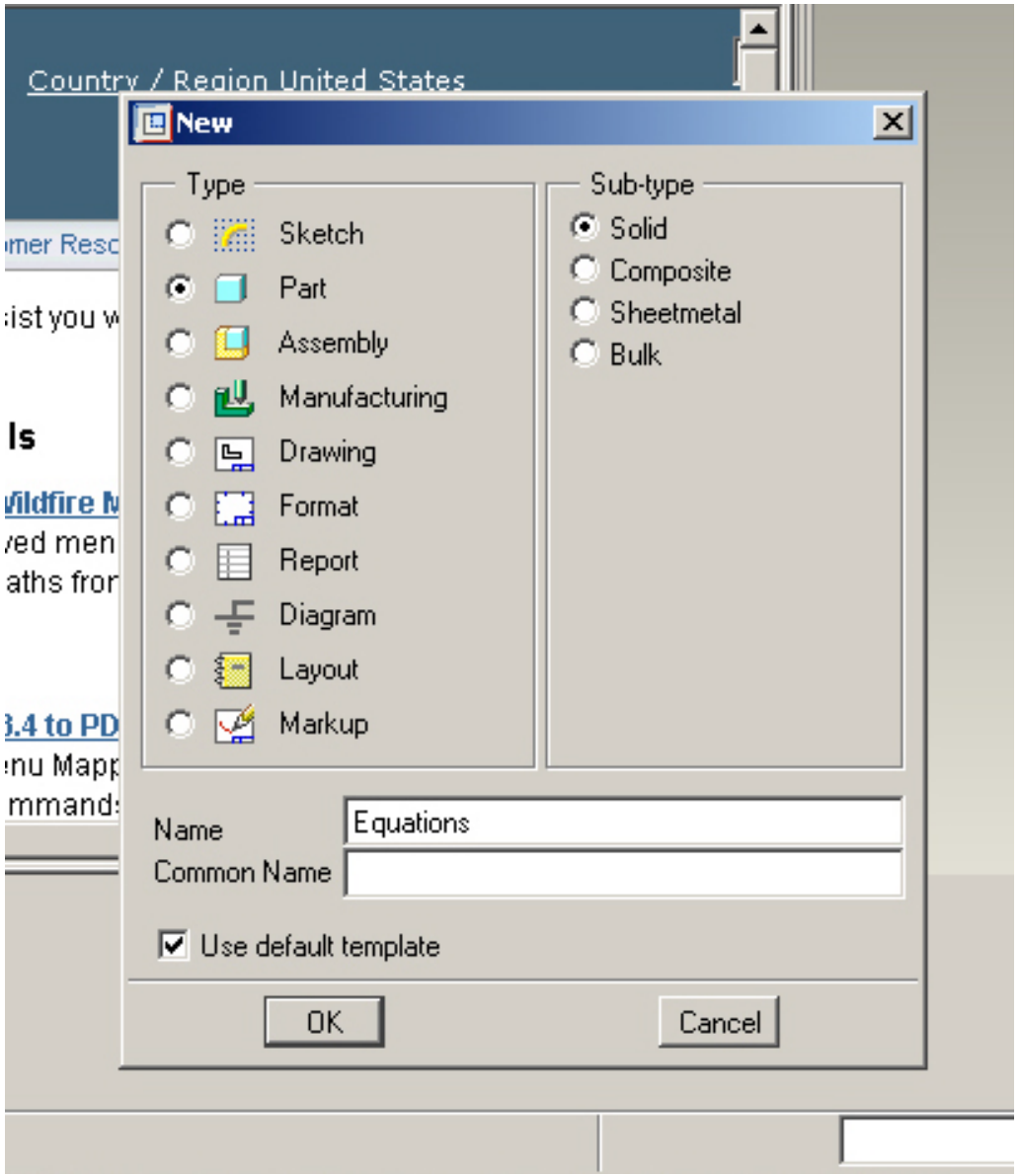
Sketching from Equations in WildFire 2.0

Creating new part

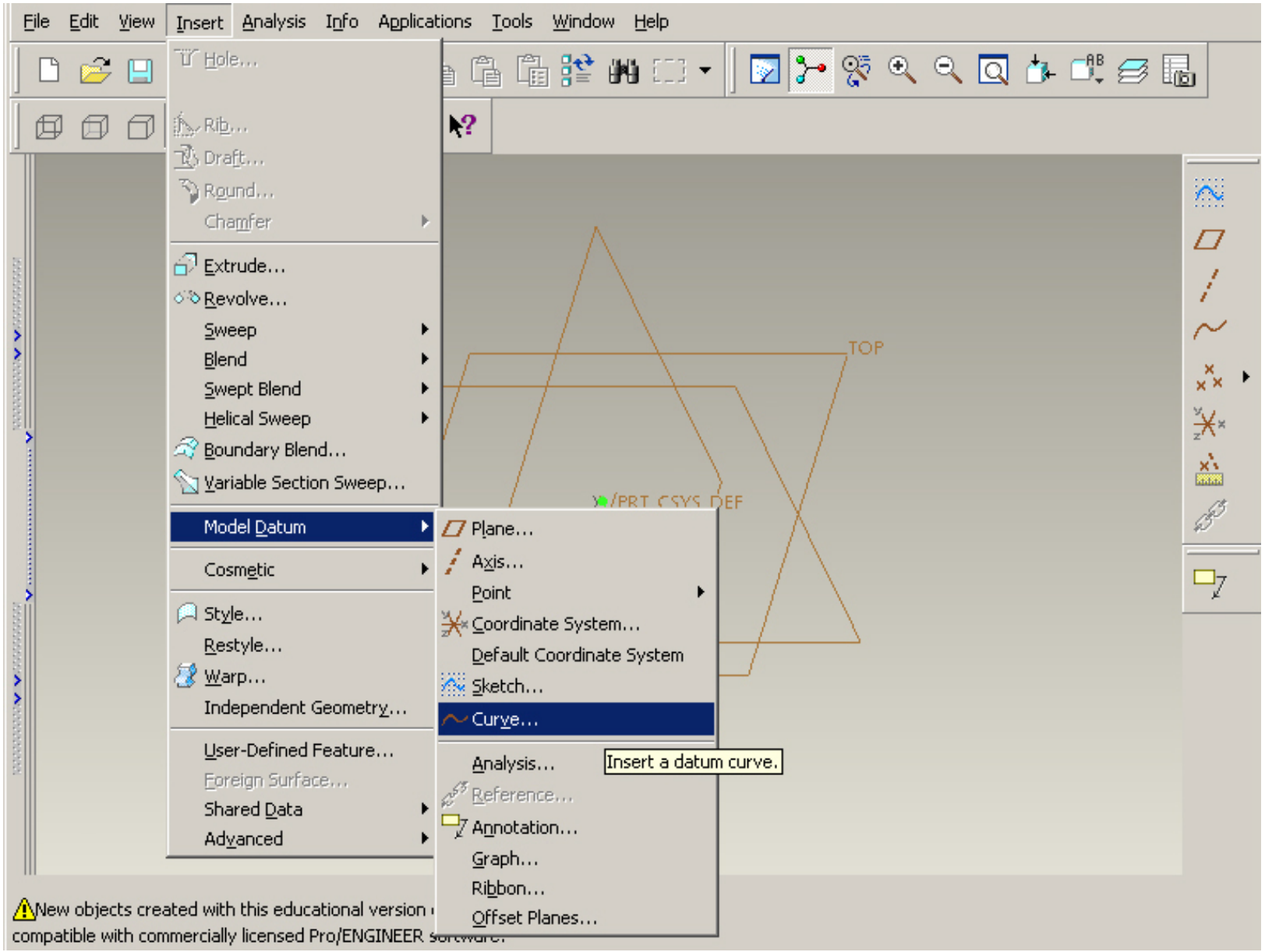
Start Pro/ENGENEER and begin a new part. Open *New document*, select *Part*. Name the part ***Equations***

Accept the default template for part.



Creating sketch

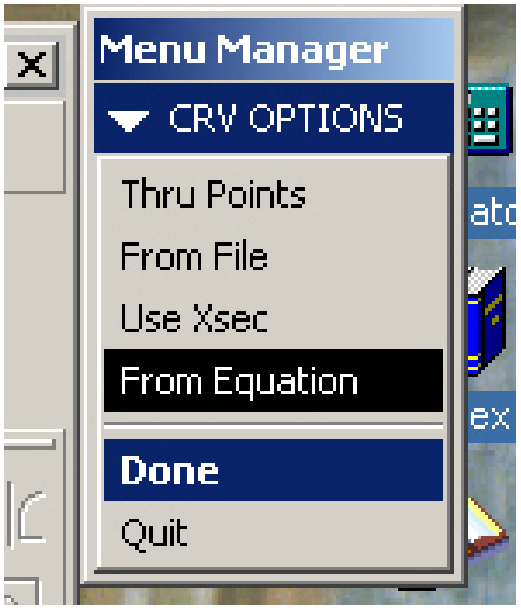
To create the sketch based on equation we have to provide a datum curve for this equation.
For this insert a Datum Curve: **Insert** – **Model Datum** – **Curve**…



Menu Manager window appears where we can select **different options for creating curve**:

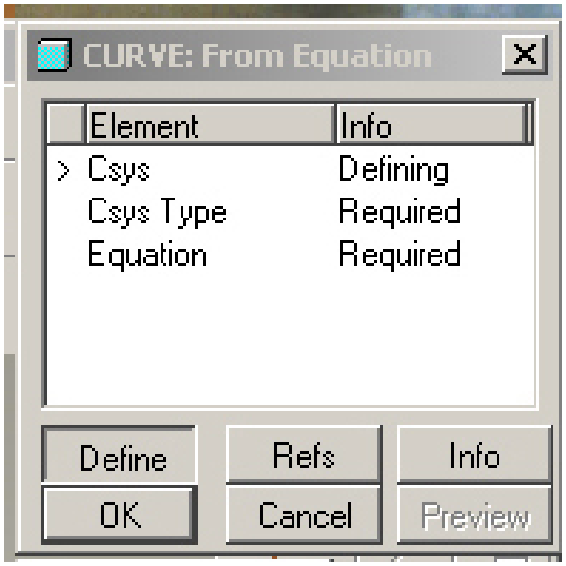
- through points
- from file
- using section line
- from equation

For our purpose we select *From Equation – Done*



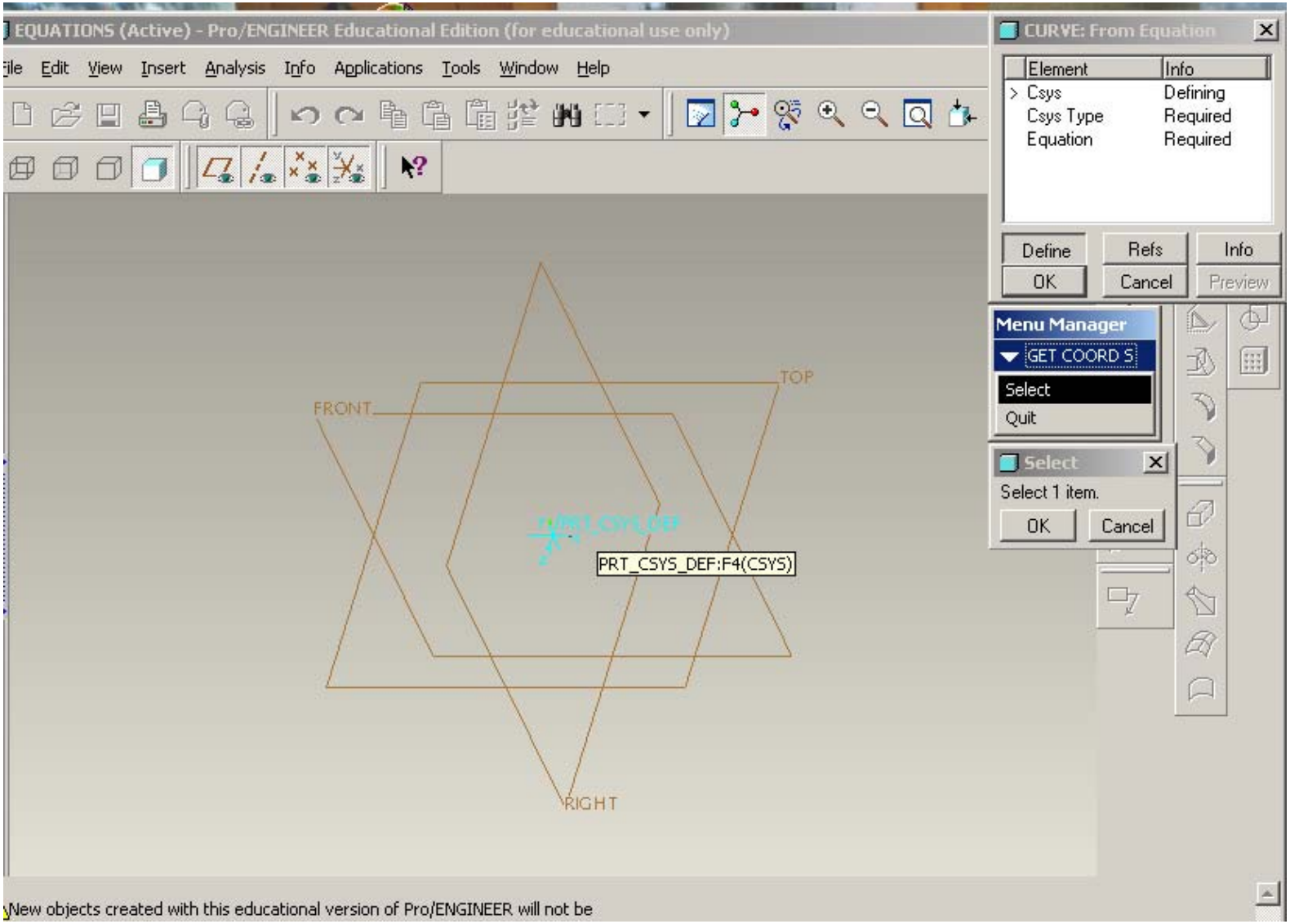
and then the window **CURVE: From Equation** pops up. Now we have to

- select coordinate system in order to relate the curve to the model
- change/keep the type of the coordinate system in which the curve will be built
- define the equation describing the curve in this coordinate system.



1st step. The coordinate system should be selected from the list of existing coordinate systems.
In our case we have only one CS in the model, and this system is offered as a default system in the window.

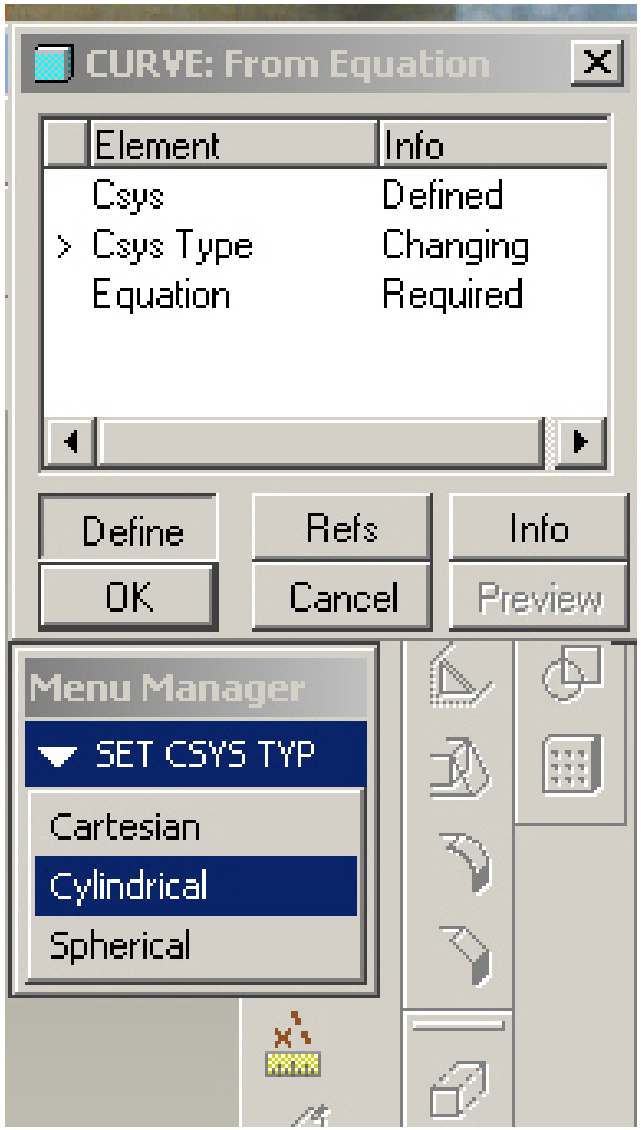
Select the **Default Coordinate System** in the Graphics window by simple clicking on it



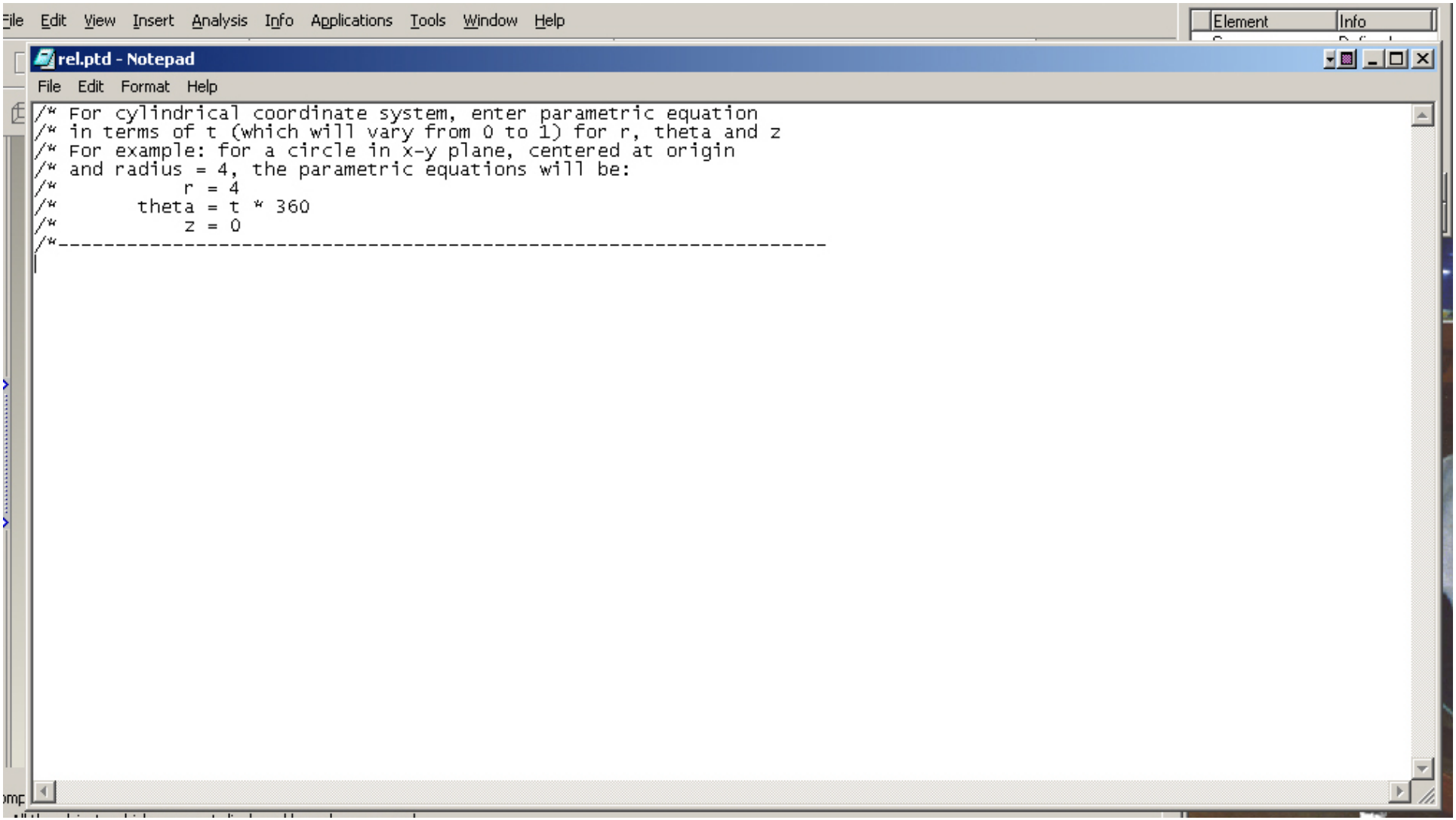
2nd step. Three types of coordinate systems are available

- Cartesian
- Cylindrical
- Spherical

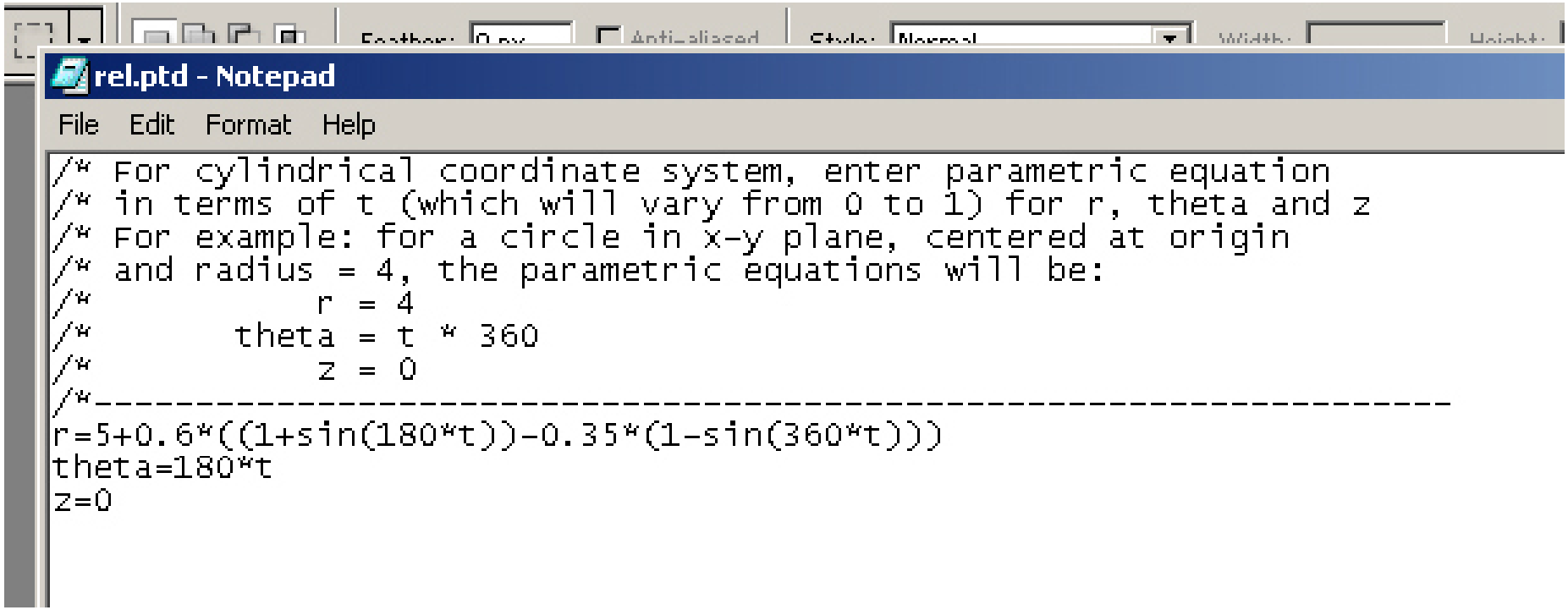
To plot the curve let's change the type of the coordinate system to cylindrical



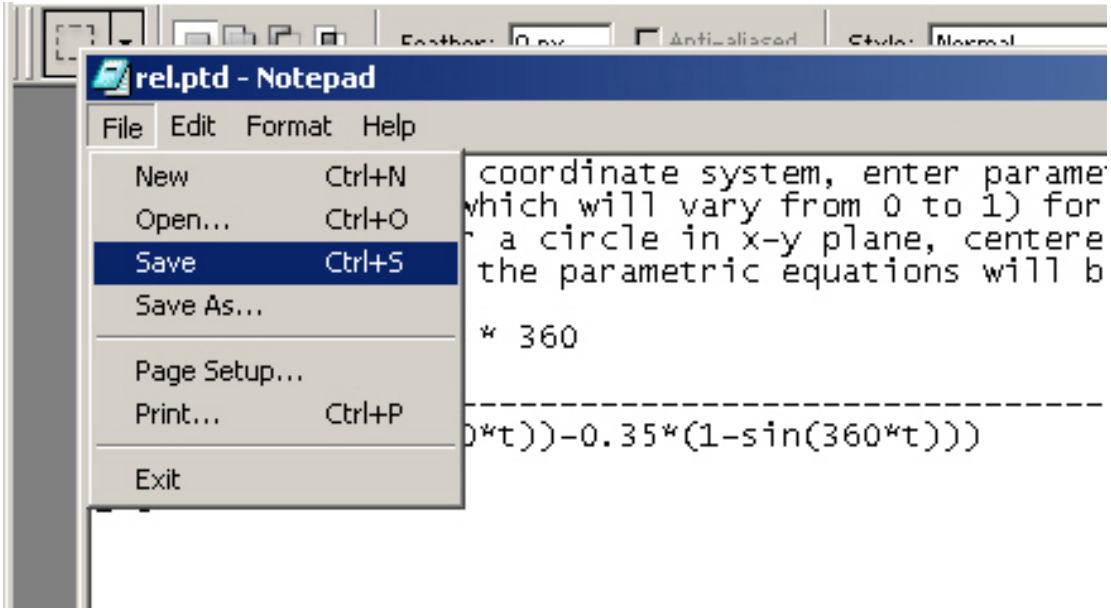
3rd step. The form for typing equation appears in the **Notebook** window.
This form describes the template of the equation in cylindrical CS.



Below the template we have to type the equation in the format shown in the form



After typing equation in the Notebook open **File** and **Save**, as in any other Windows application, then close the window.



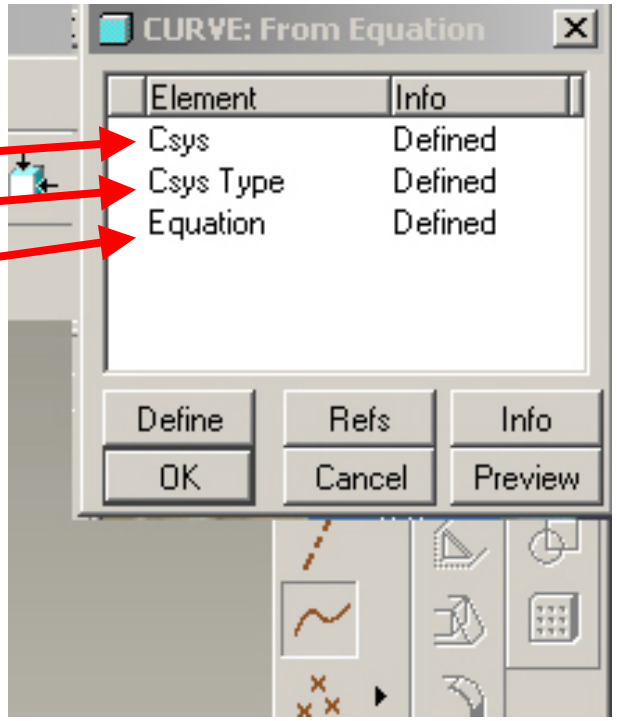
Now the **CURVE** window shows that

the coordinate system,

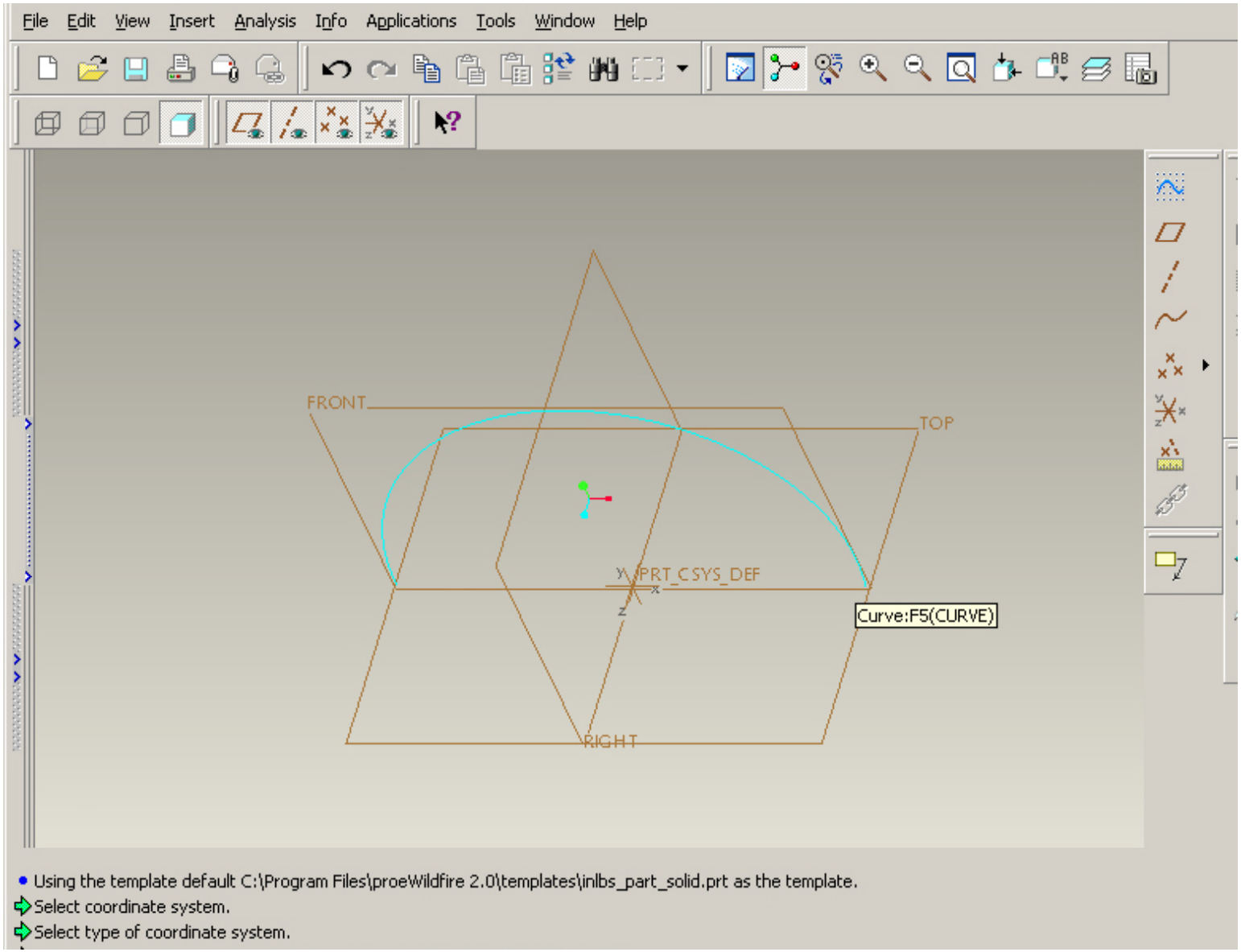
its type, and

the equation

are fully defined.



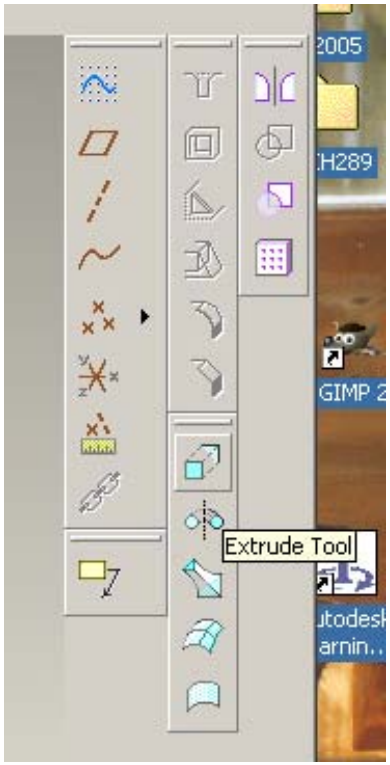
Click OK in the **CURVE** window and the Graphics window now shows the drawn curve.



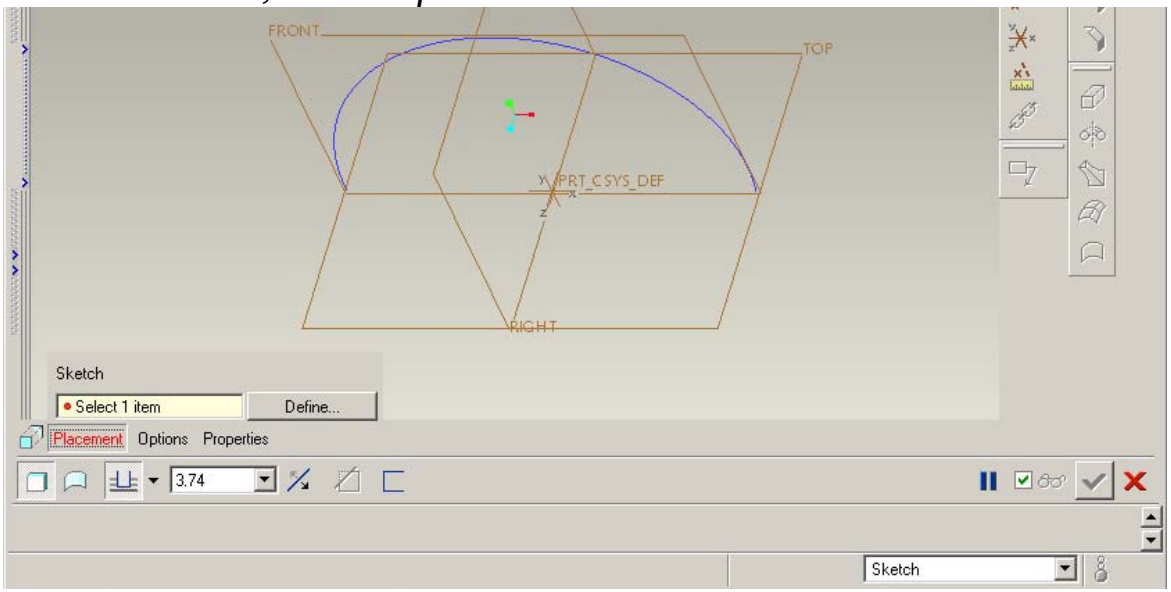
This profile has to be part of our extrude cross section. First of all, notice that the profile is drawn on the FRONT plane.

Making extrusion

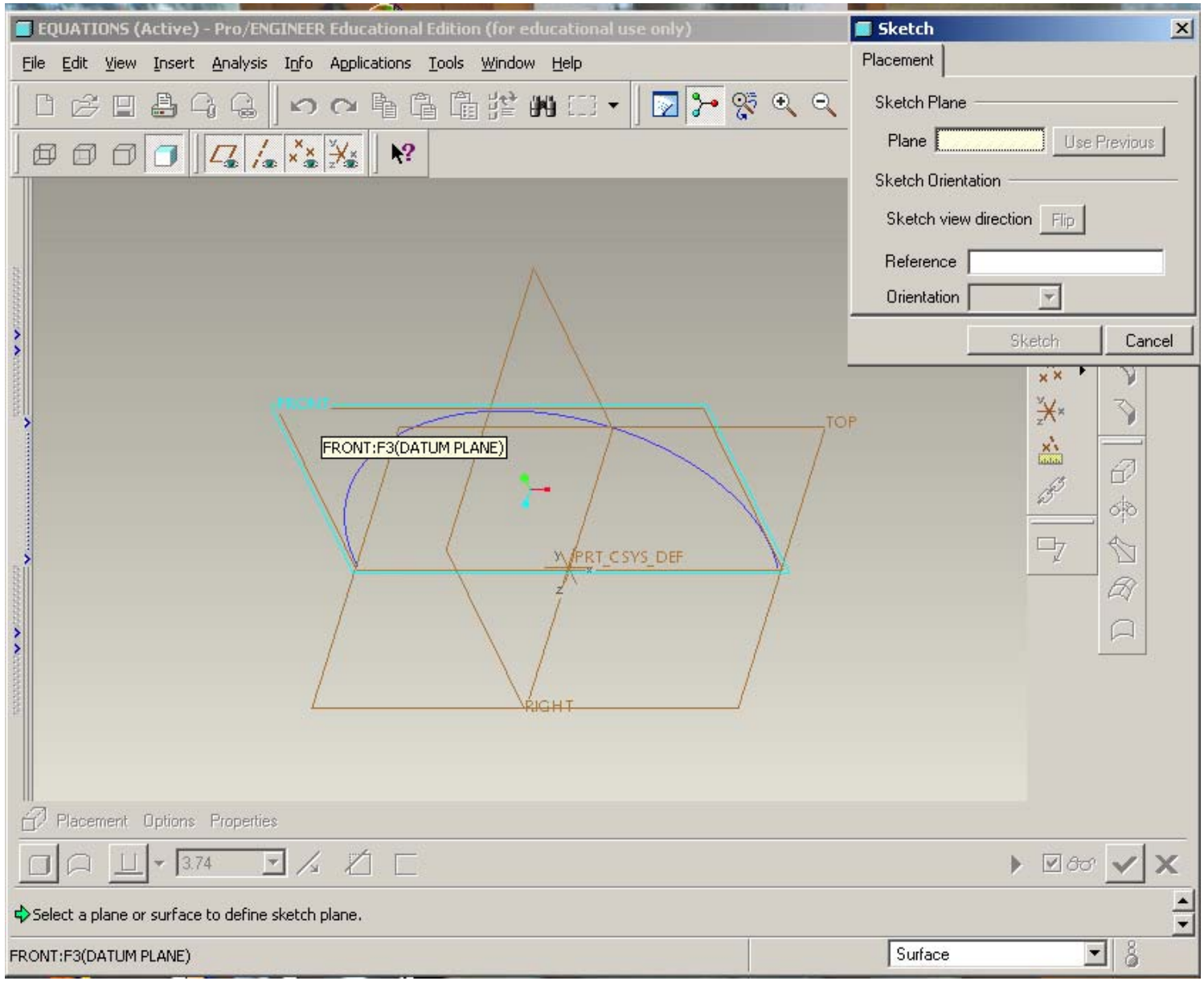
Click Extrude button on the right side of the main window.



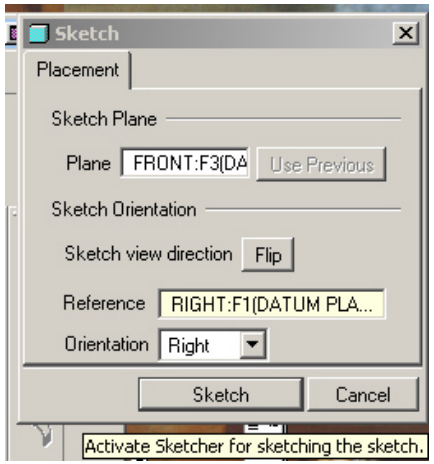
Then in the bottom select *Placement*, then *Define* button:



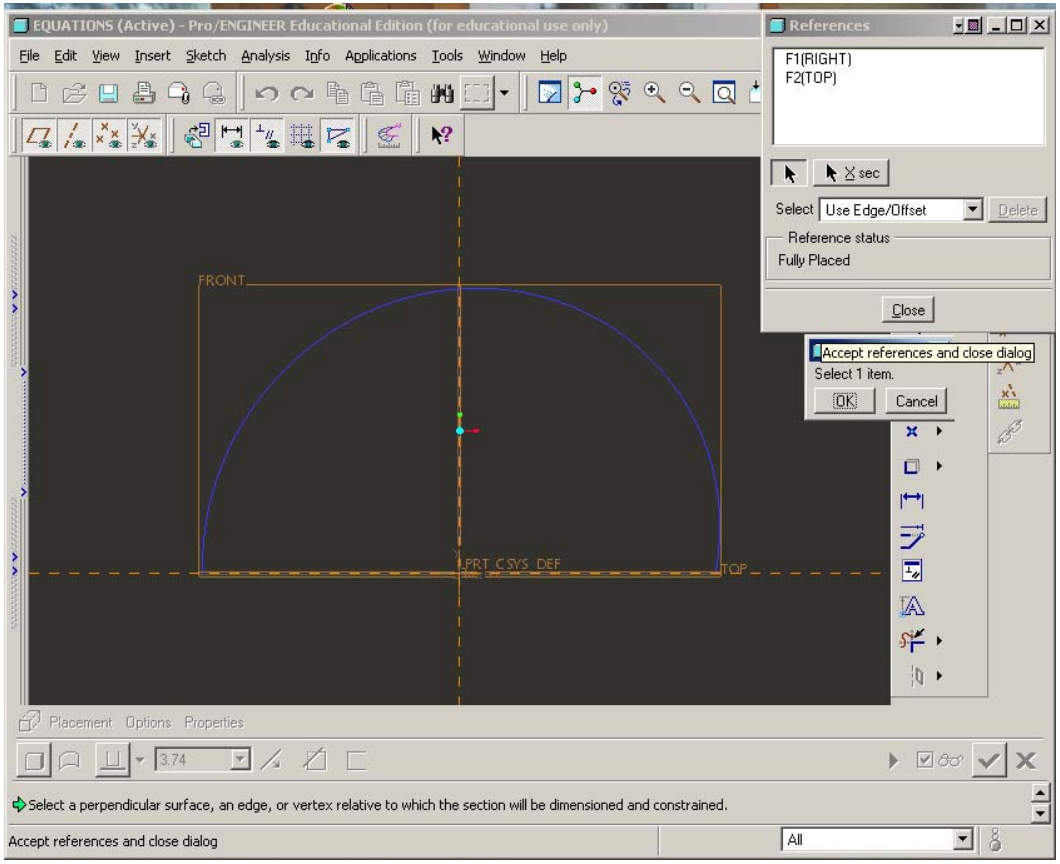
The **Sketch** window opens where the sketching plane must be defined.
Select the FRONT plane in the Graphics area as the sketching plane.



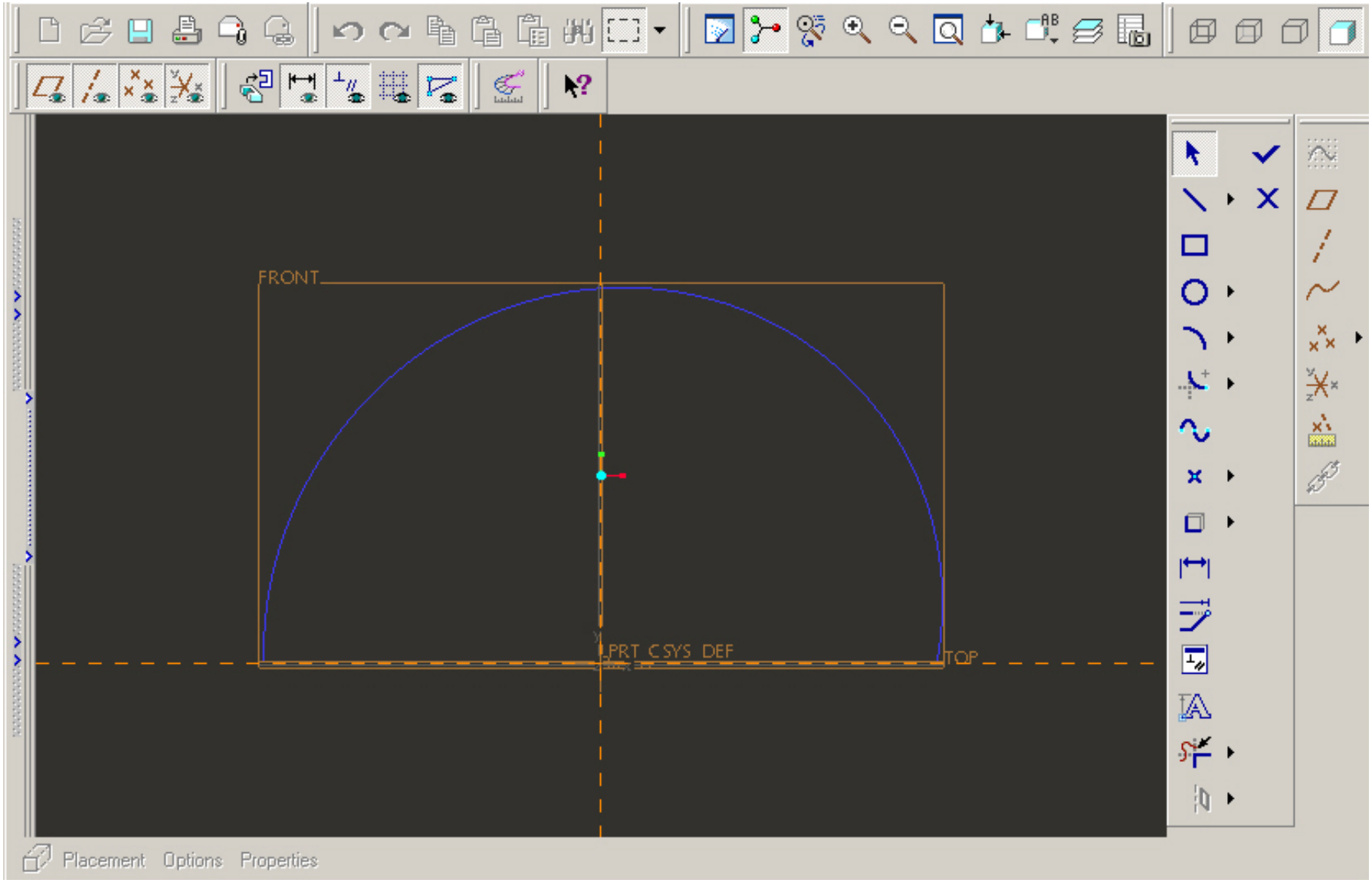
The name of the sketching plane appears in the **Sketch** window. Now click *Sketch* button in this window. This will activate Sketcher for sketching.



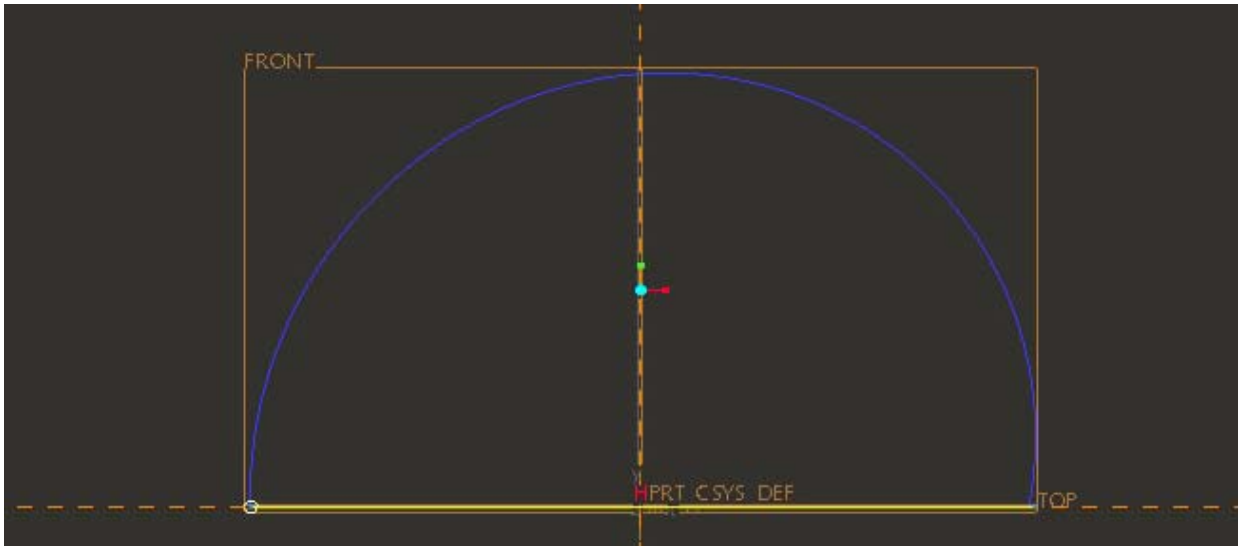
In the appearing **References** window default references will be offered. Accept these references and close the window.



Now the sketcher appears fully on the screen.

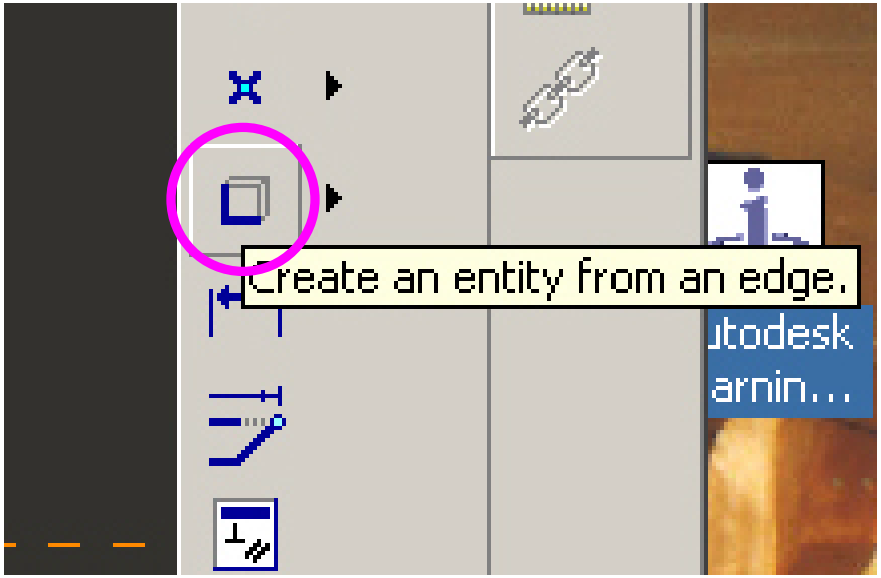


To create the sketch for the protrusion draw a line to join the ends of the profile.

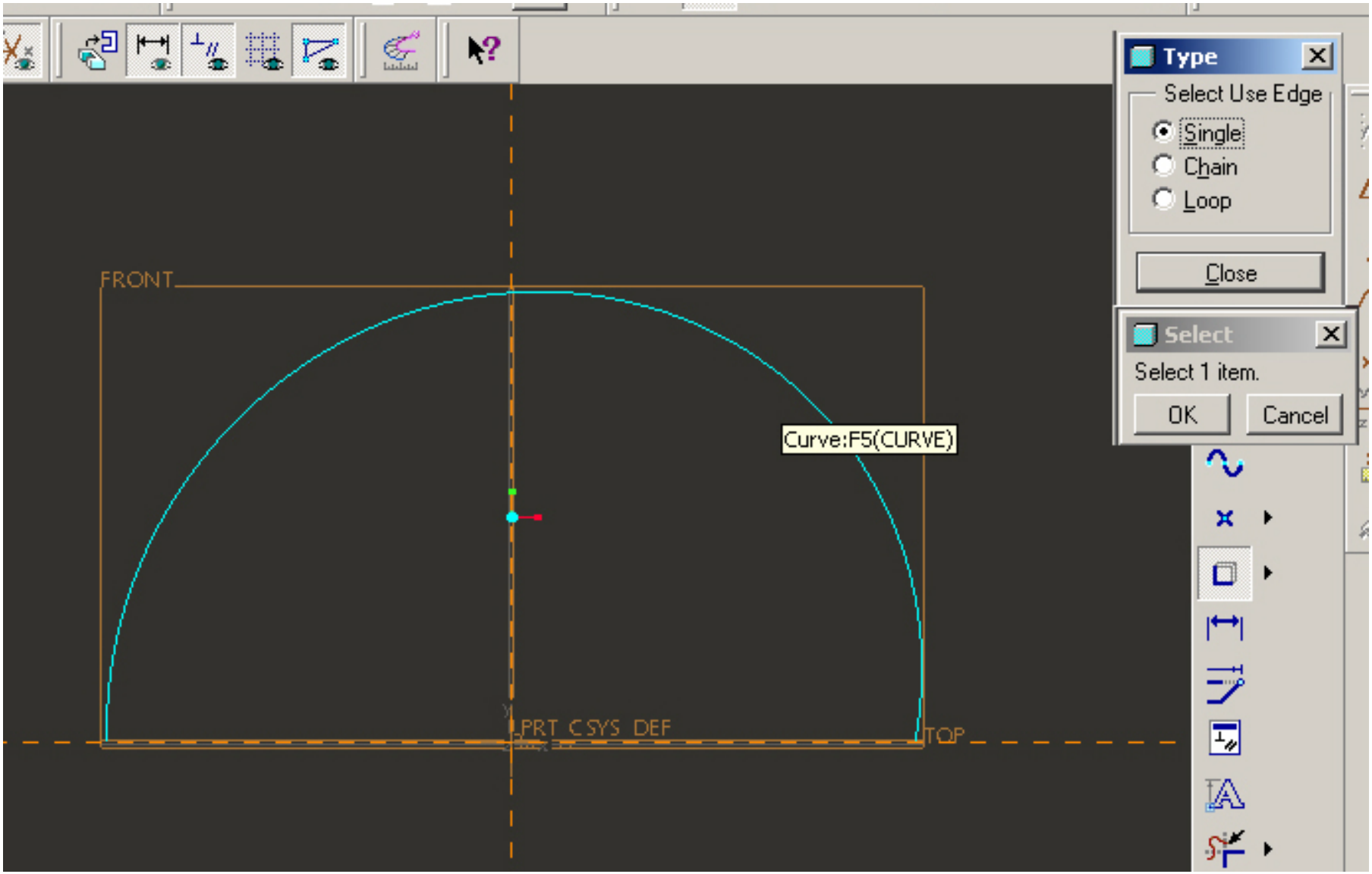


Although the entity looks closed, actually the curve is not yet part of our sketch.

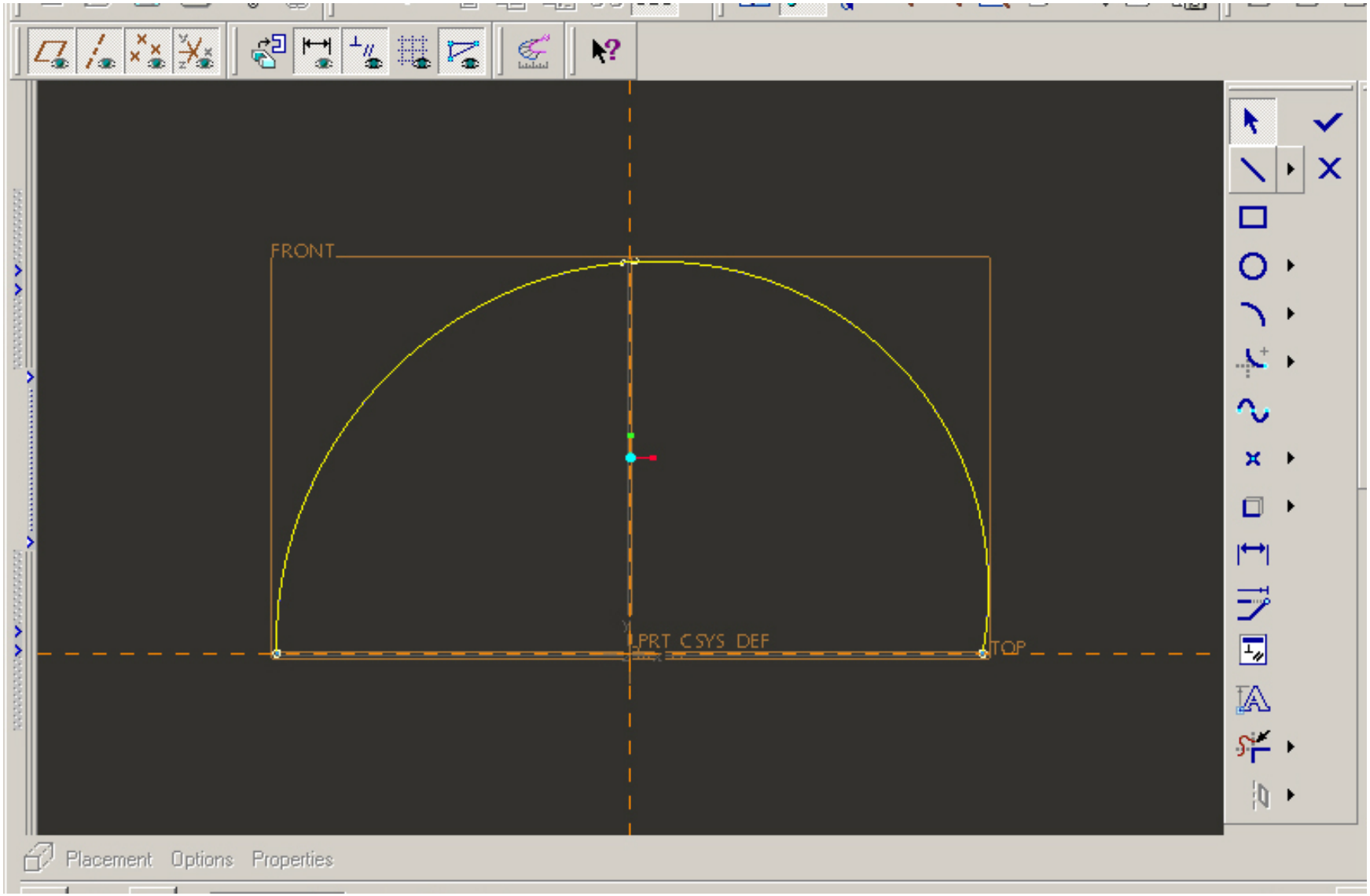
In order to make this curve part of the sketch, click *Create Entity From Edge* button in the **Sketch** toolbar on the right side of the screen.



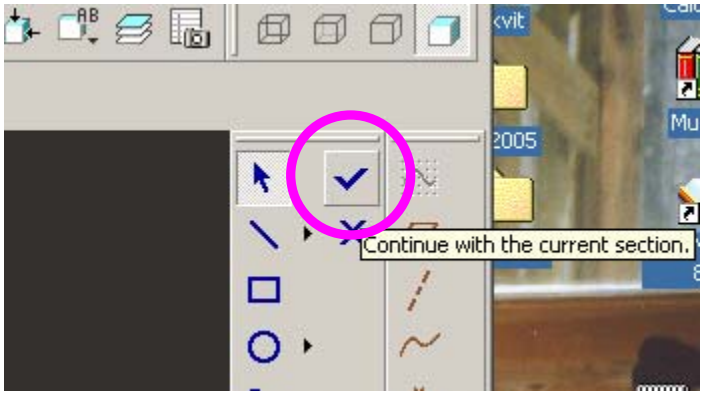
Now we have to click the profile, which we want to include, and then select *Single* option in the appeared window, and then *Close*.



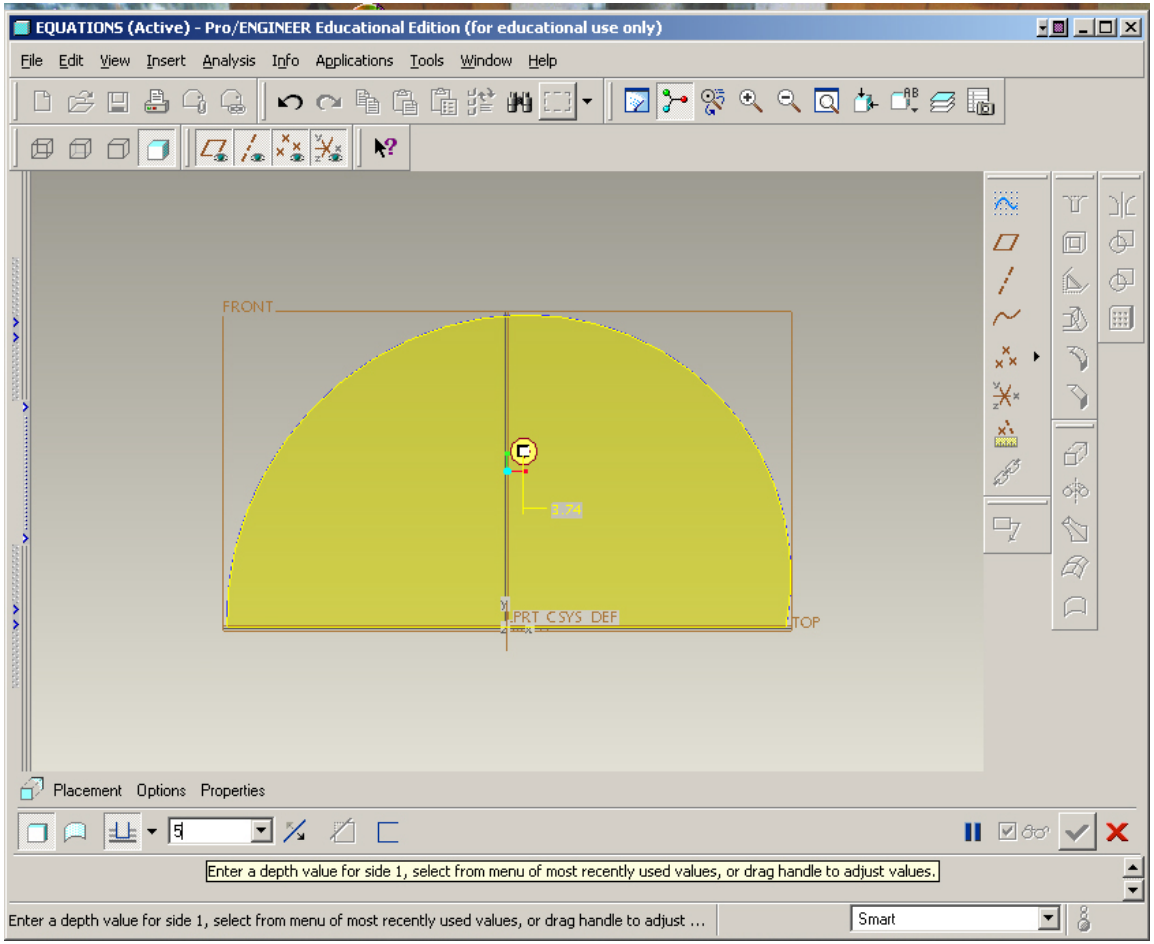
The sketch now looks different – the profile has another color. That means that it is included into the sketch.



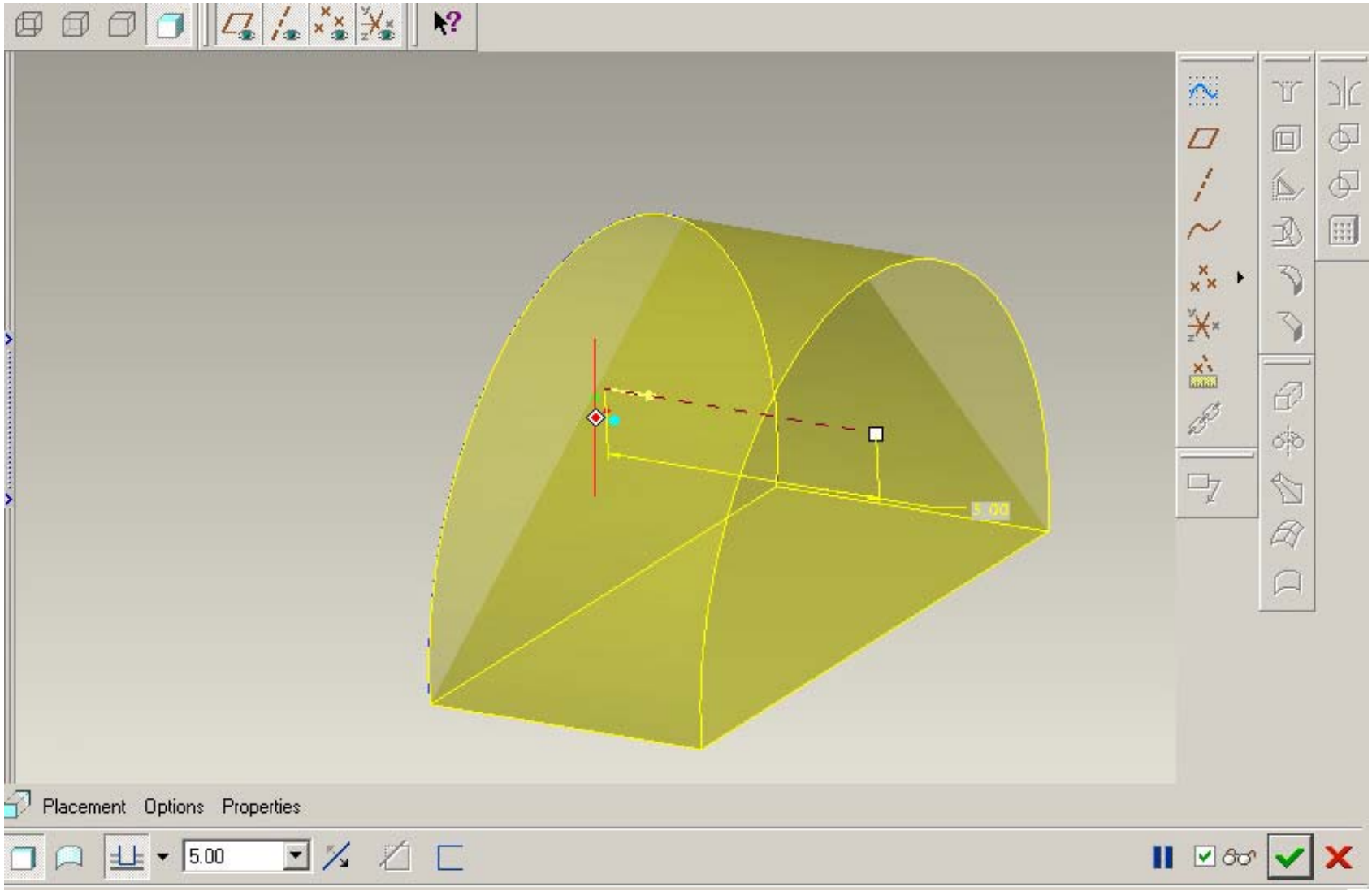
Continue the sketch with current section by clicking the tick sign



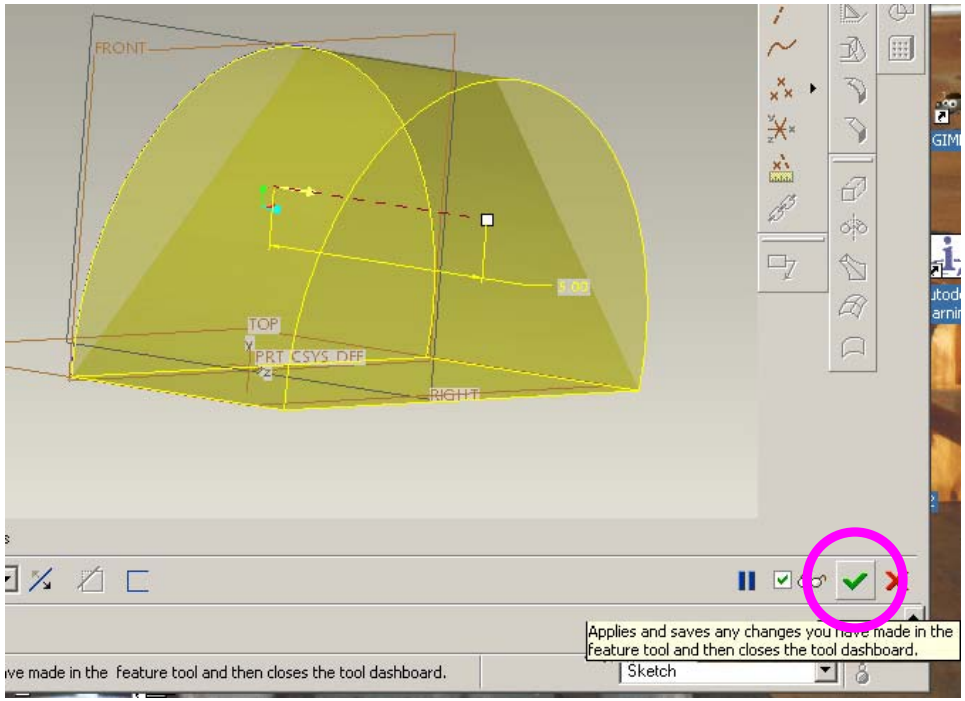
and enter the depth of extrude – 5 – in the bottom window.



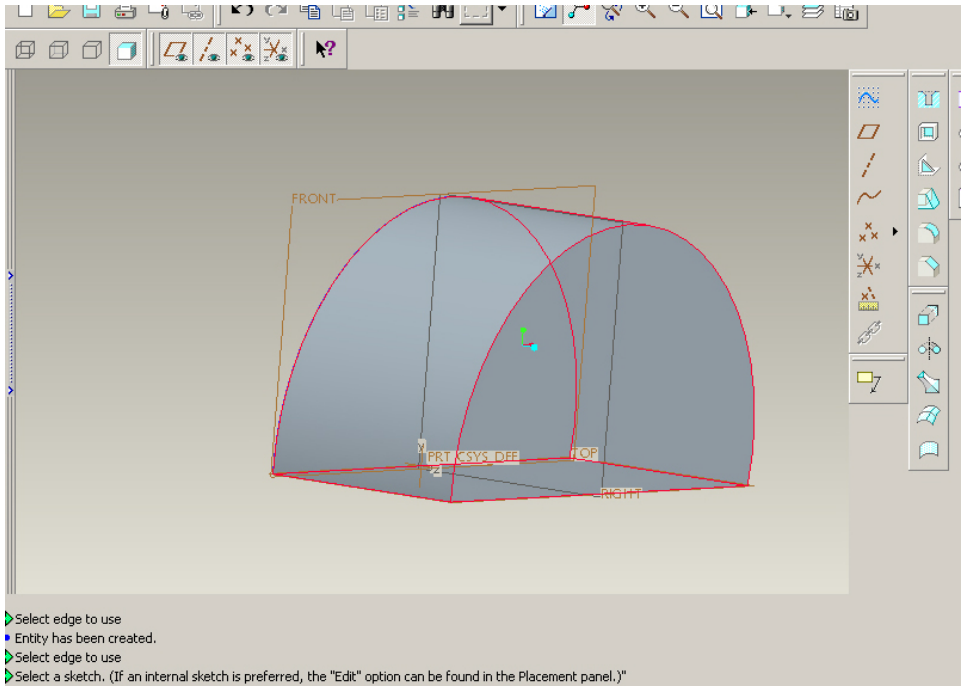
If we rotate the view we will see preliminary view of the part



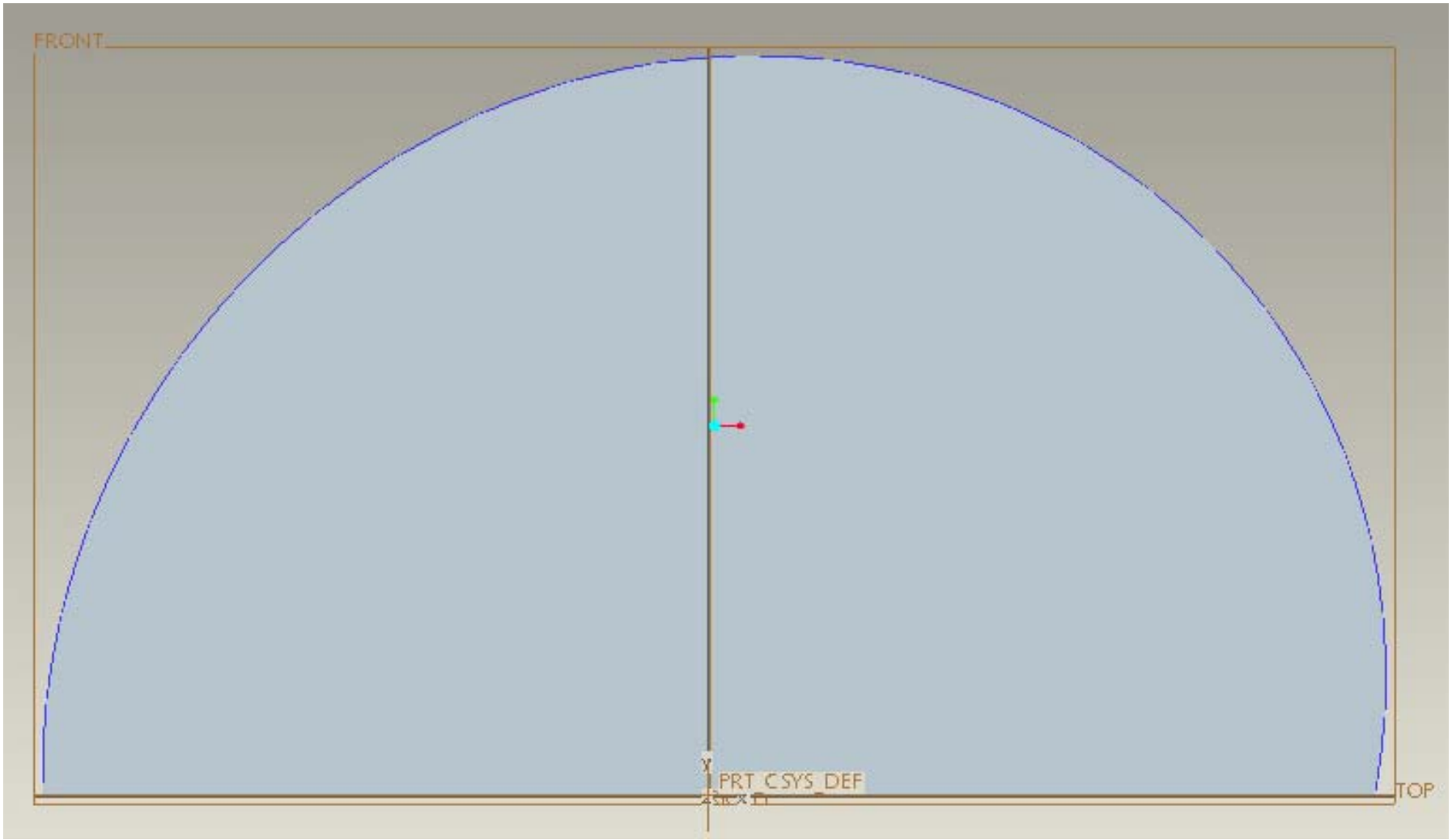
Accepting the result with the green tick sign,



We get the final view of the model:



Notice that the resulting cross section is not a semicircle as it appears, but rather defined by parametric equations.



The job is done!