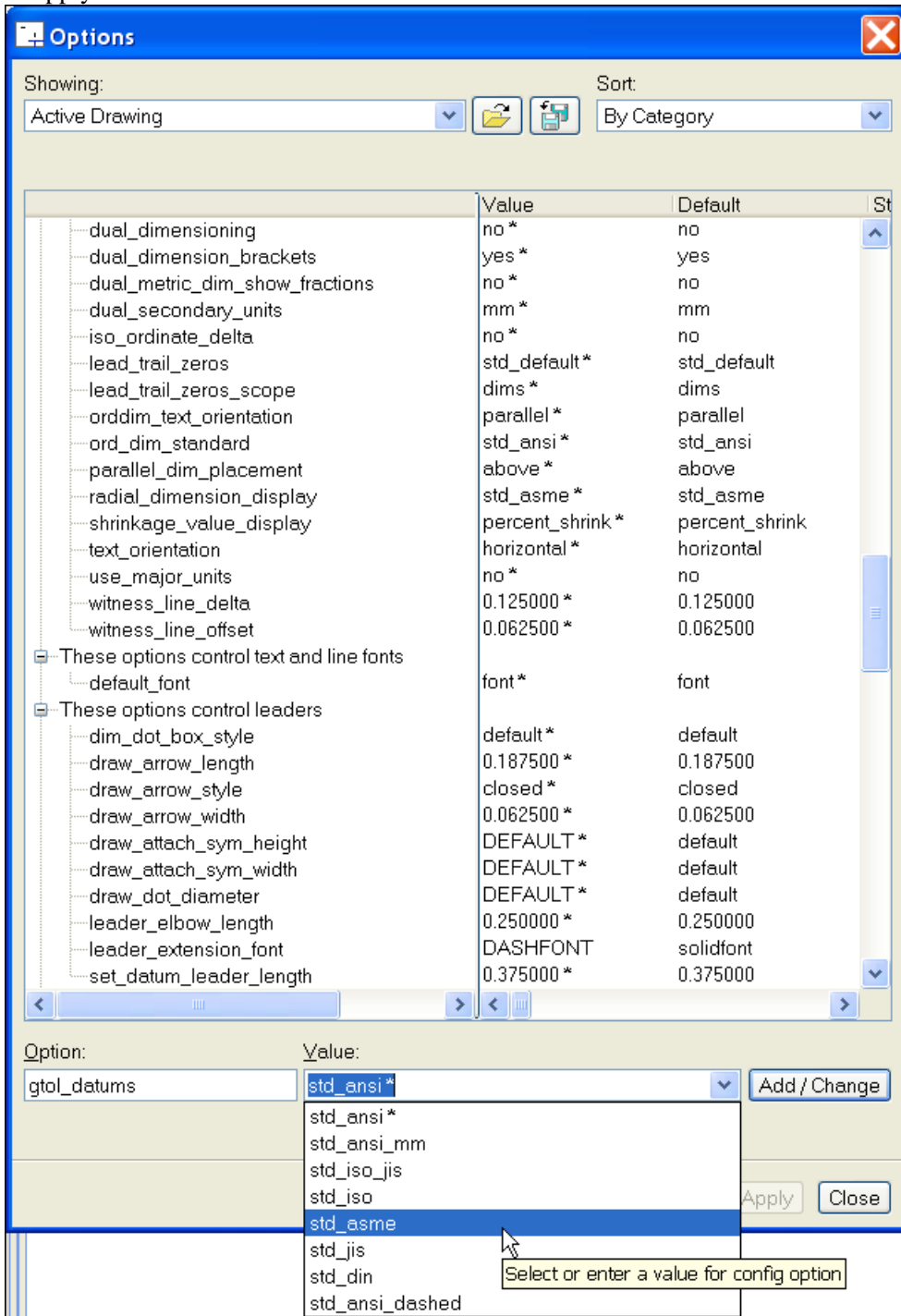
Click: File > Save > Enter

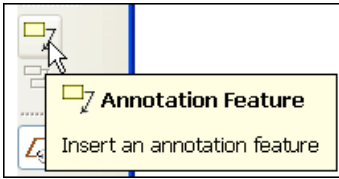


Click: File > New > Drawing > Name: annotations > OK > OK > note the datum tag style

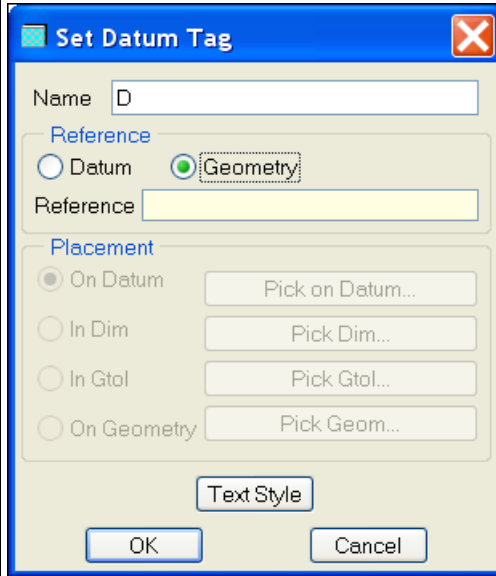
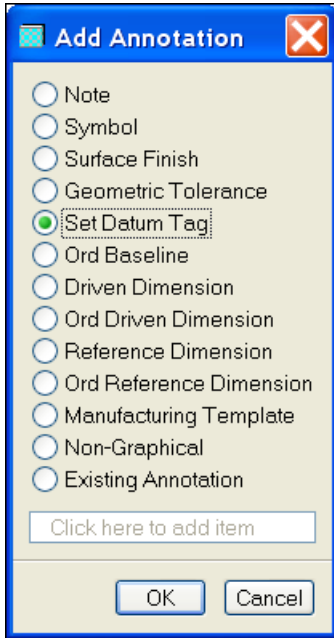


Click: File > Properties > Drawing Options > in the Option field, type: gtol > Enter > notice that the standard defaults to ANSI (1994) > in the Value field, select: std_asme > Add/Change > Apply > Close > Done/Return > File > Save > Enter > File > Close Window

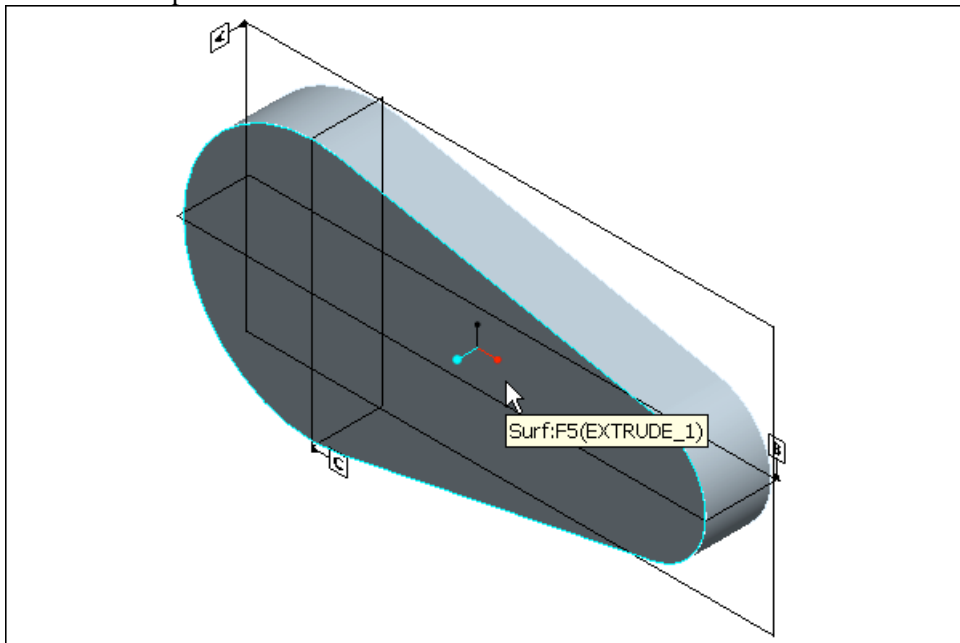




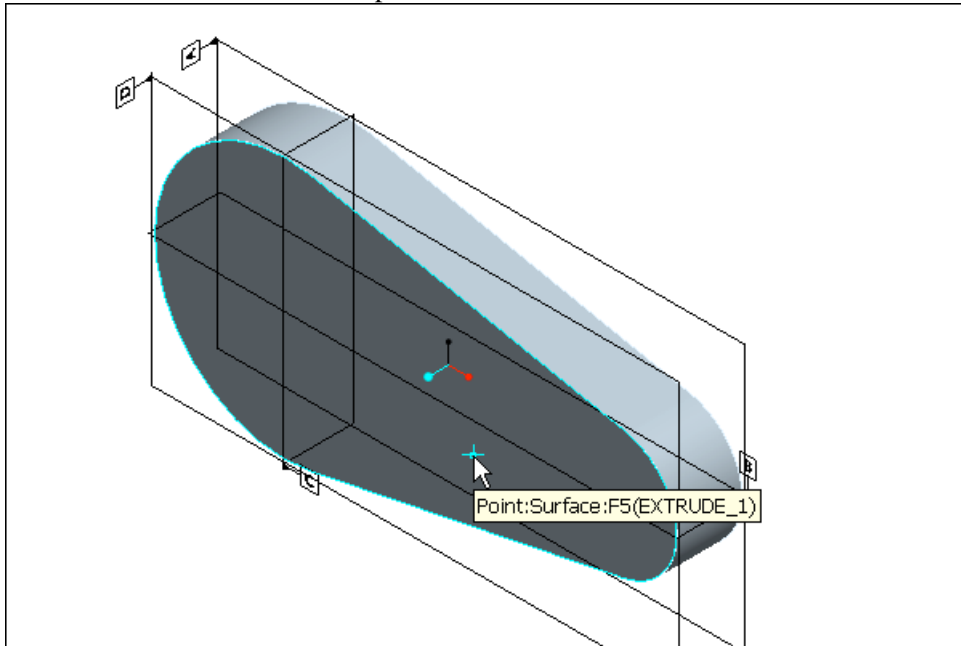
Click: > Set Datum Tag > OK > Name D > Geometry




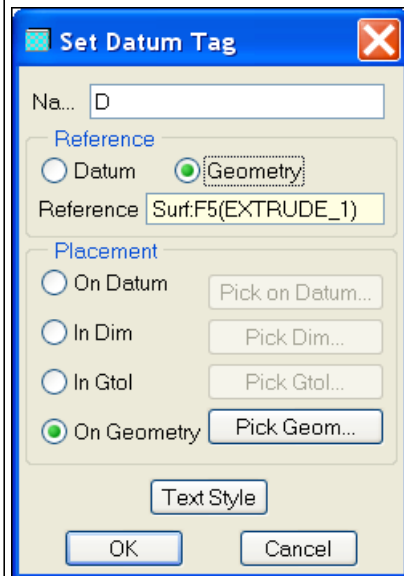
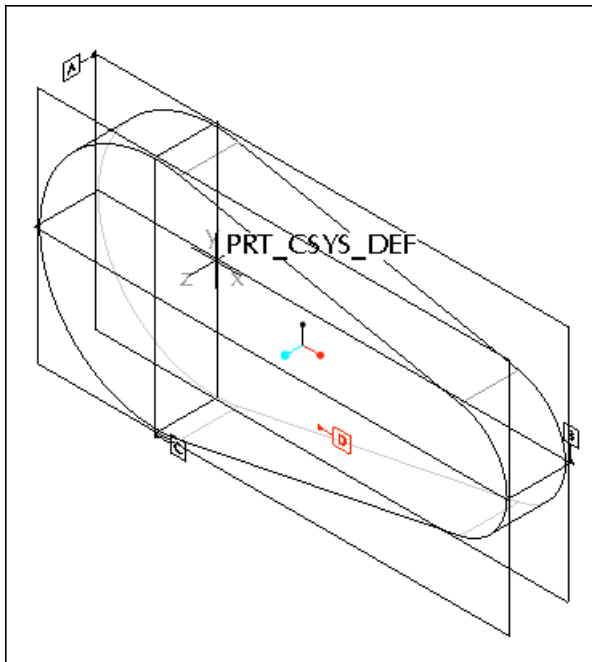
Pick the front planar face

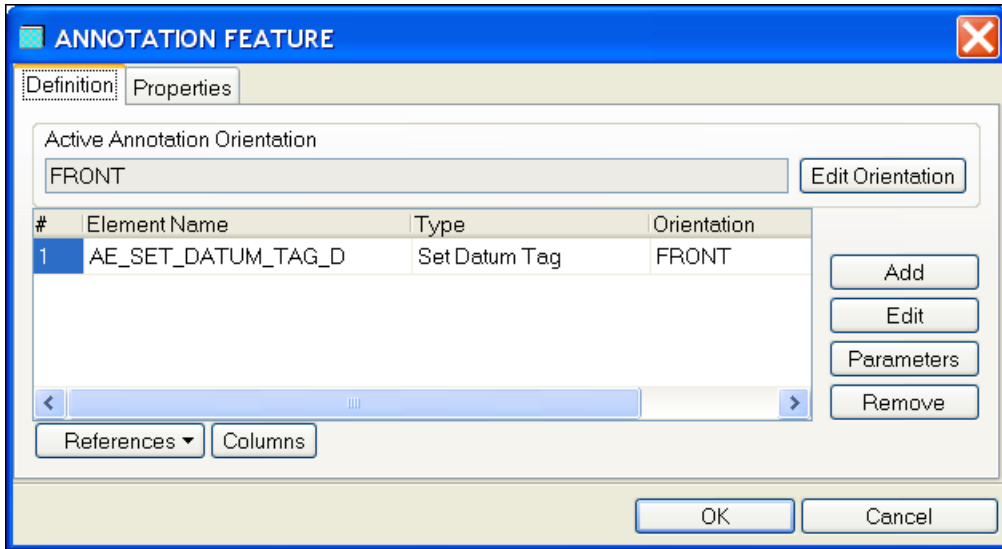


Pick on the front surface of the part



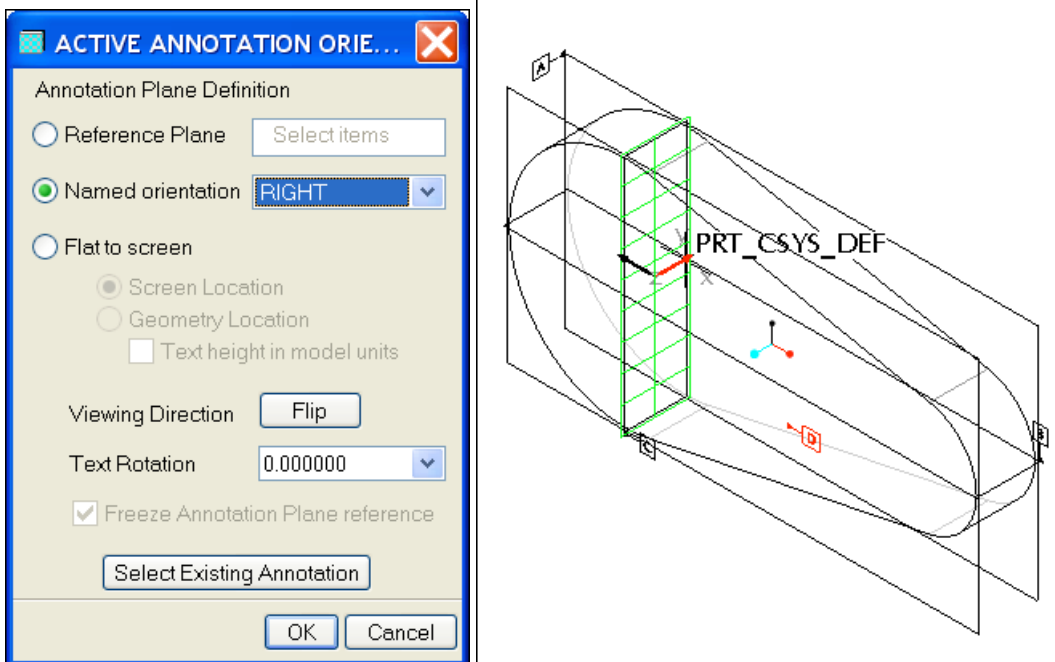
Click:  > OK





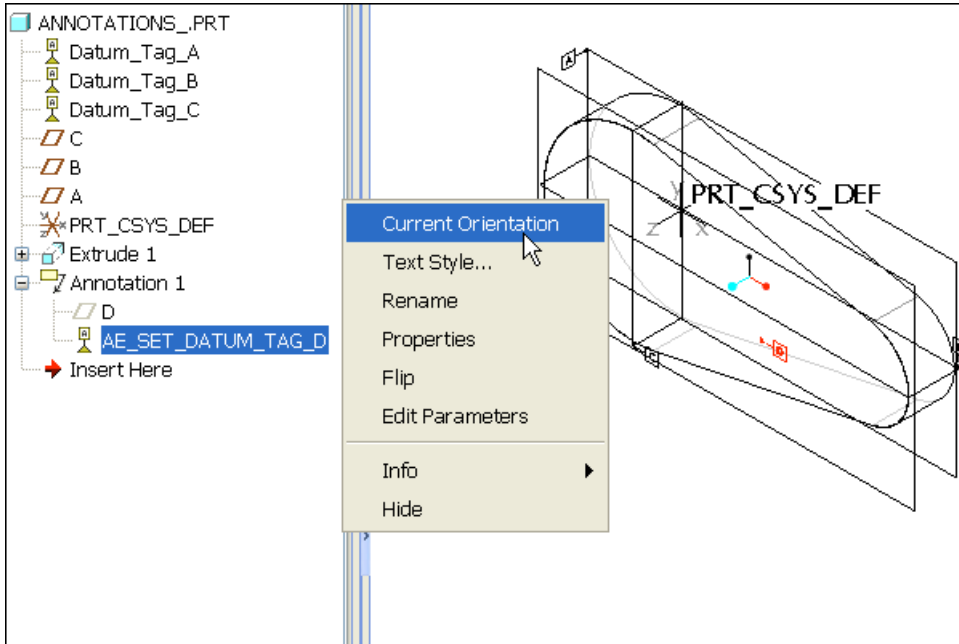
Click: OK

Click: View > Annotation Orientation > Named orientation > [v] > RIGHT

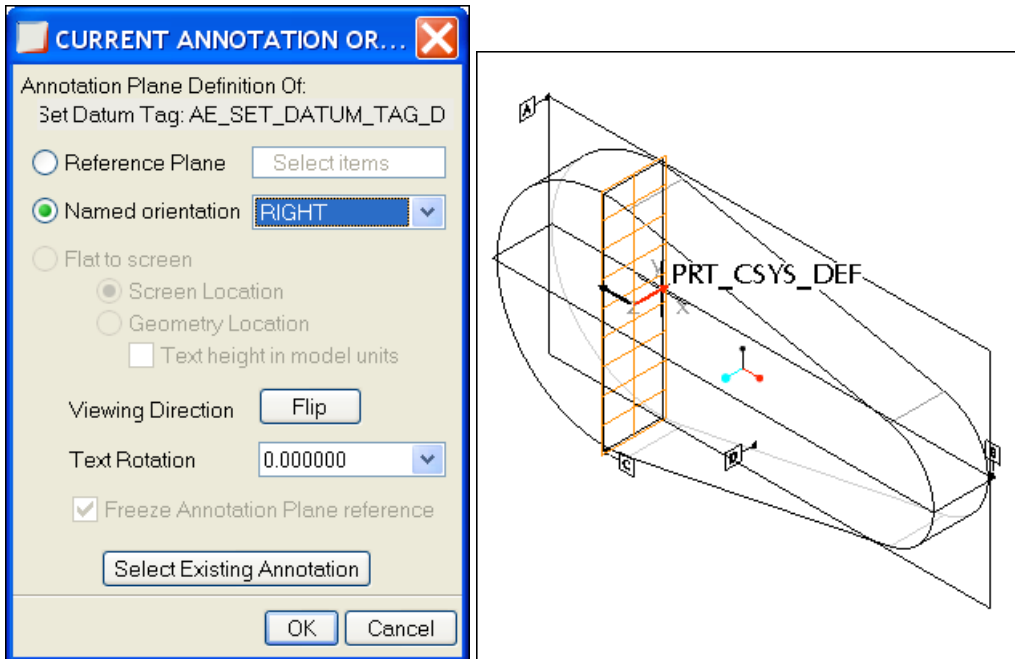


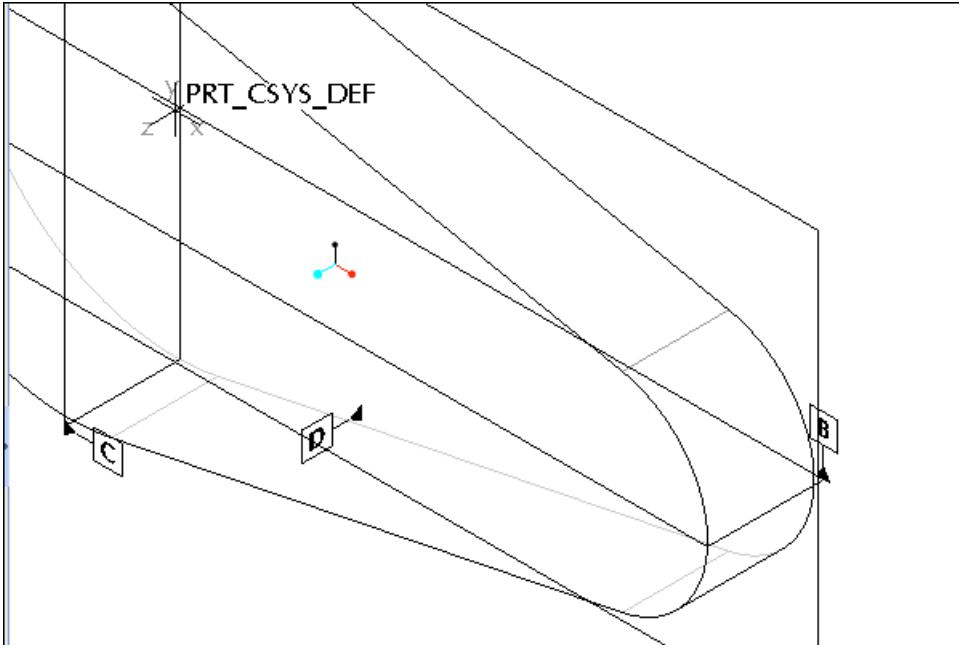
Click: OK



Select: AE_SET_DATUM_TAG_D from the Model Tree > RMB > Current Orientation

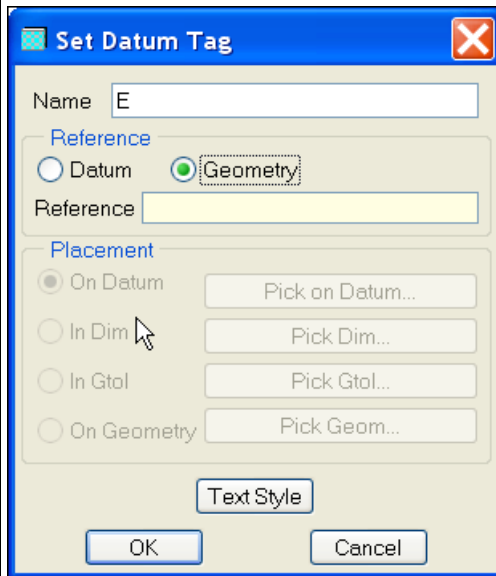
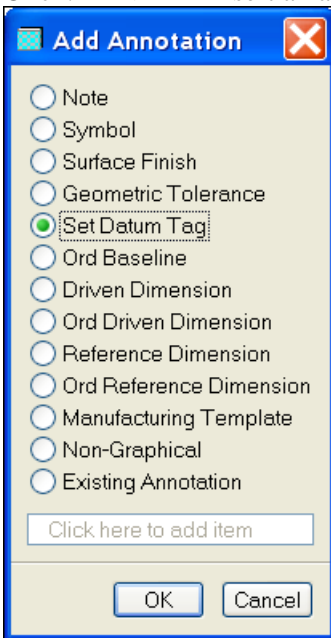


Click: Named orientation > [dropdown] > RIGHT > OK

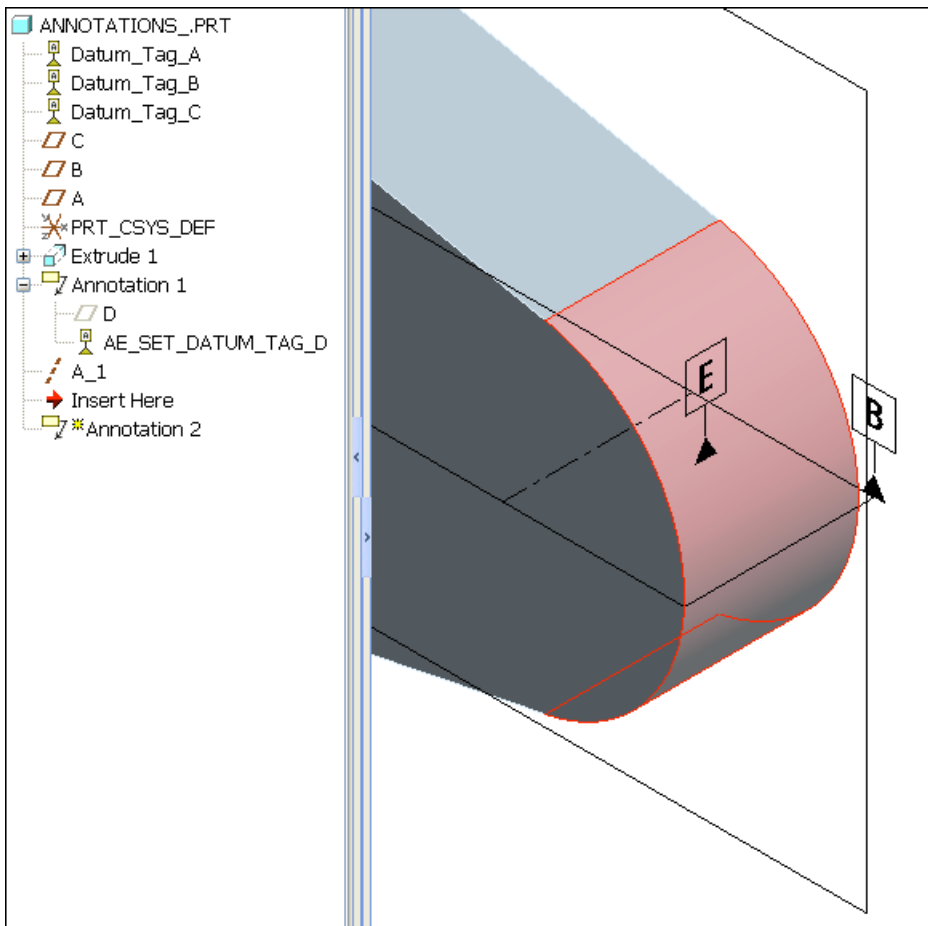
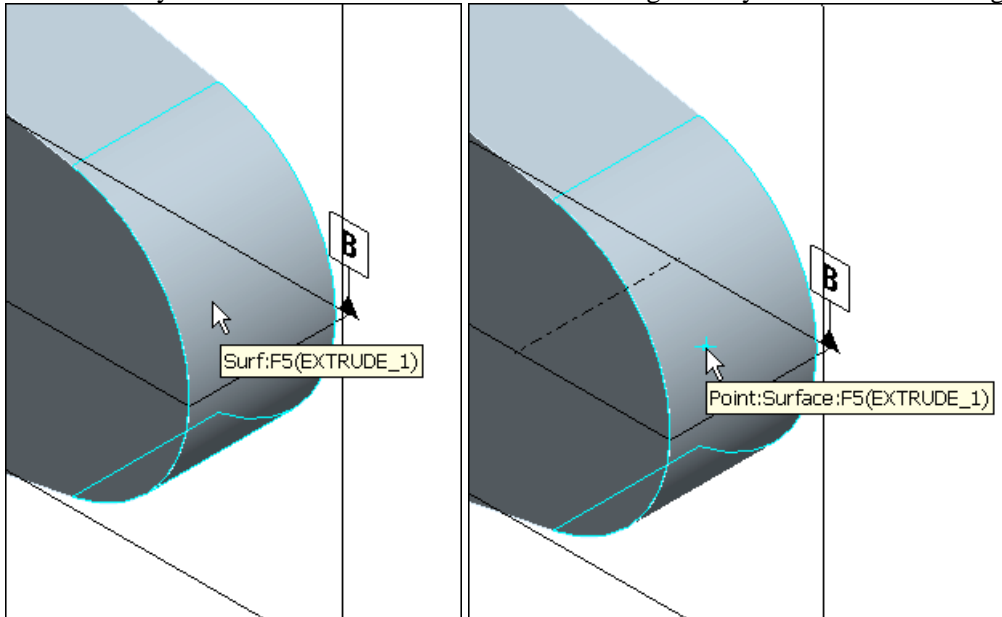




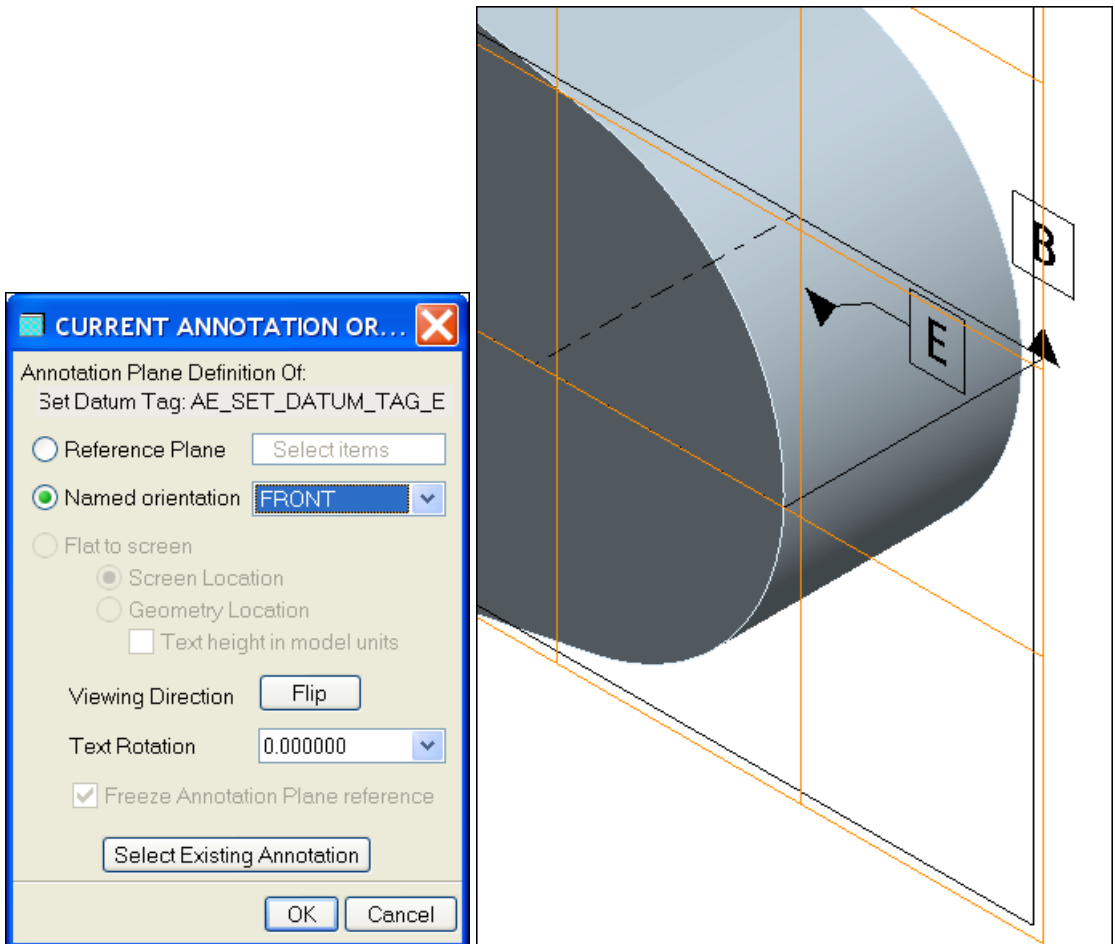
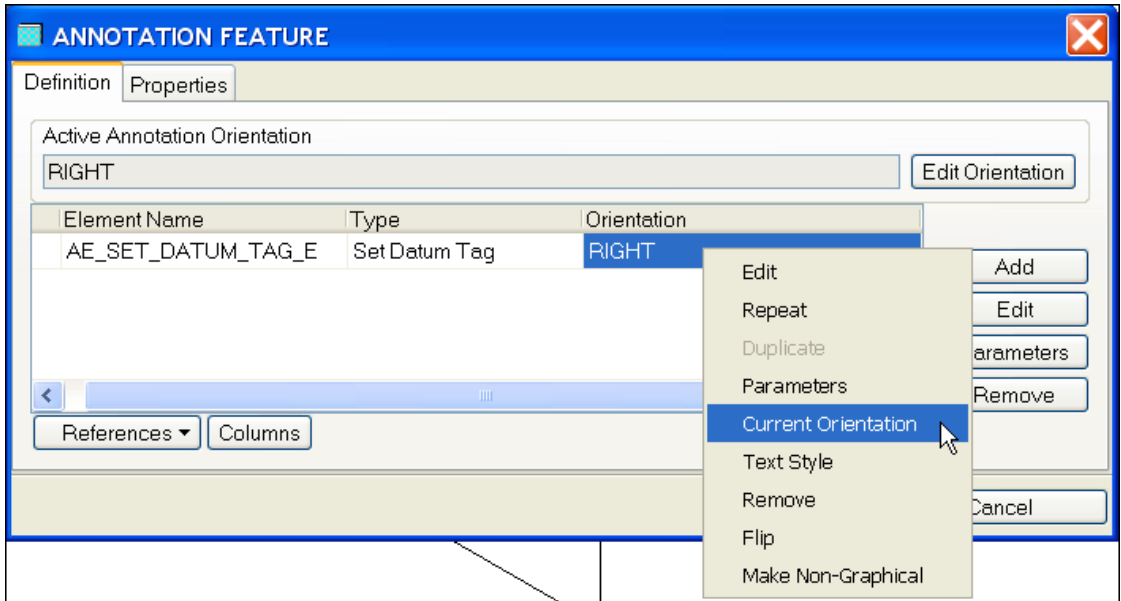
Click:  >  Insert an annotation feature > Set Datum Tag > OK > Name E > Geometry



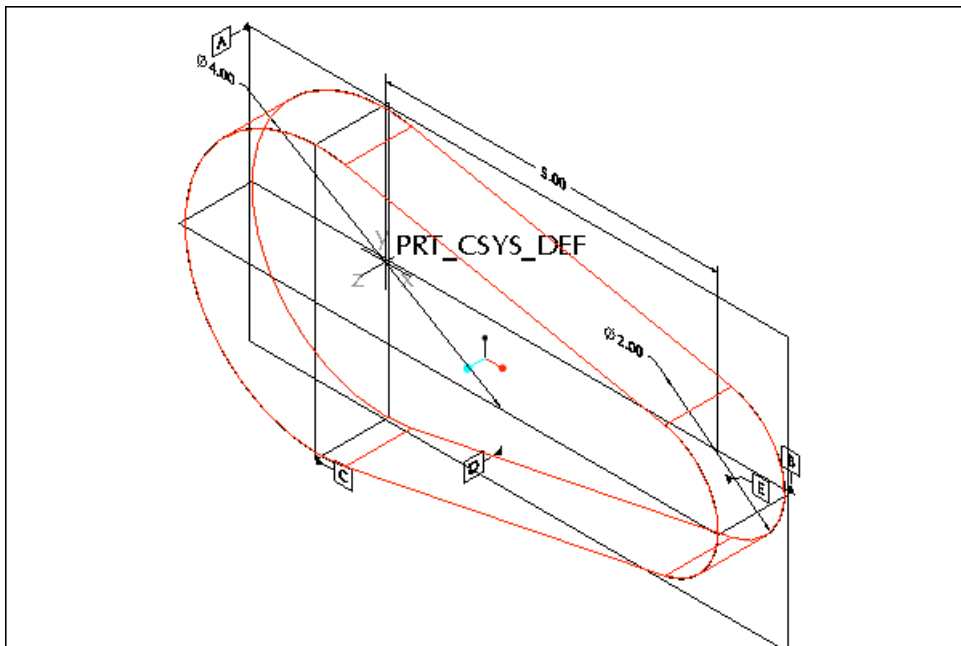
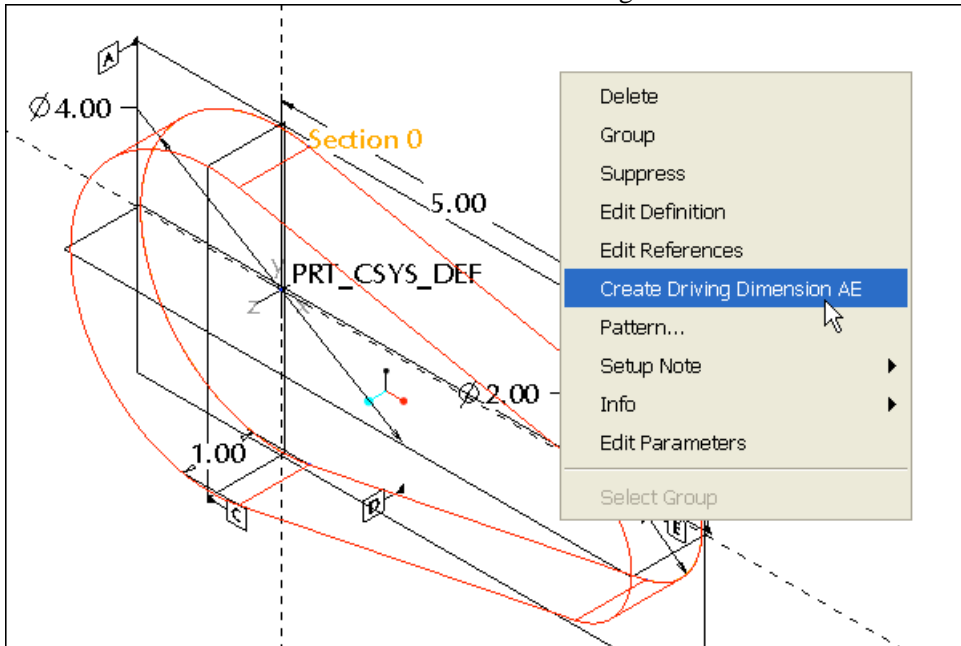
Pick on the cylindrical surface > select the attachment geometry for the set datum tag > OK




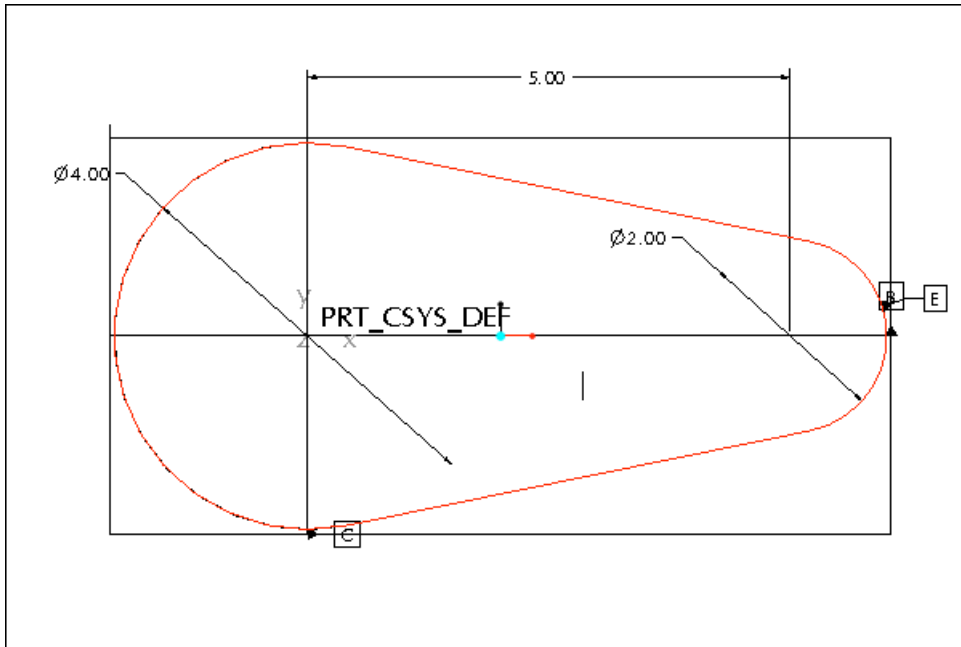
In the ANNOTATION FEATURE dialog box, click on: Orientation: RIGHT > RMB > Current Orientation > Named orientation > FRONT > OK > OK




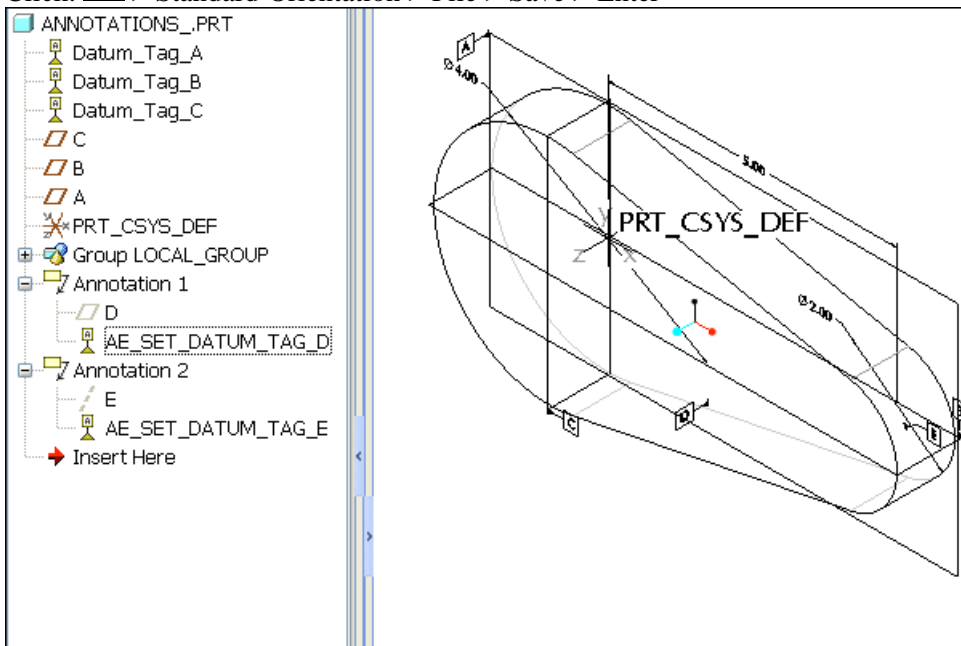
Click: View > Annotation Orientation > Named Orientation > FRONT > OK >  > double-click on the Extrude feature > RMB > Create Driving Dimension AE



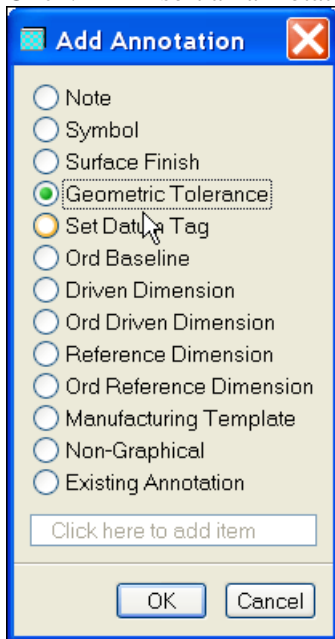
Click:  > FRONT



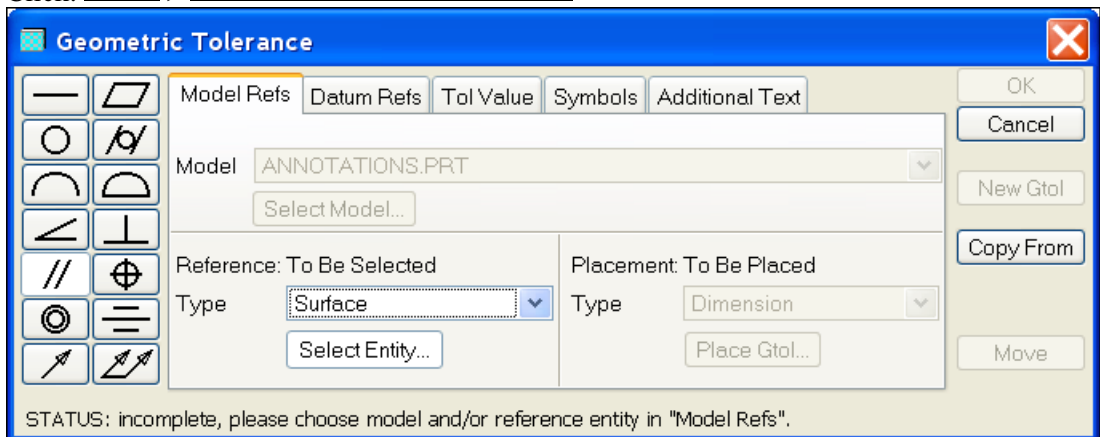
Click:  > Standard Orientation > File > Save > Enter



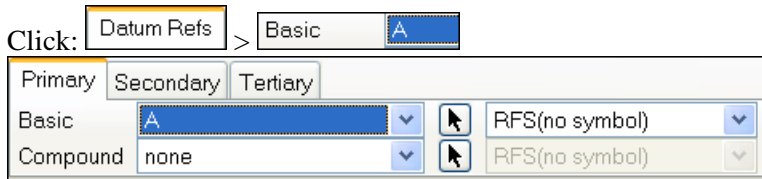
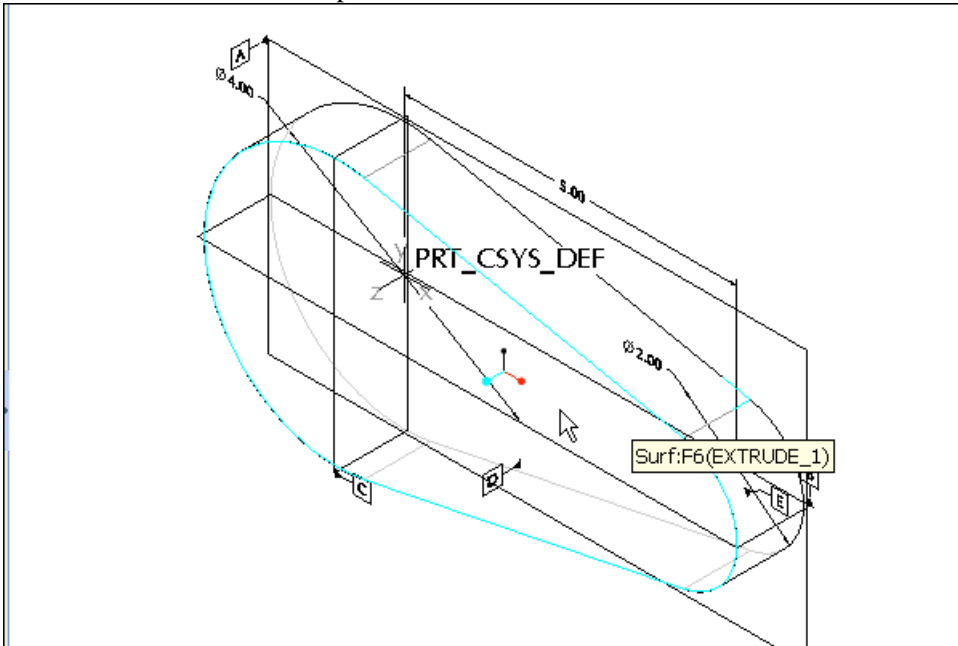
Click:  Insert an annotation feature > Geometric Tolerance > OK



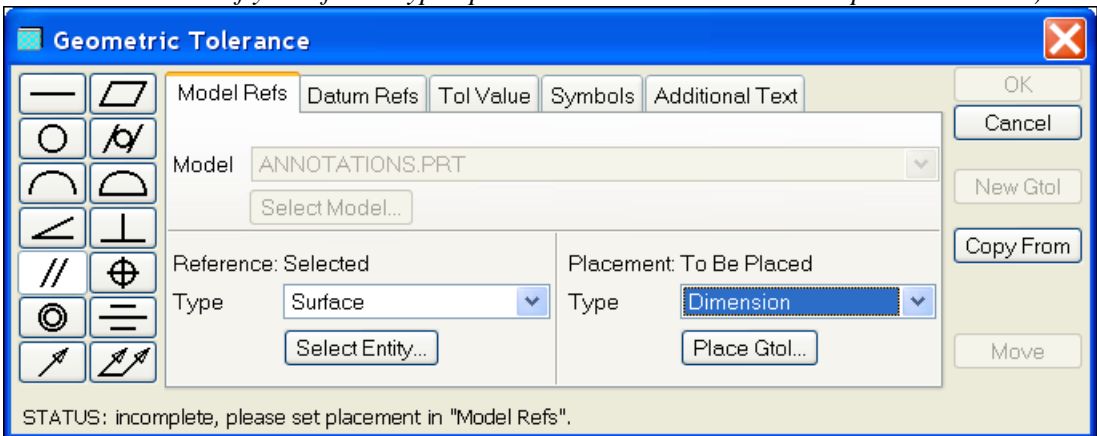
Click:  > Type:



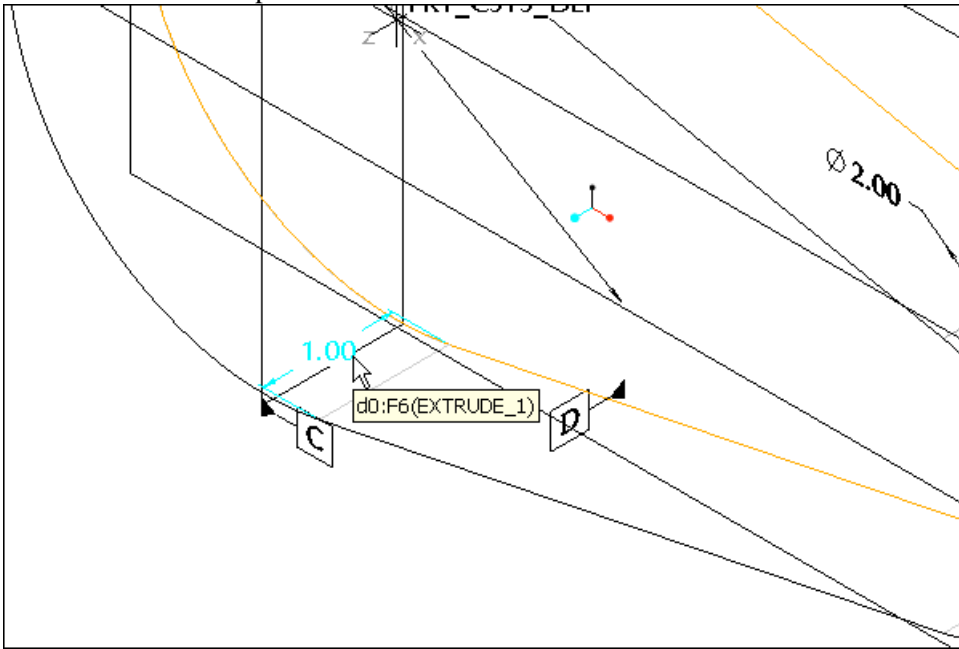
Pick the front surface of the part



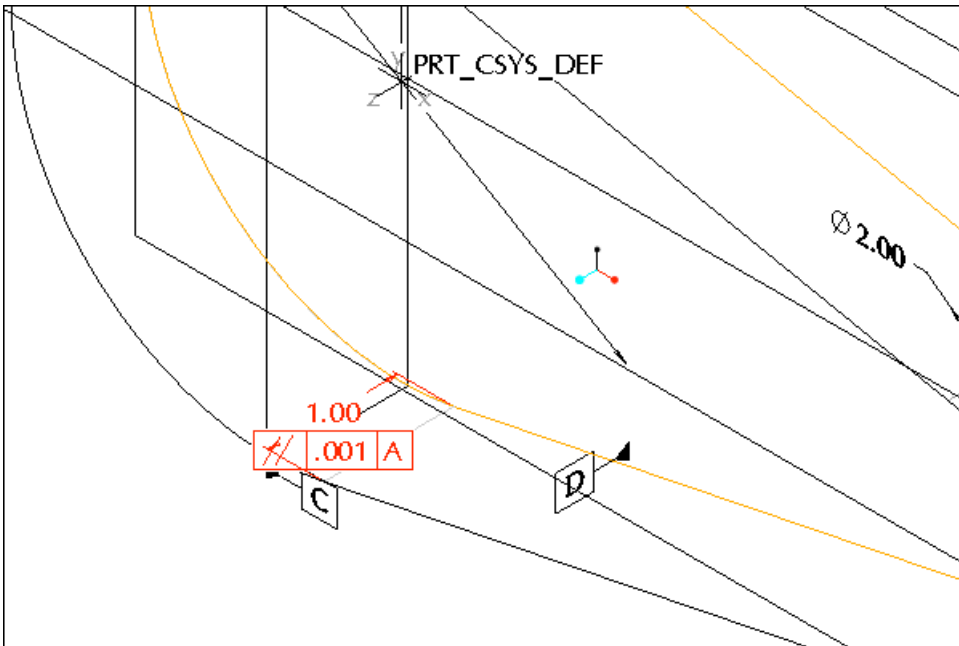
Click: Model Refs > Type Dimension (Dimension should already be selected by default. If you pick and select Dimension, the Select dialog box will open. The same result would occur if you left the Type option set to Dimension and then pick Place Gtol)



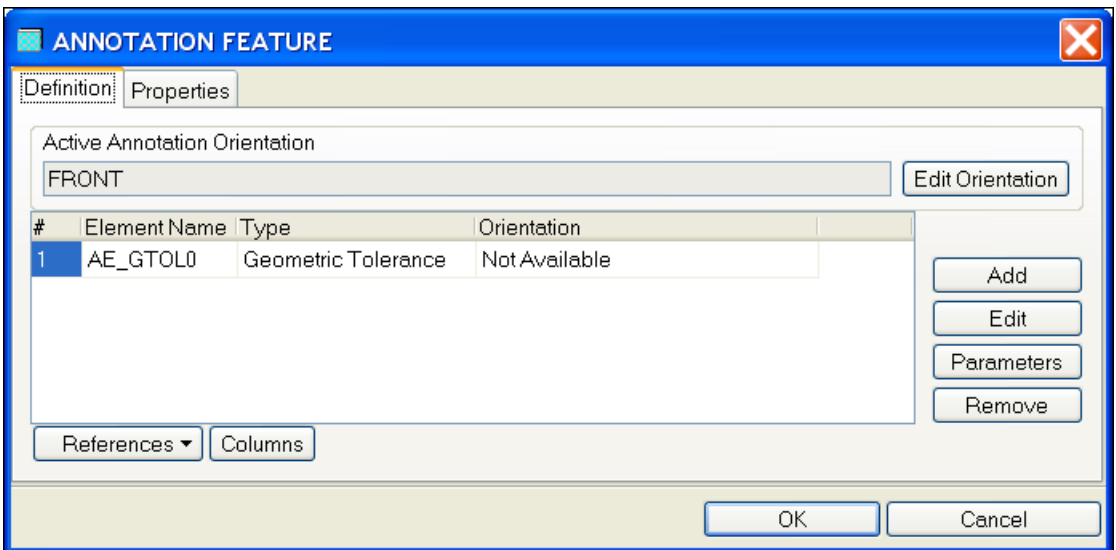
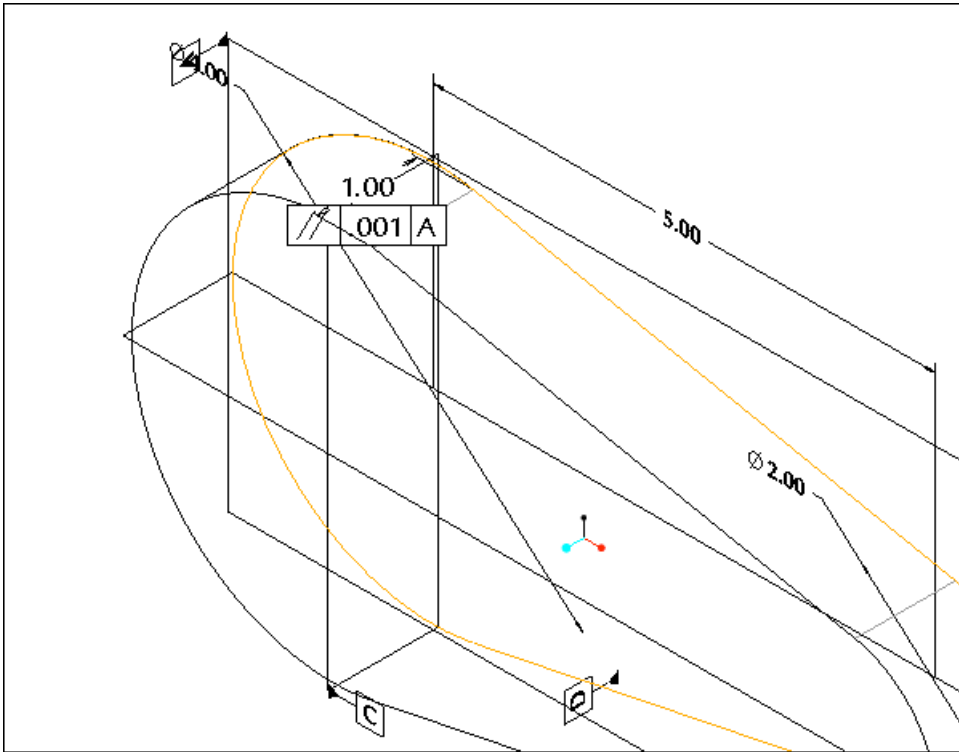
Click: > pick on the extrusion



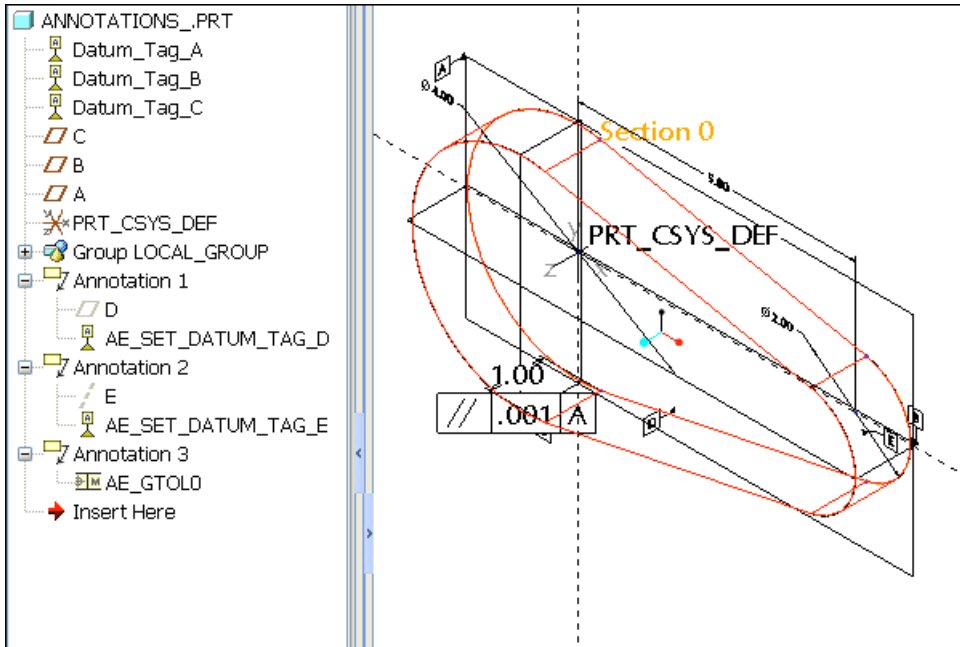
Pick on the 1.00 dimension




Click: OK > OK

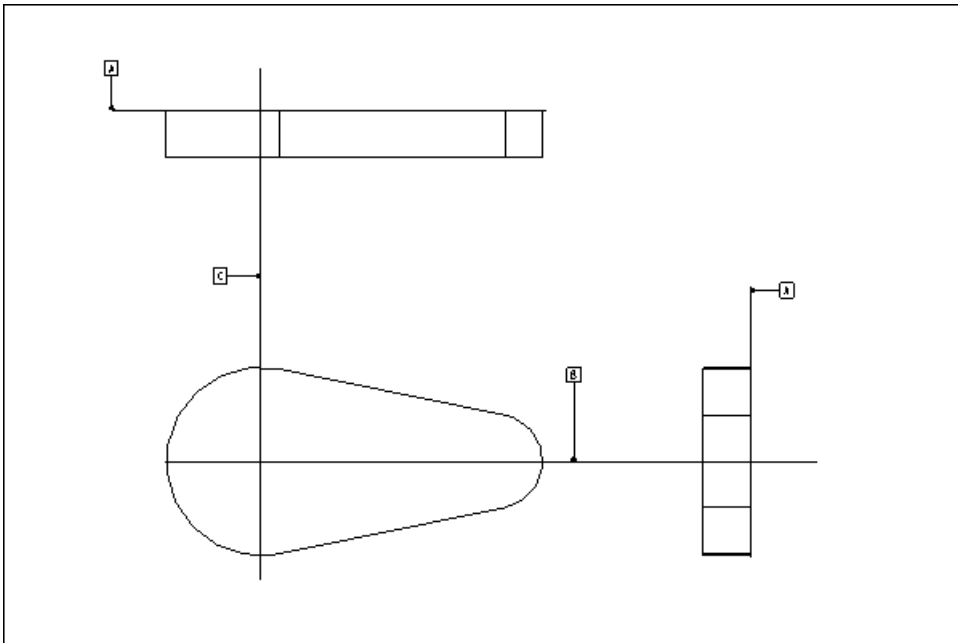


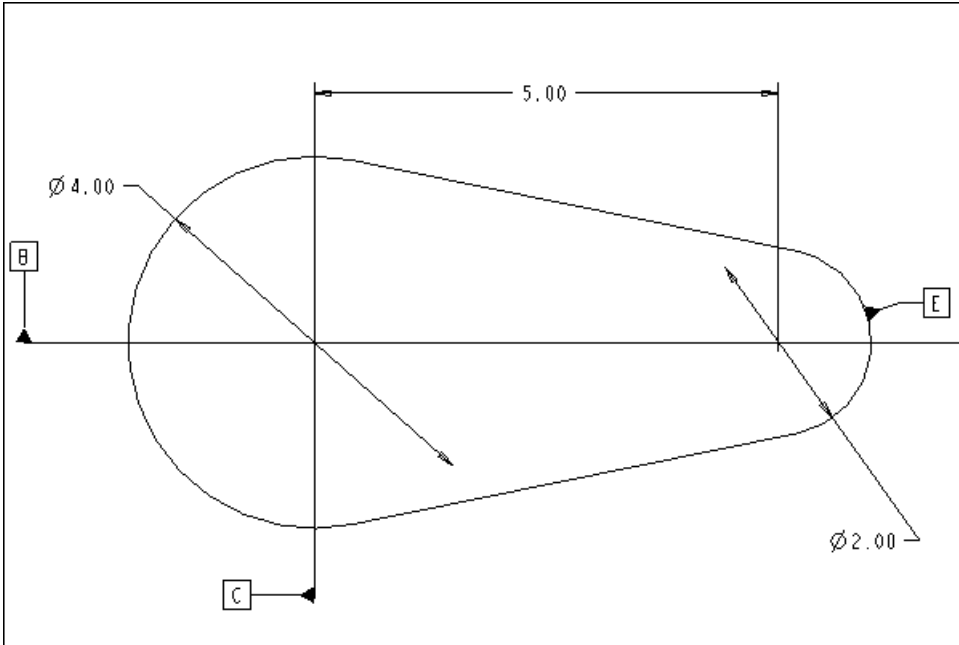
Double-click on the model > File > Save > Enter




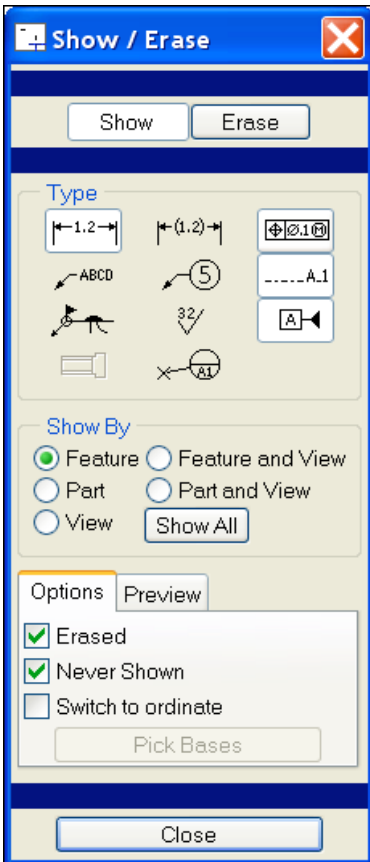
Click: File > Open > Drawing > In Session > annotations.drw > Open >  off > View > Repaint

Erase the unwanted tags and adjust the datum tags. Select a datum > move it or RMB > Erase as needed

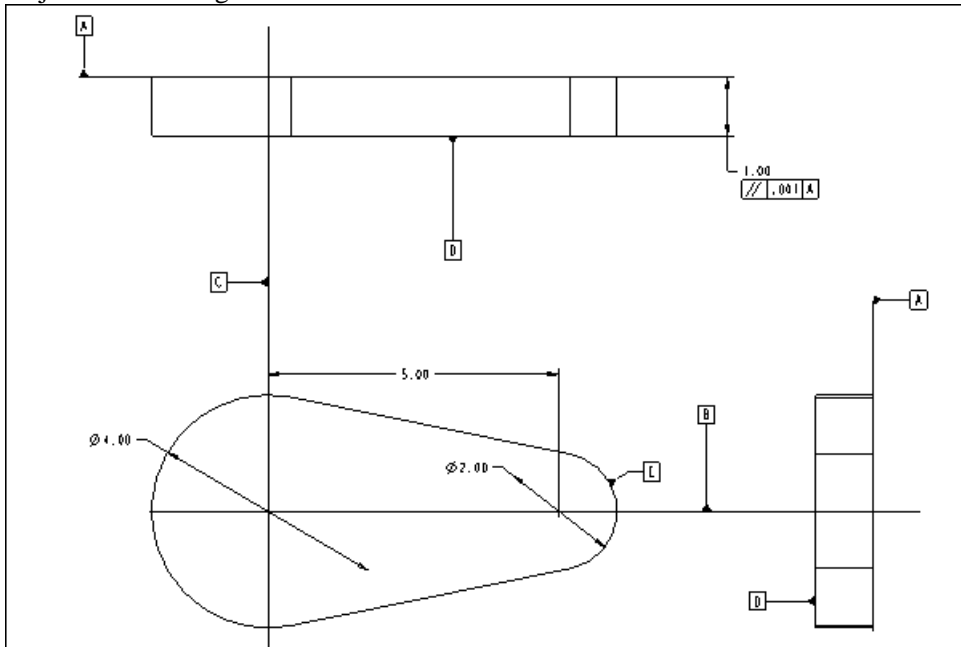




Click:  > toggle on Axis, Geometric Tolerance, Datum Plane, and Dimension > Show All > Yes > OK > Close > LMB to deselect



Adjust the drawing elements as needed



Click: File > Save > Enter > File > Close Window > Window > Close