**Solidworks Tips For
Industrial Designers**

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# Mirror bodies, not features

Insert>Pattern/Mirror/Mirror

Mirroring features, or mirroring sketch entities can be an unreliable technique and can be a pain to trouble shoot. Mirroring bodies is very reliable.

To mirror a closed sketch, you can make it a planar surface, then mirror the planar body. Then insert another sketch and convert the edge entities.

Andrew Jackson 2021
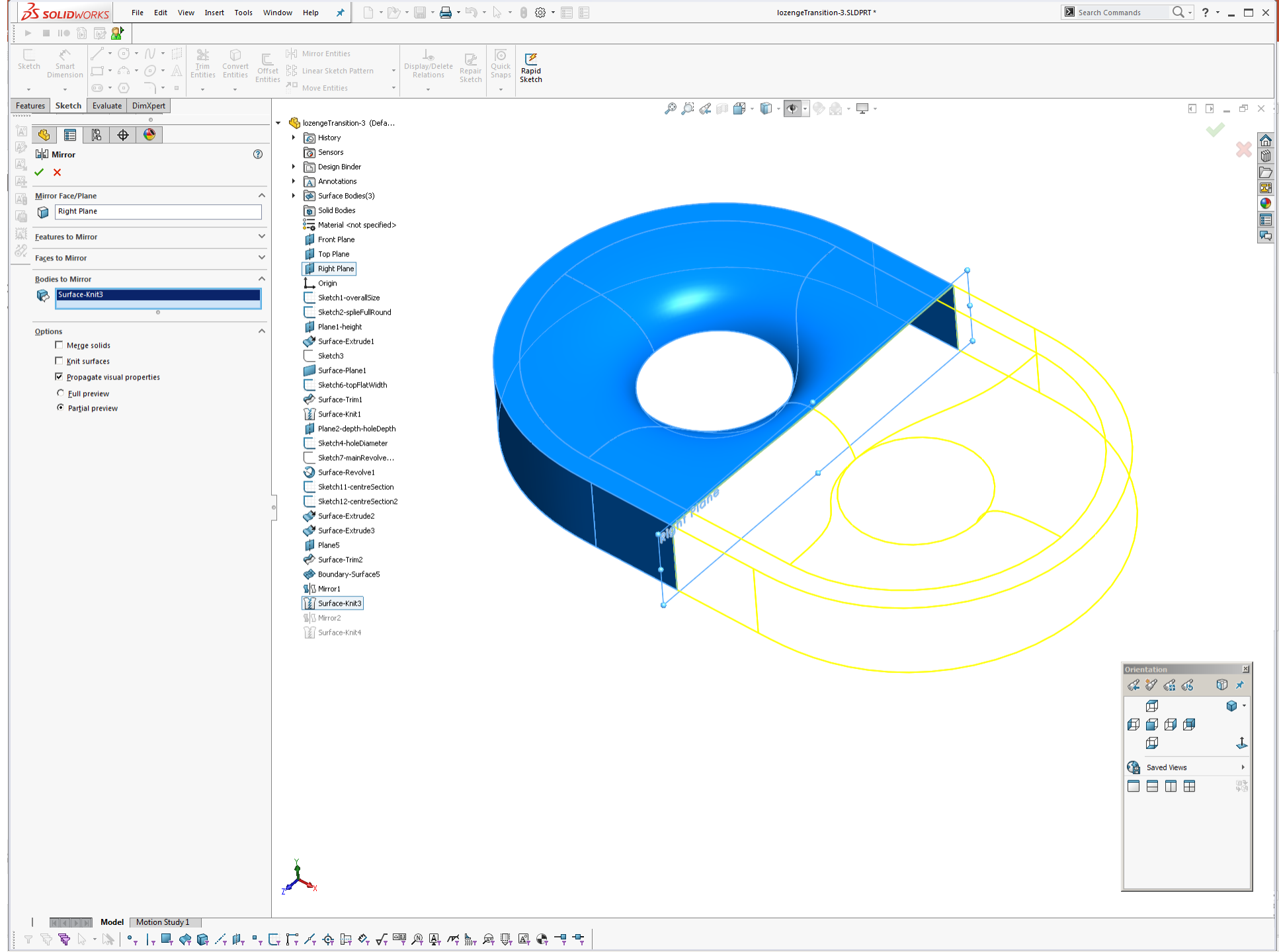
# Model sections of the form based around axes of symmetry

If the form is symmetrical around one axis (half) or two (quarter) then model a section and mirror the body. If there are asymmetric details, then add these after modelling and mirroring the symmetric forms.
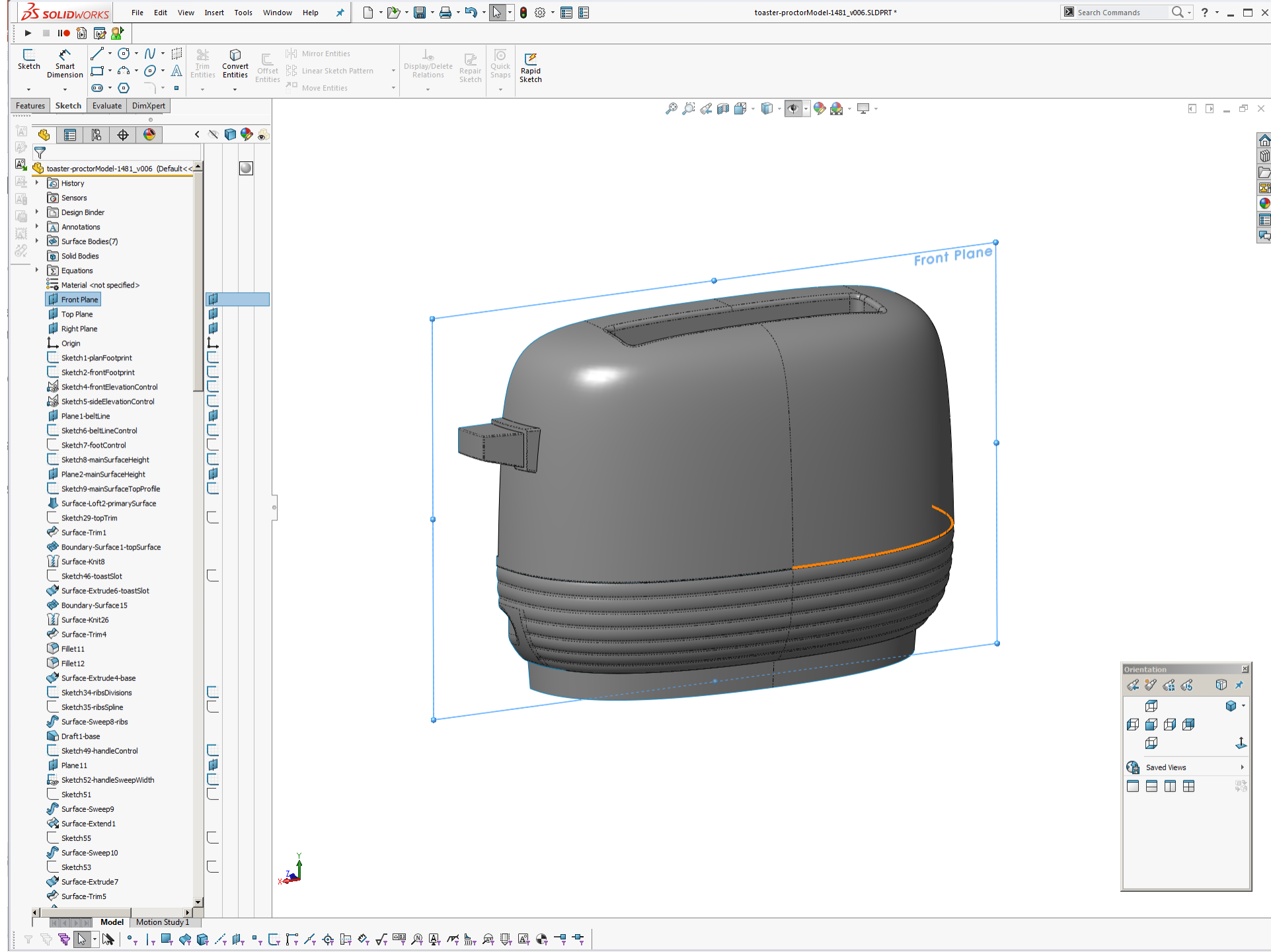
Simple features fail for no reason, or you cannot add simple features?

Tools>Evaluate>Check

If simple features like fillets and move face fall over or you cannot create new features, run the Check Entity function. You may have a general error or inconsistent face/edge (amongst other things). Solidworks will show you which face/edge etc is causing the problem, then it is up to you to roll back through the tree, repeating the Check Entity function, to figure out which feature is causing the problem and then figuring out a work around.
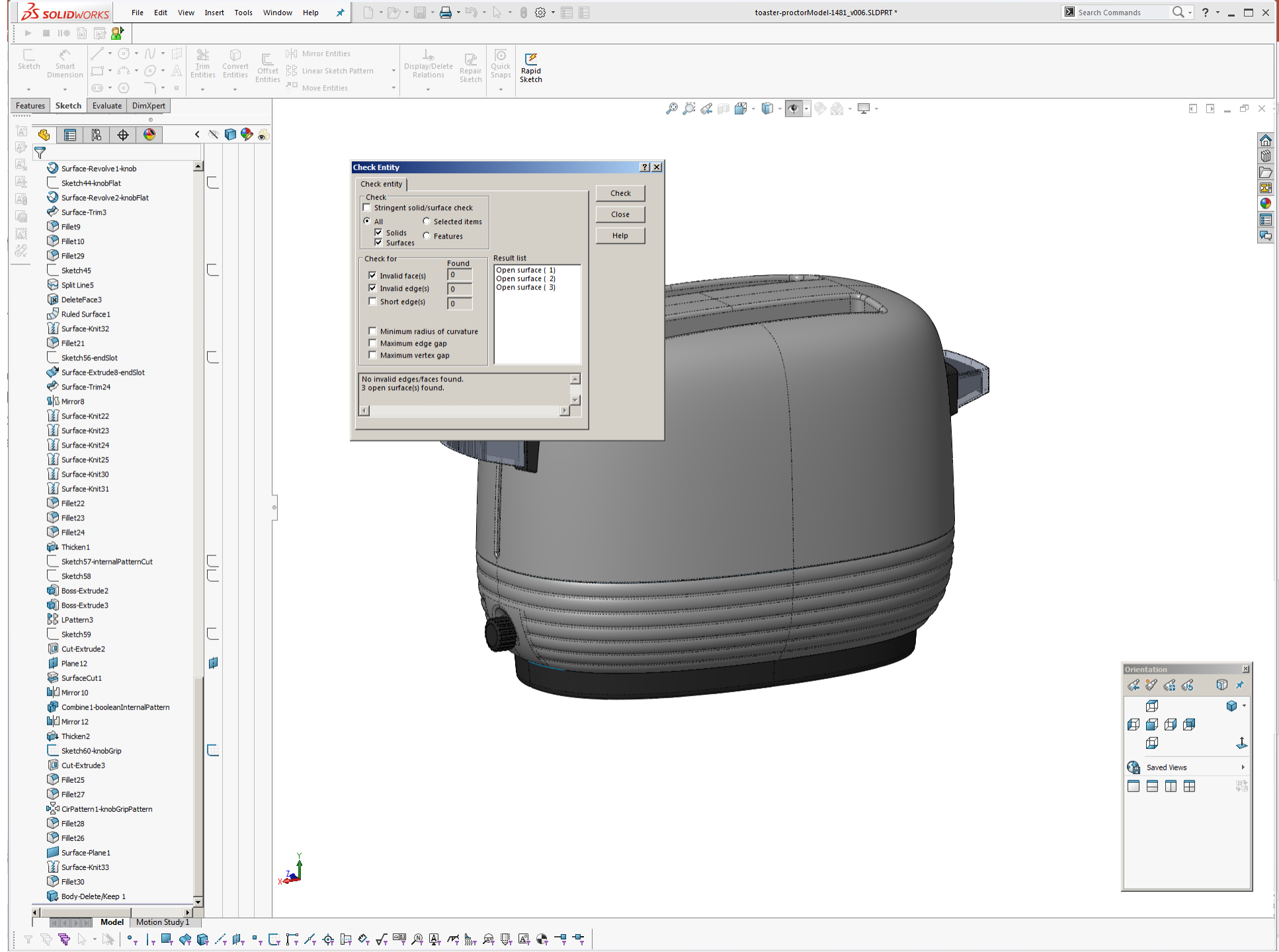
Use the style spline

The style spline is much easier to control and define than the old spline.

You can directly dimension and constrain the control polygon (could not on old spline and sometimes random things happen when you try and constrain the spline/handles).

You can also define higher than degree 3 for the spline. I frequently use degree 5 Bezier single span spline (with 6 CVs) to create curvature continuous curves between other geometry.
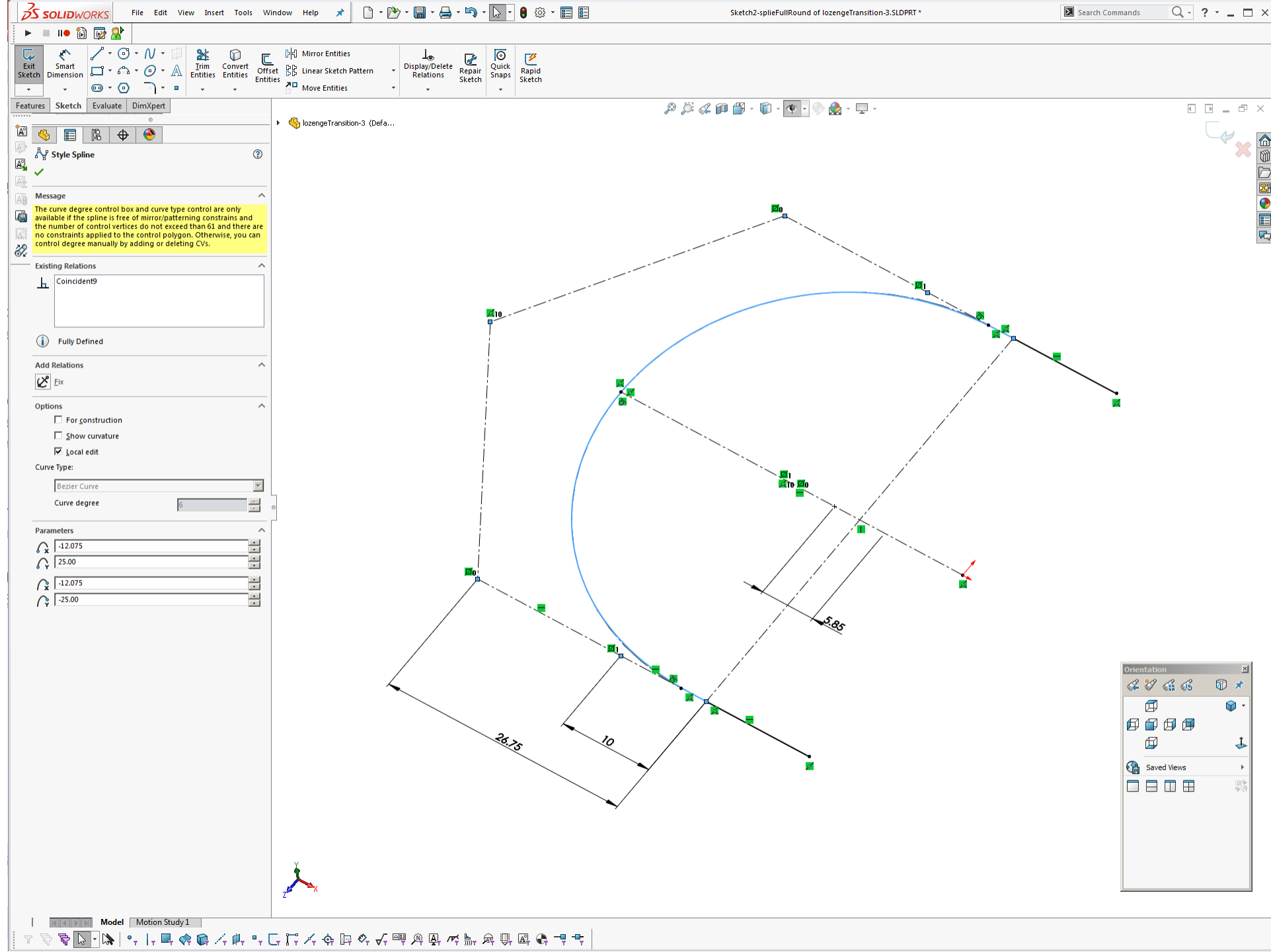
Flat view feature tree

Right click on the part name in the feature tree>Tree Display>Show Flat Tree View.

This displays the features in the order that you created them, rather than nesting features/sketches together. Firstly, this makes it far easier to trouble shoot your model when something falls over. Secondly, it makes it easier to recycle/reuse sketches in multiple features, without fishing around looking for the nested sketch. Thirdly… yes, I do have a ProE background ☺
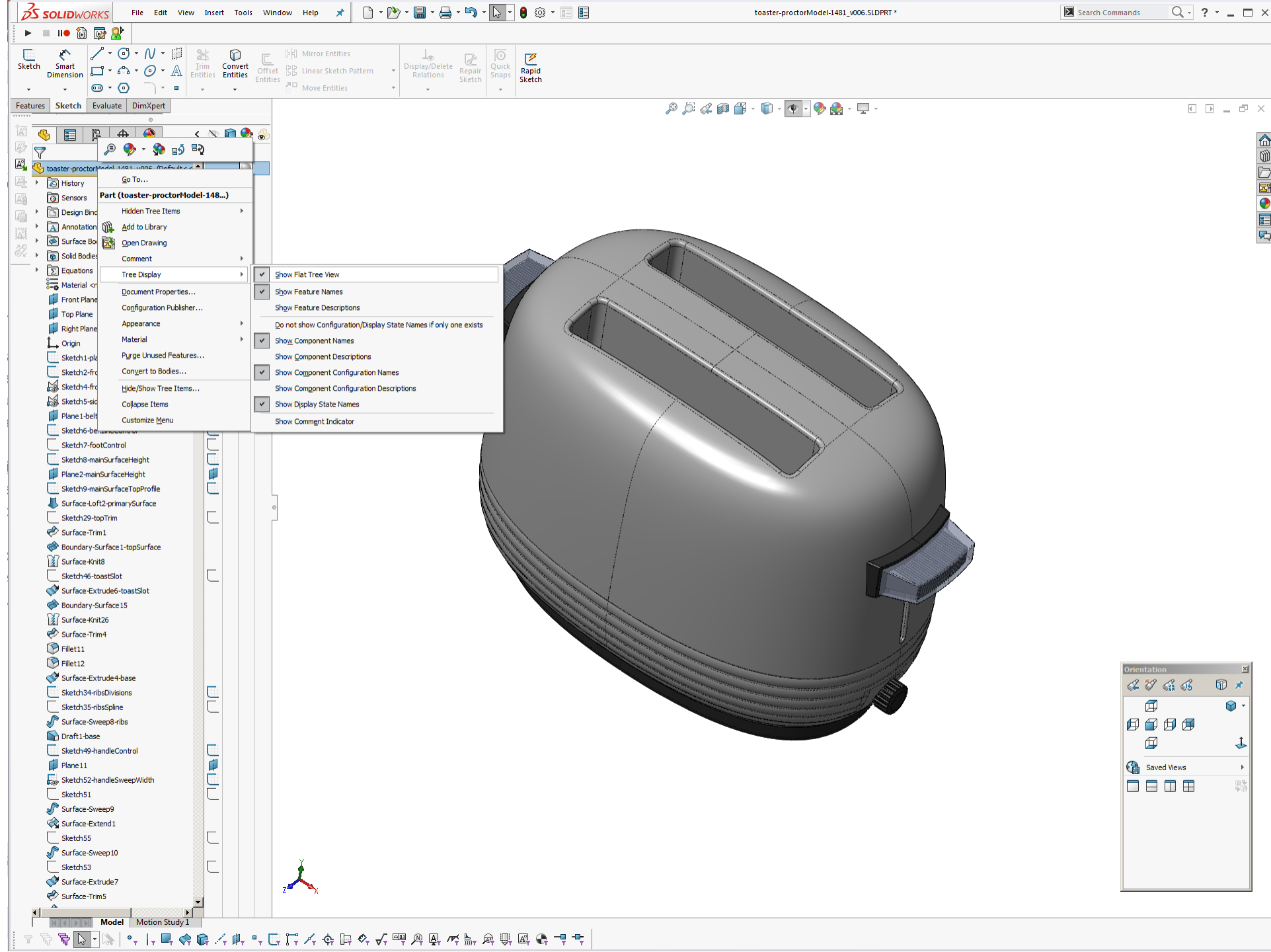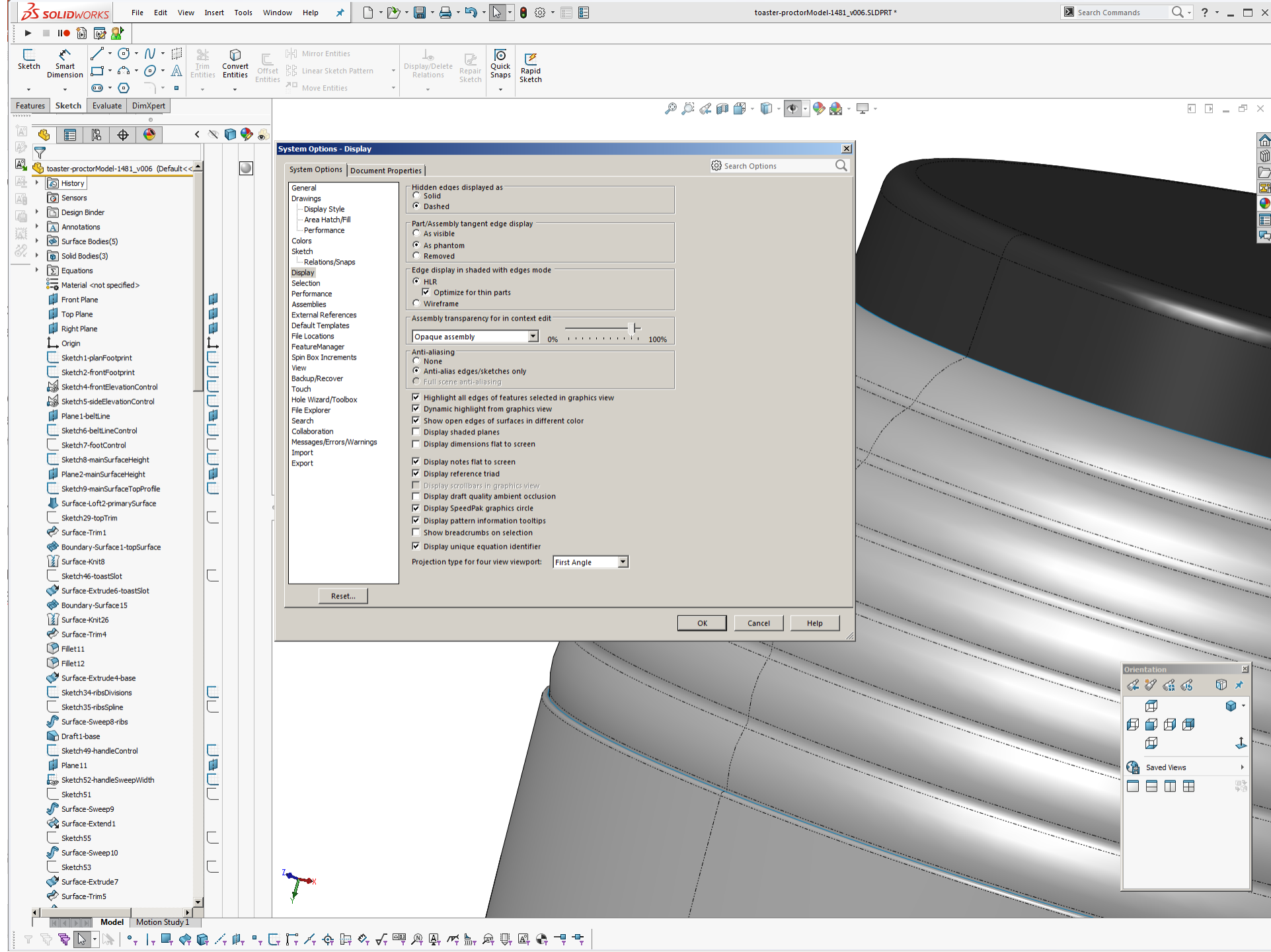
# Tangent edge display

Tools>Options>Display

If you change the Part/Assembly Tangent Edge Display to phantom, edges that are within the SW tangent tolerance are displayed with a phantom/dotted line. This makes it much easier when modelling to tell if an edge is tangent or not.

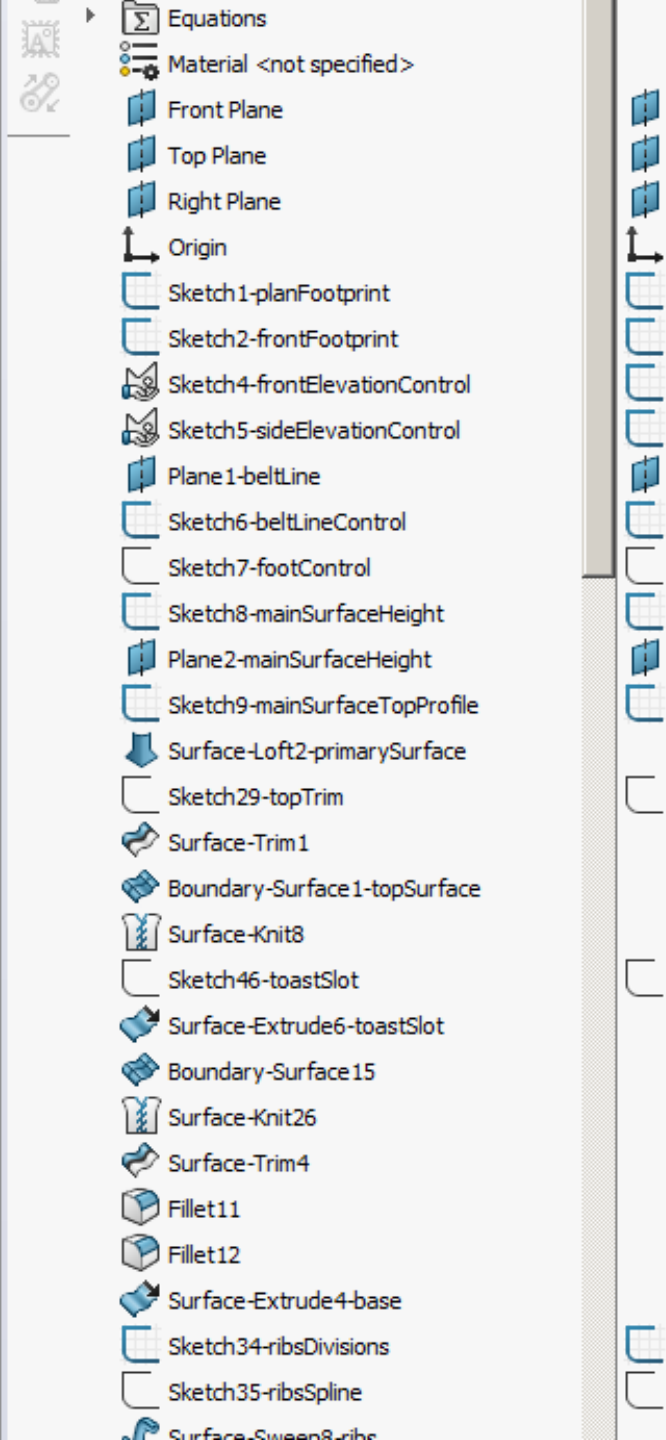Great for when you forget to set a centreline boundary condition to 'normal to profile'

Iterations/Versions of models

Make sure you save versions or iterations of your model, as you progress. This means you can have the latest version and an older version open at the same time, which is useful for trouble shooting features in the new version that may have fallen over due to alterations to the model.
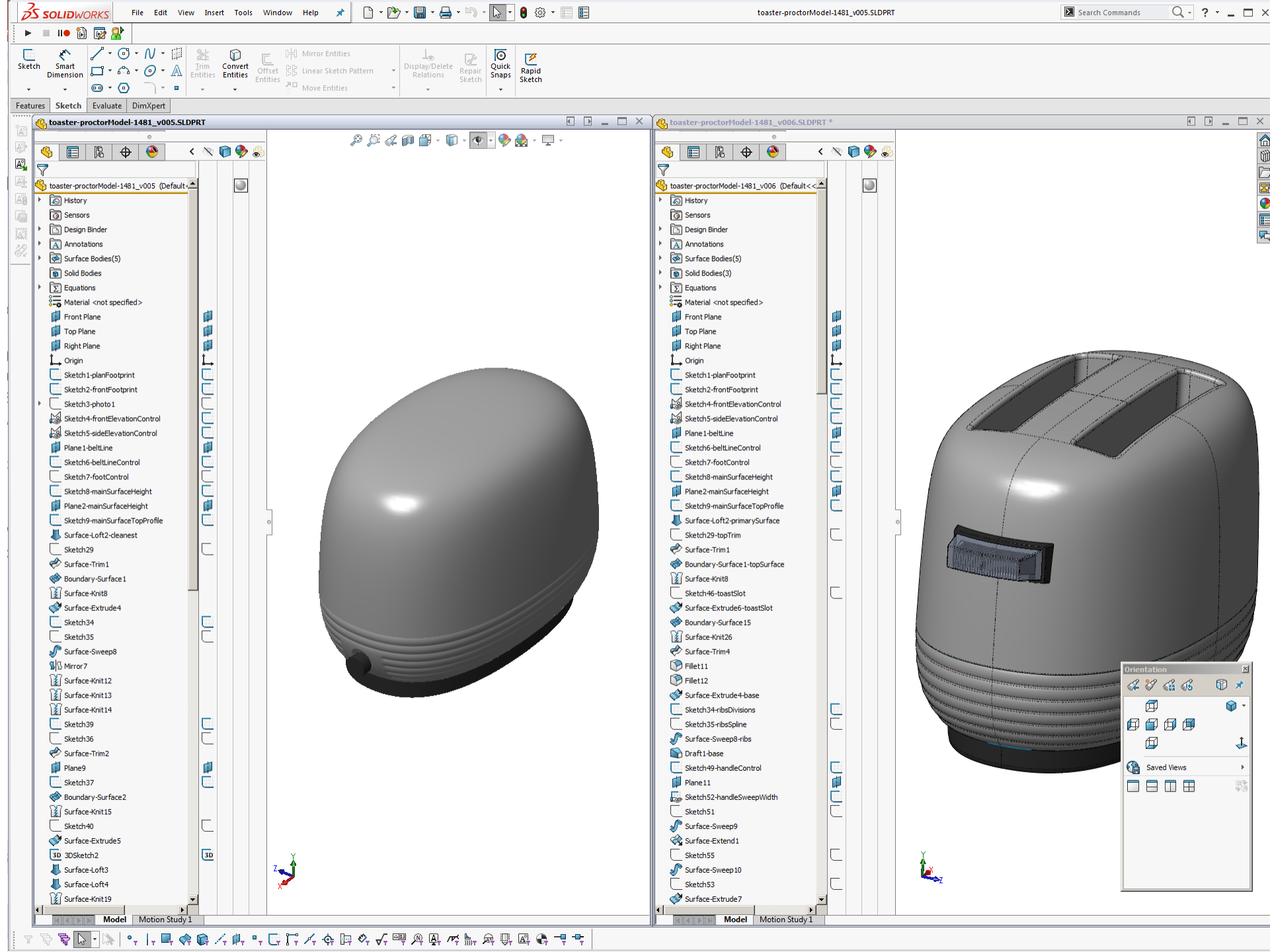
Name your features

It is useful to name features. Not necessarily all features, just ones that are frequently modified or may be heavily referenced. This makes it easier for you to work on a more complex model and also makes life easier if you are passing a model on to a client etc.

Σ Equations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Sketch1-planFootprint
Sketch2-frontFootprint
Sketch4-frontElevationControl
Sketch5-sideElevationControl
Plane1-beltLine
Sketch6-beltLineControl
Sketch7-footControl
Sketch8-mainSurfaceHeight
Plane2-mainSurfaceHeight
Sketch9-mainSurfaceTopProfile
Surface-Loft2-primarySurface
Sketch29-topTrim
Surface-Trim1
Boundary-Surface1-topSurface
Surface-Knit8
Sketch46-toastSlot
Surface-Extrude6-toastSlot
Boundary-Surface15
Surface-Knit26
Surface-Trim4
Fillet11
Fillet12
Surface-Extrude4-base
Sketch34-ribsDivisions
Sketch35-ribsSpline
Surface-Sweep8-ribs

Changing body between solid and surface

Be careful when making a solid body a surface (ie, by deleting a face) and vice versa, especially if you have downstream features.

Unfortunately Solidworks looses references when a solid becomes a surface (like Delete Face 4 in the picture) even though the actual referenced entities are still there. This can make life difficult when you have to make alterations to a complex model. So instead of using surface operations, you may be limited to solid operations and using surfaces with replace face.

Another biggie with making a solid body a surface body is any drawing that references the surface will also loose some references!
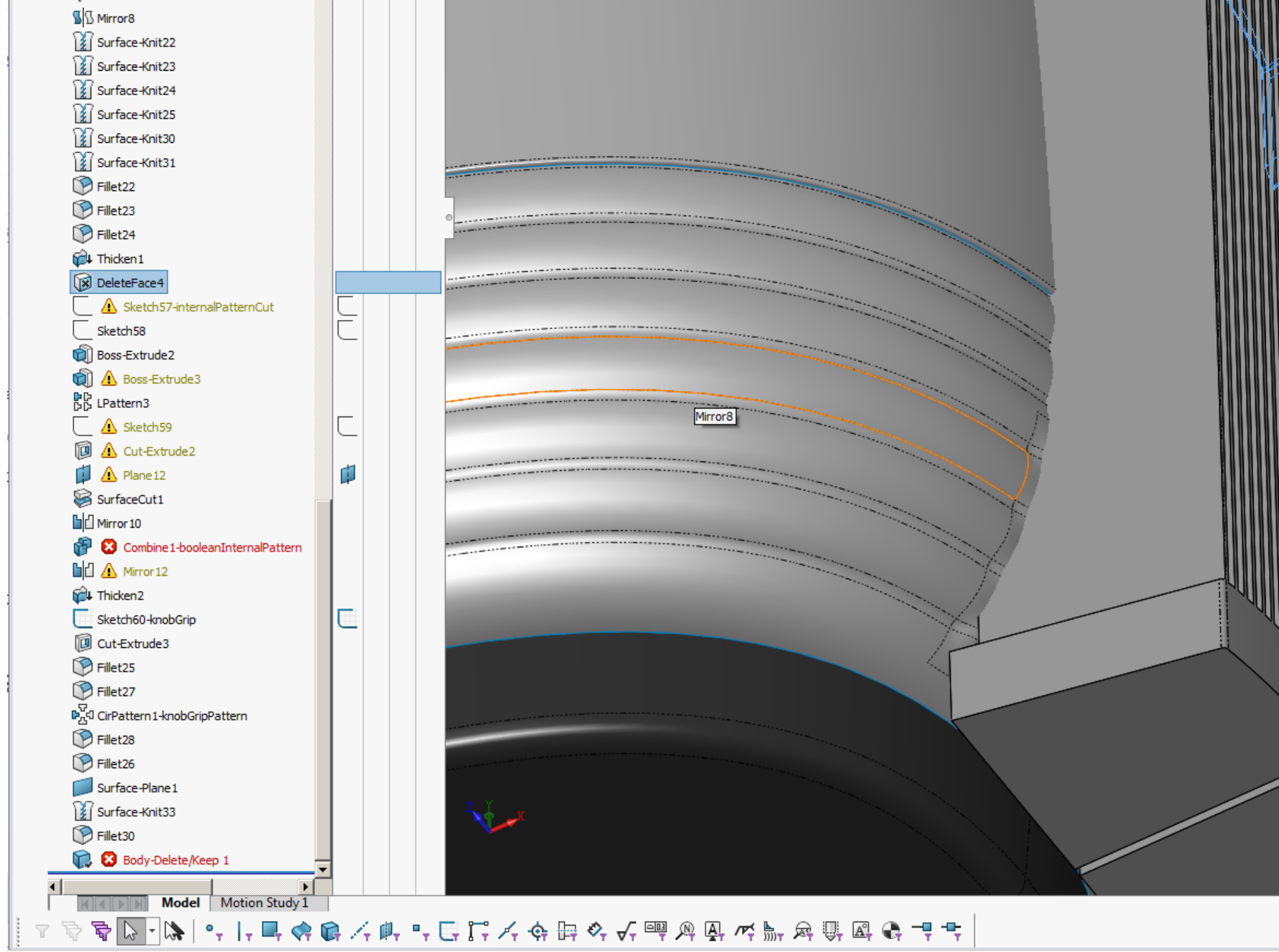
Projected curve into a 3D sketch

Projected curves (sketch onto sketch or sketch onto face/surface) are handy to use, however in the past I have sometimes struggled to create references or constraints to the actual curve or the end points.

I always create a 3D sketch directly after creating a projected curve, then convert the projected curve entities into the 3D sketch. After doing so SW is much less fussy with the 3D sketch. Also means you can extrude the 3D sketch etc if needed...
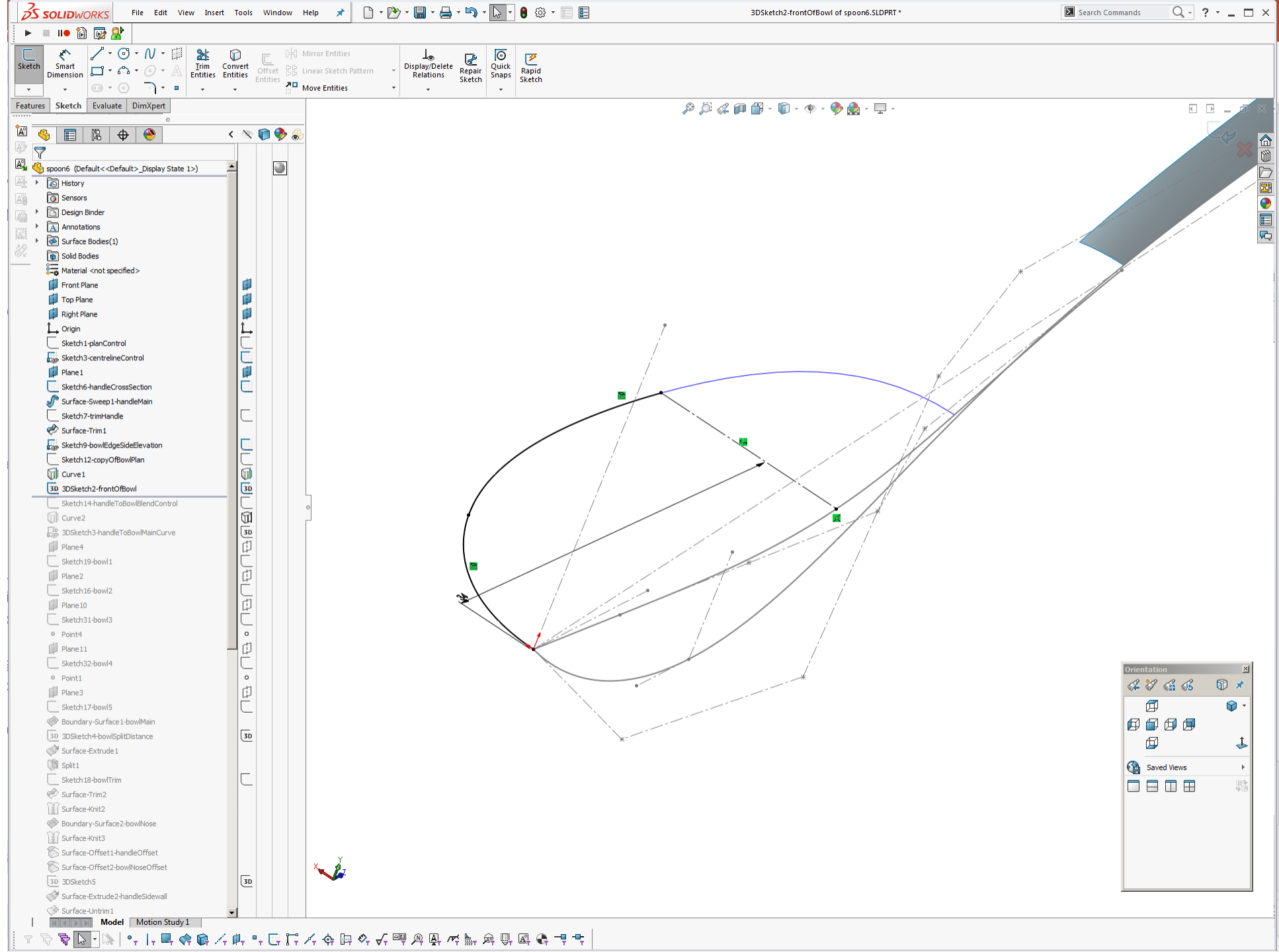
Fit Spline

If you have a sketch/3D sketch with multiple tangent sections, it is sometimes useful to combine the sections into a single spline.

Tools>Spline Tools>Fit Spline

The spline in the image is created from 3 donor splines, which still exist, just SW converts them to construction geometry when you run the fit spline command.

Be aware that you need to specify a tolerance for the fitted spline, so if you have gaps in the input geometry (ie, sometimes if you've converted entities on edges) or are converting a line and tangent arc, you can get jumps/wobbles in the curvature graph. Which makes sense, as a spline cannot exactly fit through a line and tangent arc!
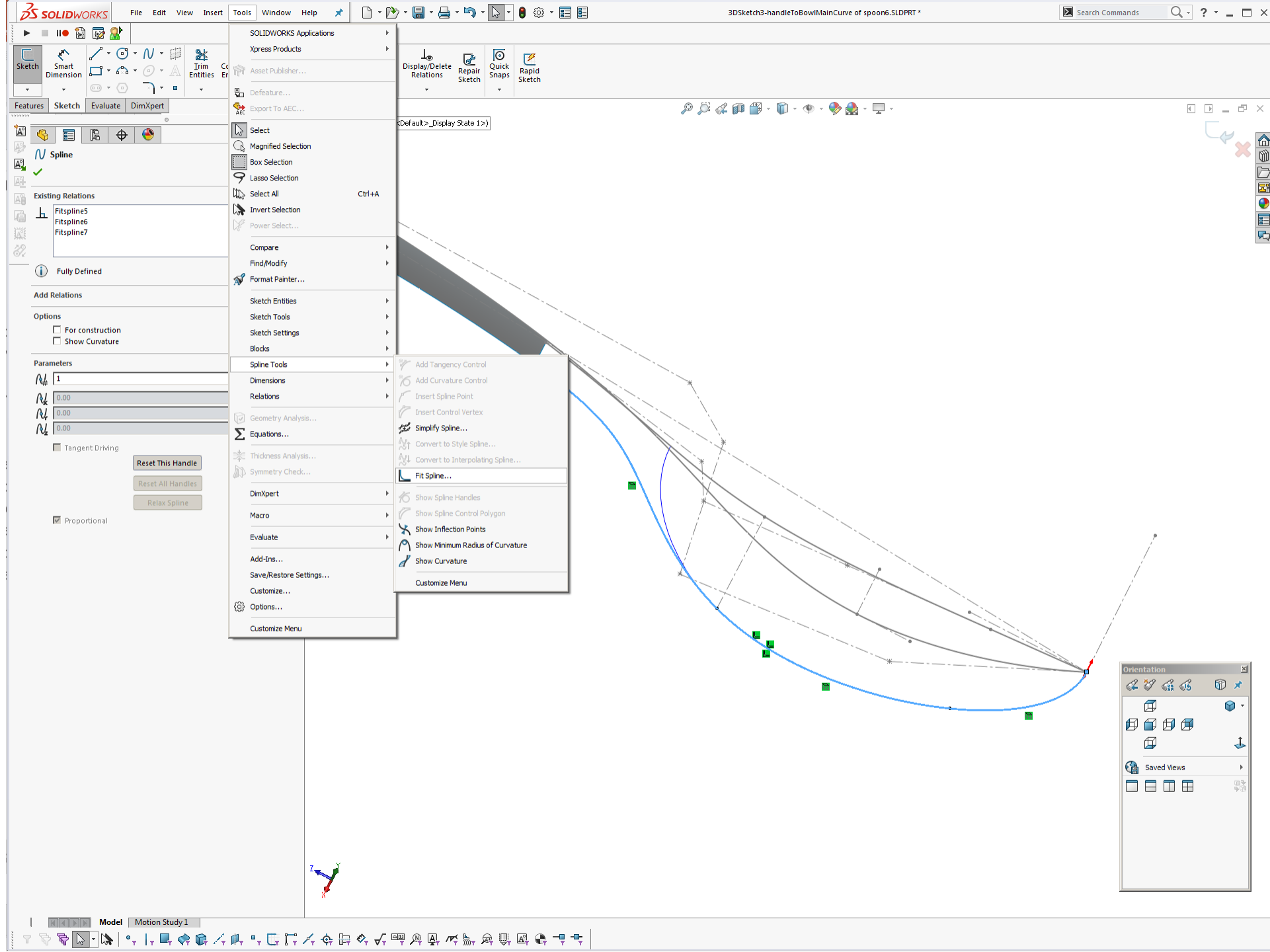
Replace entity

Tools>Sketch Tools>Replace Entity

If you need to replace an entity in a sketch it is useful to use the replace entity function (ie, replace arc with style spline). This is meant to reroute downstream references from the old entity to the new entity. Sometimes works, sometimes does not, query whether Solidworks re-references the endpoints of the entities.
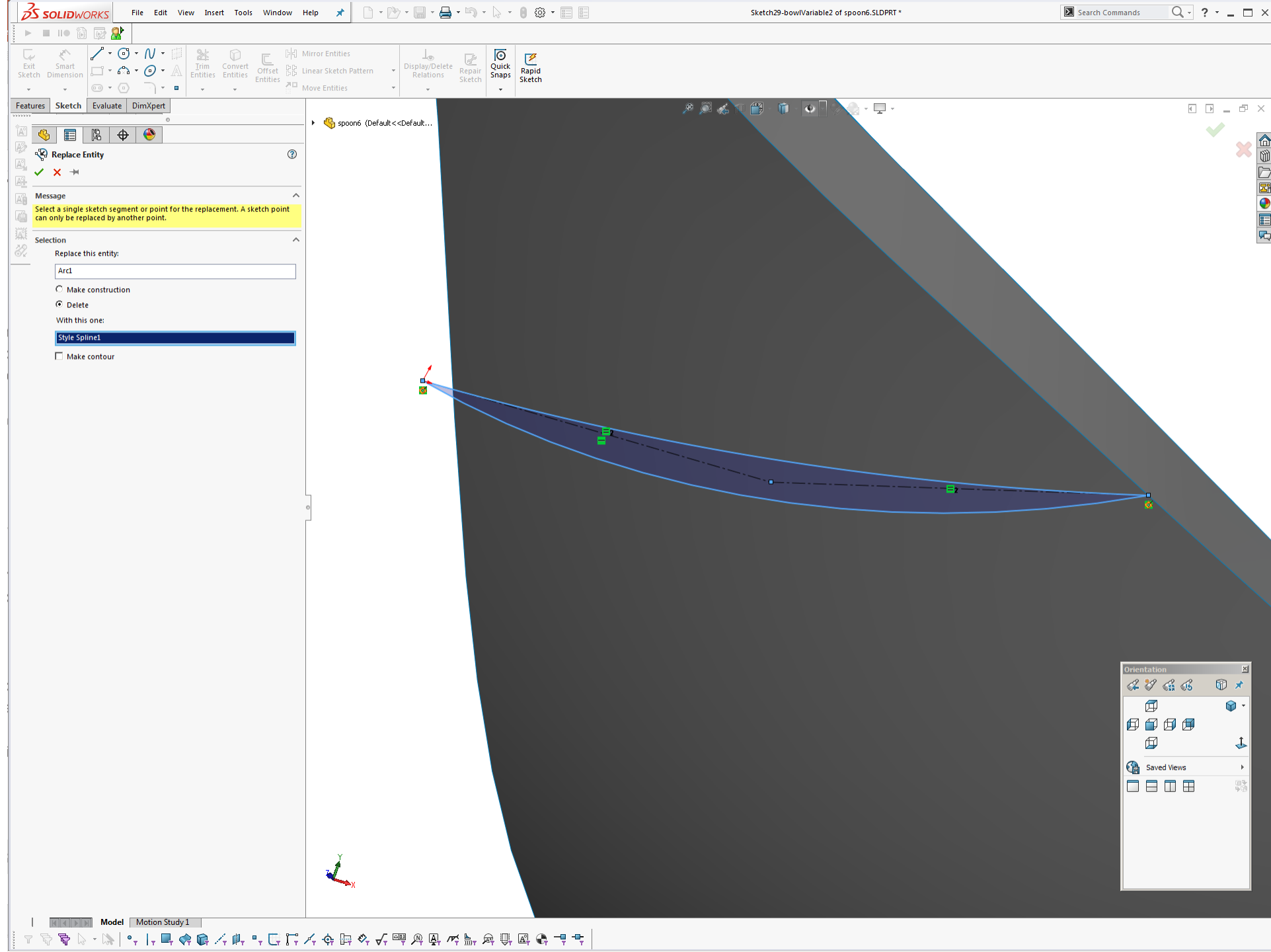
Offset entity

If you are having trouble offsetting multiple entities in a sketch, it sometimes helps to convert the entities in a prior sketch, then offset the converted entities. This is more relevant to converting and offsetting edge entities than other sketches.
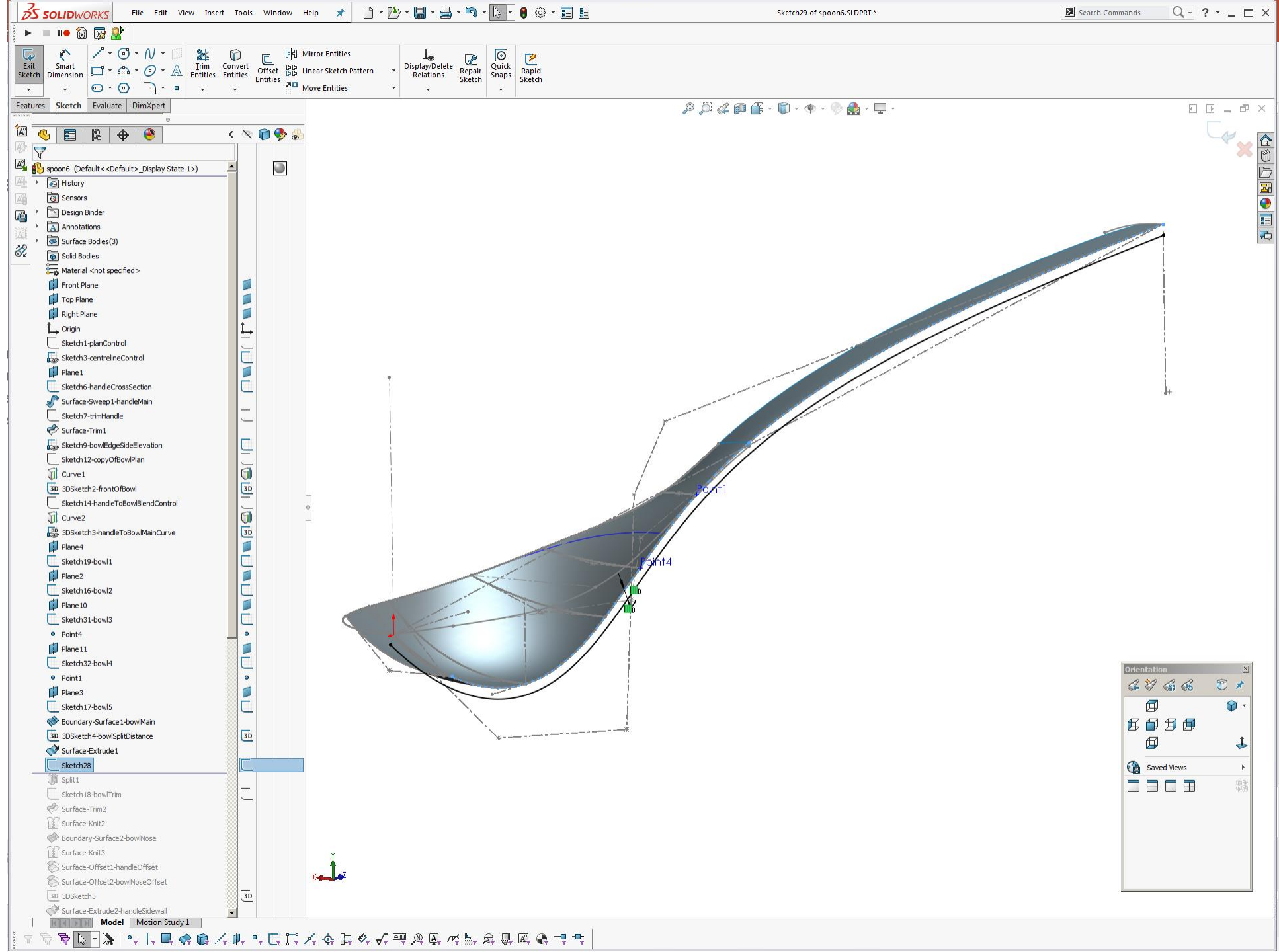
# Replace more than one face with a single surface

If you have multiple faces that you want to replace with a single surface (with no surface breaks), this quite often fails. A work around is to delete the faces and replace them with a single face using Insert>Face>Delete and using the delete and fill function. Do not select tangent fill, as your nice surface will replace the lumpy delete face fill surface, in the next feature!

There are limits on what this can achieve, as is always the case with replace face.
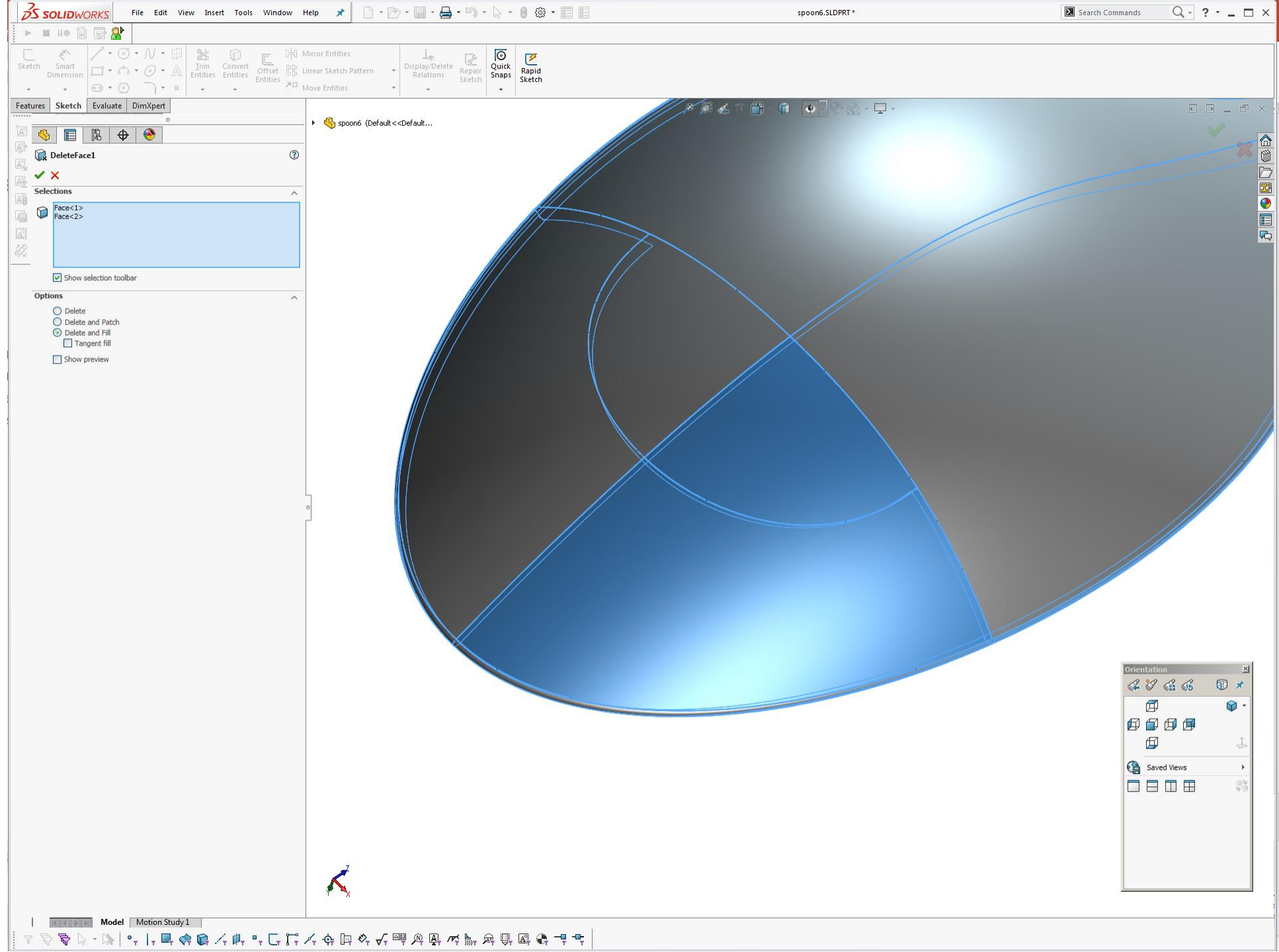
Edge won't fillet

If you cannot get an edge to fillet using the constant size fillet function, try using a face fillet instead. If the face fillet does not work, try using a variable fillet with all handles set to the same dimension. This frequently works!
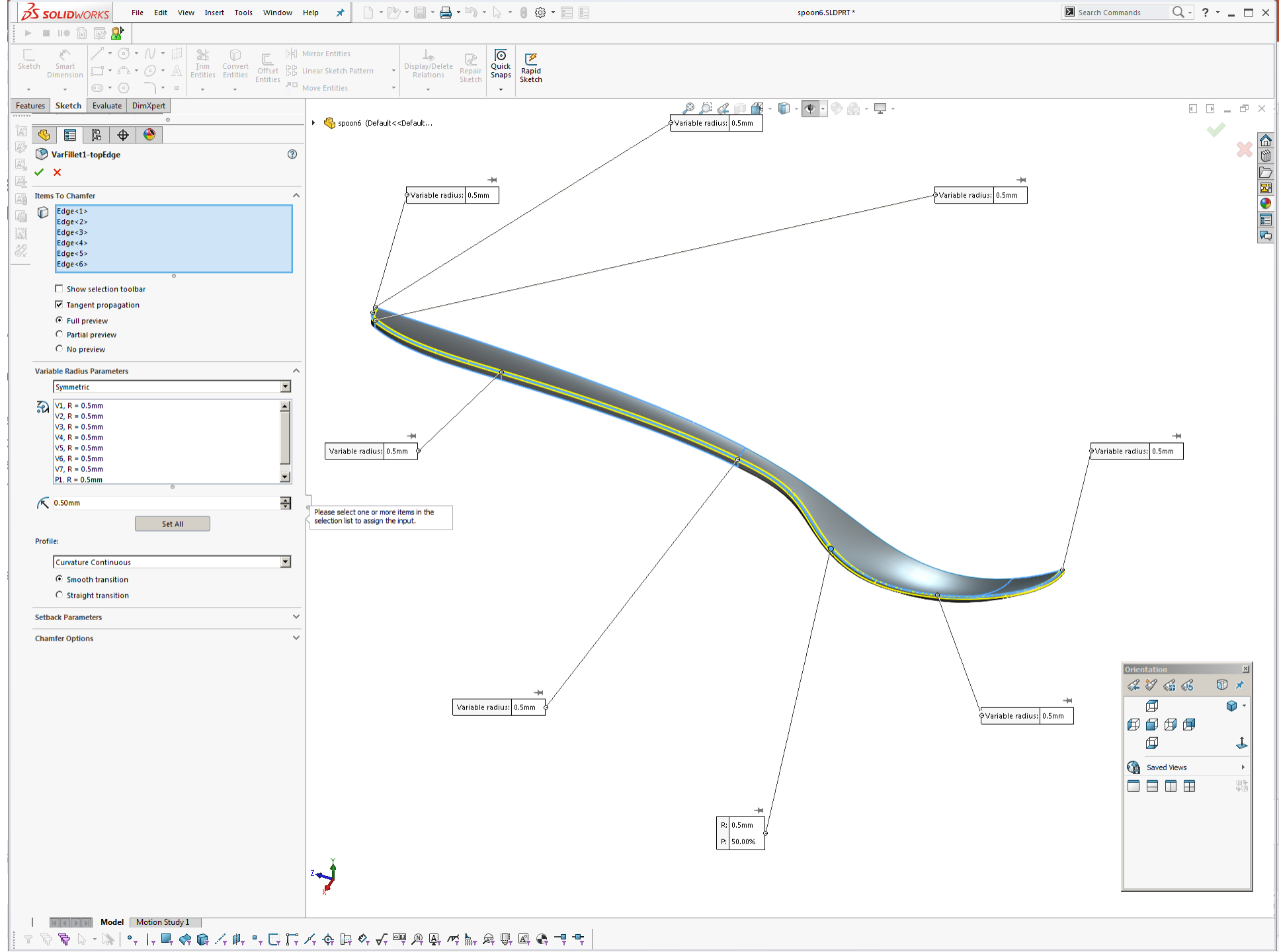
Sketch fillets

Try not to use sketch fillets and use a geometry fillet on surfaces/solids instead. Sketch fillets mean your other sketch geometry gets trimmed, which means more work if you decide later you no longer want the fillet. Using a fillet on the surface/solid edge also means you can change from a circular to curvature continuous fillet, without having to redefine a sketch and change the entity type.
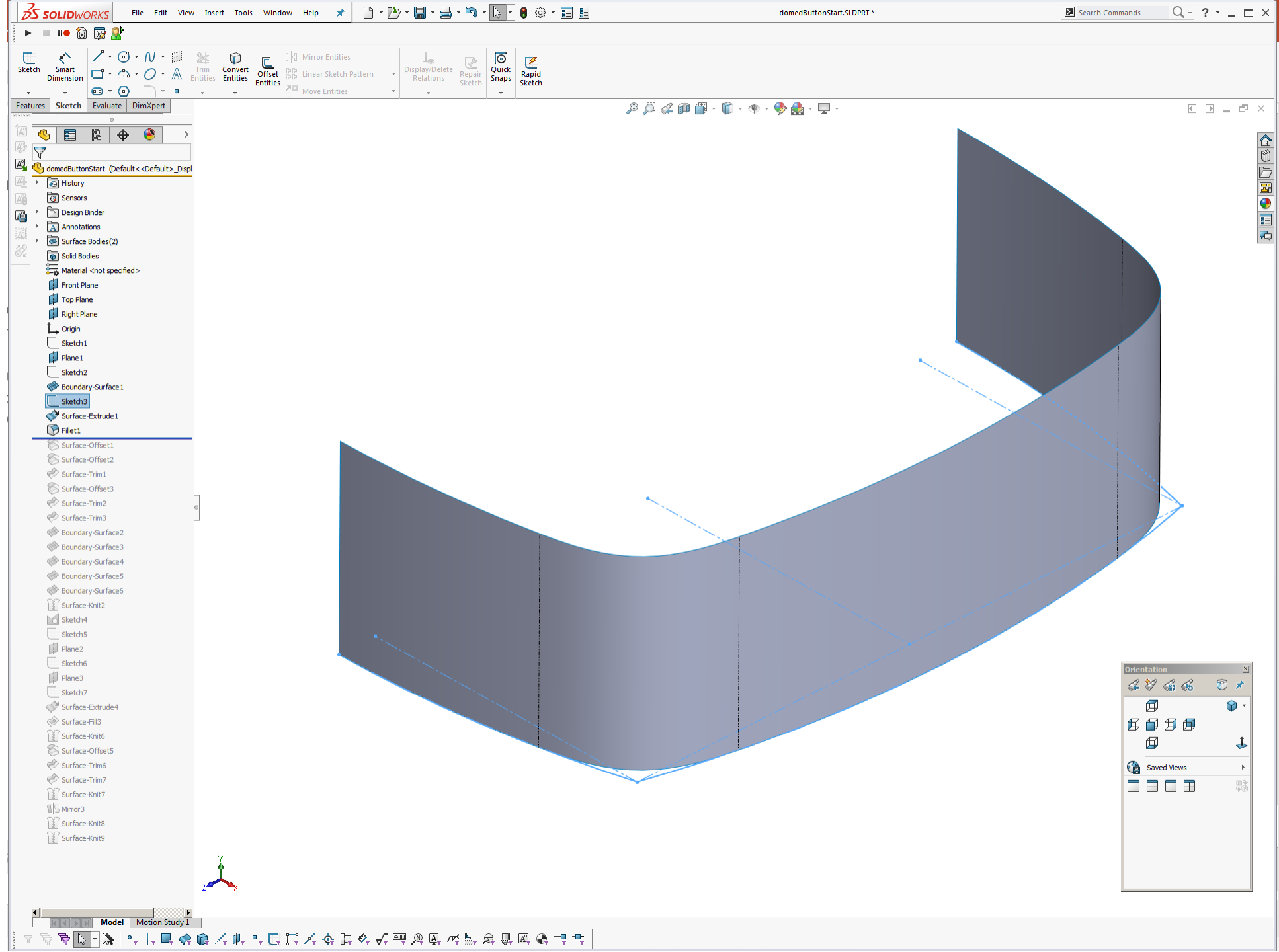
Lots of features

Use multiple simple features, rather than a single complex feature. For example, create several 3D sketches, rather than making one complex 3D sketch.

This makes it easier to trouble shoot should something fall over.
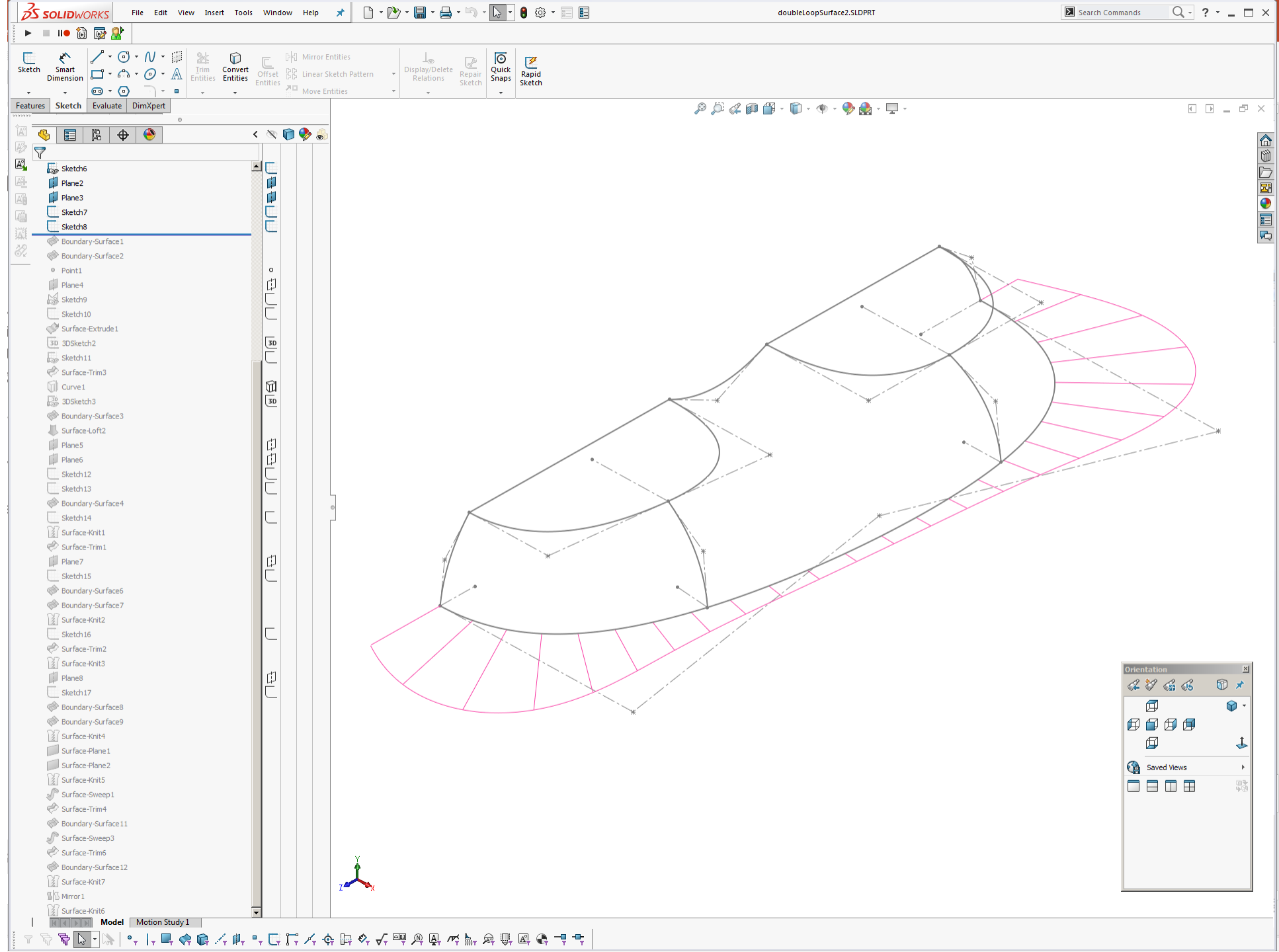
Surface Breaks

Try and keep breaks in surfaces to a minimum. Less breaks at the primary surface level means less breaks downstream (once things are trimmed etc). This has a trickle on effect, i.e. once draft is applied to a model. Multiple tangent edges being drafted results in multiple breaks in the drafted surface, which means even more complexity once any edges are filleted.

Using fit spline (as explained elsewhere) is useful to help reduce surface breaks.

Image just for effect, not necessarily a realistic situation!
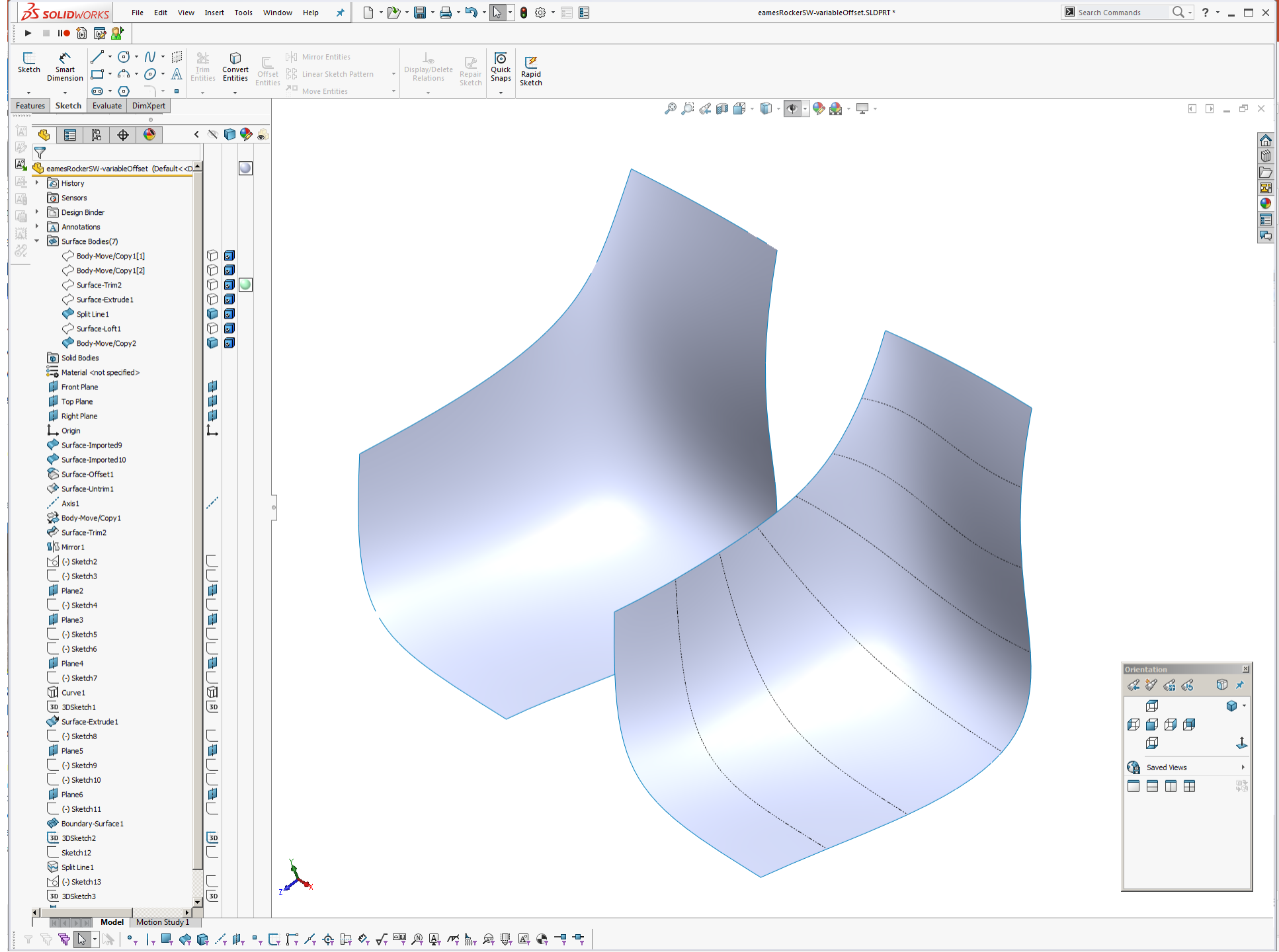
Design Intent

- Some other notes that can be useful to keep in mind when modelling – what is my design intent?

- Dimensions. What may change. How should I baseline the dimensions?

- References. How should sketches etc reference other geometry. If something upstream changes, do I want it to affect this geometry?

- What is the best way to build flexibility into the model. Is it a concept model? Create a bounding box sketch and reference that downstream so quick changes can be made to the overall size etc?

- Intended for manufacture? If so, including things like draft angles from the beginning can make life a lot easier…

- Draft BEFORE fillets, unless the fillet is a dominant feature and needs to be earlier in the feature tree

- Is this a master model? If so, try and keep features and geometry to the minimum that is needed across several parts

- Is this a concept model? If so, don't get too hung up on not using fill surfaces or boundary surfaces with a degenerate edge. If the model is developed further, the surfaces can be redefined then

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