CREATING A SHAMPOO BOTTLE USING SURFACE MODELING
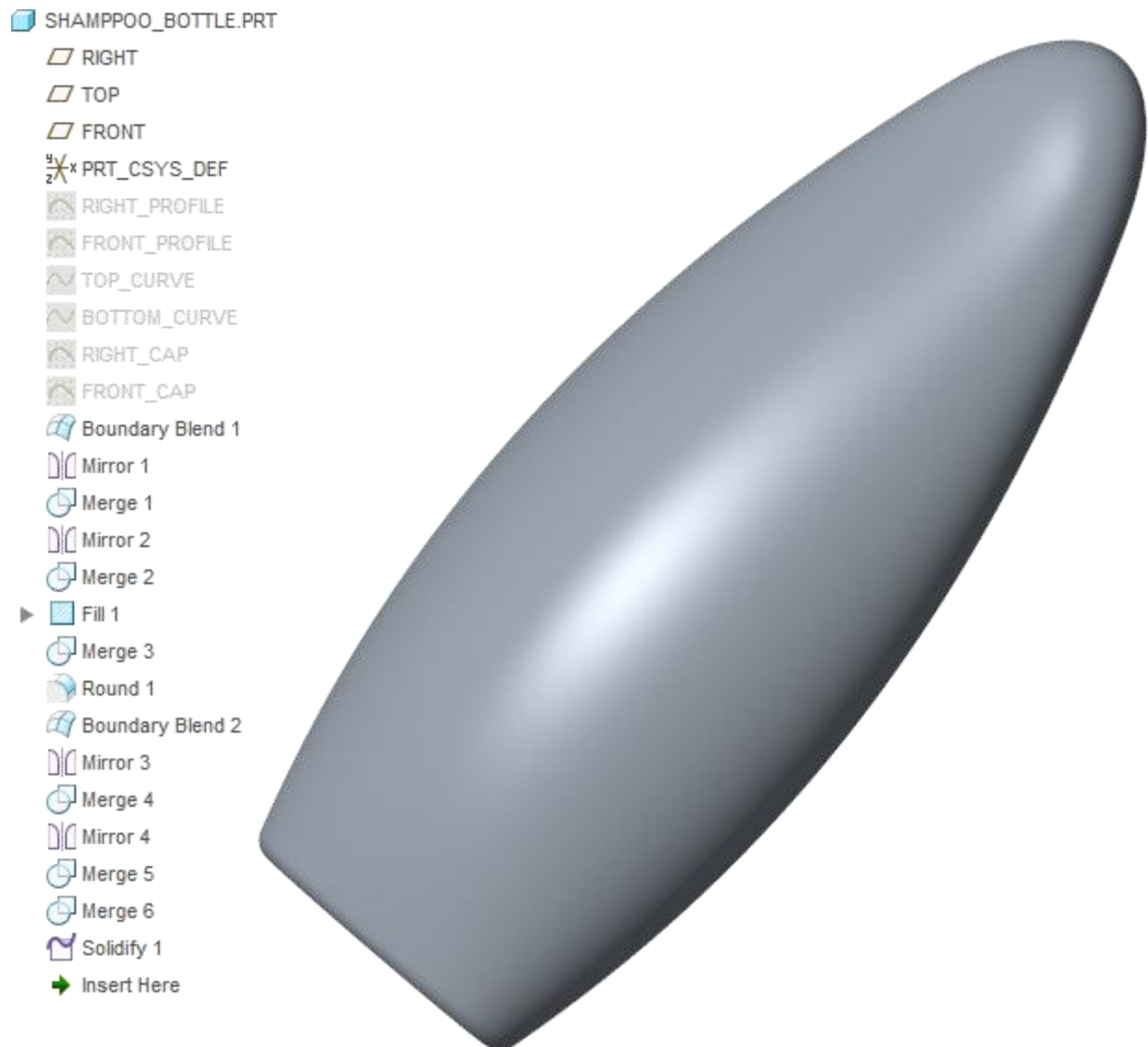
SHAMPPOO_BOTTLE.PRT
- RIGHT
- TOP
- FRONT
- PRT_CSYS_DEF
- RIGHT_PROFILE
- FRONT_PROFILE
- TOP_CURVE
- BOTTOM_CURVE
- RIGHT_CAP
- FRONT_CAP
- Boundary Blend 1
- Mirror 1
- Merge 1
- Mirror 2
- Merge 2
- ▶ Fill 1
- Merge 3
- Round 1
- Boundary Blend 2
- Mirror 3
- Merge 4
- Mirror 4
- Merge 5
- Merge 6
- Solidify 1
- ➡ Insert Here

Figure 1: Shampoo bottle and its model tree.

Learning Targets

In this exercise you will learn:

- ✓ Creating curves by using spline
- ✓ Using boundary blend
- ✓ Analyzing geometry
- ✓ Changing Creo's display options
- ✓ Exporting geometry

In this exercises, a shampoo bottle using surface modeling technique is created. In solid modeling, geometry is created using tools that add or remove material, similar to how real parts are manufactured (we have a block of material and we cut different shapes out of it). In surface modeling, we define outer surfaces of the part using curves and surface tools. Finally we merge surfaces together to get a one surface that defines all geometry of a part. This method allows creating more complicated shapes that are mostly used in customer products (a computer mouse for ex.) or where geometry is functional (turbine blades for ex.).

The used program in this exercise is Creo Parametric 3.0 M050.

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About This Exercise

This exercise illustrates curve network modeling with a simple geometry based on boundary blends.

The geometry is symmetric with respect to two planes, so only a quarter of it is modeled.

Creating Curves

Using Creo Parametric 3.0, create a **New** (⬜) solid part model and name it as *shampoo_bottle*. Next,

we create six curves to define the overall shape of the shampoo bottle.

First sketched curve

Create a **Sketch** (✎) on the FRONT plane using RIGHT plane as *Reference* and *Orientation* as *Right*.

First, create a symmetry **Centerline** (⁝ from *Sketching* group) to vertical reference line. Then, using

Spline (∿), create a spline using three points as seen in Figure 2. Press **MM**B when three points are

created to cut the spline. Create also dimensions using **Normal** (⊢⊣) as seen in Figure 2. Remember to

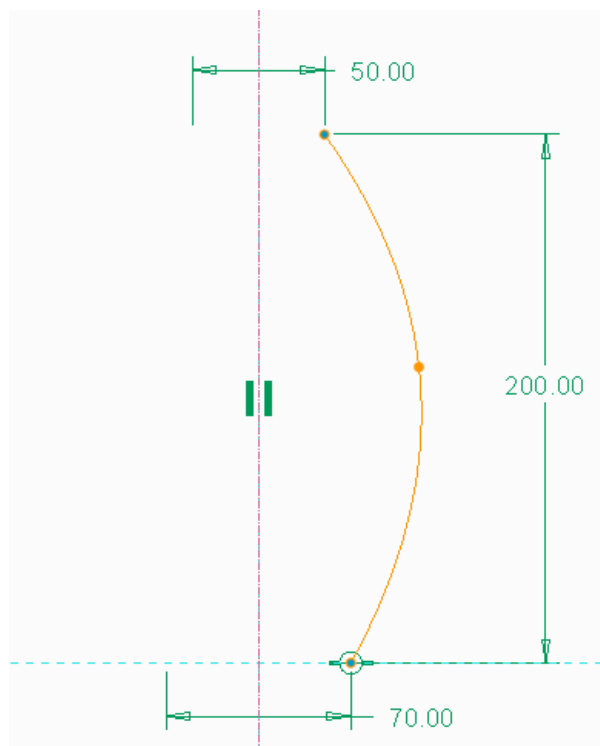create a symmetric dimension (select a point, then a centerline, then a point again and press **MMB**).

Figure 2: Result after centerline and spline created.

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As you notice, there is one interpolation point in the middle of the spline. If you select that point, you can move it around (and thus altering the spline). This is bad, if we want to define exact geometry. Luckily, we can also dimension that point. Using **Normal** (↦) tool, select that point, then centerline, then point again and press **MMB** to place a symmetric dimension. This also creates a height dimension to that point. Give values as seen in Figure 3. When ready, accept the sketch (✔). Rename the sketch as RIGHT_PROFILE.
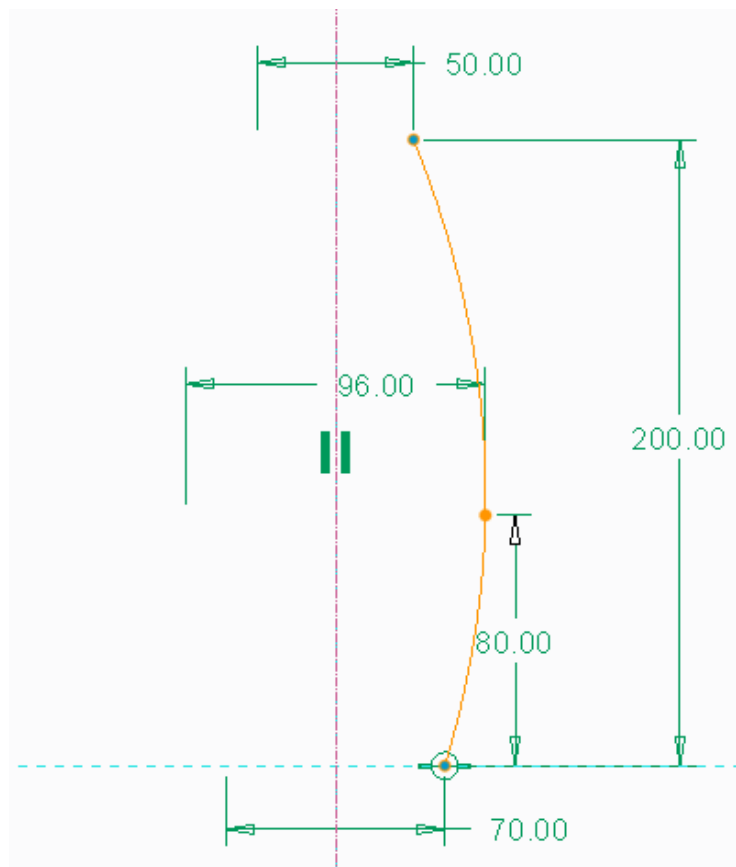

Figure 3: Sketch with final dimensioning.

Second sketched curve

Create a new **Sketch** (⟳) to the RIGHT plane using TOP (*Orientation Top*) as *Reference*. Select the topmost end point of the RIGHT_PROFILE sketch as reference using **References** (⊡, *Setup* group). Using that reference point, place a horizontal **Centerline** (⦙). Create also a vertical centerline. Sketch on the left side of the vertical centerline a spline using three points (Figure 4).
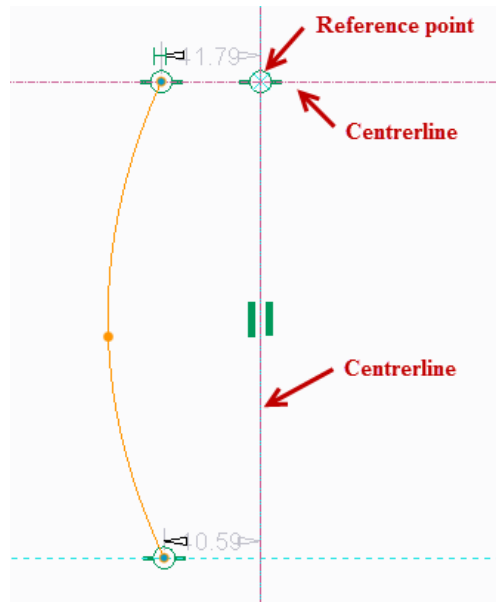
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Figure 4: Sketched spline. Notice the horizontal reference line (highlighted in red) and reference point.

Double-click the spline (or select it, hold **RMB** and select **Properties**) to access *Spline* dashboard. With this tool you can modify spline. Click on the middle interpolation point (three points together, marked with black circles) to select it, hold **RMB** and select **Delete Point** to remove this point (Figure 5).
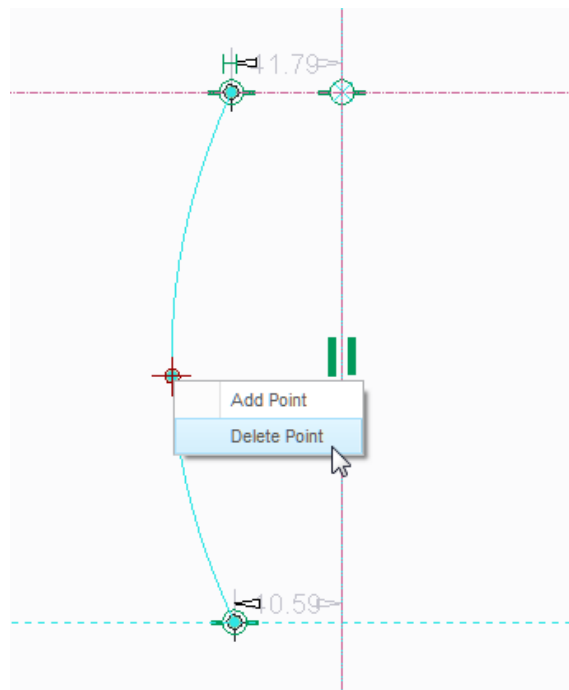

Figure 5: Point selected (highlighted in red) and selecting *Delete Point* from RMB menu.

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Then click ⬚ on the dashboard to switch to polygon mode. Now there are four points (to be exact, three lines!) that control this spline. Move some of the middle points to see their affect to the spline (Figure 6).
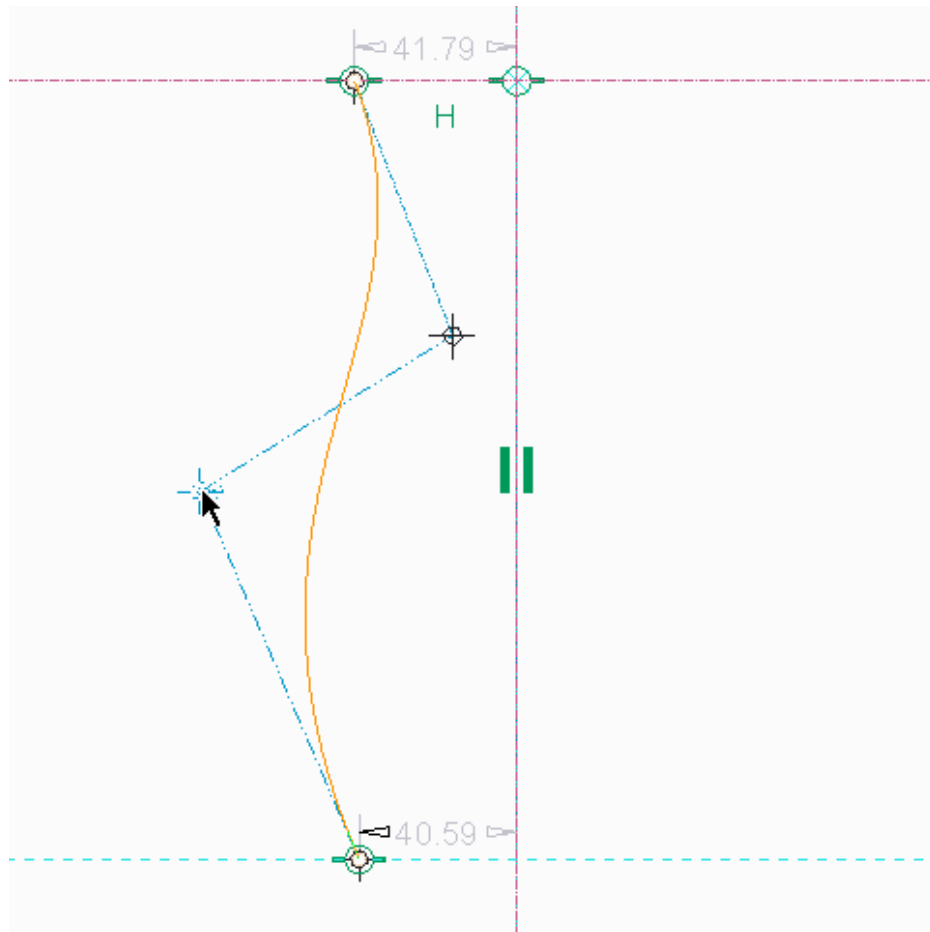

Figure 6: Moving a control point.

Click **MMB** to accept changes and to close the tool. Now those lines are dimensioned and we can define exact geometry to the spline. Using right type dimensions and their values, modify sketch to as seen in Figure 7. Notice, that besides dimensions, you can also define geometric constraints to the construction lines. When ready, accept the sketch (✔) and rename it to FRONT_PROFILE.

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Figure 7: Final sketch with two symmetric dimensions, one height dimension and three angular dimensions. Spline in yellow, control lines in turquoise.

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First datum curve

Next we create two datum curves. Select **Curve Through Points** (〰) from *Datum* group. Select the topmost endpoints of both sketches. Open **End Condition** tab, change **End condition** of the *Start Point* to **Normal** and select RIGHT plane as reference (Figure 8). Then change *End Point* also to **Normal** and select FRONT plane as reference. If needed, click red arrow(s) to change the direction of the curve.
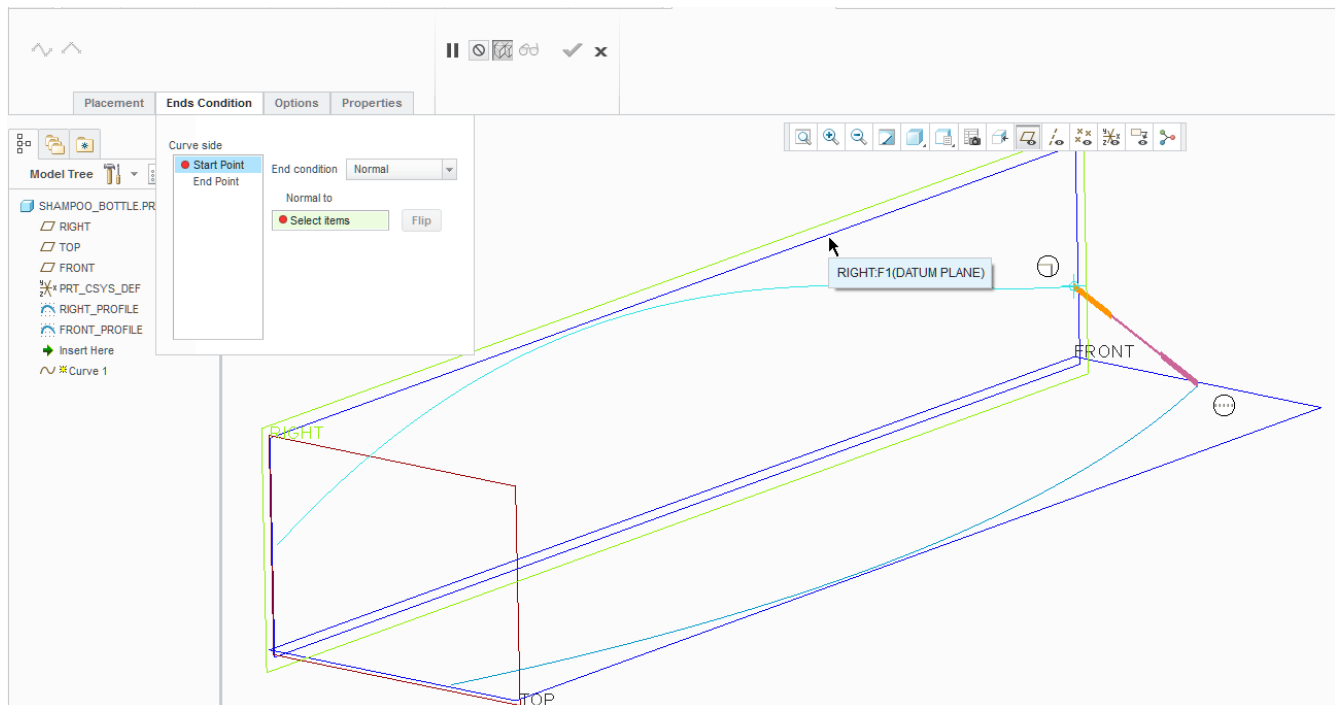

Figure 8: Two point selected, normal chose as end condition and choosing RIGHT plane.

When preview (👓) looks like in Figure 9, accept the feature (**MMB**). Rename curve as TOP_CURVE.
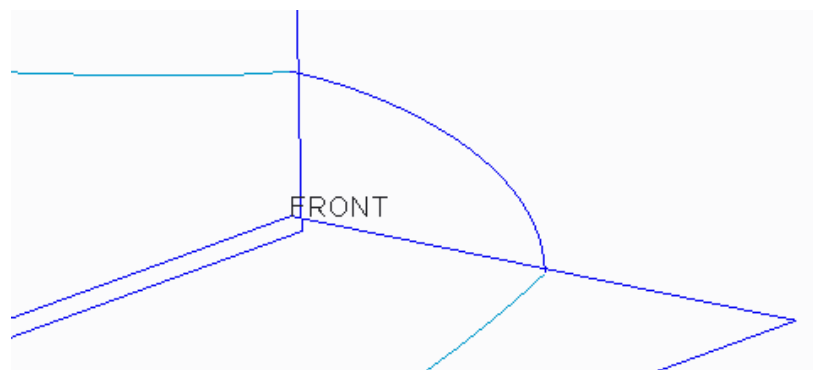

Figure 9: Preview of the curve (darker blue).

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Second datum curve

Using the same method as with the first datum curve, create a curve to the bottom of the first two

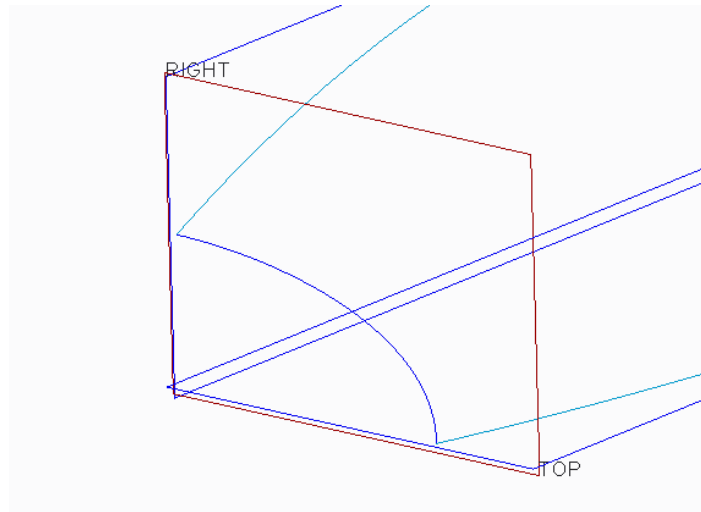sketches (Figure 10). Rename curve as BOTTOM_CURVE.

Figure 10: Second datum curve (darker blue).

Third sketched curve

Create a new **Sketch** (⬚) to the FRONT plane using RIGHT as *Reference* (*Orientation Right*). Select the

first sketched curve as **Reference** (⬚, the one that is also sketched to the FRONT plane) and create a

sketch using spline as seen in Figure 11. Remember **Tangent** (⬠) constraints (from *Constrain* group)!

You will need a centerline as a horizontal line to define the spline's tangency. When ready, accept
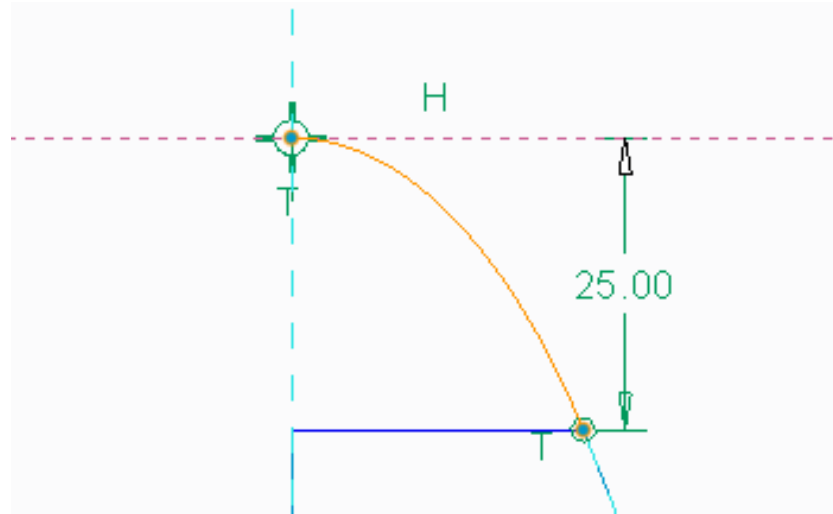
sketch and name it RIGHT_CAP.

Figure 11: Sketched spline. Notice the horizontal centerline (red dashed line) and curve as reference.

Fourth sketched curve

Using the same method, create also another sketched spline on the RIGHT plane. Remember to take also the end point of the RIGHT_CAP as reference (Figure 12). When ready, accept sketch and name it to FRONT_CAP.
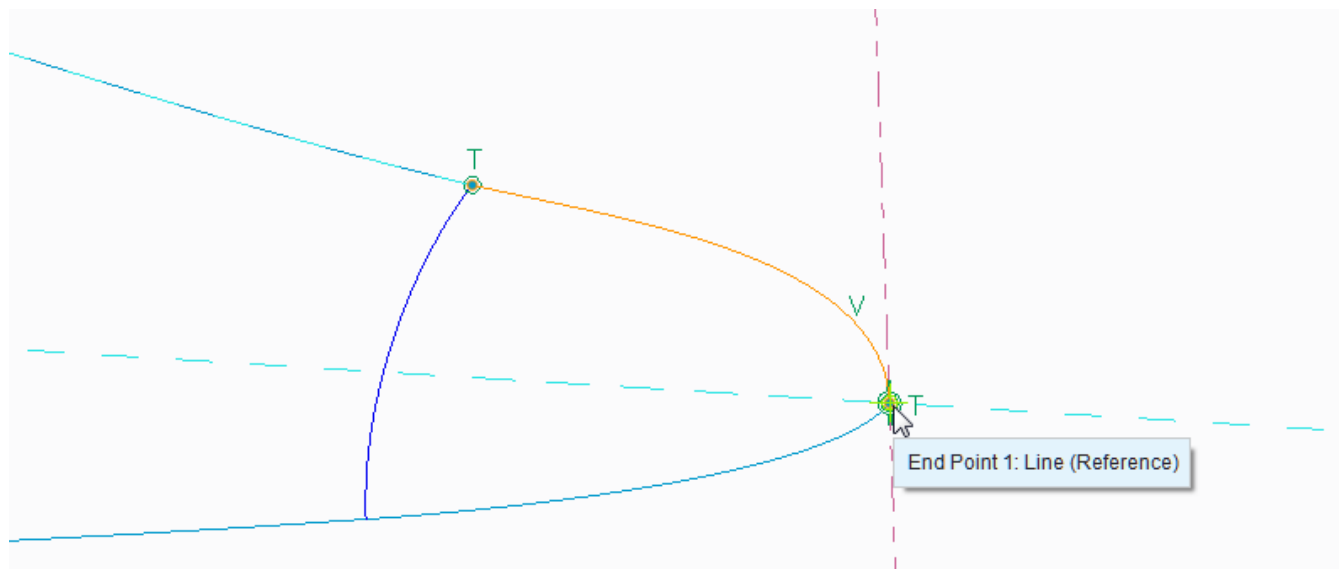

Figure 12: Sketch for the fourth sketched curve. Reference point highlighted with mouse pointer.

Names of the curves

Our model has now six curves. To make the rest of the exercise easier, Figure 13 shows the names of the curves. Press **CTRL+D** to see the logic behind the naming policy used.
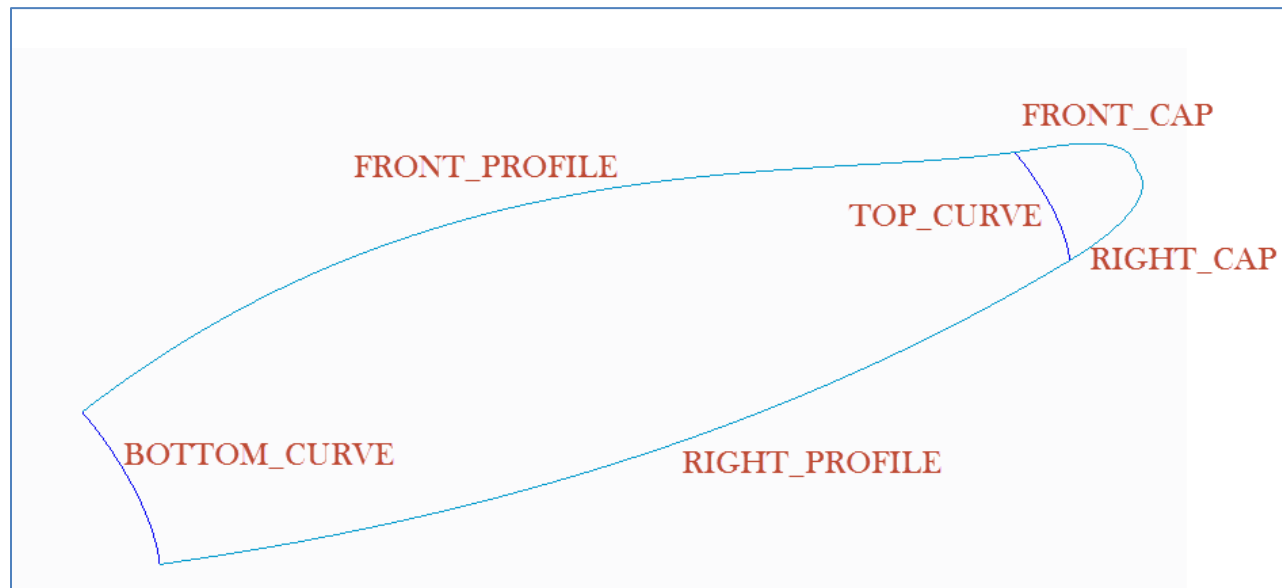

Figure 13: Names of the curves.

Creating Geometry

Using Boundary Blend

Activate the **Boundary Blend** ( ) tool from the *Surfaces* group. The *Boundary Blend* dashboard appears. When we select curves, select them from <u>graphical area</u>, <u>not</u> from model tree. In the model tree, <u>features</u> are presented, but we want to select <u>geometry</u>! Select RIGHT_PROFILE and then holding **CTRL** select FRONT_PROFILE. Now the tool fits a surface between these two curves. Hold **RMB** on graphical area and select **Second Direction Curves**. Then select BOTTOM_CURVE and then holding **CTRL** TOP_CURVE. You should get a preview of a quite good-looking surface (Figure 14). Now the tool creates a surface between four selected curves; notice the difference when only two curves were selected! Accept the feature (**MMB**).
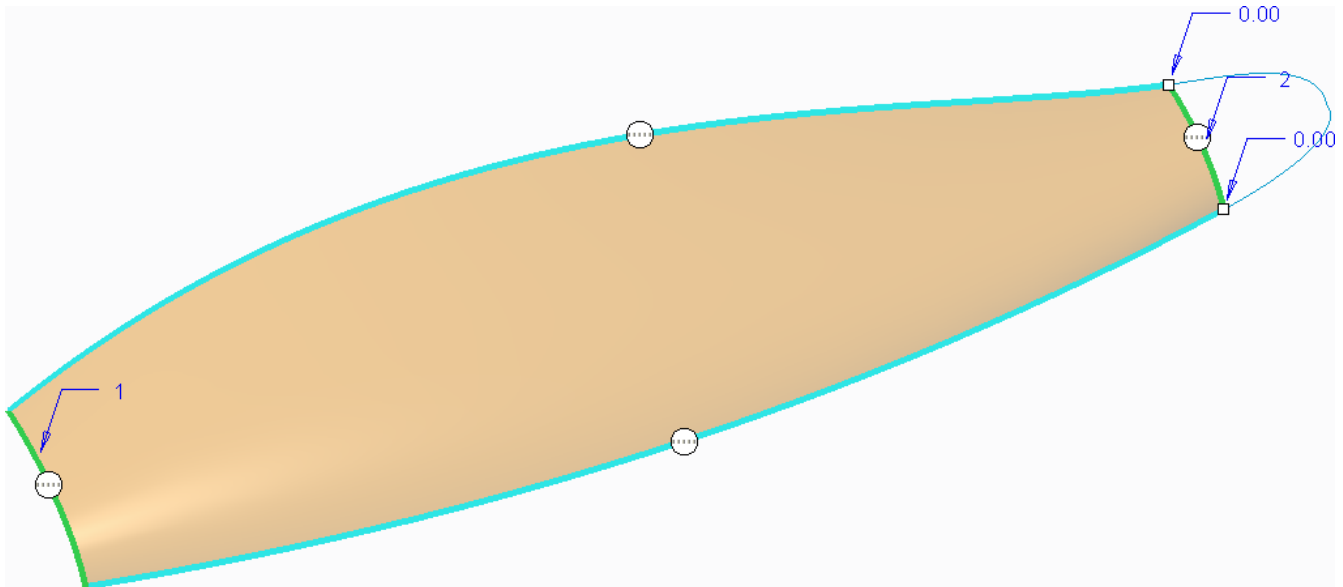
Figure 14: Last curve for Second Direction is selected and preview is showed.

Mirroring a surface

Select the underline surface geometry (by first selecting the feature and then clicking again on the surface, it should turn solid green). Click **Mirror** (⌈⌉) from *Editing* group and select the FRONT datum plane to mirror the geometry about it (Figure 15). (In addition to geometry, it is possible to mirror surface, curve, point etc. features as well. Some experts say that geometry is preferable, though.) Accept the mirror feature.
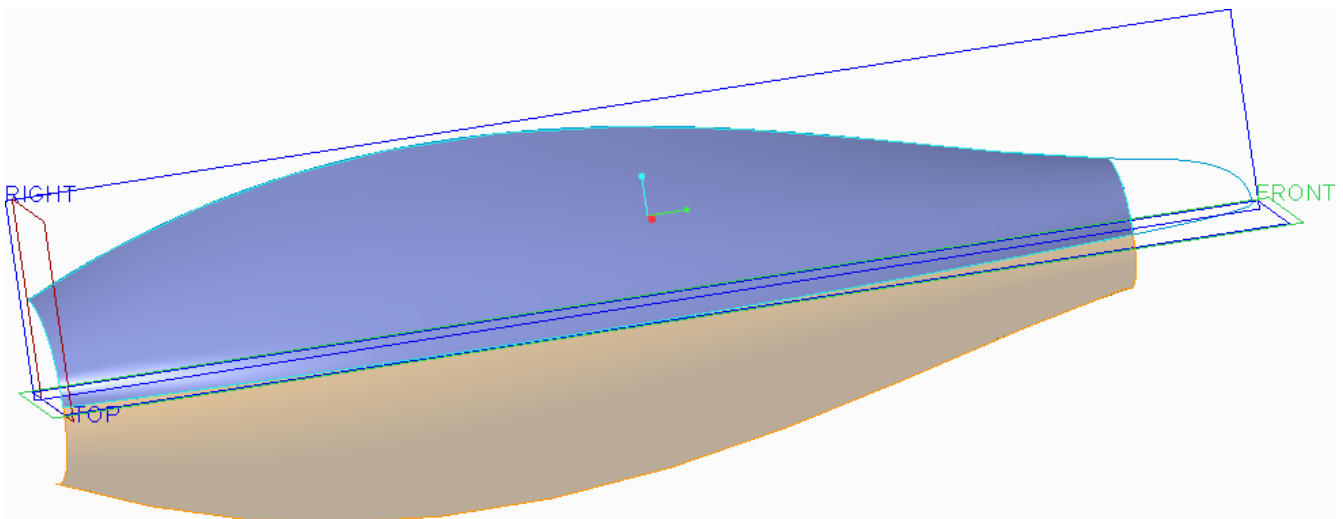

Figure 15: Preview of a mirror.

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Geometry analysis

Select **Analysis** tab and select **Reflection** (🖼) from *Inspect Geometry* group (there is a black triangle, click that!). Select the two surfaces as references (the second one with CTRL!). Spin the view to notice that the reflections are not continuous on the surface boundary (Figure 16). When ready, close the *Reflection Analysis* tool.
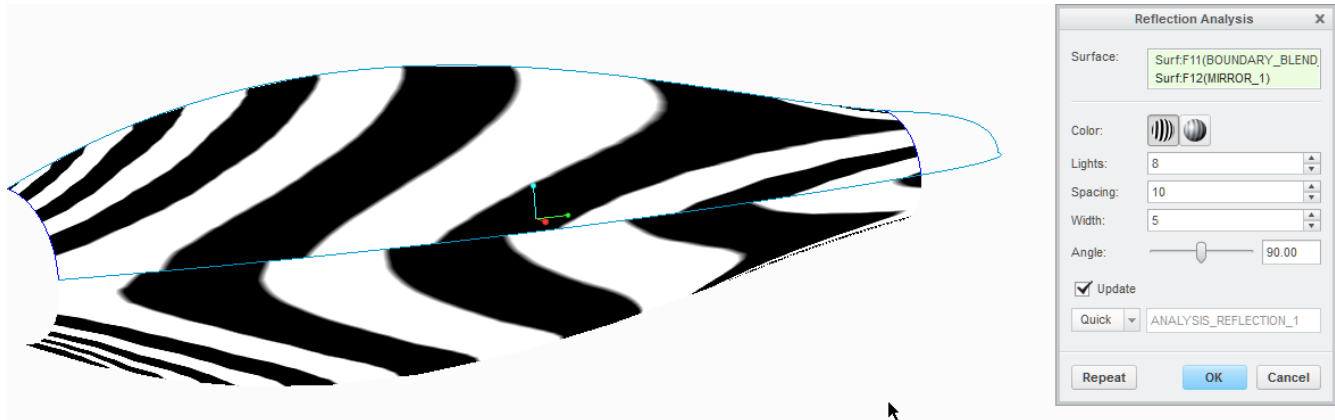

Figure 16: Performing a Reflection analysis.

Changing continuity

The boundary curves are (G2) continuous, but that is only a necessary condition for similar surface continuity. The boundary conditions have to be set on the whole surface boundary. Since the surfaces are symmetric, it is quite clear that their tangential (G1) continuity is ensured by setting them normal to the symmetry plane (the boundary curves are set normal, check their definitions if you wish). But how about curvature continuity (G2)? If you think about it for a moment, it is clear that G1 continuity at symmetry plane also leads to G2. This is because everything about the surfaces is symmetric, including curvature; thus, the boundary curvatures are always equal!

Let's set the surface boundary conditions. Right-click Boundary Blend feature from model tree and select **Edit Definition** (🖌). **RMB** on ⬭ symbol on FRONT_PROFILE and change Free to **Normal** (the reference plane for normal condition is usually correct by default, but they can be manually set under Constraints tab). Do the same for RIGHT_PROFILE and accept the feature. Do another **Reflection**

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analysis (⬚) to see that the reflections are now continuous (Figure 17). (Note! If you have mirrored the feature, it won't work! Remove the mirror and mirror the <u>surface</u> this time!)
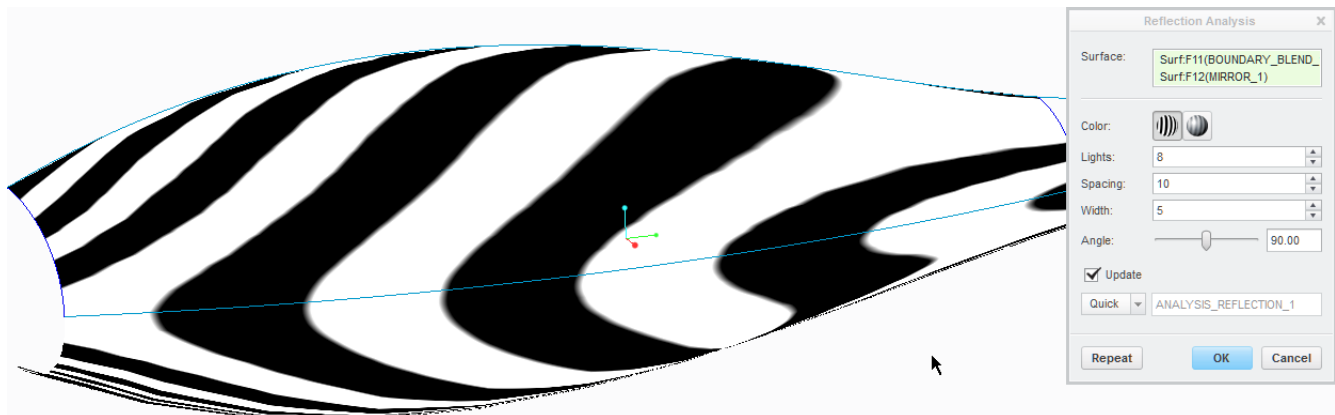

Figure 17: Reflection analysis after changing the continuity of the boundary blend.

Merging surfaces

Select both surfaces (should be highlighted in geen) while holding CTRL and then select **Merge** (⬚, *Editing* group in **Model** tab). Click on **Options** tab and select **Join**. Now there is only on surface in the model.

Mirroring merged surfaces and merge them

Select the surface geometry of the new merge feature (highlighted in green on graphical area), noticing that there is only one surface. **Mirror** (⬚) the surface about RIGHT datum plane and **Merge** (⬚) the resulting halves (again, select surface geometry).

Creating a fill

Next we add a surface on the bottom of our shampoo bottle. Select **Fill** (⬚) from *Surfaces* group. Select TOP plane. In sketcher, hold **RMB** and from **Section Orientation**, select **Flip sketching plane**. Select **References** (⬚, from *Setup* group) and select all four edges as seen in Figure 18. Click **Close** to accept these new references.

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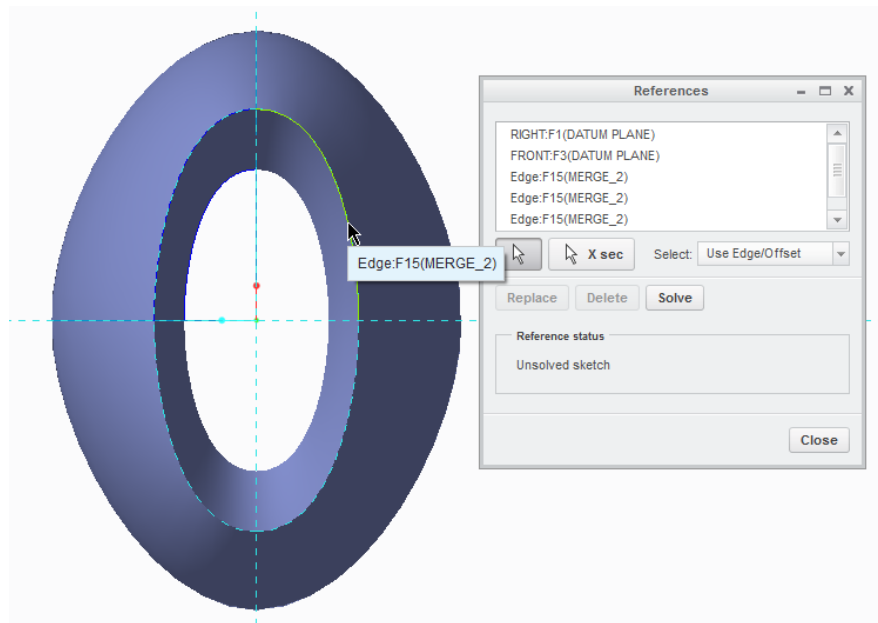School of Engineering**

Figure 18: Four edges of the surfaces selected.

Using **Center Rectangle** (🖼, *Sketching* group, under *Rectangle*), select the center of the sketch as a

starting point and move the cursor until rectangle snaps to the geometry (Figure 19) and click **LMB**.

There should be no dimensions in the sketch. When ready, accept the sketch (✔) and the feature
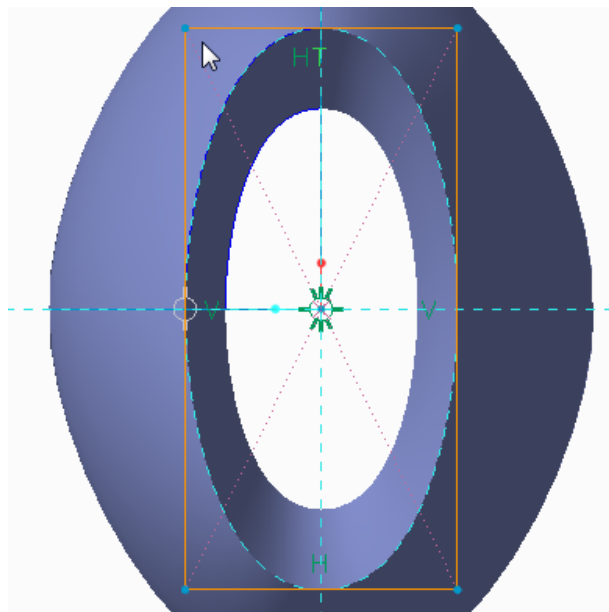
(**MMB**).

Figure 19: Rectangle snaps to the geometry.

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Merge and hide

Merge (⬡) the surface created with Fill feature with the rest of the surfaces. You may need to flip the merging direction, so that the bottom will be part of the new surface (flip the pink arrow).

Hide curves RIGHT_PROFILE, TOP_PROFILE, TOP_CURVE and BOTTOM_CURVE (select those four in the model tree (hold CTRL), hold **RMB** and select **Hide** (👓)).

Round

Select **Round** (🔽, *Engineering* group) and select the edge in the bottom of the bottle. The normal round is G1 continuous, but we want G2 round. We can do it, from **Sets** tab, select **G2 Continuous** as round type (Figure 20) and accept the round feature.
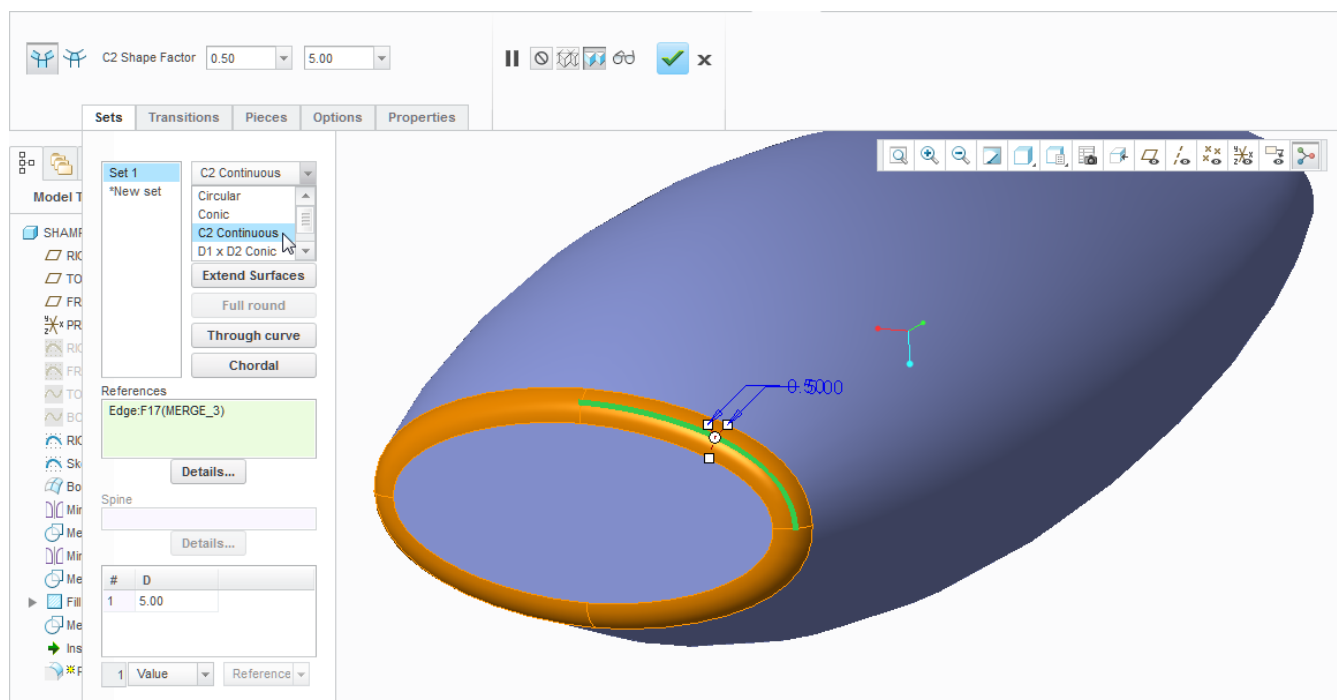

Figure 20: Changing round type from Circular to G2 Continuous. Rounding value of 5.

Time to save your model?

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Using Boundary Blend to create a cap

To create the "cap" of the bottle, start another **Boundary Blend** (⌗). Select curves RIGHT_CAP and TOP_CAP as first direction boundaries. As second direction boundaries (hold **RMB** and…), select only the surface boundary corresponding to curve TOP_CURVE (it is always better to select the surface boundaries than their reference curves in a case like this, although it leads to deeper dependency between the features).

Set boundary conditions of curves RIGHT_CAP and TOP_CAP to **Normal** (⊖). Enter to the verify mode (👓), to check that the geometry can be constructed and exit the verify mode (▶). Try and set the boundary condition of TOP_CURVE to **Curvature** (G2, ⊖). Verify (👓) the feature to see that the geometry cannot be constructed. Change the problematic constraint to **Tangent** (G1, ⊖). Verify and check that geometry is OK. Accept the feature.

Why was it not possible to set G2 continuity between the surfaces? That is simply because curves RIGHT_CAP and TOP_CAP were only tangential at the endpoints. As said earlier, the continuity level of the boundary curves is a necessary condition of surface continuity.

Finalizing the model

Mirror the new surface <u>geometry</u> about FRONT plane, merge, mirror the merge about RIGHT plane, merge again and merge the cap with the rest of the bottle. When ready, set **No Hidden** (⬜) display type on from *Graphical* toolbar (Figure 21).
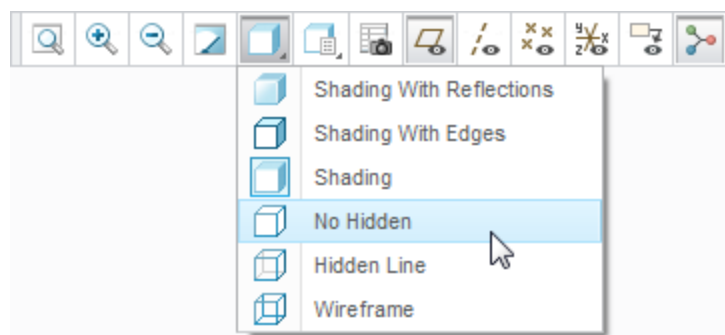

Figure 21: Setting view type to No Hidden.

Be free to hide all curves in the model. If there are only purple edges (Figure 22), your model geometry is ready.
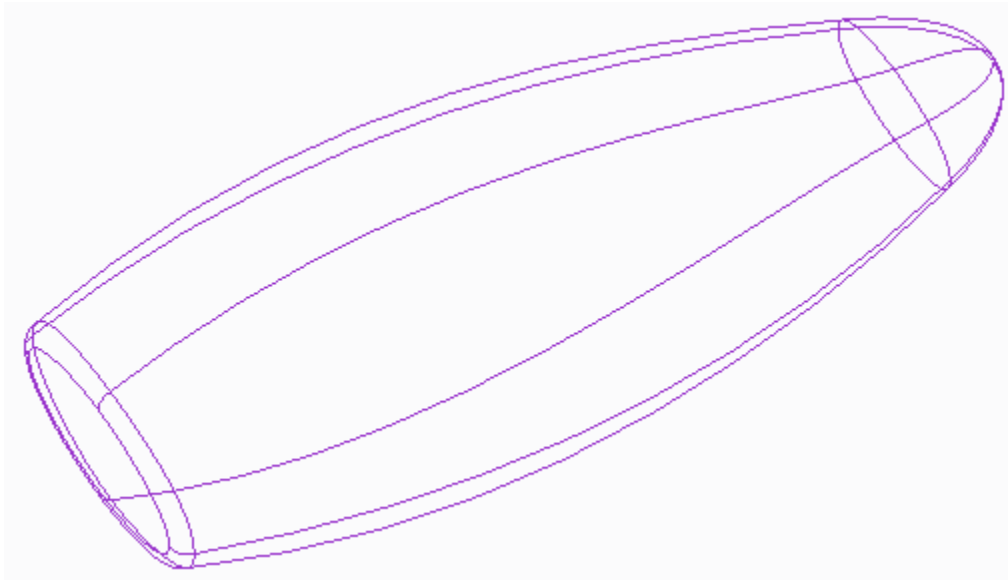

Figure 22: No Hidden display on and all curves hided.

Purple color in the edge means, that a surface continues to both side of the edge. If there is a disconnected surface, program shows it in yellow (Figure 23).
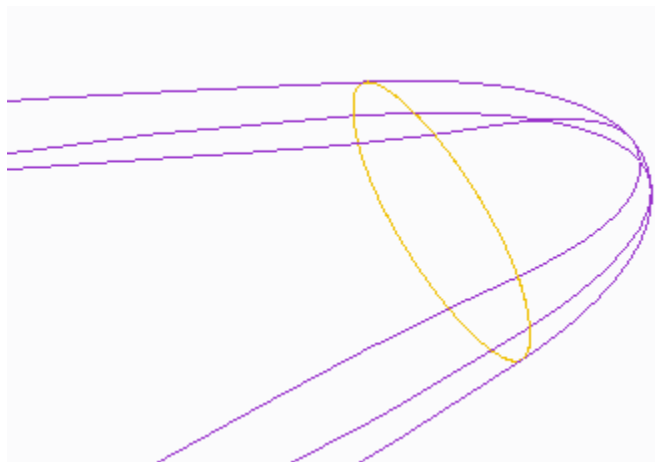

Figure 23: Surface in the cap is not connected to the main surface.

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Shaded Curvature analysis

Do a **Shaded Curvature** (⬛) analysis for the cap surfaces to see how three-sided patches cause wrinkles (Figure 24) in Creo (and many more surface modelers). According to PTC, this is not a big problem.
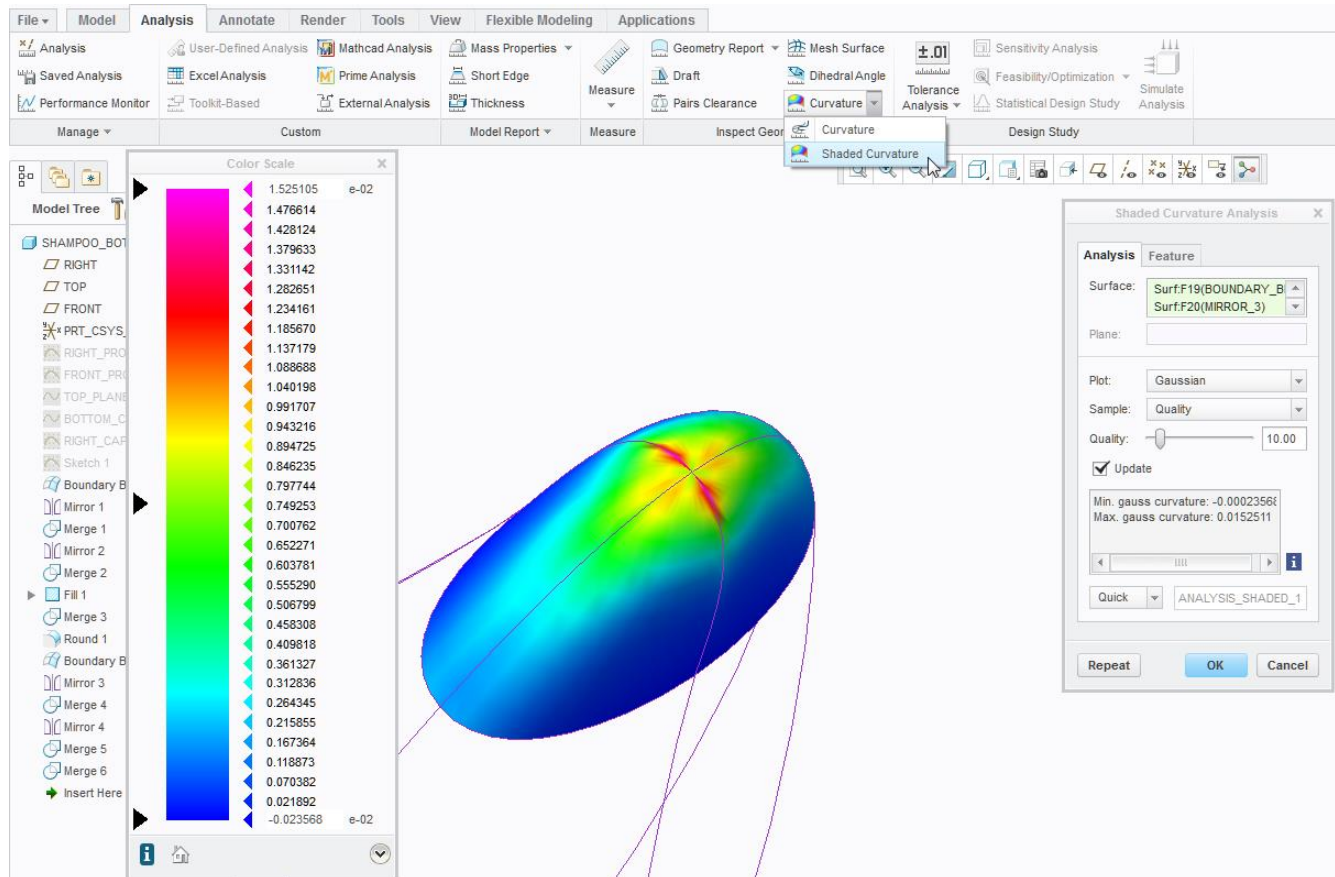

Figure 24: Shaded Curvature analysis and location of its button.

Setting Display Settings

If you look closer to your model, you can see, that the edges are very sharp and ugly (Figure 25).

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Figure 25: A closer look to an edge.

This is not because our geometry is so, but because Creo tries to save computing power and shade edges (and other geometry) in a rough level. Normally this is a good thing (free RAM!), but sometimes it is needed to see the more exact geometry (e.g. rendering, surface modeling).

Select **File → Options** ( ).

Select **Model Display** tab. From *Shaded model display options*, set options *Set shade quality to* **30** (maximum is 50) and **check** *Shade very small surfaces* option. Press **OK** and select **No** (otherwise this setting will be always active, painful with larger assemblies…). As you can see, the shape of the bottle are smoother than before (Figure 26).
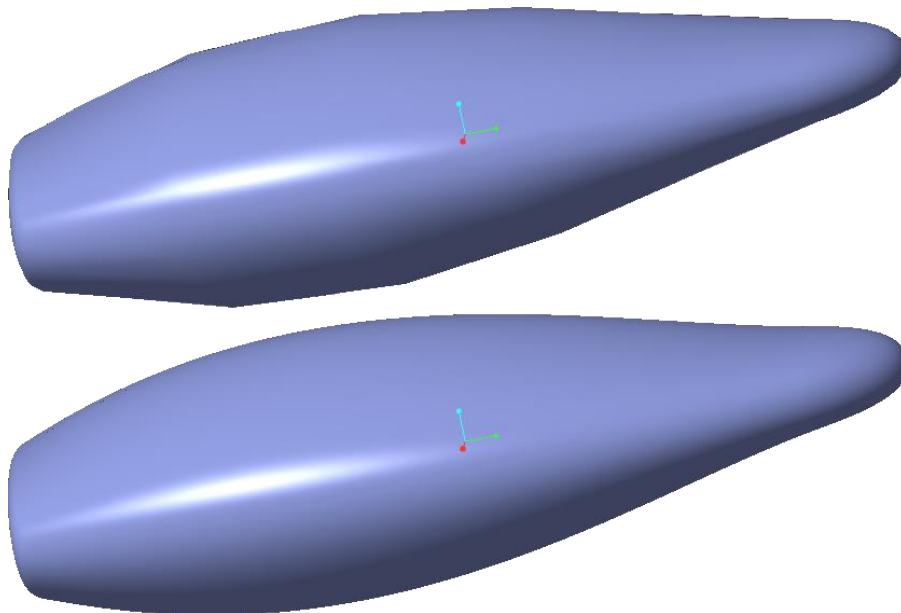

Figure 26: On top: before changing *Model Display* options; On bottom: Options changed.

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Select **File → Options** (▣) again. Select **Entity Display** tab. Set *Edge display quality* to **Very High** and *Anti-Aliasing* to **16 X**. **OK** and **No** like before. Edges are now smoother (Figure 27).
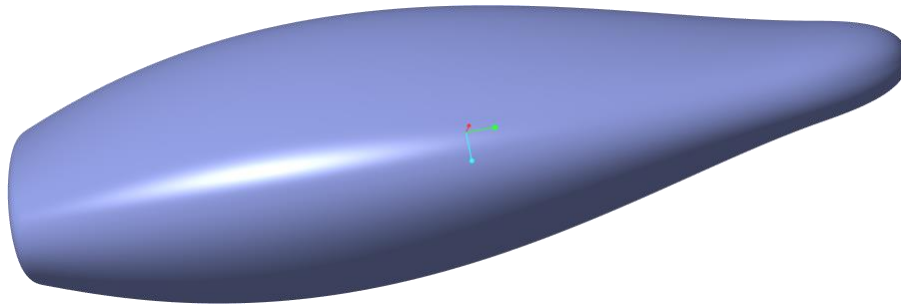

Figure 27: After *Entity Display* options changed.

Solidify

Our model is a surface model, so there is a one surface that bound a volume in the space. Sometimes a surface model is fine, but mostly we want to have a solid model (mass and mass center are very interesting properties for mechanical engineers). To transfer surface to solid, select the geometry and select **Solidify** (▱, *Editing* group). Now your model is solid.

Exporting Geometry

To export this created geometry to some other CAD programs, a neutral file format is needed. There are several neutral file formats, we use two of them.

1) **File → Save As → Save a Copy**.

2) Change Type to Stereolithography (*stl).

3) Select suitable folder and **OK**.

4) Settings should be fine, click **OK**.

As you can notice, Creo created triangle-shape surfaces. Press **Ctrl+R** to repaint the geometry. The geometry were exported STL-file type that for ex. most of the 3D-printers can understand. Also, STL export demands that the model is solid. You can test it by suppressing the last *Solidify* feature.

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Export another file (**Save a Copy**), this time using SPEP *.stp) as a file format. Default settings are fine.
STEP is an international standard (ISO 10303), that mostly all other CAD programs can understand it.

Opening Exported Files

STL

Select **Open** (📂), change *Type* to **All Files** (*) and open the previously created STL-file. Click **OK** to the *Import New Model* window. You can see what this export has done to the smooth shampoo bottle shape…

STEP

Select **Open** (📂), change *Type* to **All Files** (*) and open the previously created STP-file. Check that *Type* is *Part* and click **OK** to the *Import New Model* window. As you can see, the shape is much better than with STL-file. This is because STEP exports the outer shape of the model as surfaces, but STL is using solid approximation (triangle) of the shape.

Notice, that in both cases, there are no features in model three that were used to create the geometry. Therefore, it is not possible to change the geometry by utilizing the features. Only by modifying the original native CAD file and exporting the geometry again, changes can be made.

When exporting geometries to other CAD programs (Solid Edge, Catia, NX, Inventor, SolidWorks, Vertex…), STEP or IGES file formats are recommended.

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