SOLIDWORKS WORLD 2015
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PHOENIX CONVENTION CENTER | PHOENIX, AZ

SURFACES & SPLINES

THE FUNDAMENTALS OF MODELING FOR CONSUMER PRODUCT DESIGN

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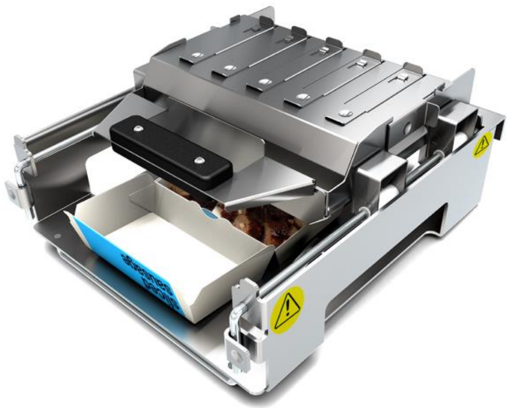
DIMONTE GROUP

And Who Am I?

An Industrial Designer at the DiMonte Group; a product development consultancy working across multiple industries, materials and processes. CAD can't dictate our designs.





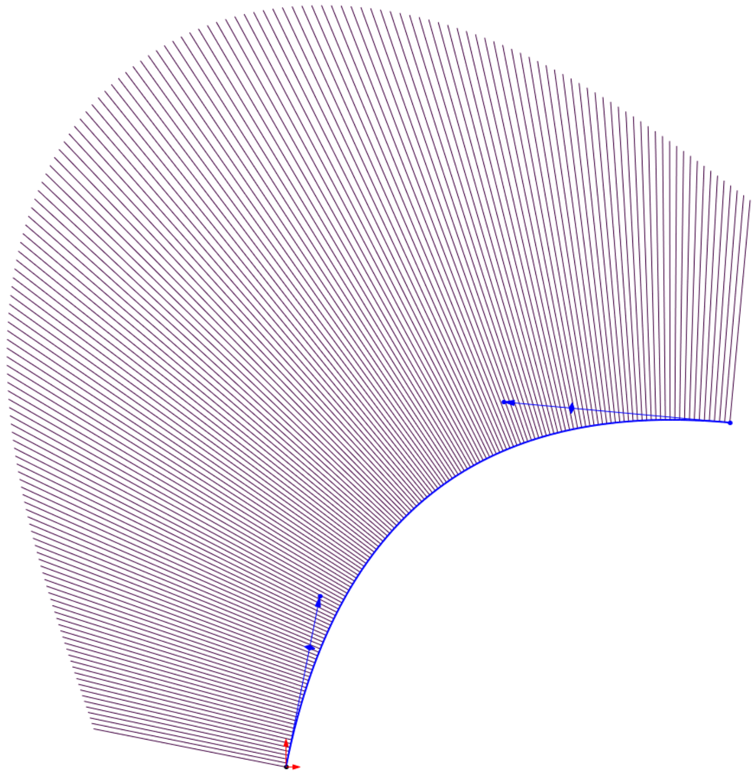



Design Details

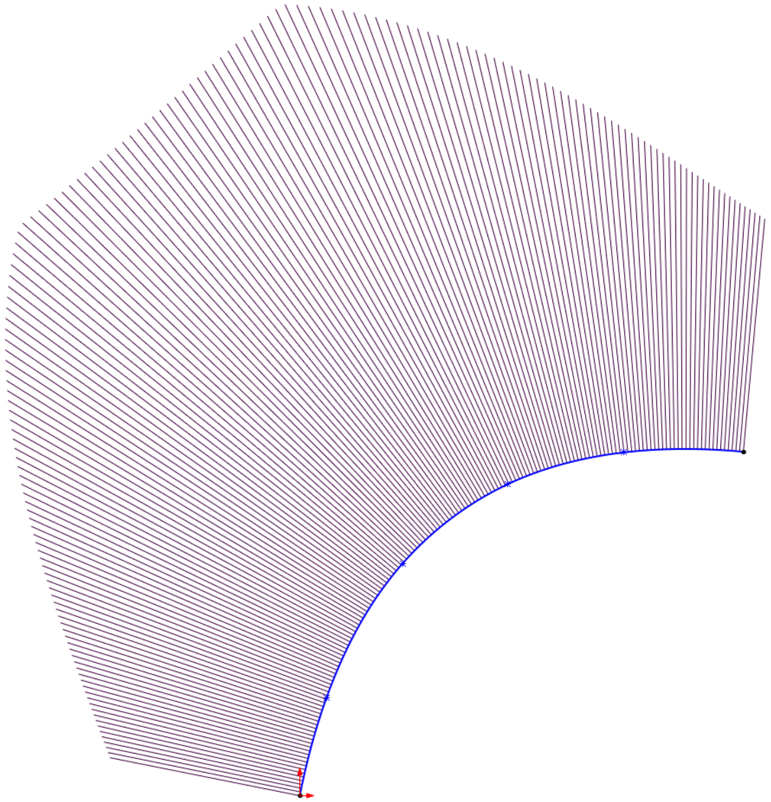
SPLINES

Analyzing Splines and Surfaces

The key to great surfaces is great underlying curves. The curvature comb tool is critical to evaluating the difference between what looks smooth, and what is actually smooth.



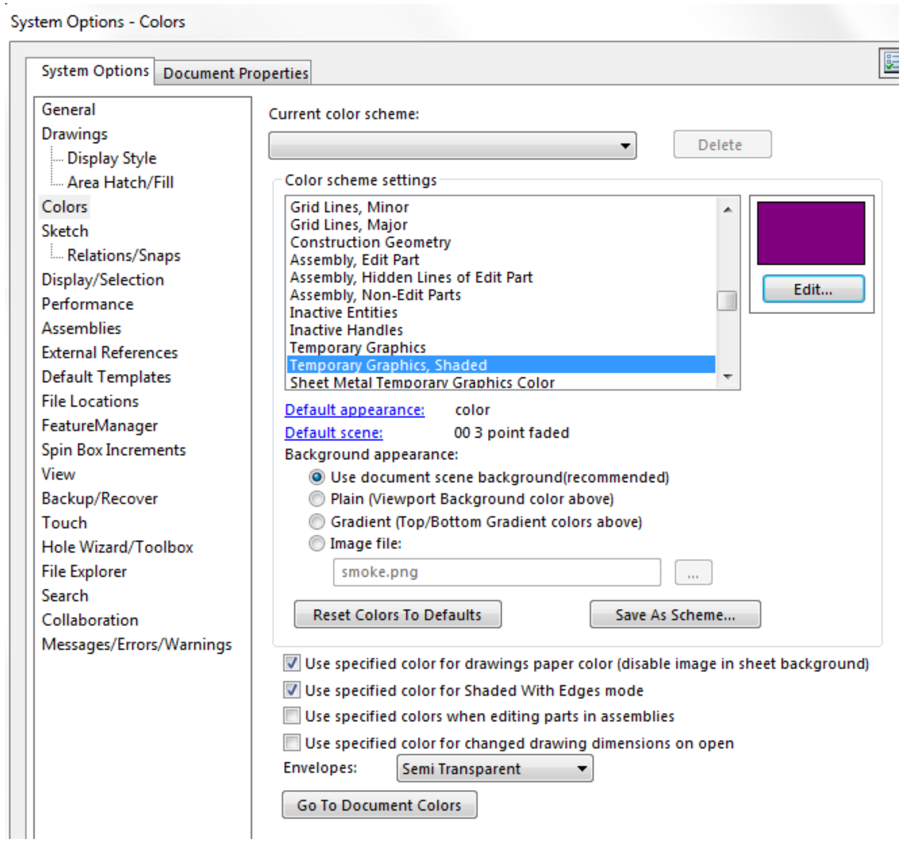
Created by manipulating spline handles

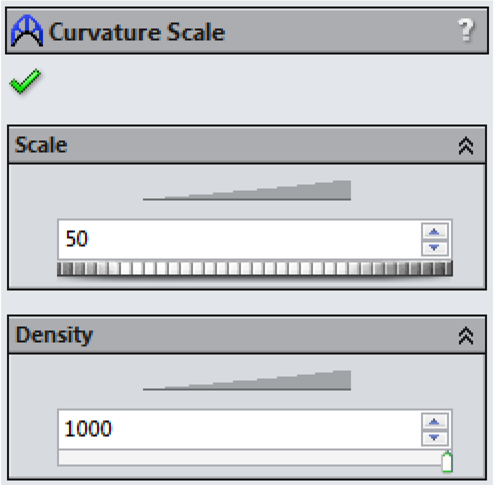


Created by adding multiple spline points

Change the Defaults

The default yellow curvature comb color can be difficult to read. Likewise, the default curvature comb settings can be optimized.



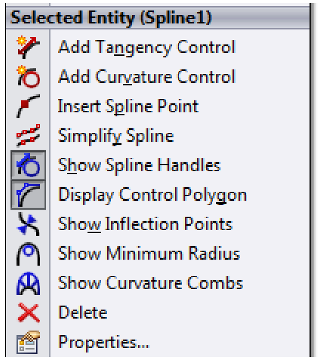


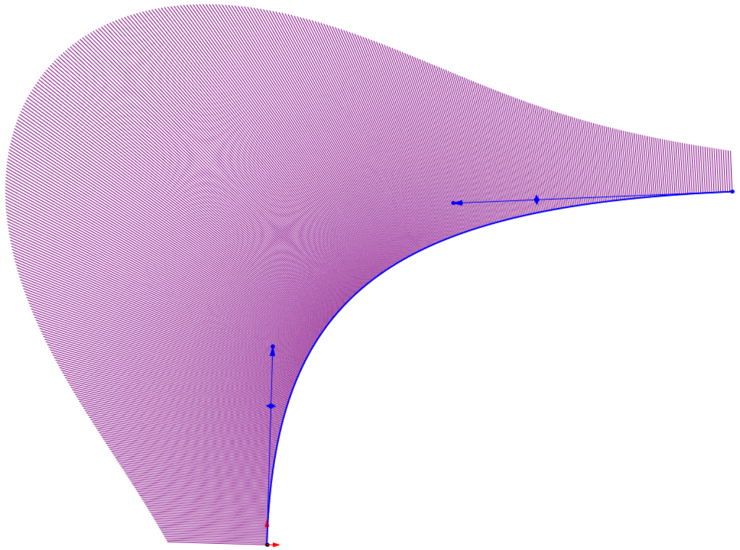
NOTE:

The density slider cannot be dragged to 1000. It must be manually typed in.

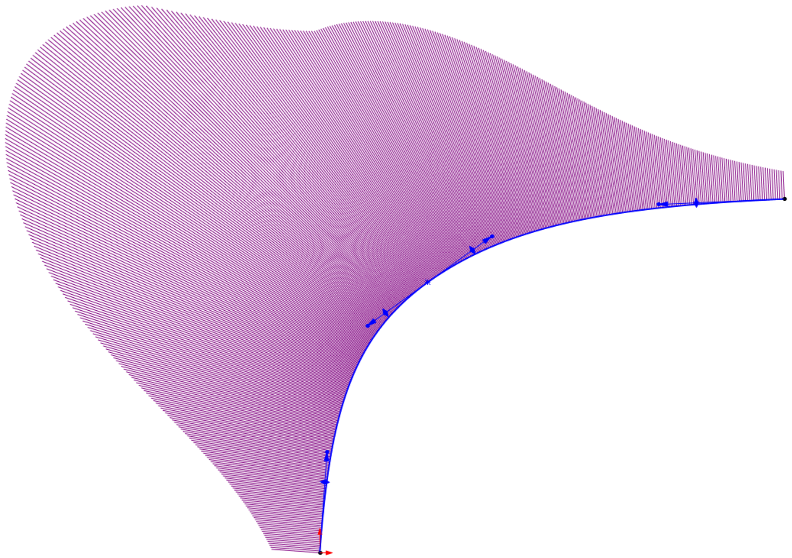
Interpolated Spline Tool

The key to using the interpolated spline tool is to use as few spline points as possible. Manipulate the spline via the end points. 2 point create the smoothest curves. If a 3rd point is required, avoid manipulating its handles. Use the control polygon instead.

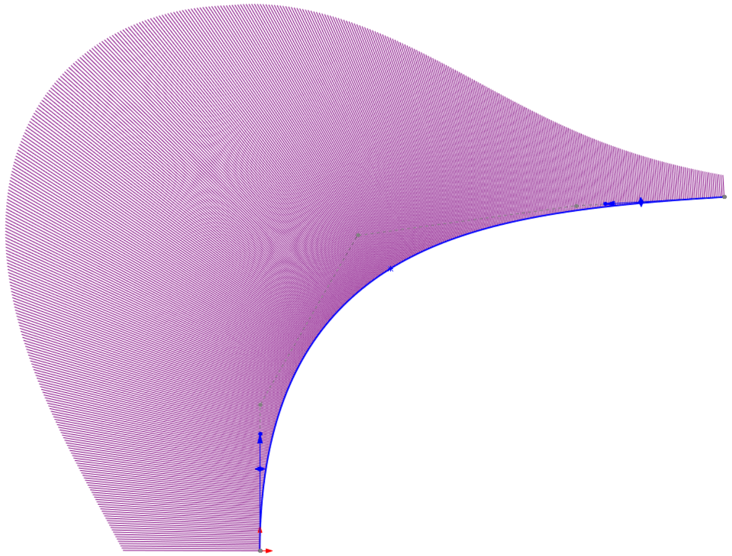




2 Point Spline



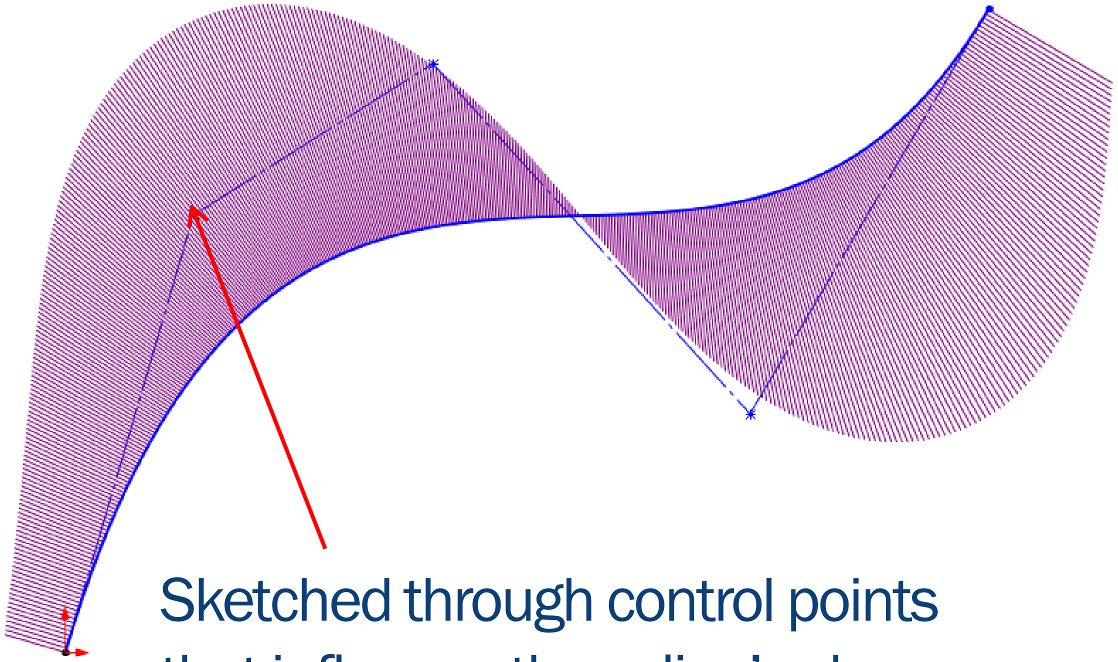
3 point, manipulated handles



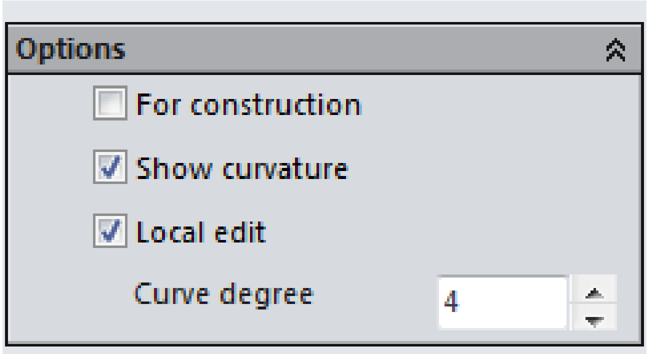
3 point, control polygon

The New 2014 Style Spline

The Style Spline is a GAME CHANGER in surface modeling. It uses a different set of math than the interpolated spline. This means smoother curves with less spikes and dips in curvature. It is also created differently from the interpolated spline.



Sketched through control points that influence the spline's shape.

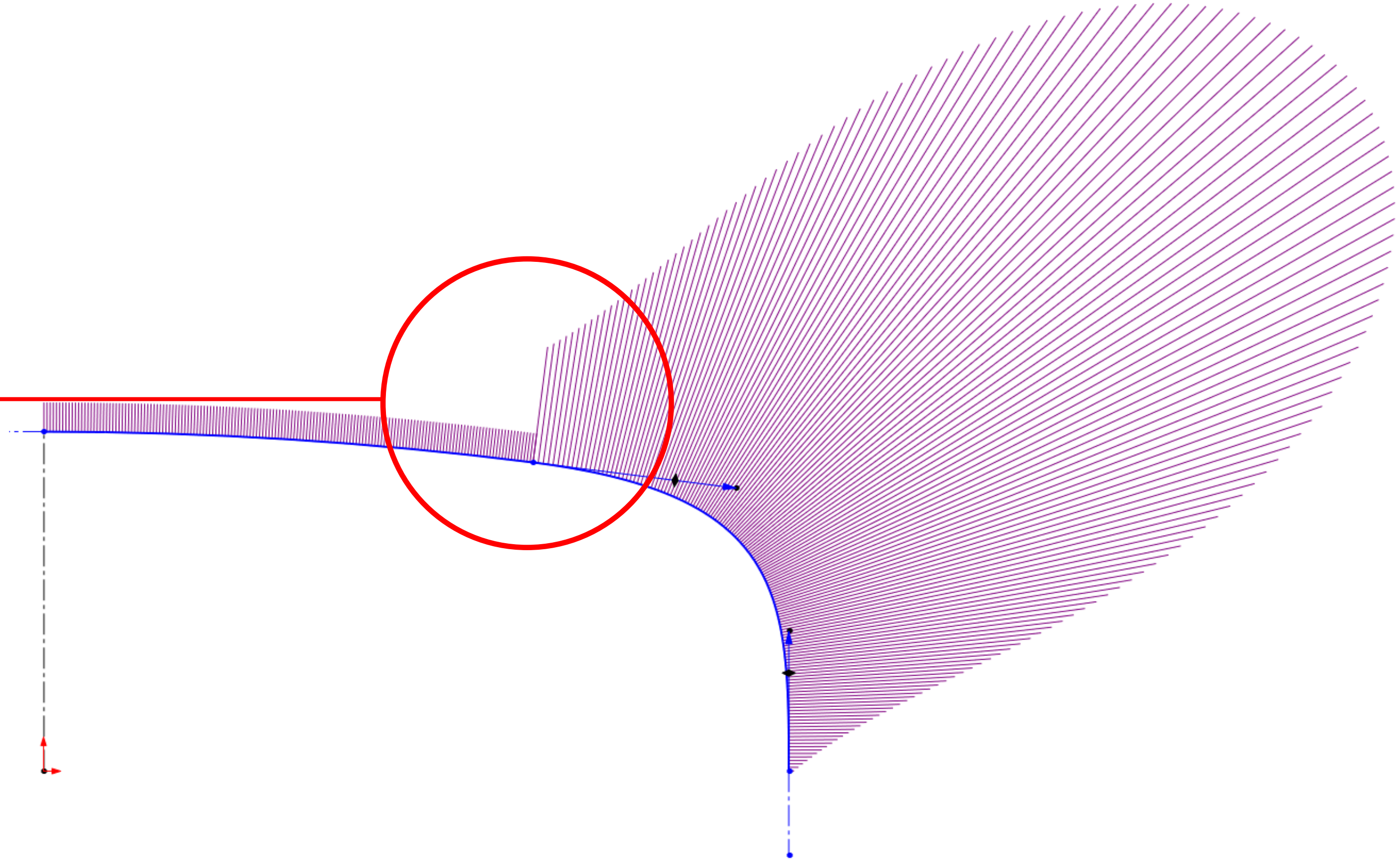


The curve's degree can be changed. The degree = N control points -1. Adding more points decreases each point's influence on the curve.

Tangency (G1)

When two curves are G1, the angle of the two curves at the junction is equal. While there is no change in angle in the curves, the transition can visually be seen.

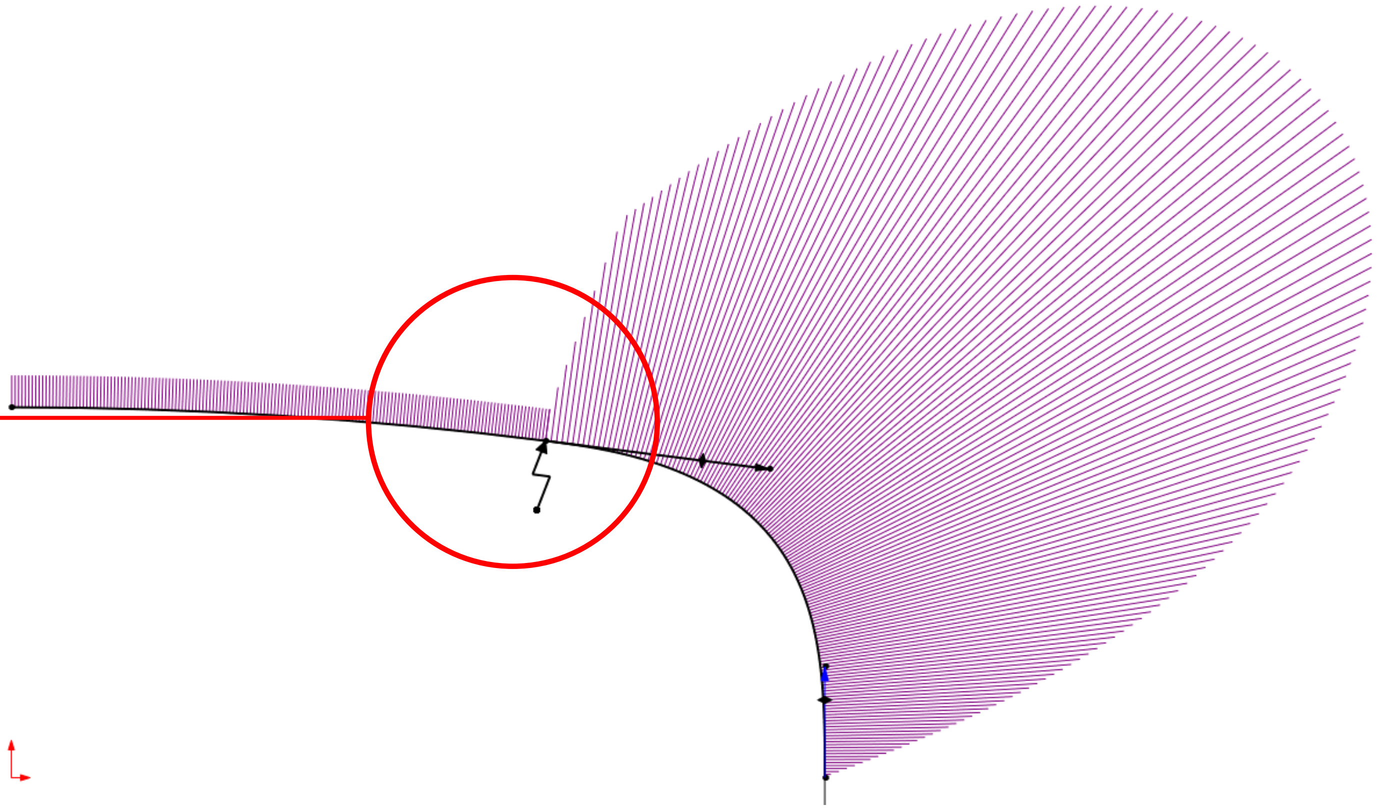
The angle of the curvature combs are equal, but the length of the curves is not.



Curvature Continuous(G2)

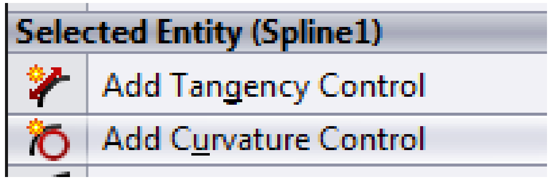
When two curves are curvature continuous to each other, the radius of the two curves at the junction is equal. This is a smoother connection than G1 as there is no sudden jump in the radius between the two curves

The angle and length of the curvature combs are equal at the junction.

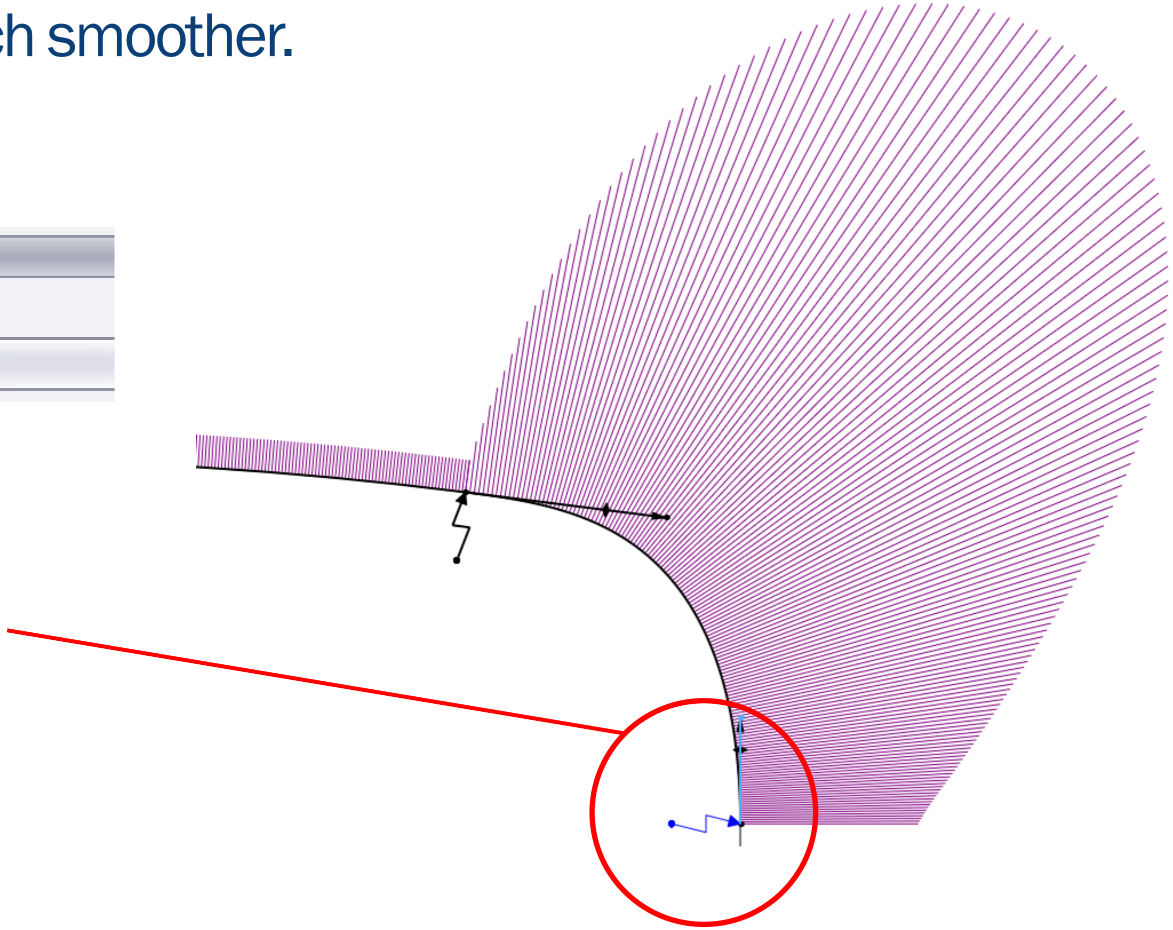


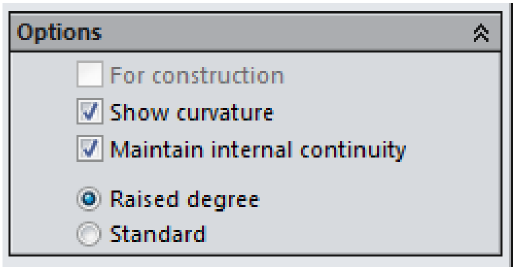
Better Curvature Continuous Connections

G2 connections can be improved compared to just adding the equal curvature relation. By adding a curvature control "widget" to an interpolated spline and enable the raised degree option the transition is much smoother.



The curvature control widget is added to the spline.



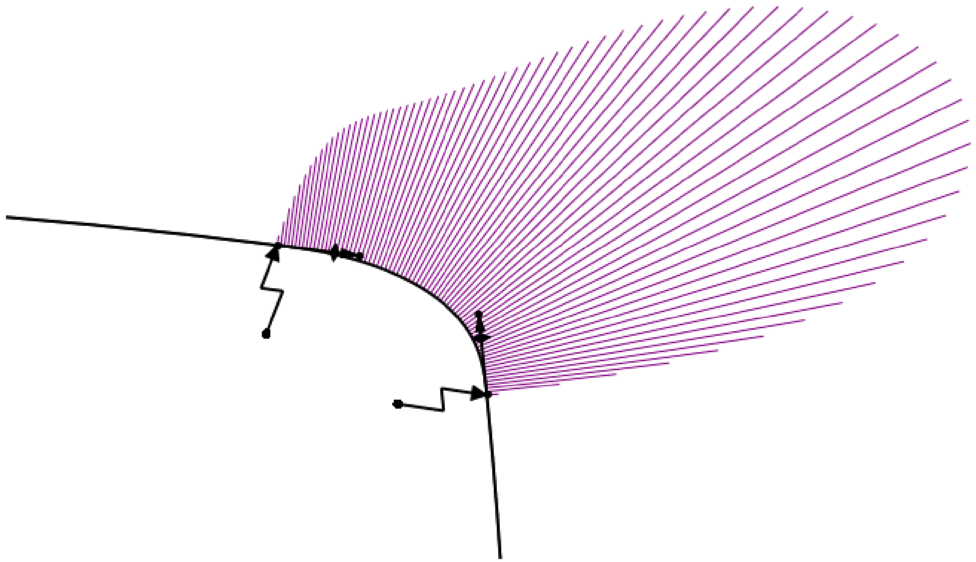


The curve is changed to a raised degree curve.

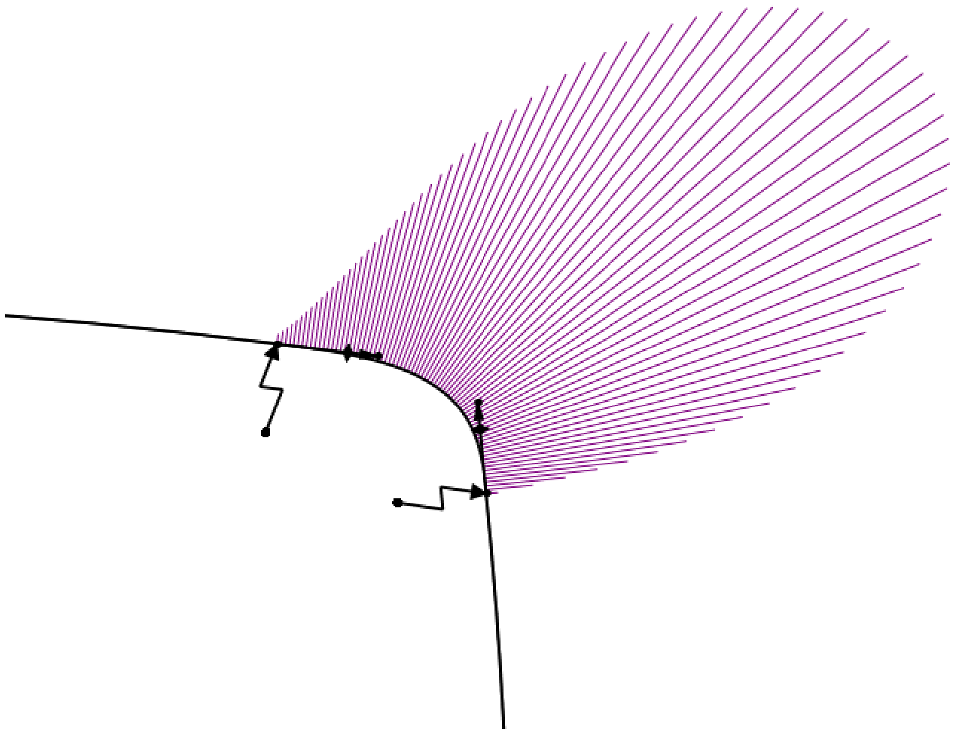
NOTE: Splines with the equal curvature relation applied to both ends are Raised Degree by default.

Better Curvature Continuous Connections

By adjusting the tangent handles from the default positions, the shape of the new spline is greatly improved.



The default handle lengths usually do not produce the smoothest curve.

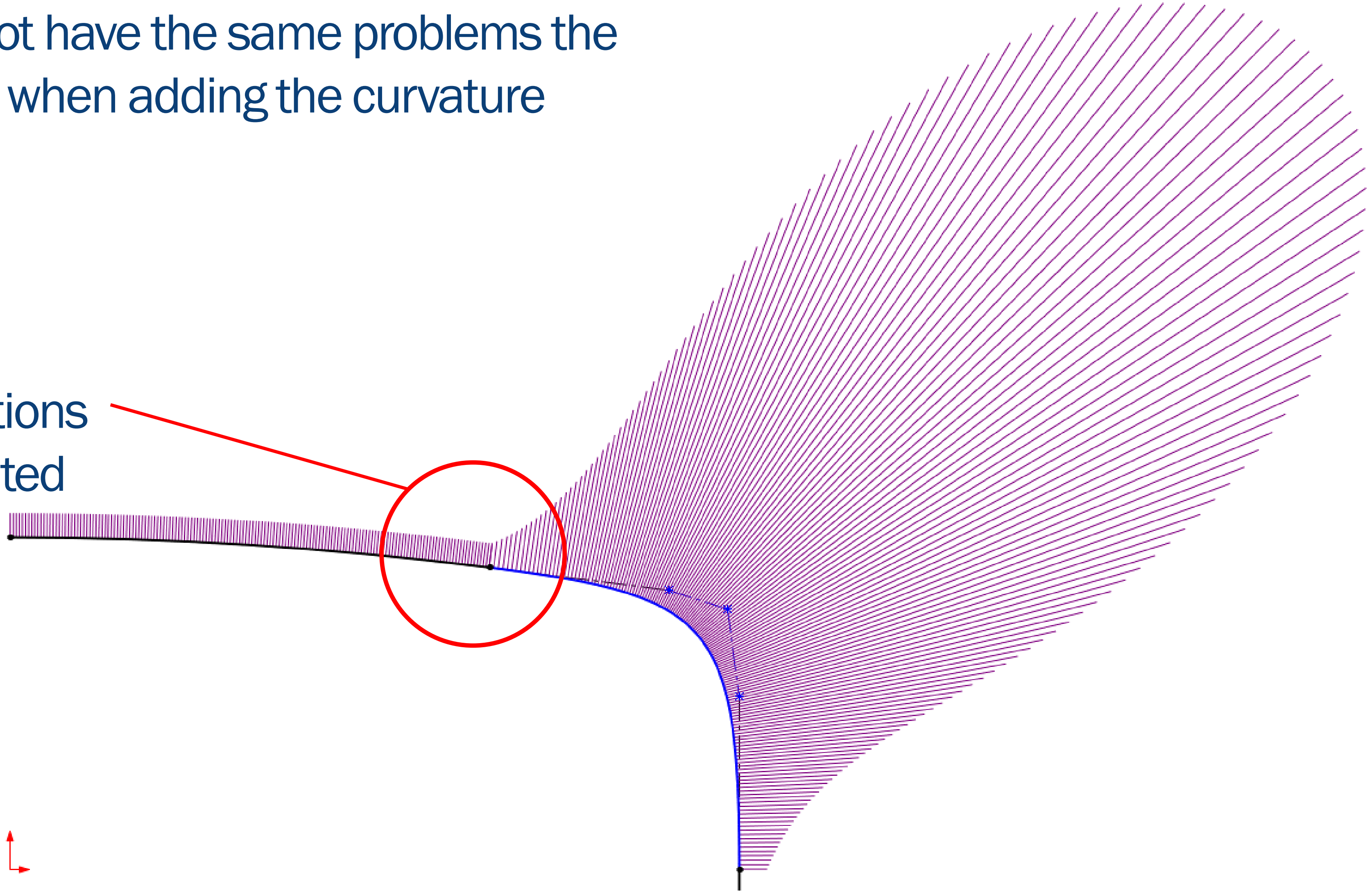


Minor tweaks to the handles greatly improve the splines shape.

Better Curvature Continuous Connections

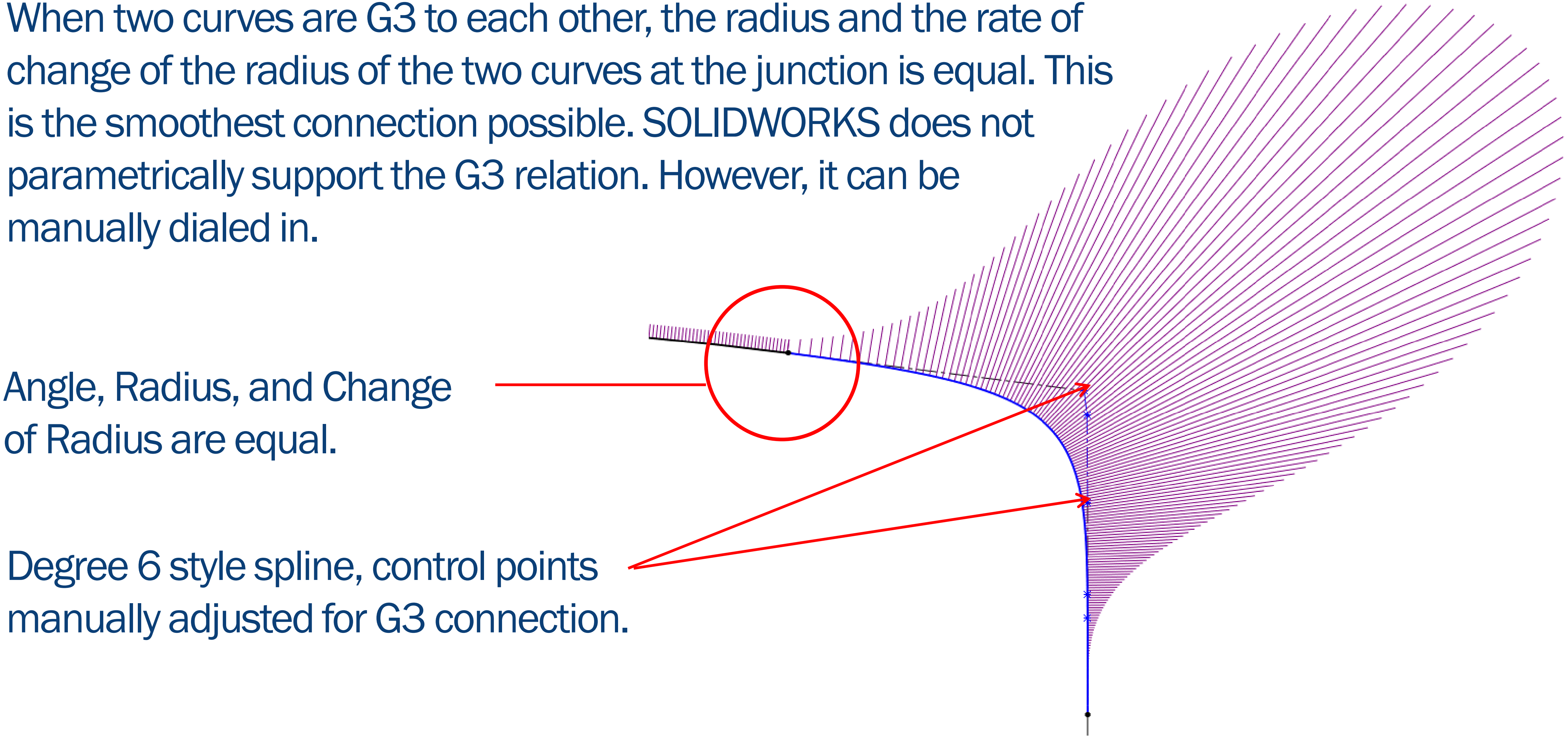
The style spline does not have the same problems the interpolated spline has when adding the curvature continuous relation.

The style spline handles equal curvature connections better than the interpolated spline.



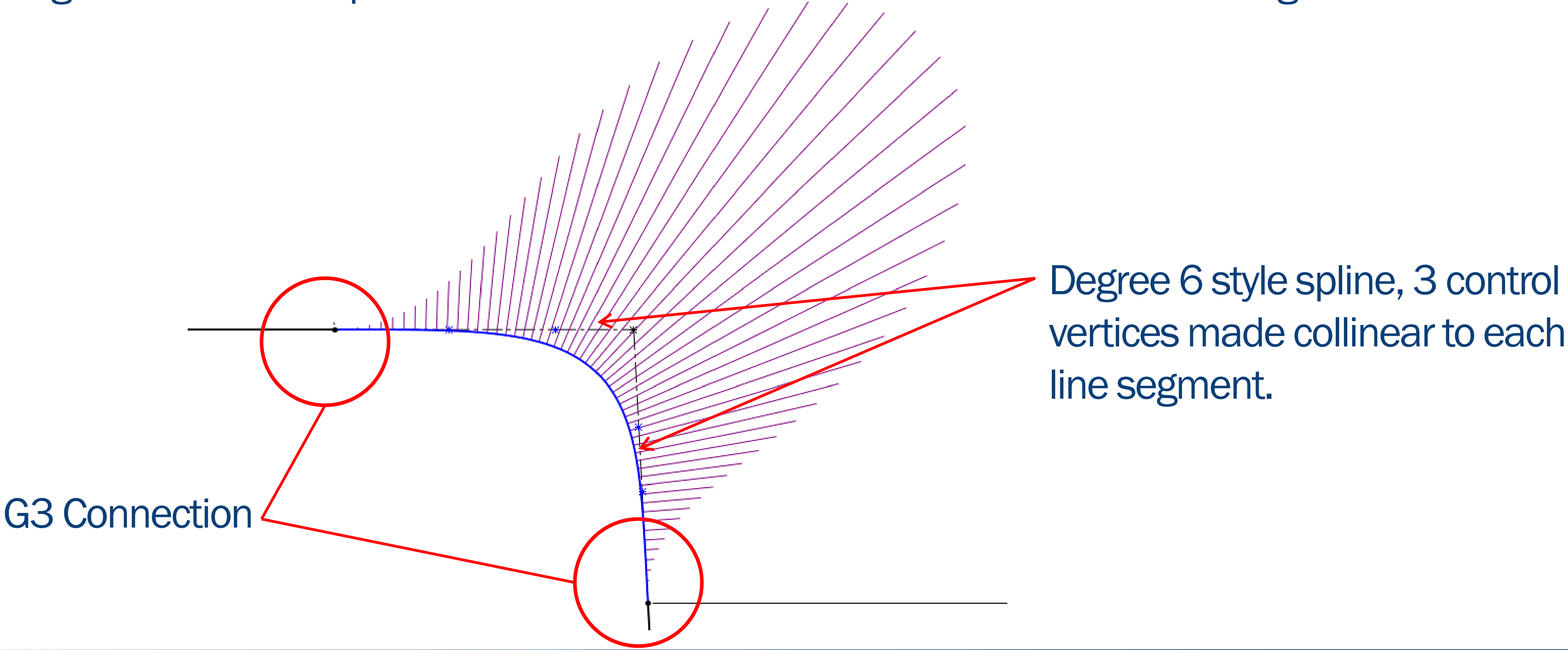
G3 Continuous Connections

When two curves are G3 to each other, the radius and the rate of change of the radius of the two curves at the junction is equal. This is the smoothest connection possible. SOLIDWORKS does not parametrically support the G3 relation. However, it can be manually dialed in.



Parametric G3 Connections

SOLIDWORKS supports G3 connections to straight line segments with the new Style Spline. A degree 6 curve is required. 3 control vertices are made collinear to each straight line.



SURFACES

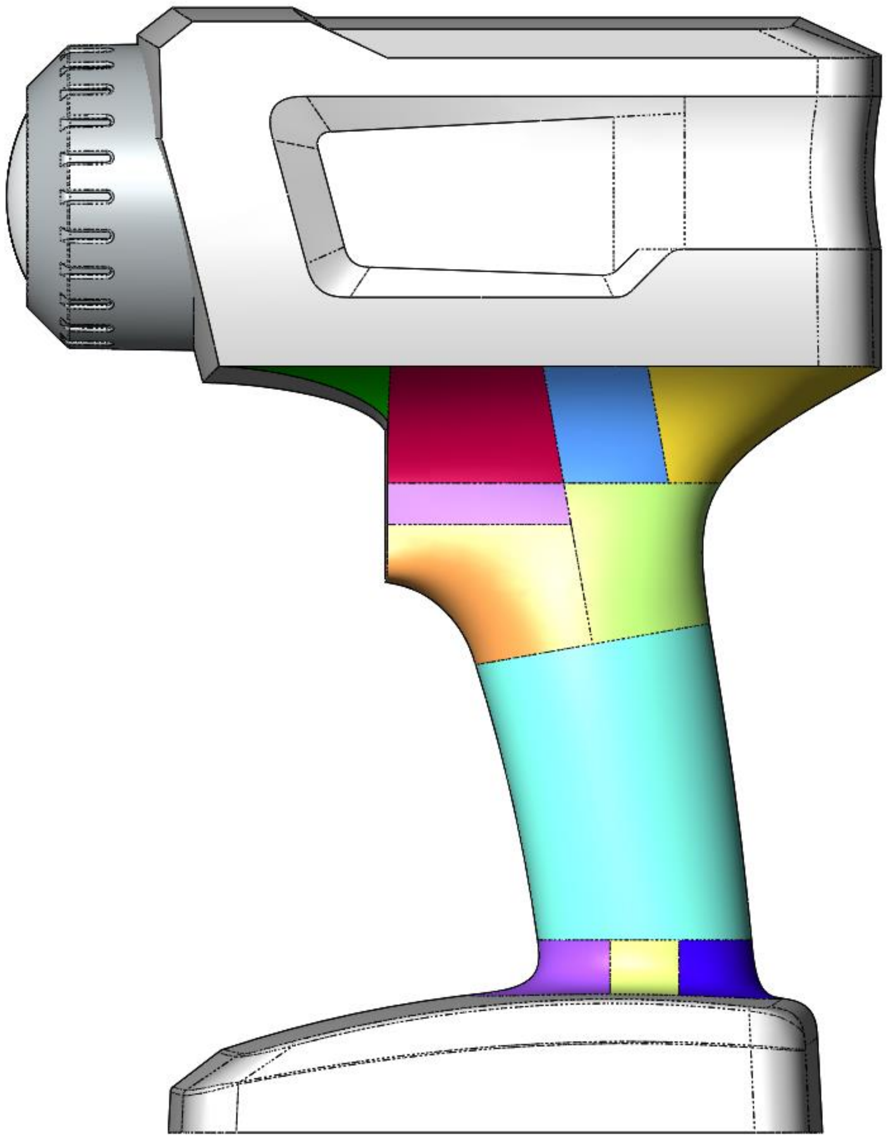
Patch Layout

Before starting to model, layout the surfaces you'll use to construct the shape. The strategy you use will dictate where planes, sketches and projected curves/ 3D sketches will be positioned.

It is ALWAYS better to break patches into smaller surfaces. Don't try to do too much with one surface.

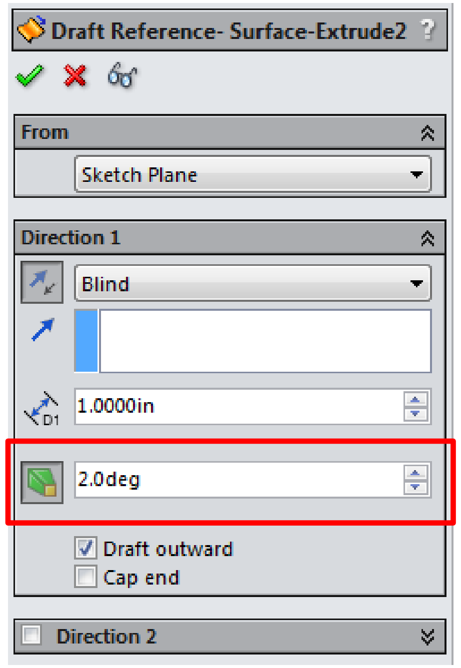
Break the shape into as many 4 sided patches as required. These will become Boundary Surfaces.

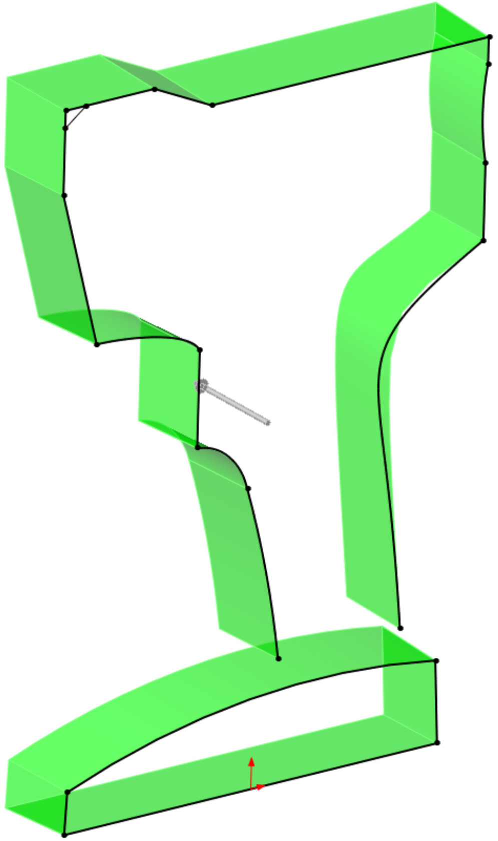
The remaining N-sided patches will be completed with Surface Fills.



Surface Extrude

Surface-Extrude works the same as solid extrudes. They are critical for creating reference geometry that will be used to define the shape of more complicated surfaces.



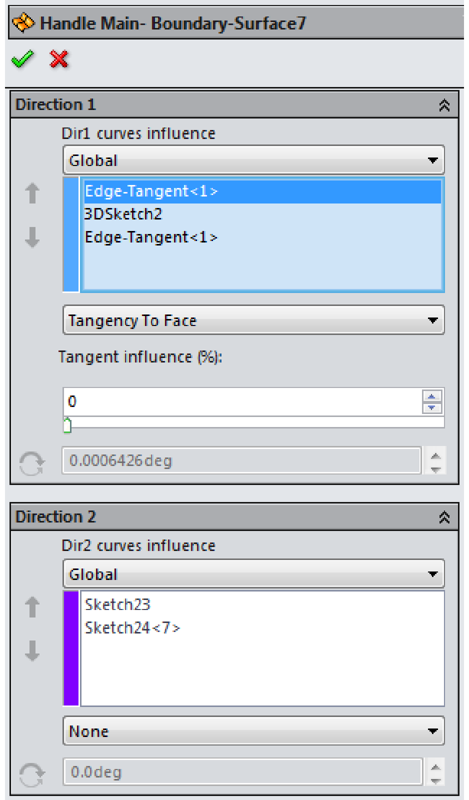


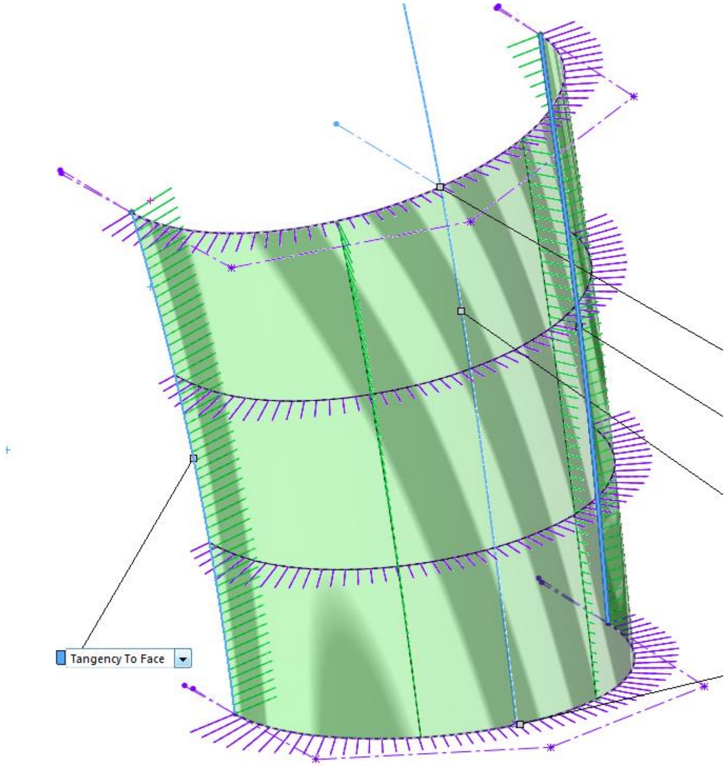
Reference surfaces have two major functions.

1. Creating draft reference surfaces. These ensure surface features have the correct draft at the parting line.
2. Creating tangency reference surfaces. These are used to ensure a flat surface along a mirror plane when creating symmetric parts.

Boundary Surface

The Boundary Surface is the workhorse of my surface modeling in SOLIDWORKS. It generates new surfaces from a selection of profiles in either one or two directions. It supports G1 & G2 continuity in both directions.





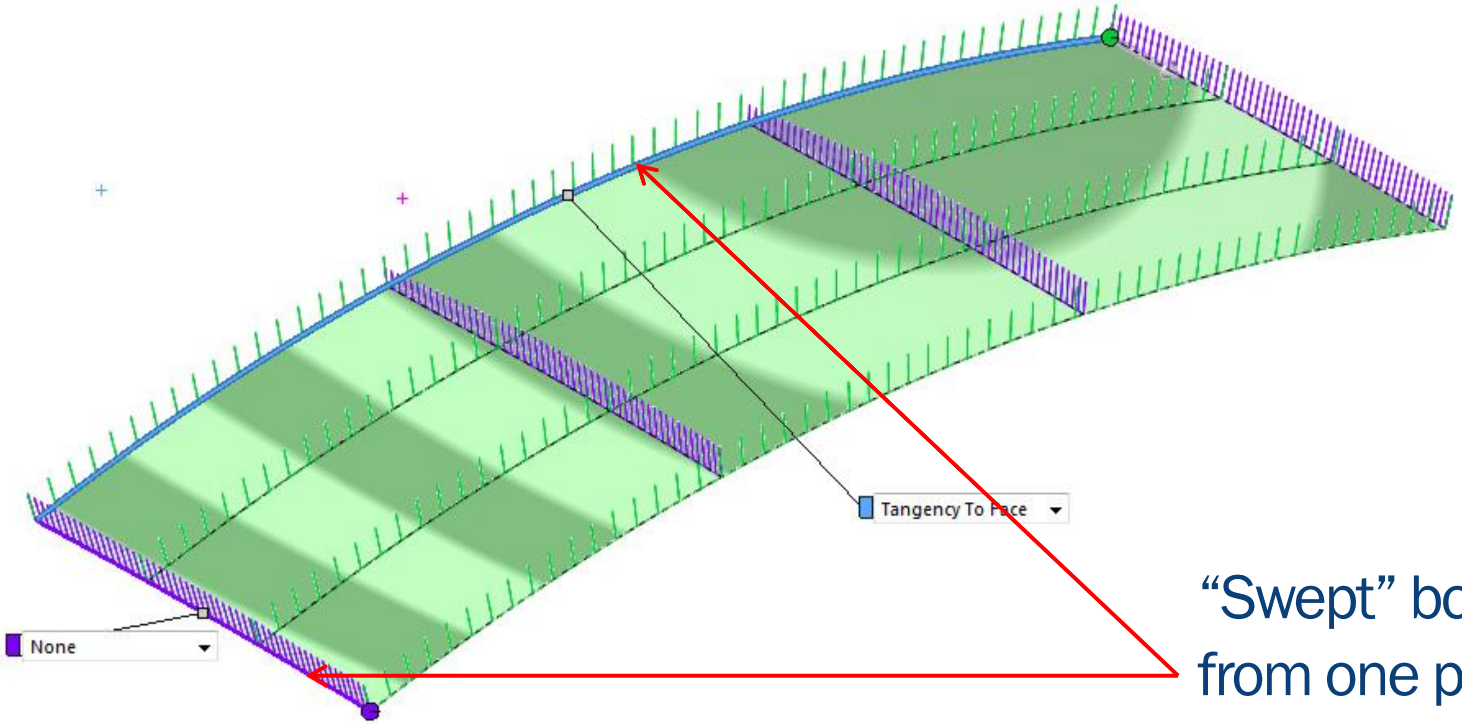
BEST PRACTICE:

Always create Boundary Surfaces that have 4 sides. 3-sided Boundary Surfaces introduce a “degenerate point” where the two edges of the surface converge in a single vertex.

Degenerate points cause issues with Shells, Fillets, Surface Offsets and can corrupt downstream operations.

Boundary Surface

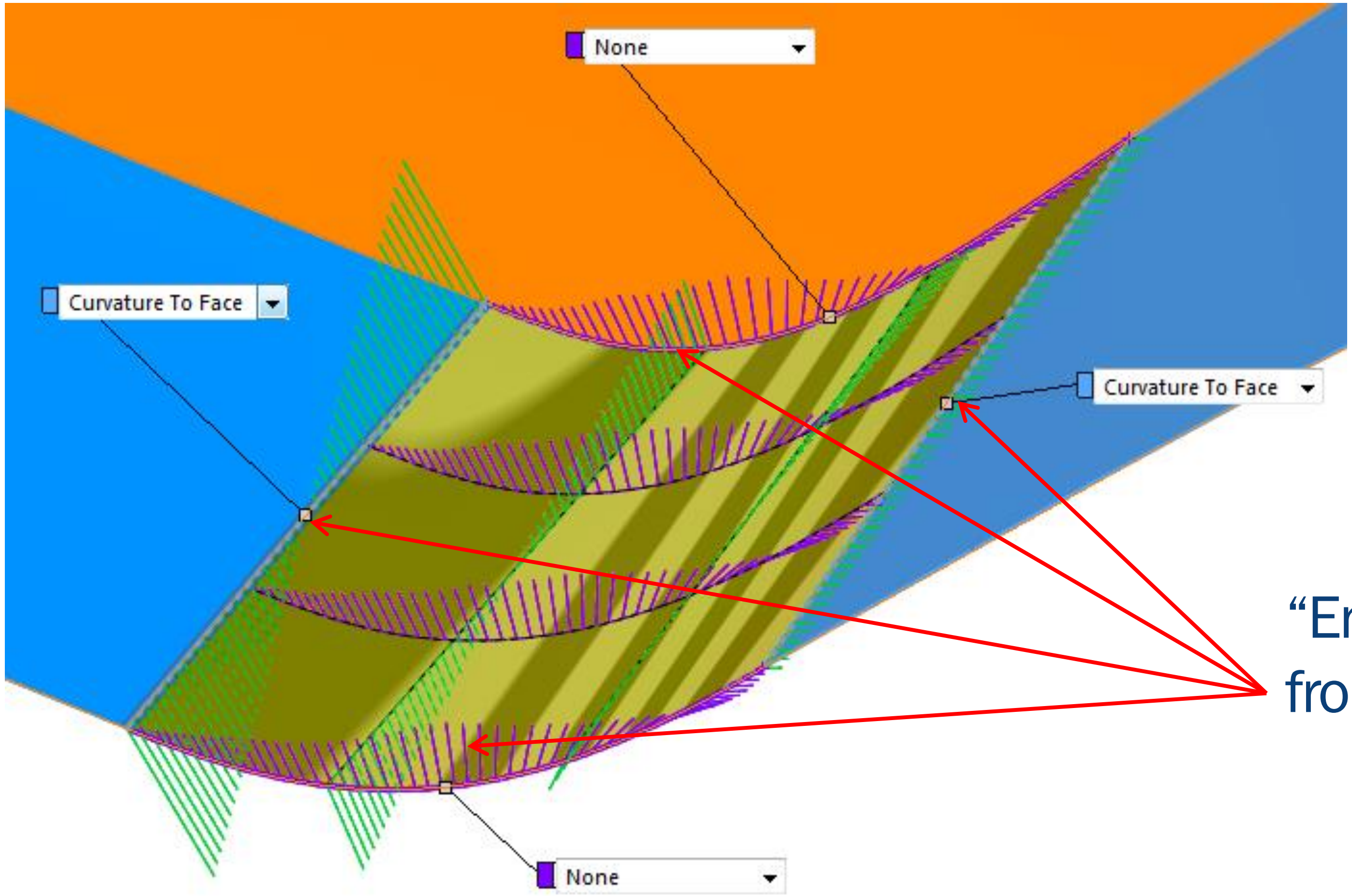
The Boundary Surface can create surfaces from a number of profiles.



"Swept" boundary surface created from one profile in each direction.

Boundary Surface

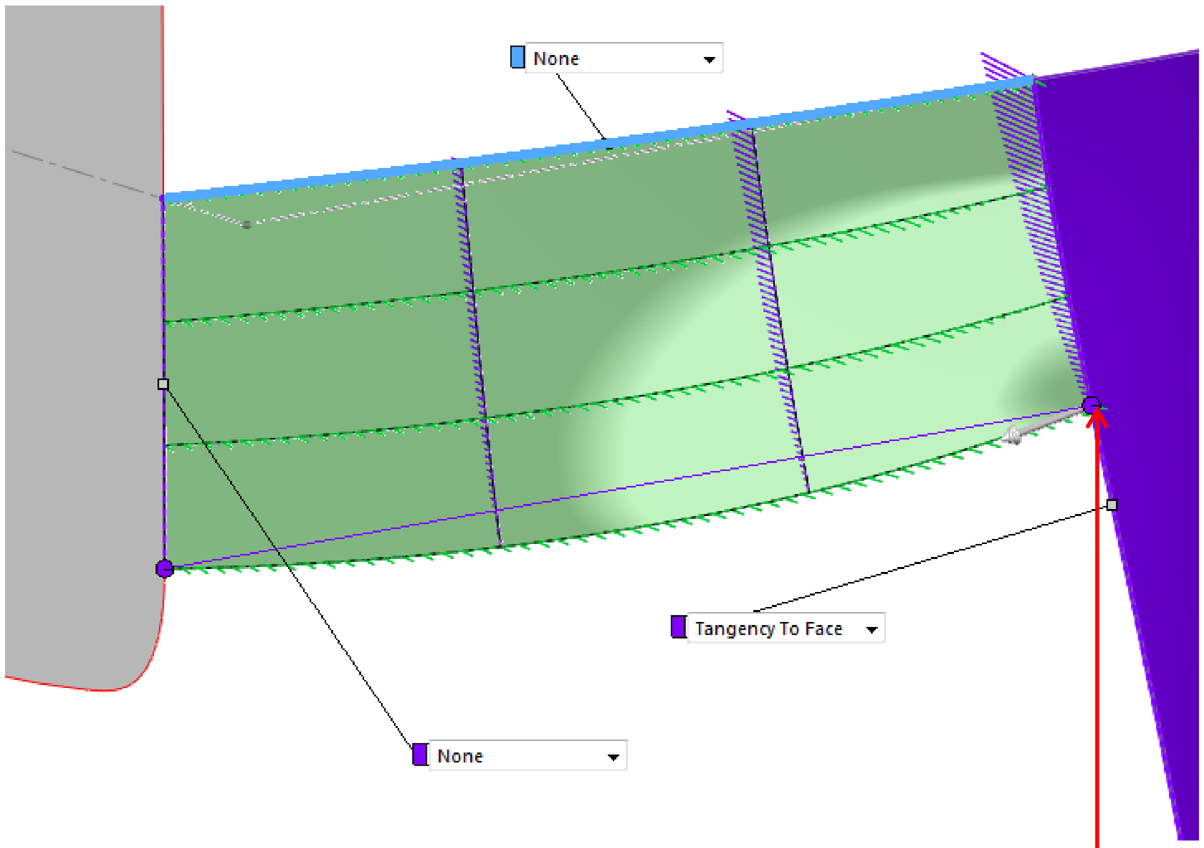
The Boundary Surface can create surfaces from a number of profiles.



"Enclosed" Boundary Surface created from a 4 sided perimeter.

Boundary Surface

The Boundary Surface can create surfaces from a number of profiles.



Connector has been dragged back

"Short edge" Boundary Surface where a connector has been dragged back forming new, partial edge.

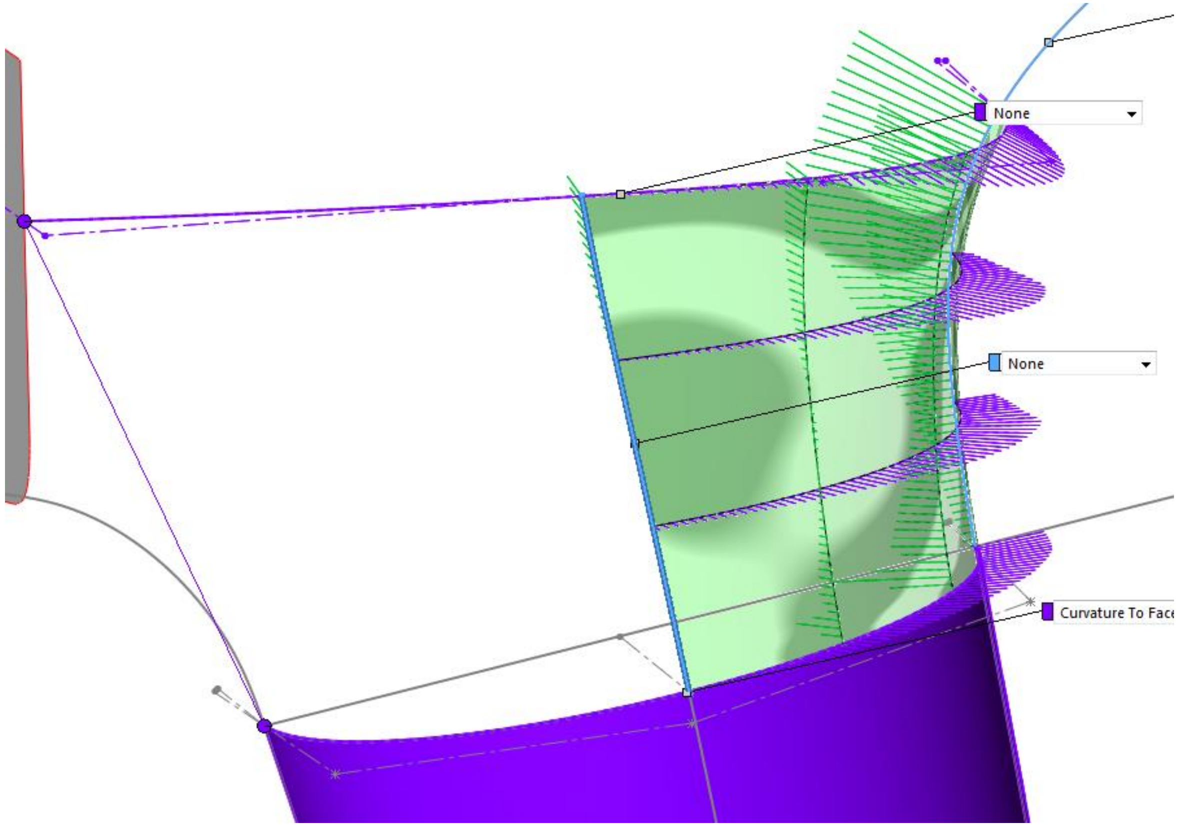
The ability to adjust the start and end of a boundary profile is one its of greatest strengths.

NOTE:

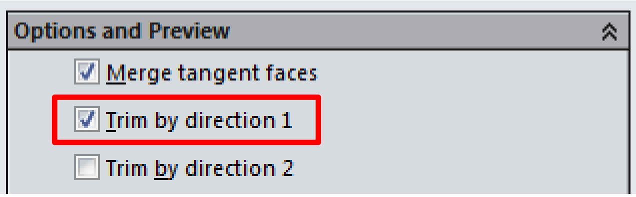
Connector edits are NOT parametric and may revert to original positions on major design changes.

Boundary Surface

The Boundary Surface can create surfaces from a number of profiles.



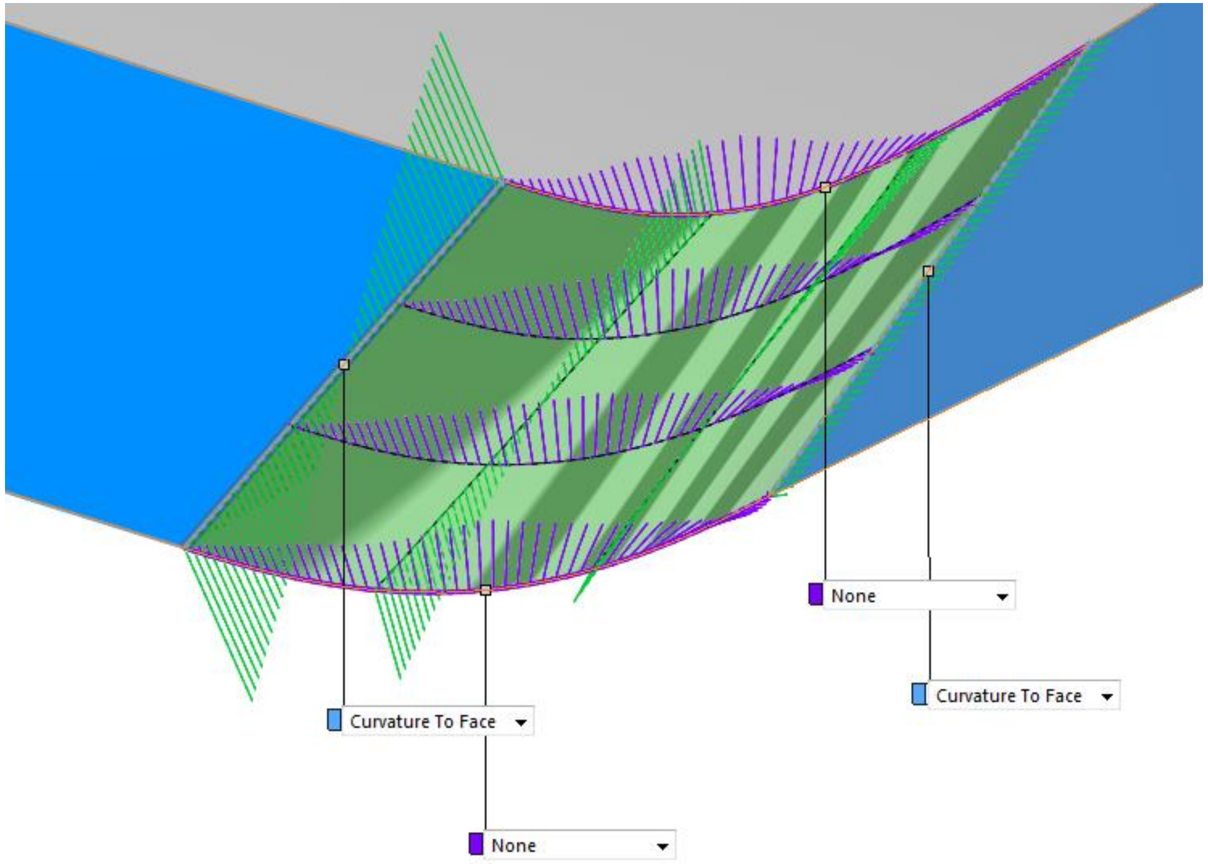
“Trimmed” Boundary Surface where the profiles in one direction extend past the profiles used. The Boundary surface is then trimmed back.



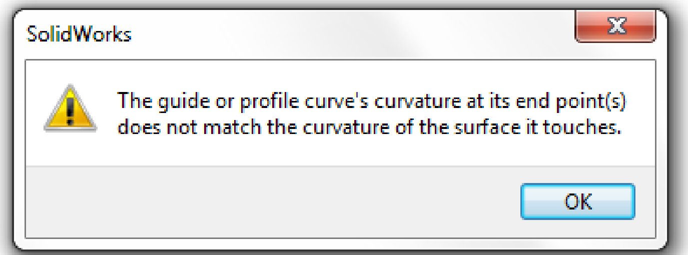
Direction 2 (PURPLE) extends past the profiles in Direction 1 (BLUE). Enabling “Trim by direction 1” allows the larger profile to be used.

Boundary Surface

The Boundary Surface supports tangent and curvature continuity in both directions.



Curvature continuity has been applied to the profiles in direction 1. This ensures the resultant surface is G2 to the adjacent faces.

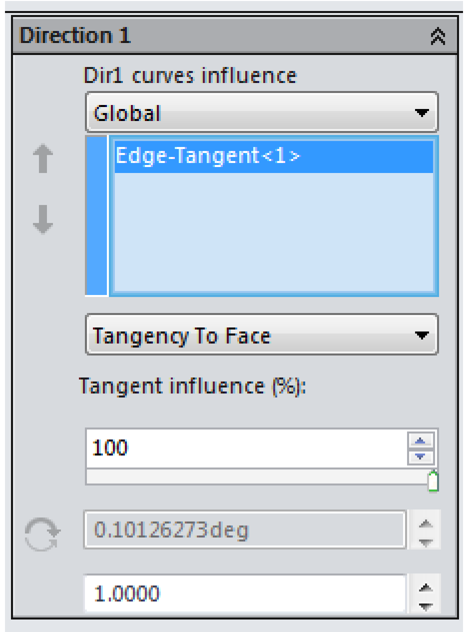


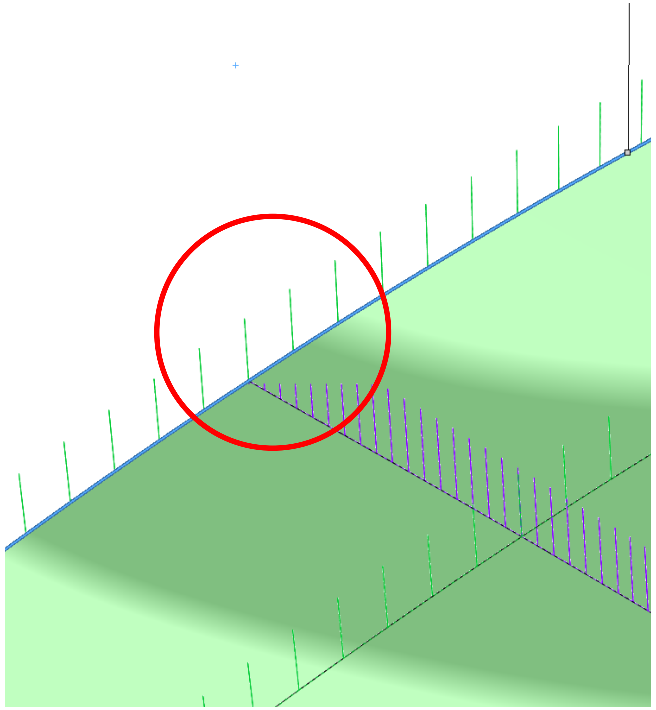
NOTE:

This error message frequently pops up when adding the curvature relation. It is caused by slight mismatches in the adjacent faces, normally due to knitting tolerances. It can be ignored.

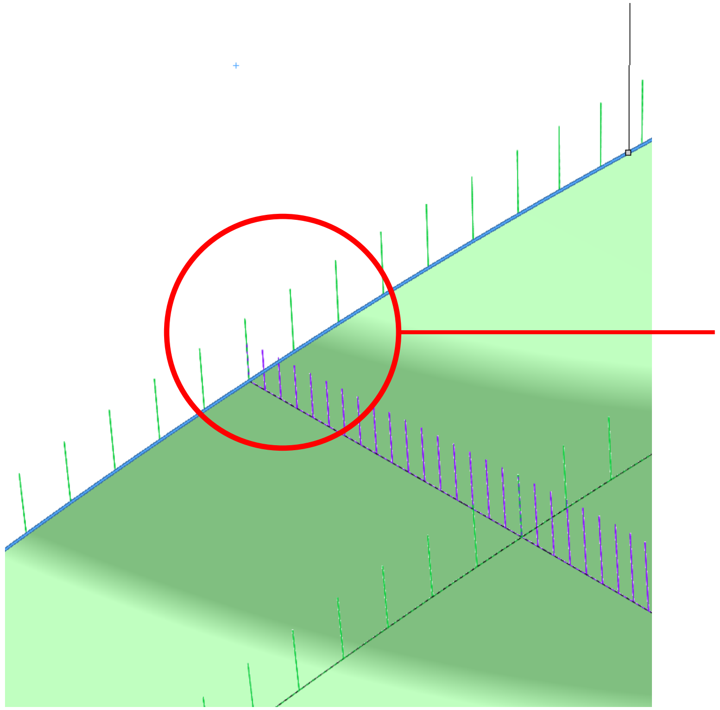
Boundary Surface

The tangent influence slider is used to allow one profile to affect the shape of the boundary more than other profiles. It can be used to help “inflate” the shape of the surface.





Tangent Influence 0%

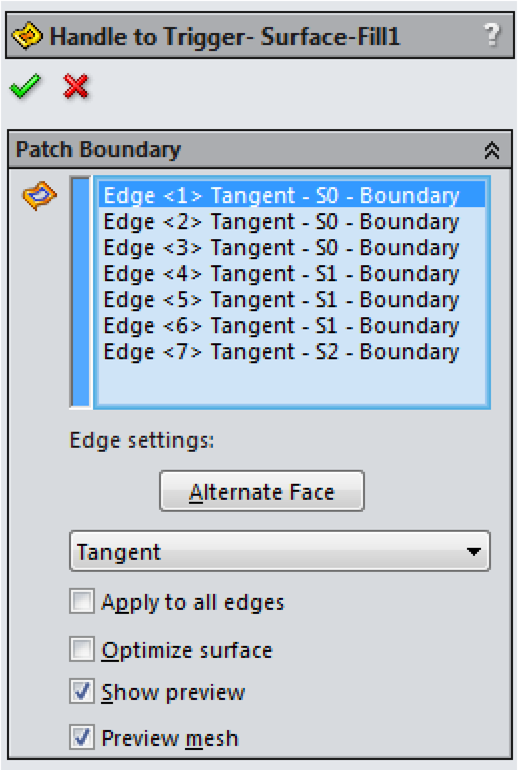


Tangent Influence 100%

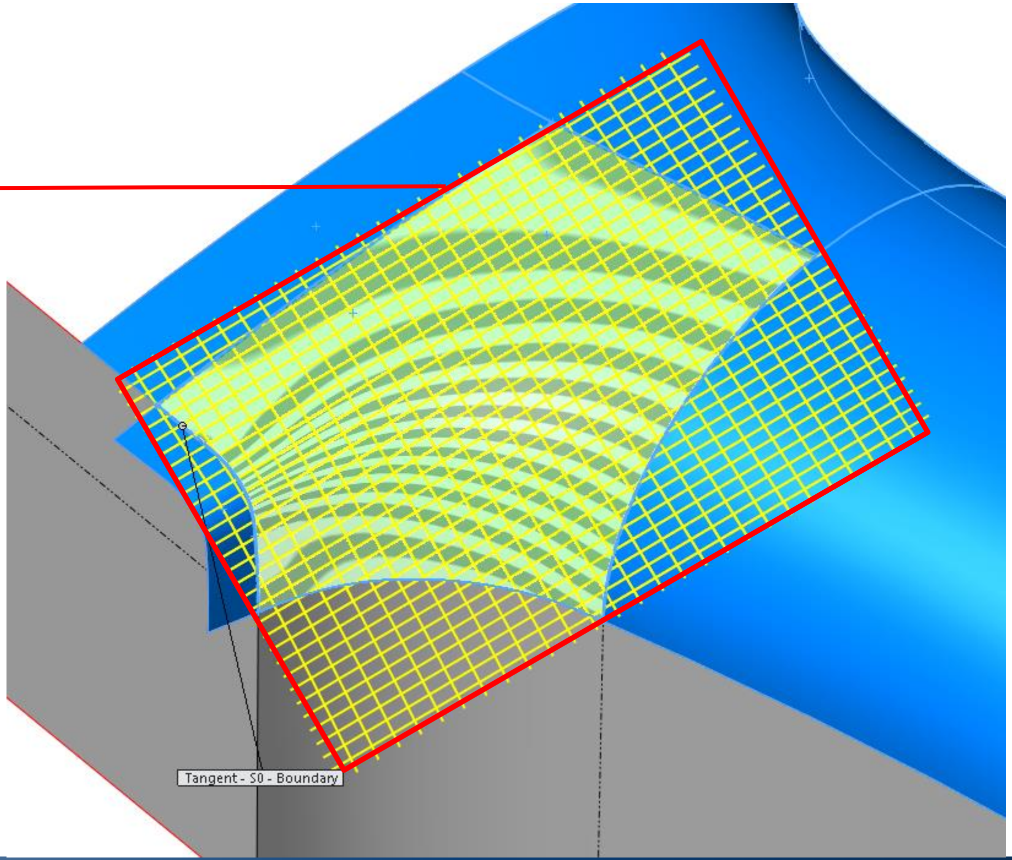
100% tangent influence corrects dip in curvature.

Surface Fill

Surface Fill is one of the most powerful features in SOLIDWORKS. Surface Fill has the ability to create a patch within N number of surface edges. Tangent and curvature relations can also be added to the edges.



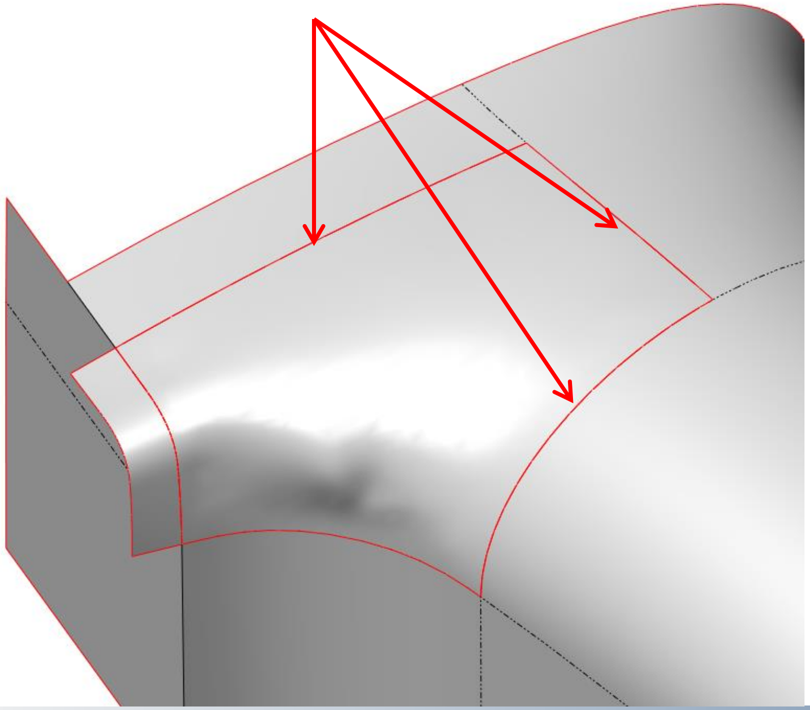
Surface Fill generates a 4-sided surface and trims it to fit within the patch boundary. This gives it the power to patch N sided boundaries.



Surface Fill

While it may make sense to use the Curvature relation on all required edges, it can actually produce detrimental results.

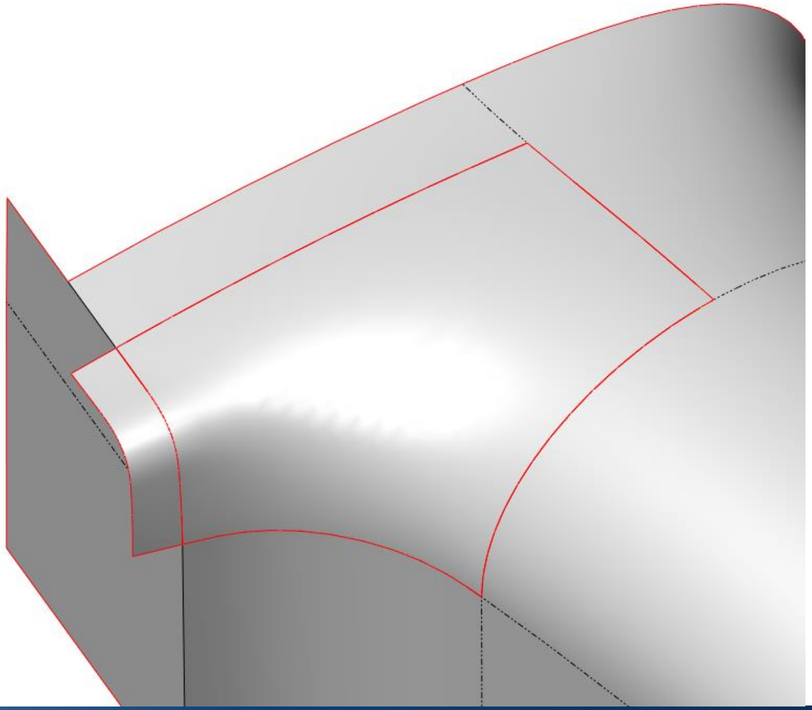
Curvature on these edges,
Tangency on remaining edges



Best Practice:

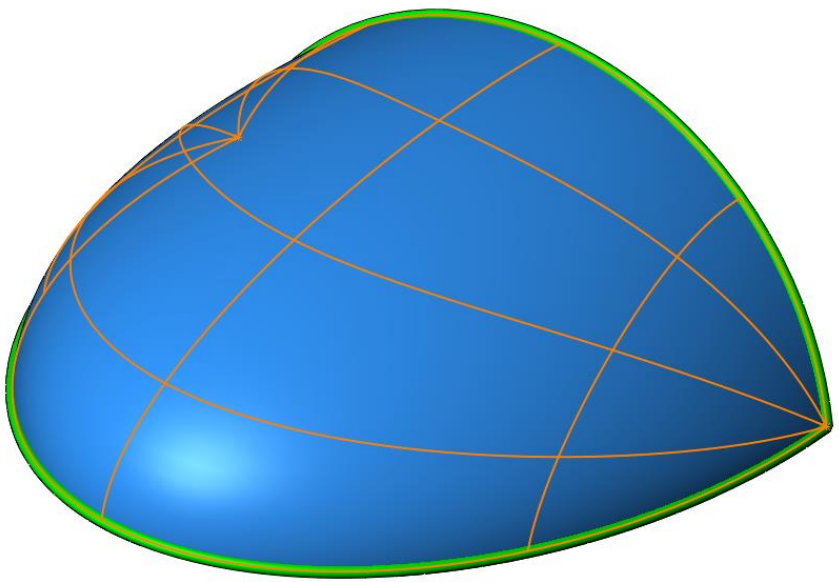
Apply tangency to all required edges.
Evaluate the surface.
Add curvature only as required.

Tangency on all edges

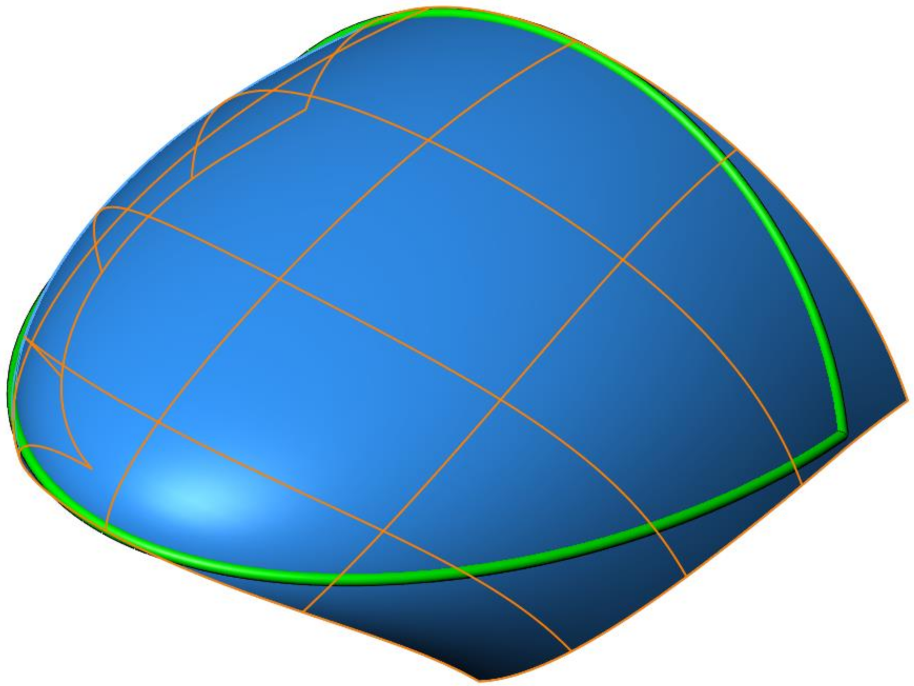


Boundary Surface vs Surface Fill

Imagine the outline of the patch as a frame with a rectangular tarp stretched over. The tarp can be any shape and size, but always starts with 4 sides.



Trying to fit the tarp over this shape results in bunched up corners.

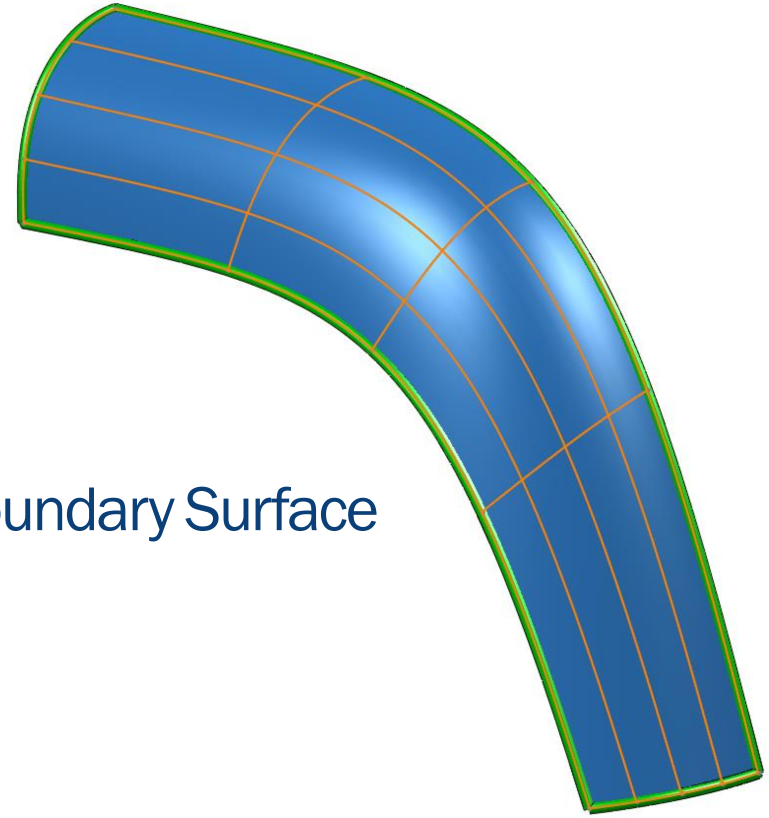


Draping the tarp over the frame and then trimming prevents the corners from bunching.

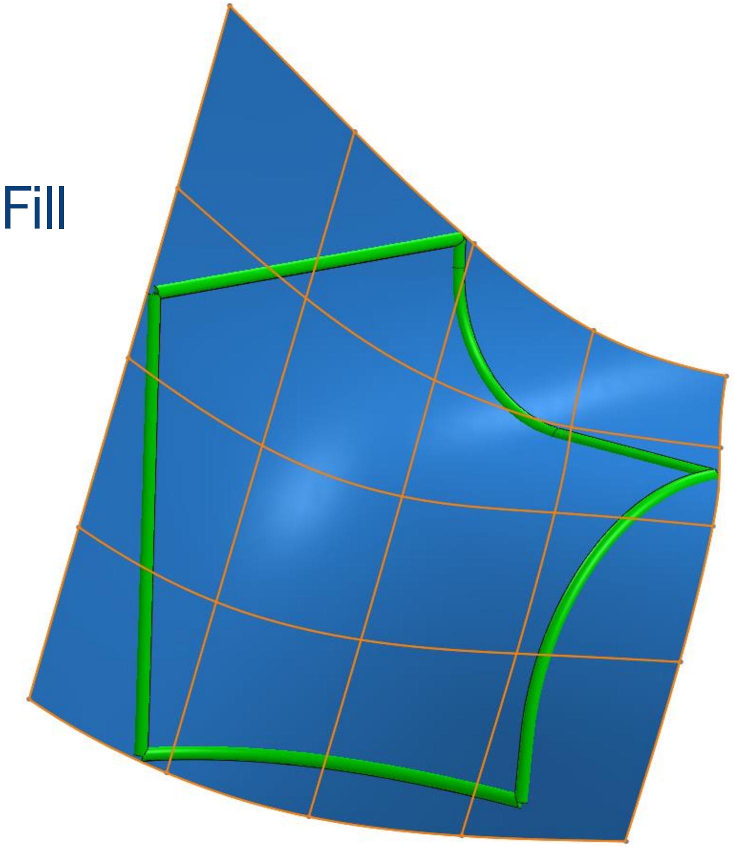
Boundary Surface vs Surface Fill

When the surface to be created is 4 sided, use Boundary Surface for more control.

When the surface to be created is 2,3 or 5+ sided, use Surface Fill due to its ability to trim back.

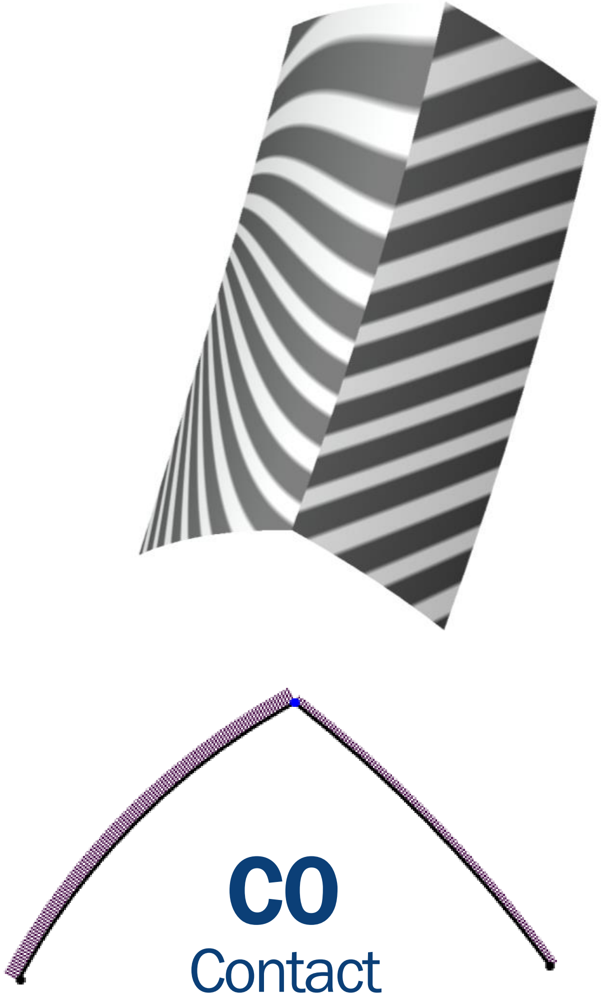


Boundary Surface



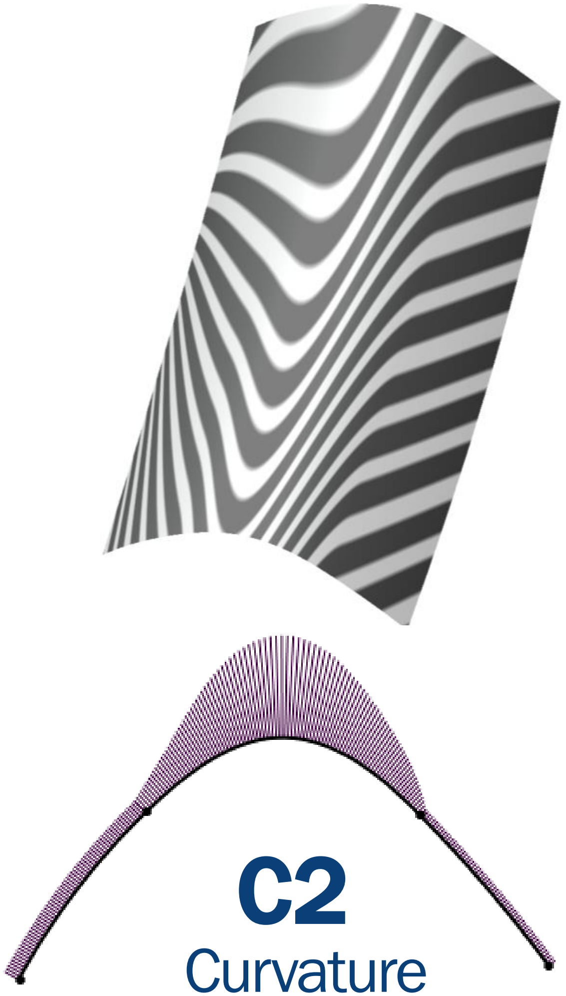
Surface Fill

Surface Analysis | Zebra Stripes



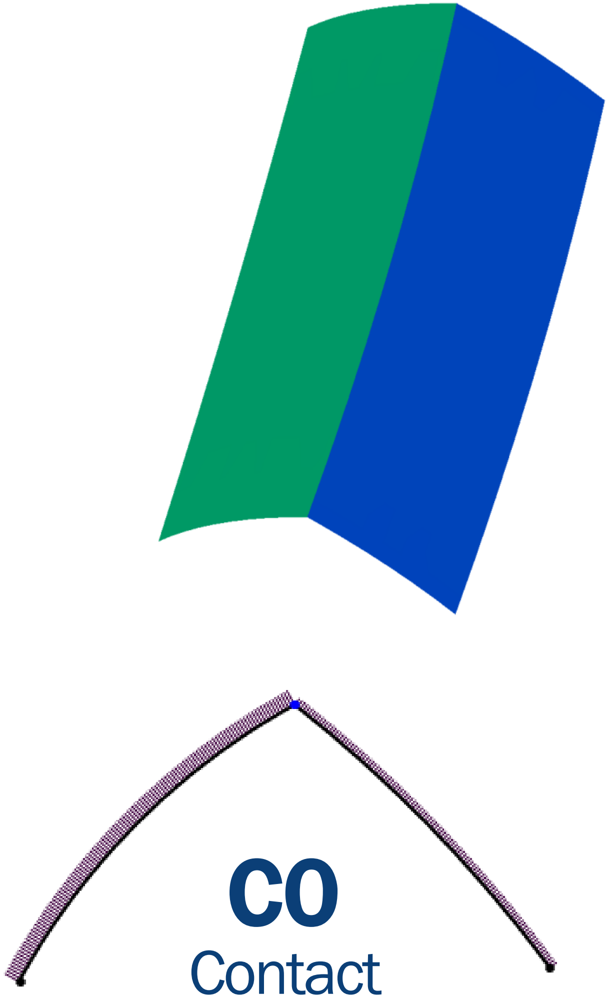
C0
Contact

C1
Tangent

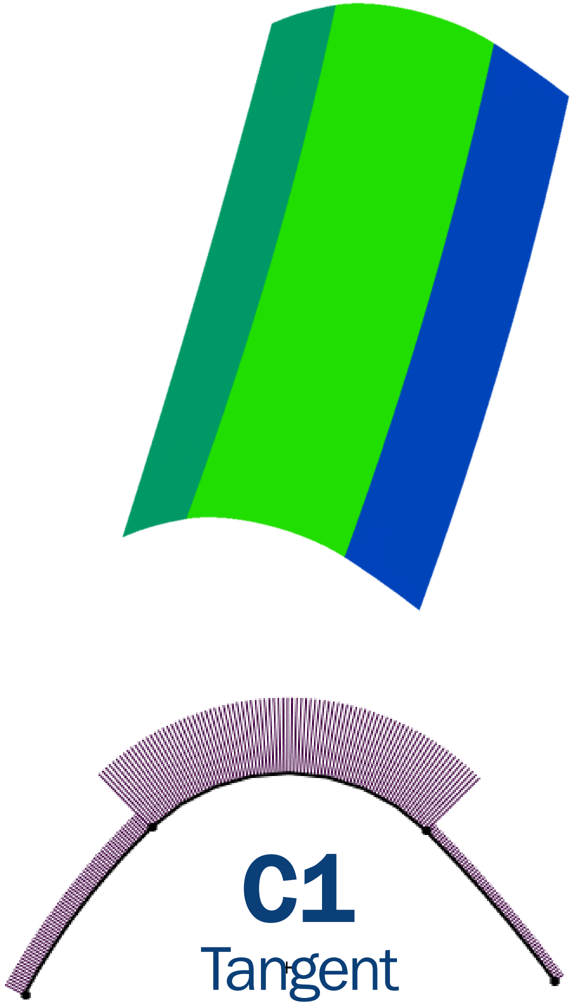


C2
Curvature

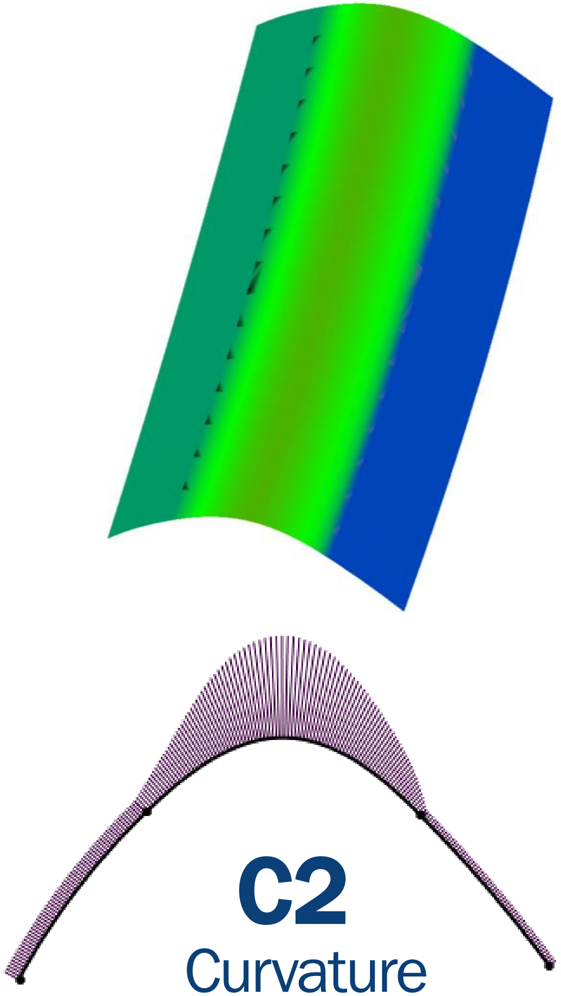
Surface Analysis | Curvature Display



C0
Contact



C1
Tangent



C2
Curvature

LET'S MODEL!

Develop a Modeling Strategy

Top

Extruded mass, filleted corners, surfaced chamfers.

Indent

New “diving” surface, surfaced fading chamfer.

Handle

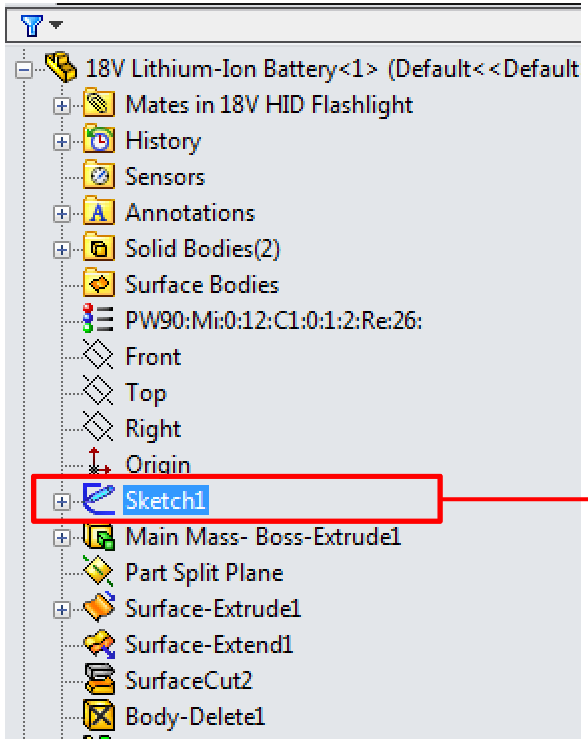
Patchwork of surface features.

Base

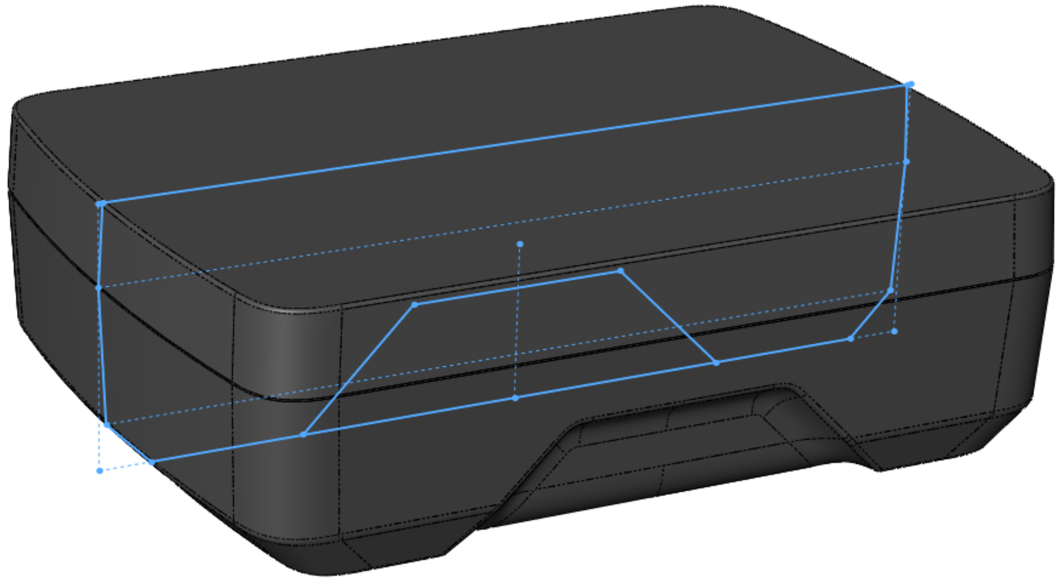
Extruded mass, filleted corners, surfaced top face, drafted chamfer, mounts to existing battery.

Begin With What You Know

This work flashlight uses an existing 18V battery that is already in production. Because the part won't change, we don't need to build any in-context relations to the battery. But its profile should still be captured.

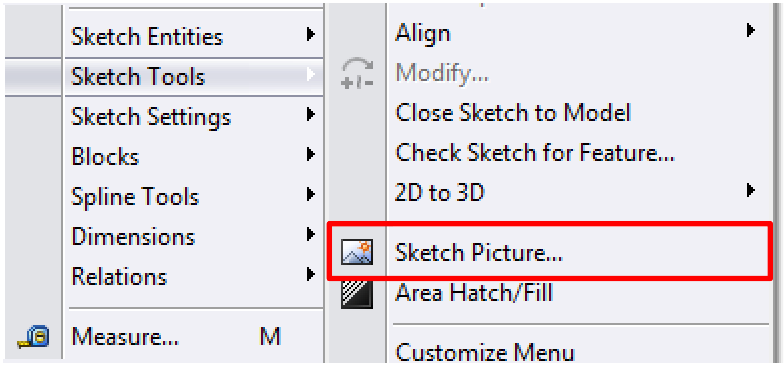


Sketches can be copied from one part another with Ctrl-V and Ctrl-C. The intended plane needs to be selected in the recipient part.



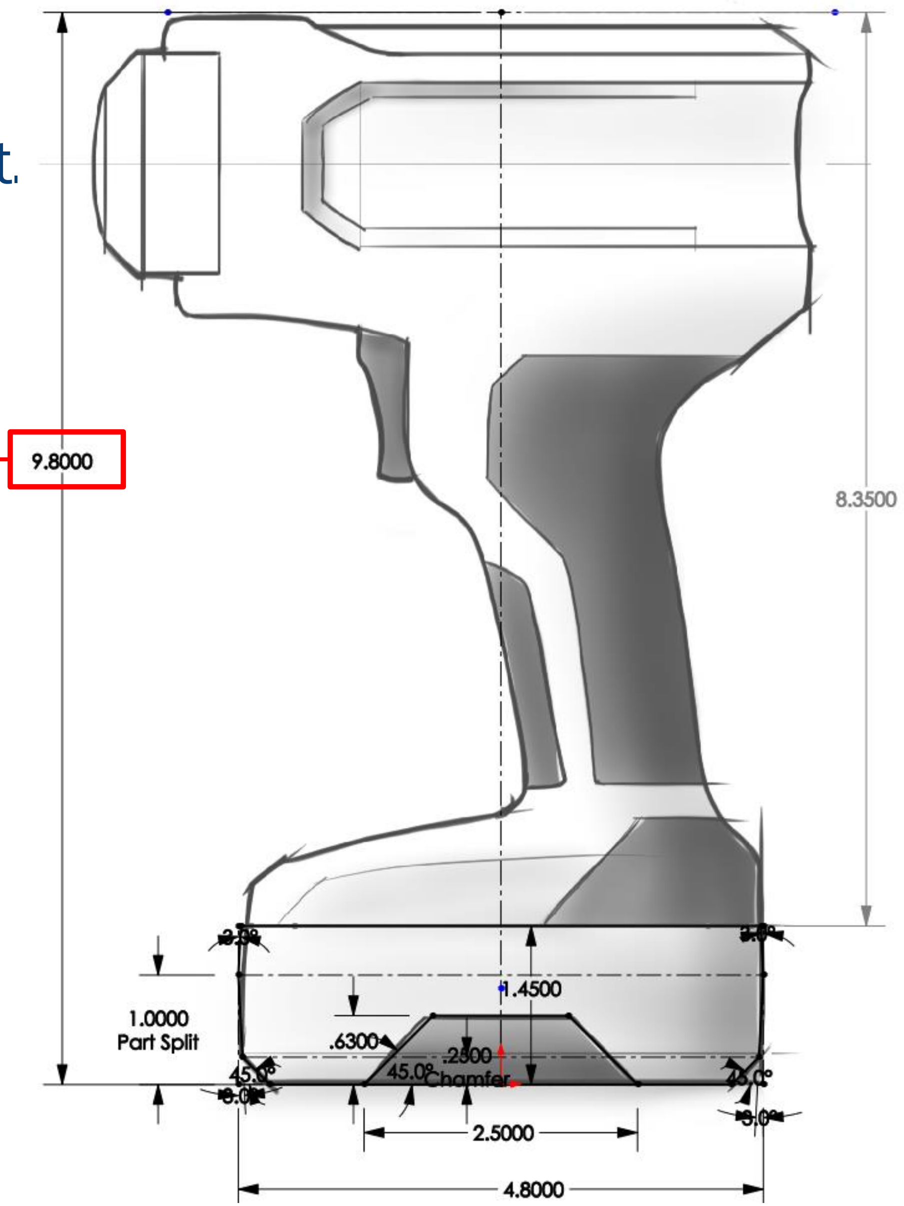
Capture the Industrial Design Intent

An Industrial Designer sketched out a concept for this flashlight. Our job is to now replicate the design in SOLIDWORKS, complete with all details required for manufacturing.



Tools>Sketch Tools> Sketch Picture

Create a new sketch on the right plane. Insert a sketch picture. Create a construction line with a dimension and scale the sketch picture.



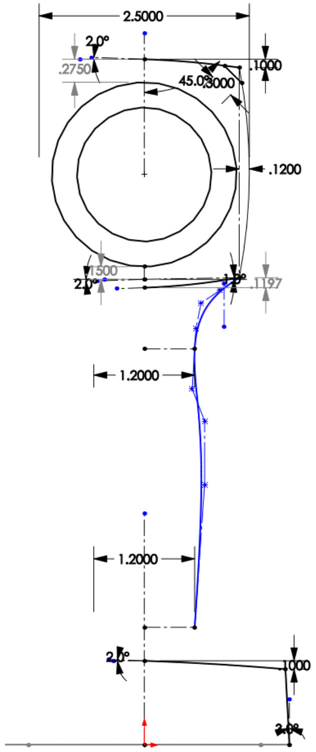
Create Layout Sketches

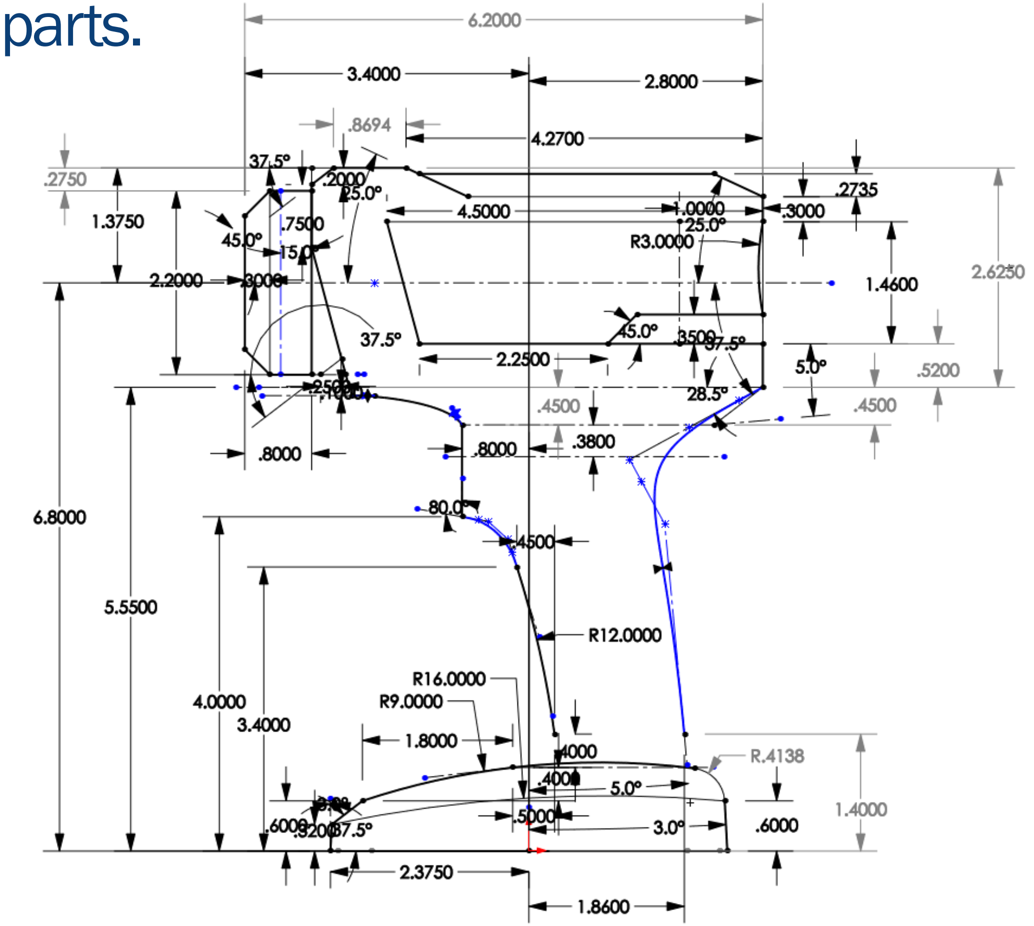
Layout sketches are used to capture all of the critical dimensions of the part. Building intelligence into sketches at this stage will save time when design changes are required. Creating layout sketches can take several hours on some parts.

BEST PRACTICE:

Never directly reference any entity in the layout sketch to prevent it from being absorbed into new features.

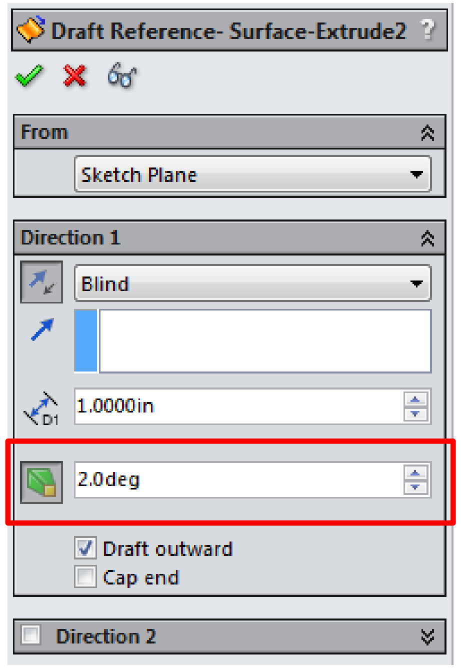
Convert entities from the layout sketch into new sketches and use these to build features.





Define the Parting Line

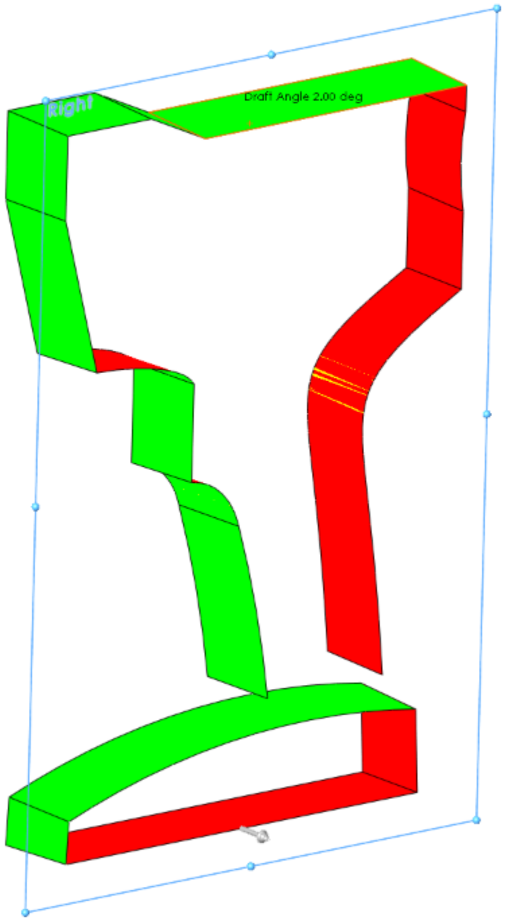
The housing of the flash light will be injection molded. When modeling any part that requires draft, the first step is to define the parting line. It is difficult to add draft to surface features after the fact; it must be modeled in from the start.



A surface extrude feature with draft is used to create a draft reference surface. The sketch is converted from the layout. Subsequent surface features will use this reference to ensure correct draft.

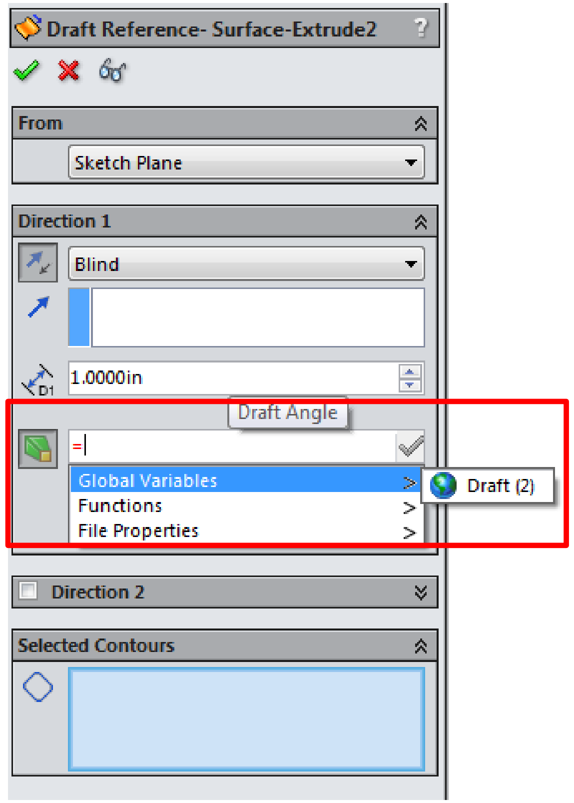
TIP:

If the parting line is non-planar, use a 3D sketch with the pull direction plane as an extrude direction.



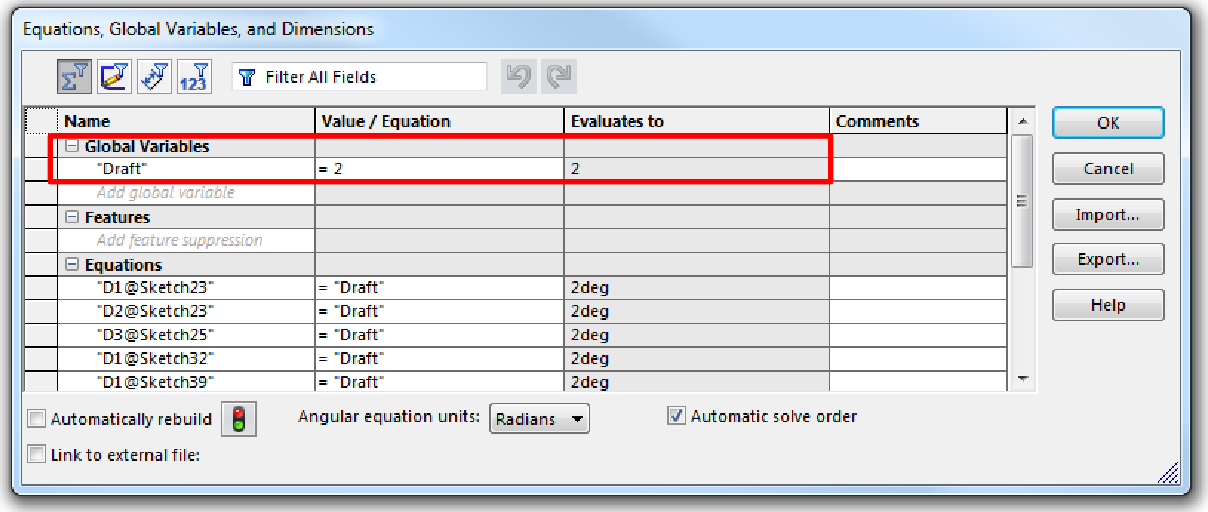
Create a Draft Angle Global Variable

The part requires draft for injection molding. The drat angle will frequently be referenced throughout the part. Create a global variable defining the draft angle and use this whenever a draft dimension is required.



Tip:

As of SW2013, equations and variables can be entered directly into the property manager.

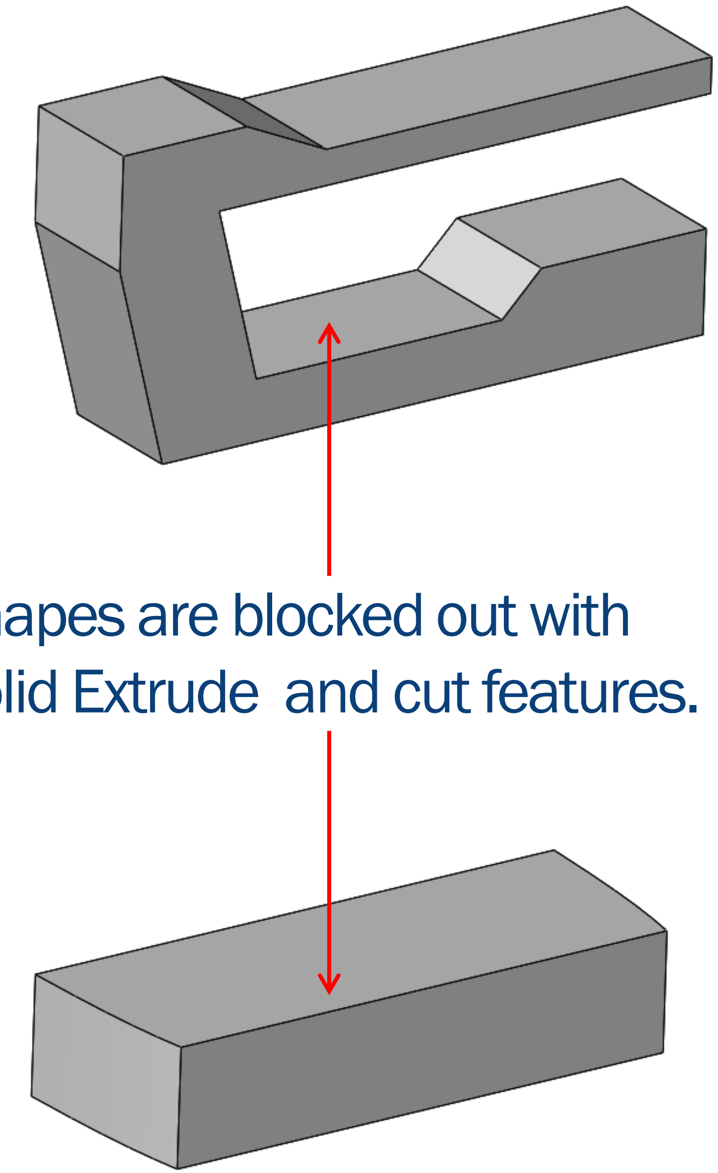


Use Solid Features to Start the Shape

Consider the faces required in the design. Is the form relatively blocky or round? If the base shape can be defined with a solid extrude or revolve feature, use these tools to create initial geometry.

A solid feature generates a closed volume with more faces than a surface feature. Cuts remove material and leave a closed volume.

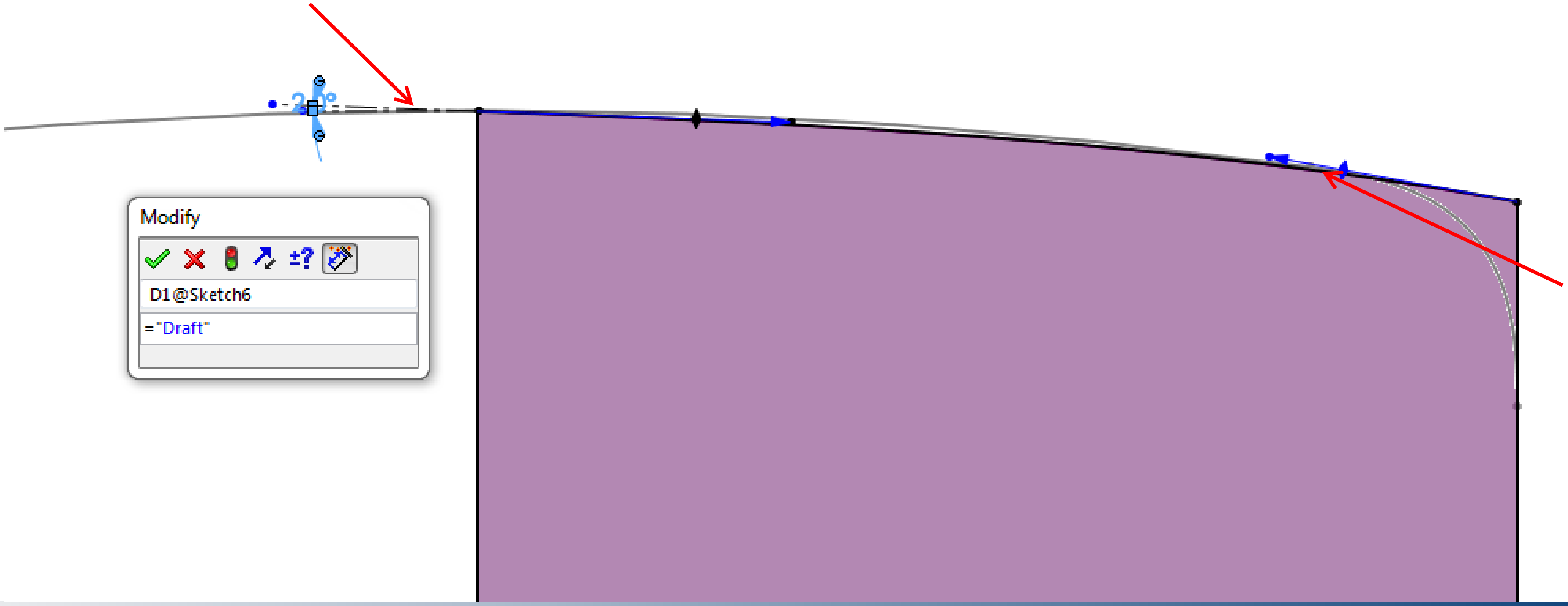
Use solid features whenever possible to sculpt the initial shape of the design. Additional techniques will be used to sculpt the shape into its final form.



Build Draft Into Features

The Draft feature creates ruled surfaces (which are flat along their length). It does not work on curved faces. Draft must be built into these features from the start.

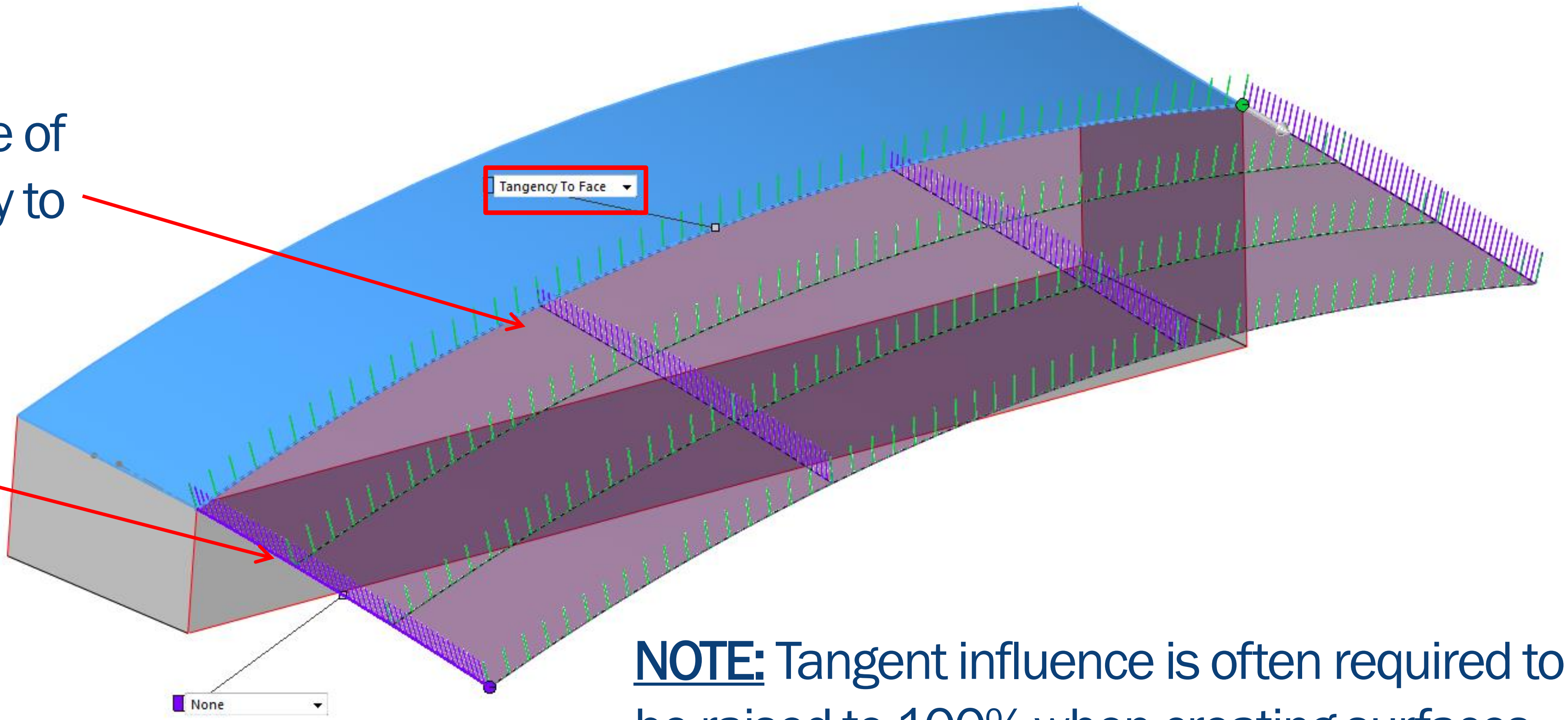
Construction line with angle made equal to draft global variable



Spline is shaped with handles to match arc of battery. A perfect match is not possible due to different pull directions.

Sculpt Solids with Surfaces

The top face of the battery receiver is curved in both directions. A Boundary Surface is used to create this face.



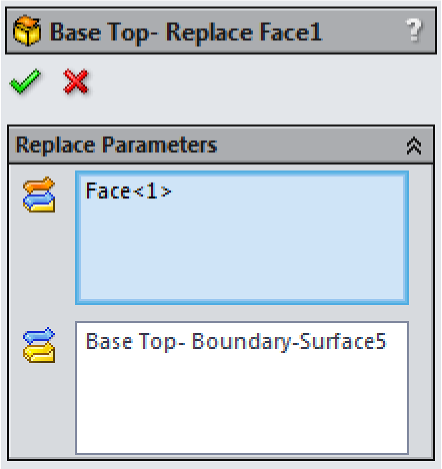
Direction 1: Surface edge of draft reference. Tangency to face applied.

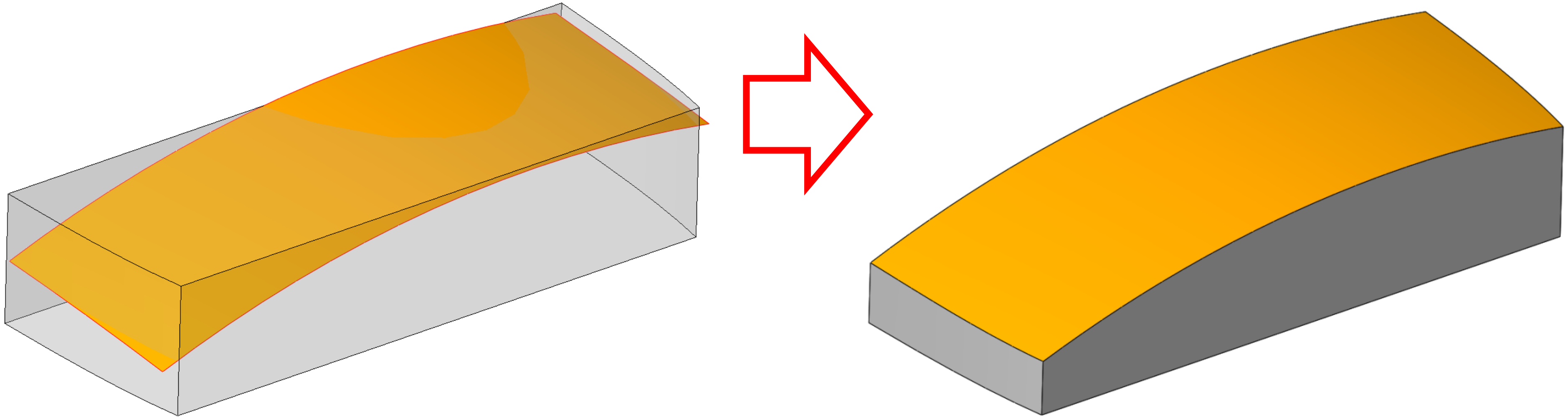
Direction 2: Sketched arc made concentric to a profile in the front layout.

NOTE: Tangent influence is often required to be raised to 100% when creating surfaces tangent to draft or mirror references.

Replace Face

Replace Face is a power tool for sculpting solid bodies with surfaces. It also offers more flexibility than a solid cut or revolve, as the input surface can be changed.



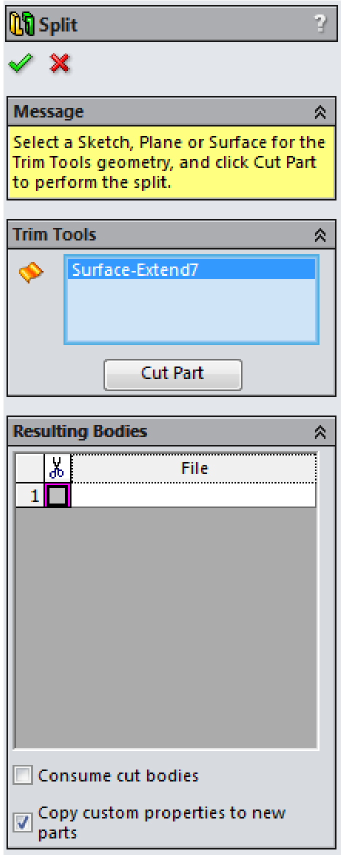


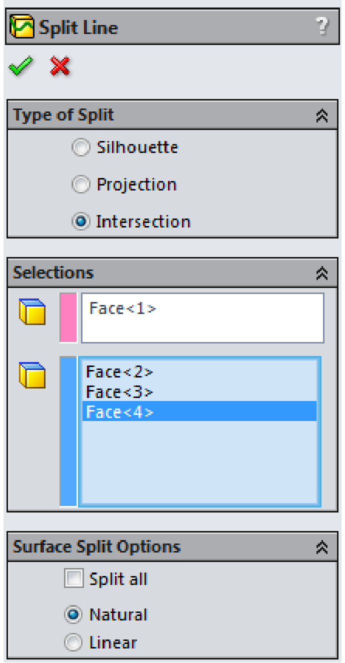
Surface passes above and below solid body

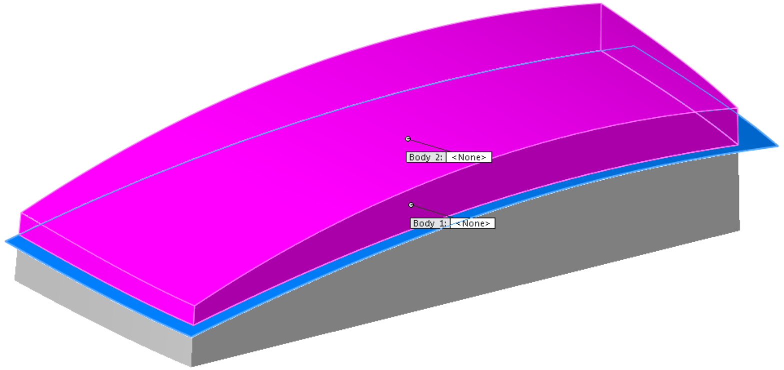
The top face is replaced with the surface body

Build Better Chamfers

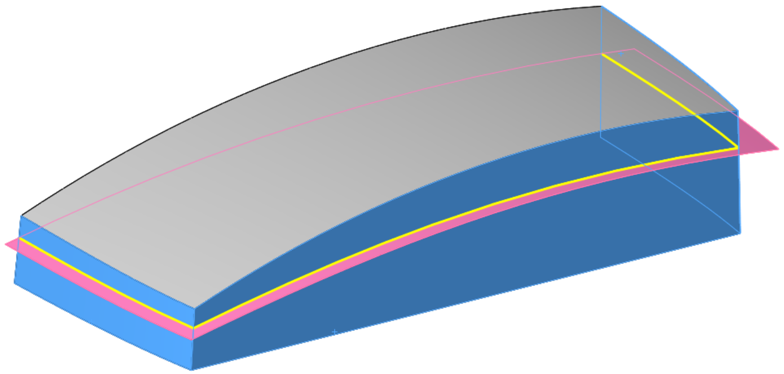
The Chamfer tool is great for breaking edges on simple geometry. Aesthetic chamfers can be better built by generating new edges and the draft tool.







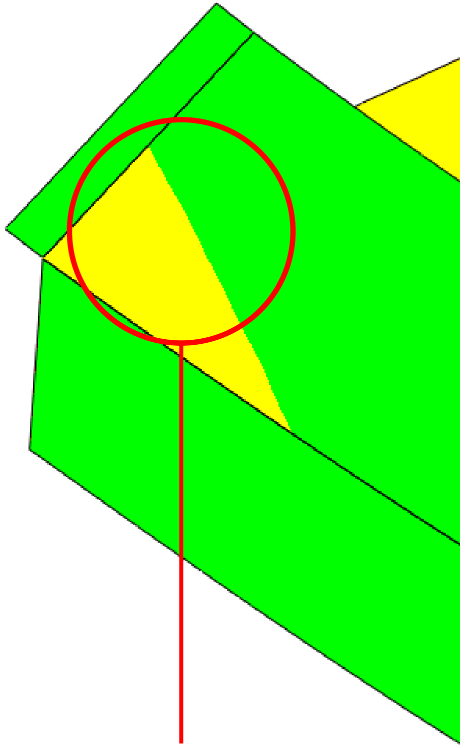
The solid body is split in two with the Split Tool.



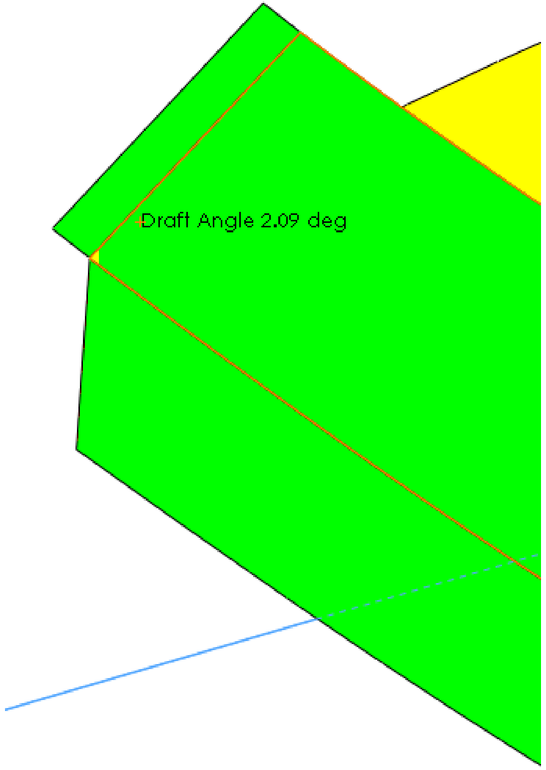
Split line also generates new edges, but can cause parent child issues after design changes. Split is recommended.

Be Aware of Pull Direction

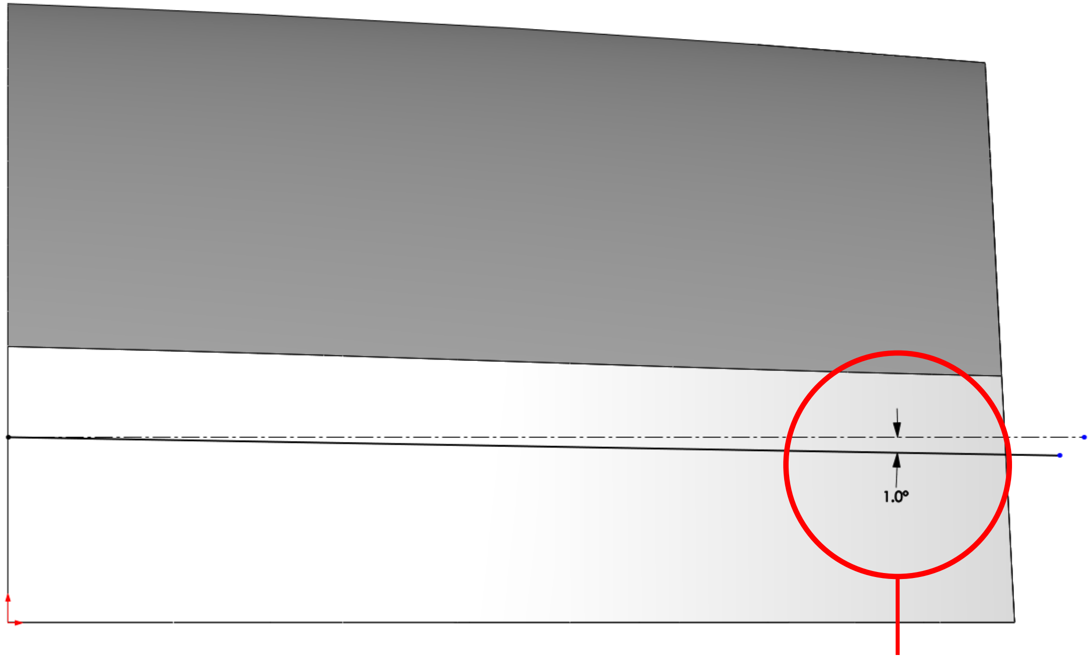
The splitting surface needs to have a slight kick when viewed from the right plane. This is to ensure correct draft at the parting line.



No Kick (insufficient draft)



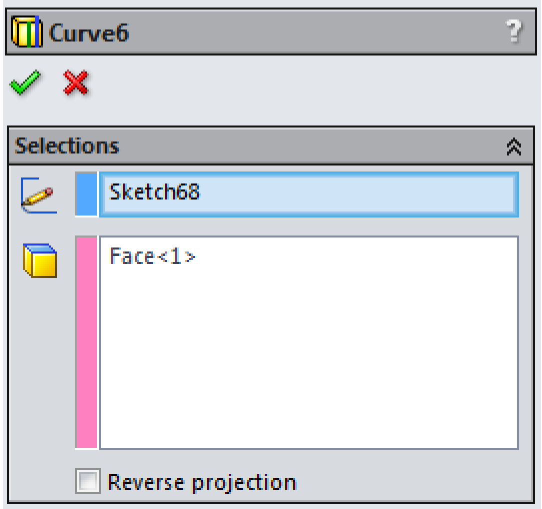
With Kick



Draft required in splitting surface. Note that as the surface is also drafted when viewed from the top plane, a shallower angle will suffice.

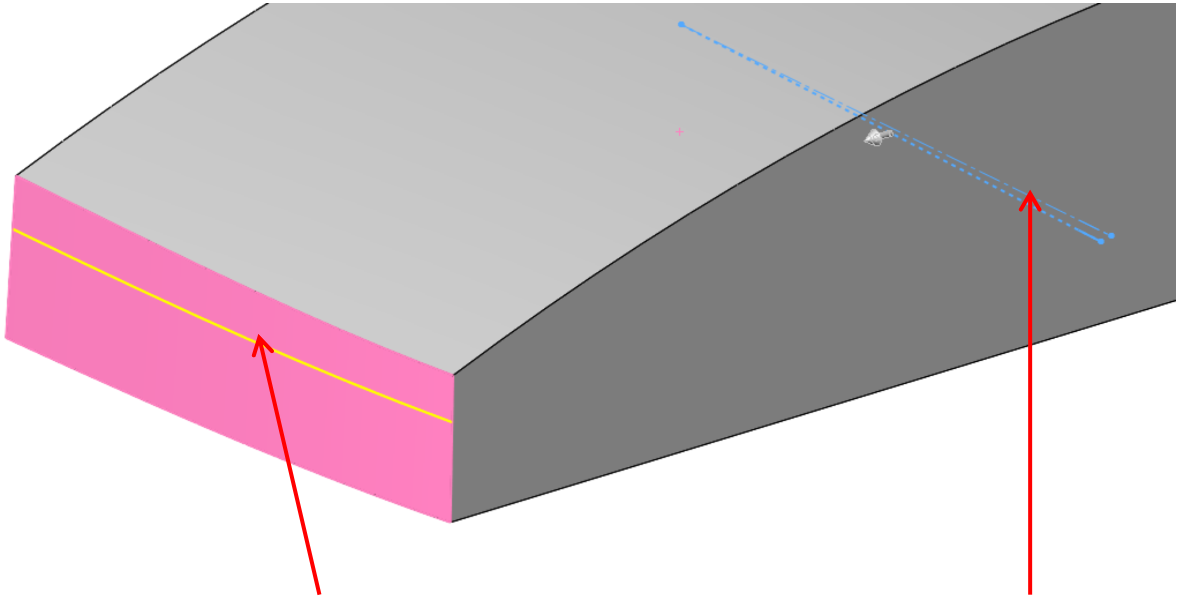
Projected Curves

Projected Curves are invaluable when creating complicated geometry. They can project two 2D sketches onto each other creating a new 3D curve. Or they can project a 2D curve on a face creating a 3D curve.



BEST PRACTICE:

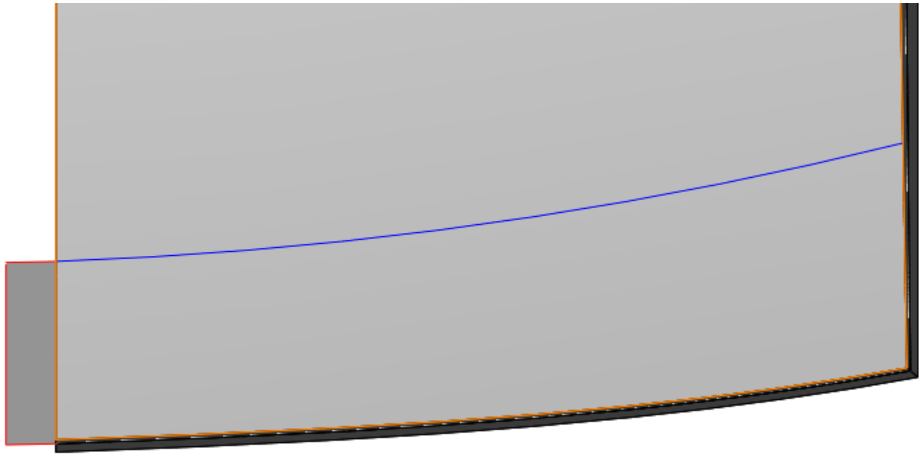
If the projected curve is to be used or referenced in any other features, convert it into a 3D sketch. There is a bug in SW where absorbed projected curves cannot be selected in the graphics window or used in any other feature.

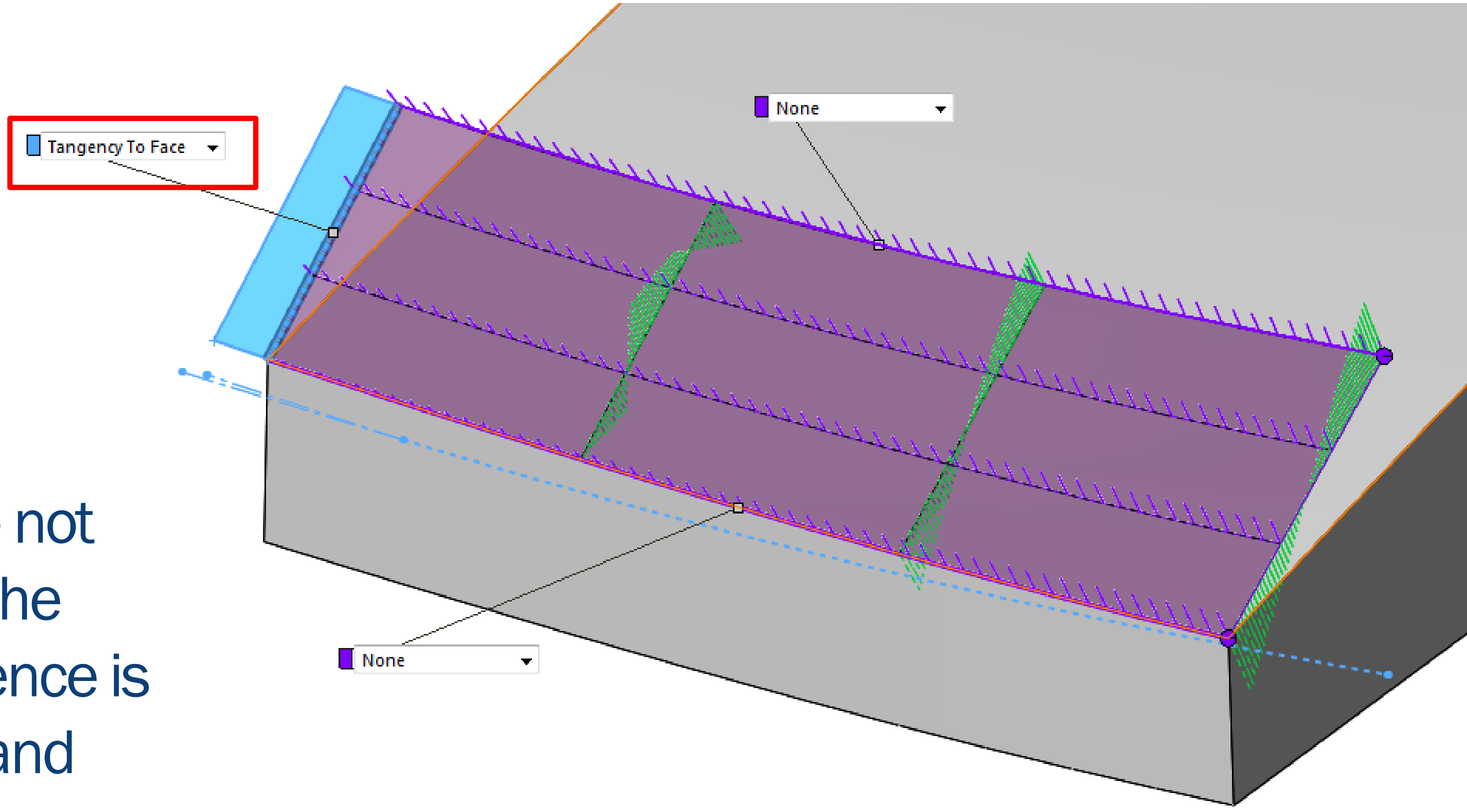


3D Curve is generated by projecting 2D sketch on curved front face.

Build the Chamfer

The two arcs of the chamfer are not concentric. Draft is also required at the parting line. A Boundary Surface will be used to construct this face.

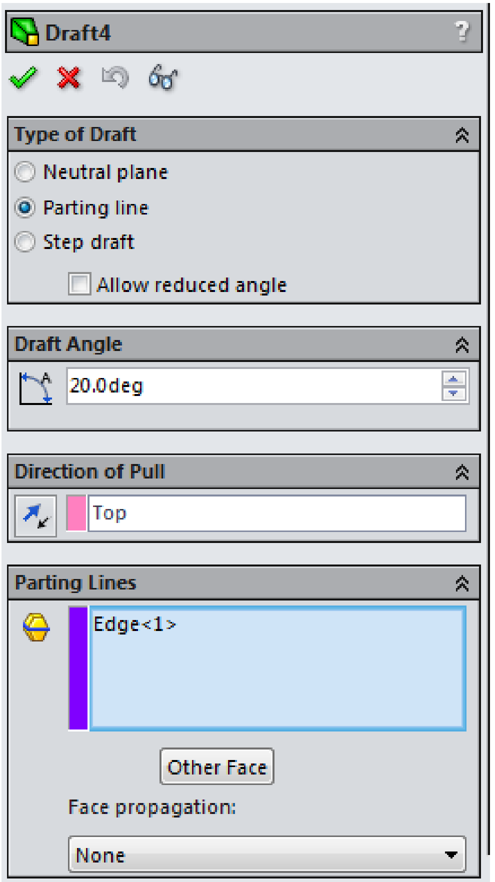


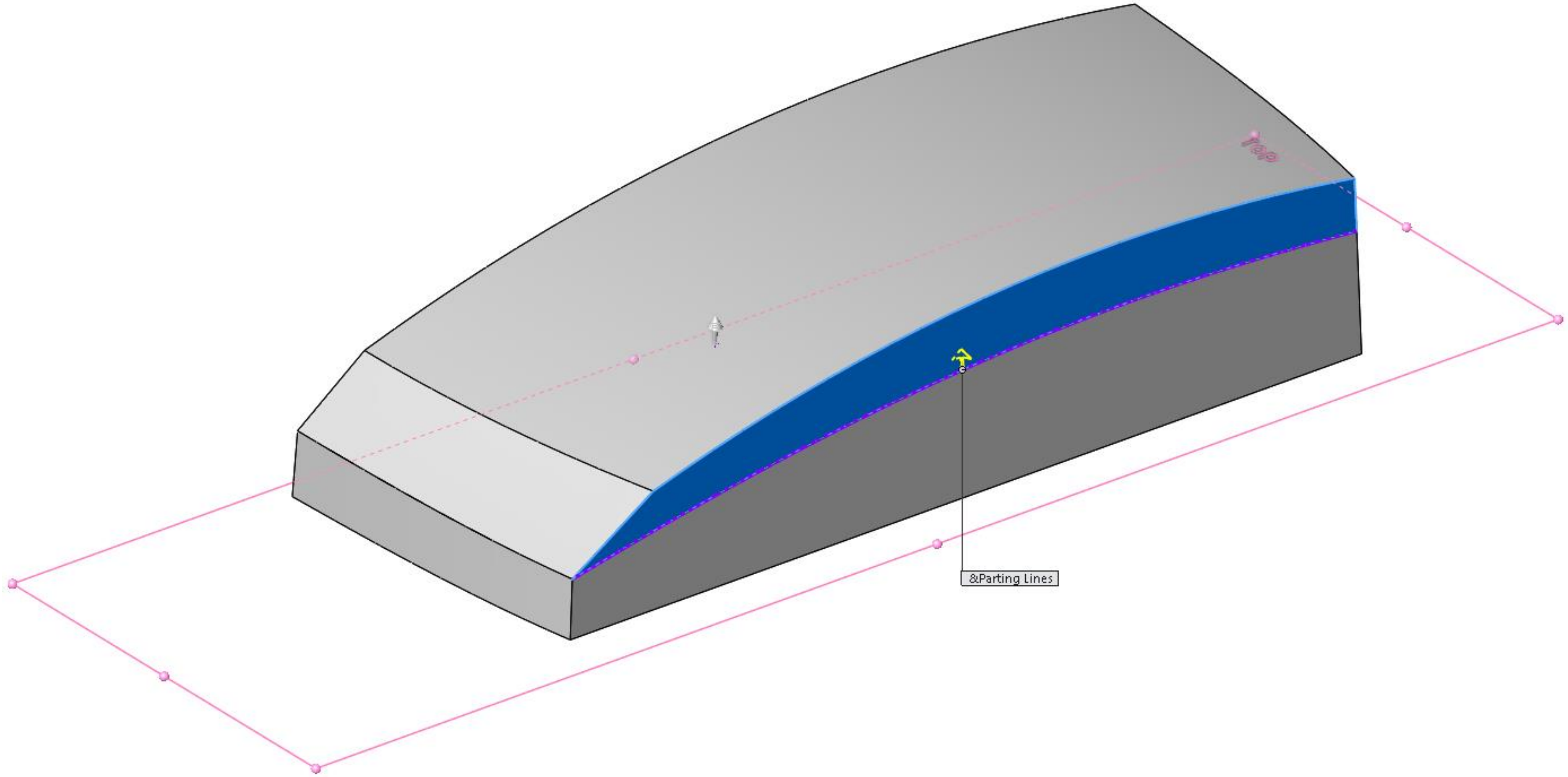


Top and bottom edges of the chamfer are not concentric. A Boundary Surface creates the chamfer face. The edge of the draft reference is used in D1, the bottom edge of the solid and projected curve in D2.

Build the Chamfer

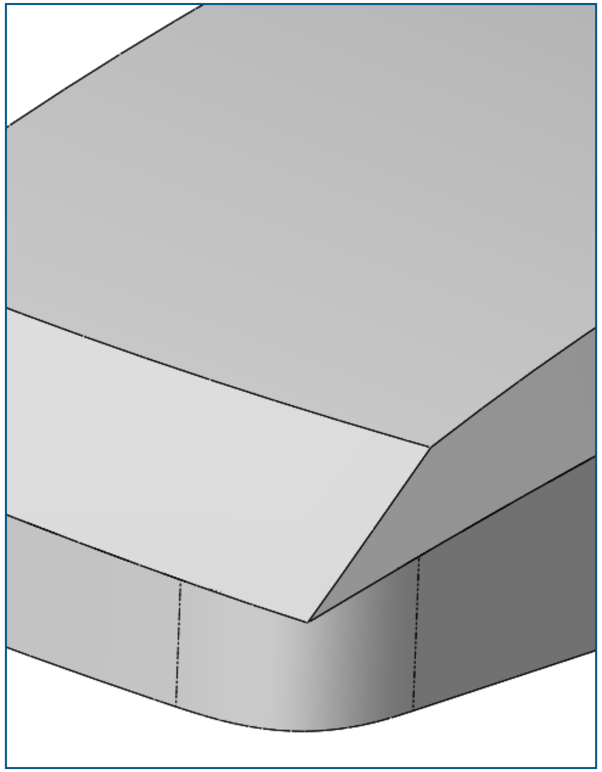
The draft tool allows better definition of the chamfer. The exact start of the chamfer can be defined when using a parting line draft feature.



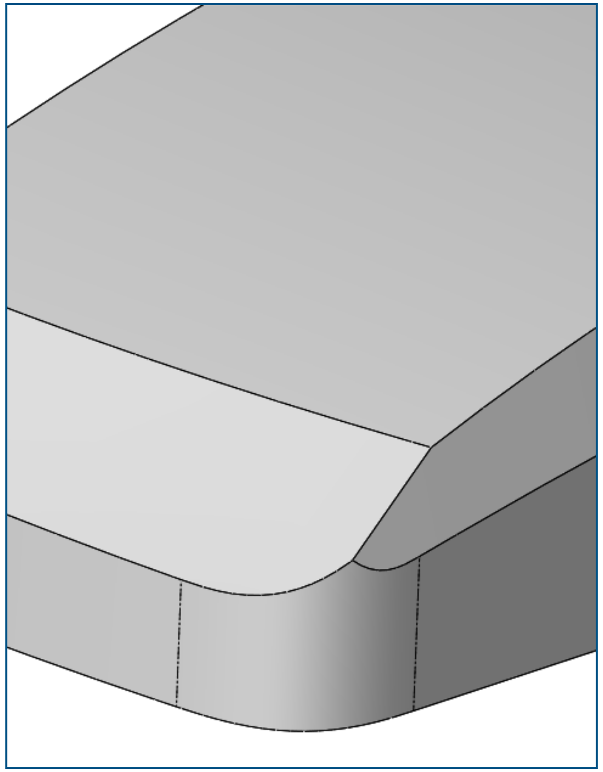


Complicated Corner Fillets

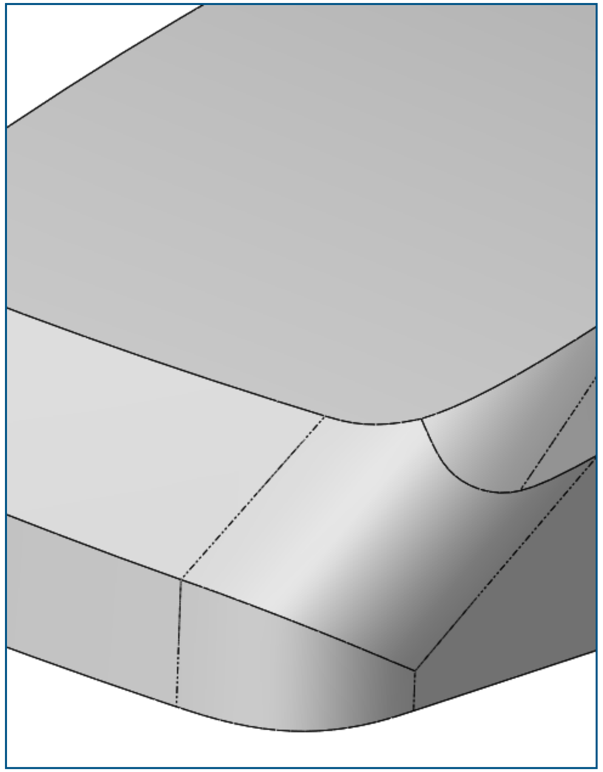
One benefit to splitting the body in two is controlling the flow of fillets. The fillet stops at the split between the body. The fillet propagates awkwardly when the bodies are combined.



Split Body



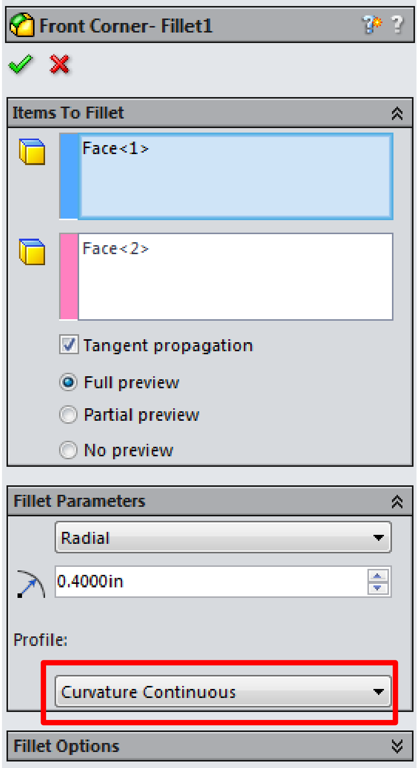
Combined Body, 1 Face Set

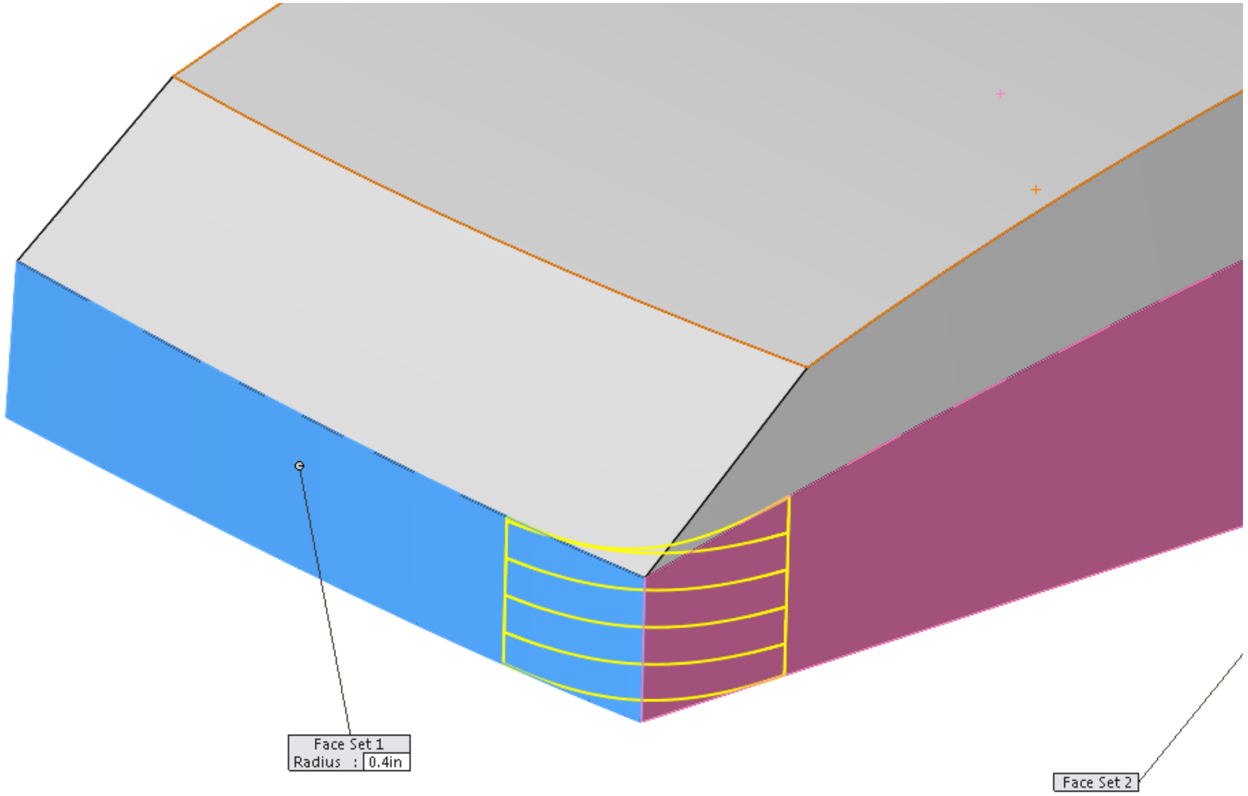


Combined Body, 2 Face Sets

Define the Fillet Region

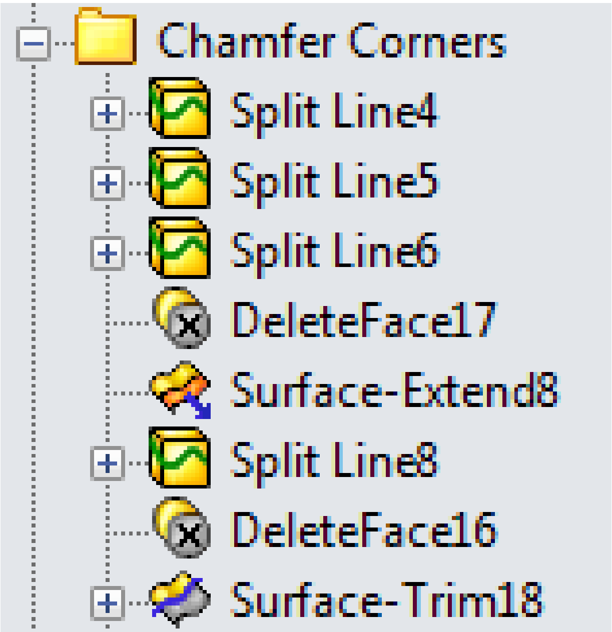
The Face Fillet tool with the Curvature Continuous option enabled is an excellent way of quickly creating C2 connections between surfaces. Note that only one fillet can be constructed at a time.

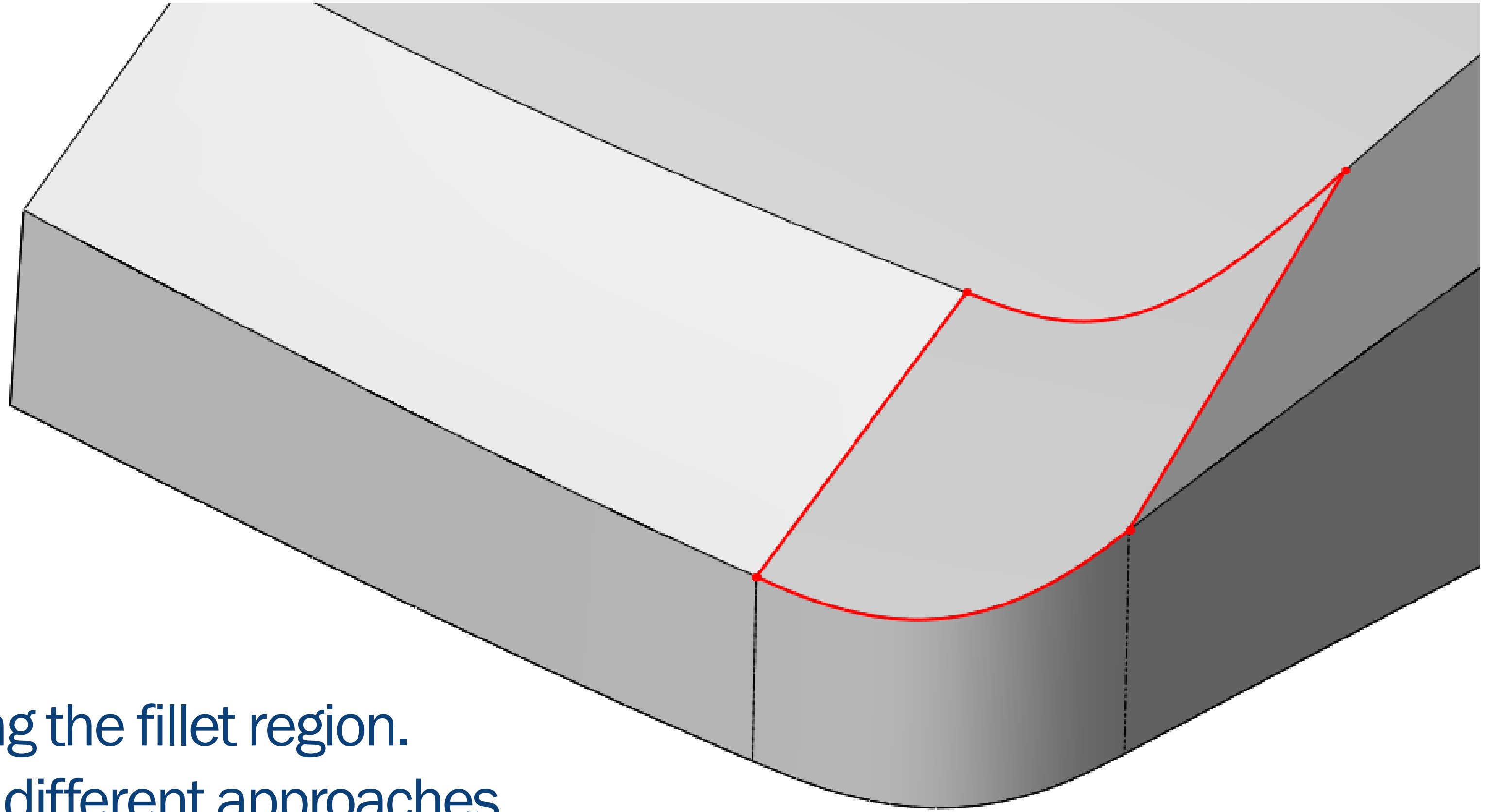




Finish the Fillet Manually

Use a combination of Split Lines, Delete Faces, and Surface Trims to manually create a new boundary for the fillet.

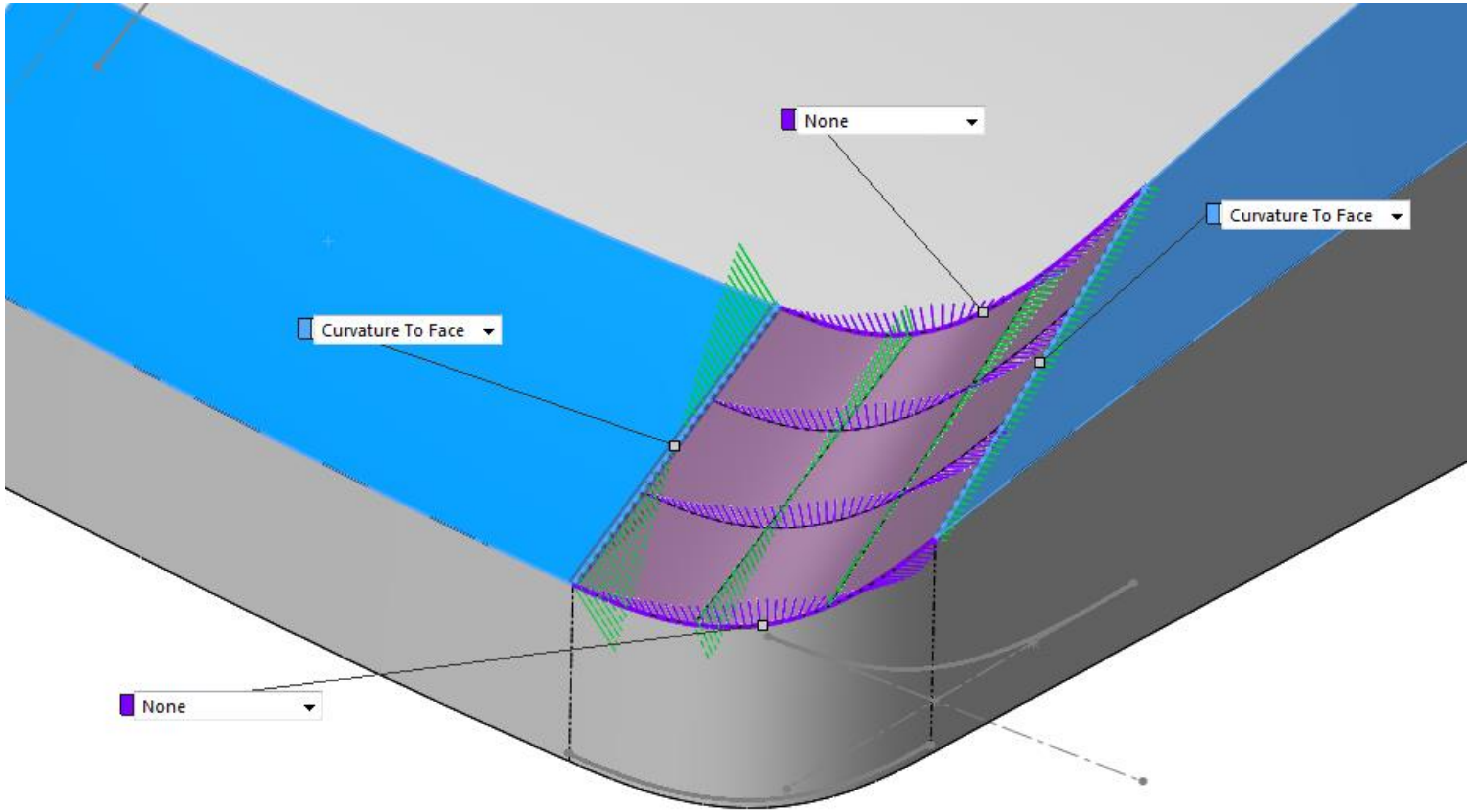


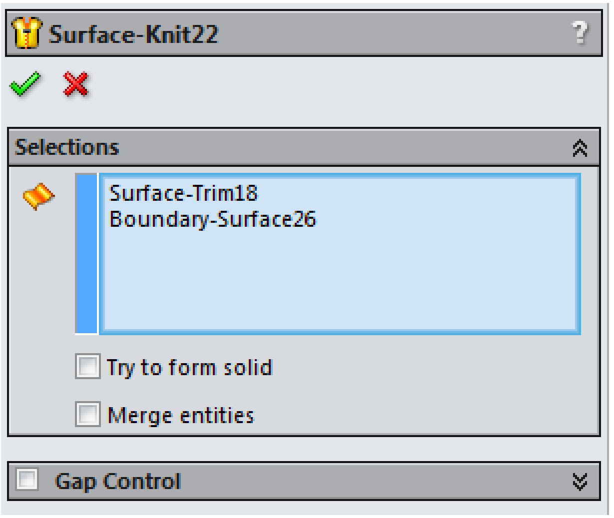


There are multiple ways of defining the fillet region.
Different situations might require different approaches.

Finish the Fillet Manually

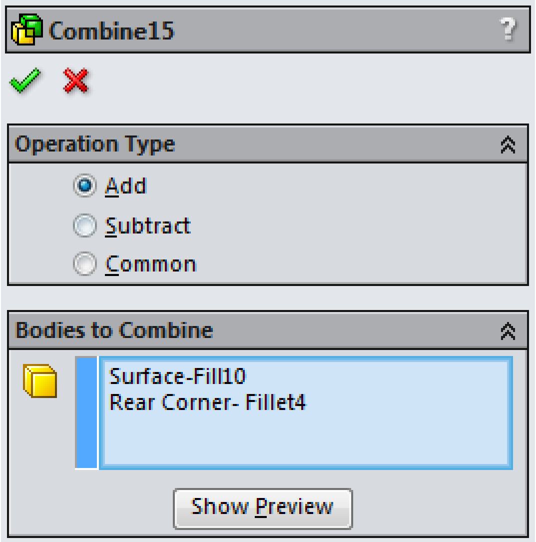
Finish the fillet with a new Boundary Surface. Note that curvature to face is being used on both profiles in Direction 1. Finish the fillet by knitting it into the model.

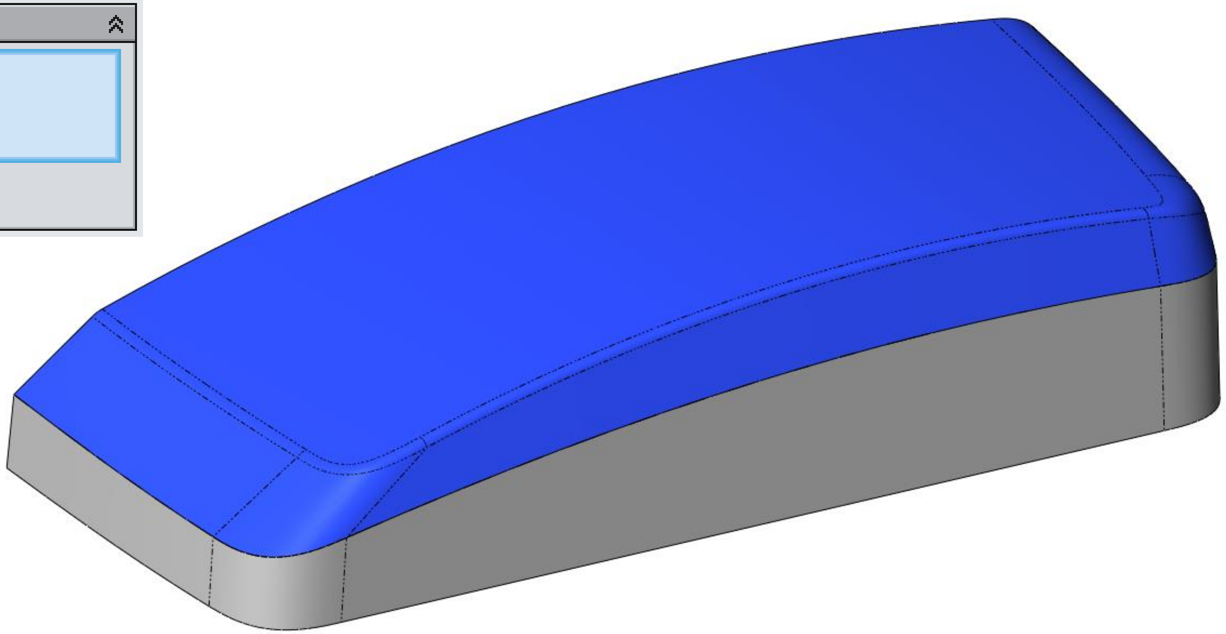




Combine the Halves

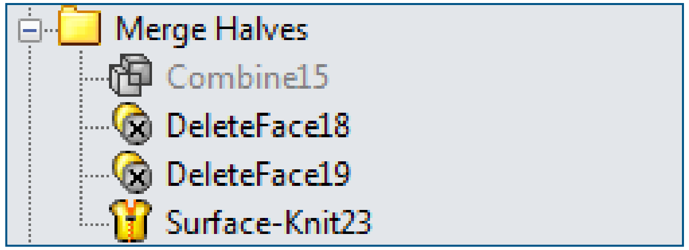
The two halves of the body are put back together with the Combine tool.





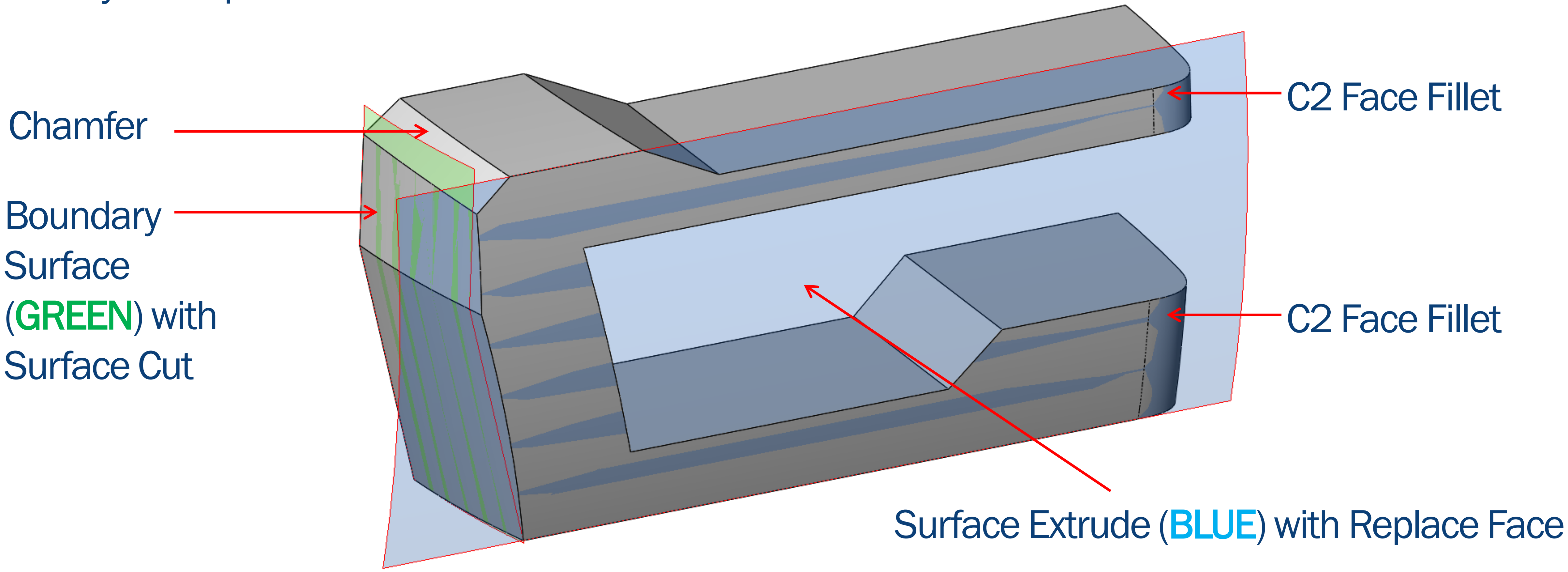
TIP:

Due to microscopic differences in faces and edges and the way Parasolid interprets them, the combine may fail. In this case, delete the two common faces of the bodies and knit them together.



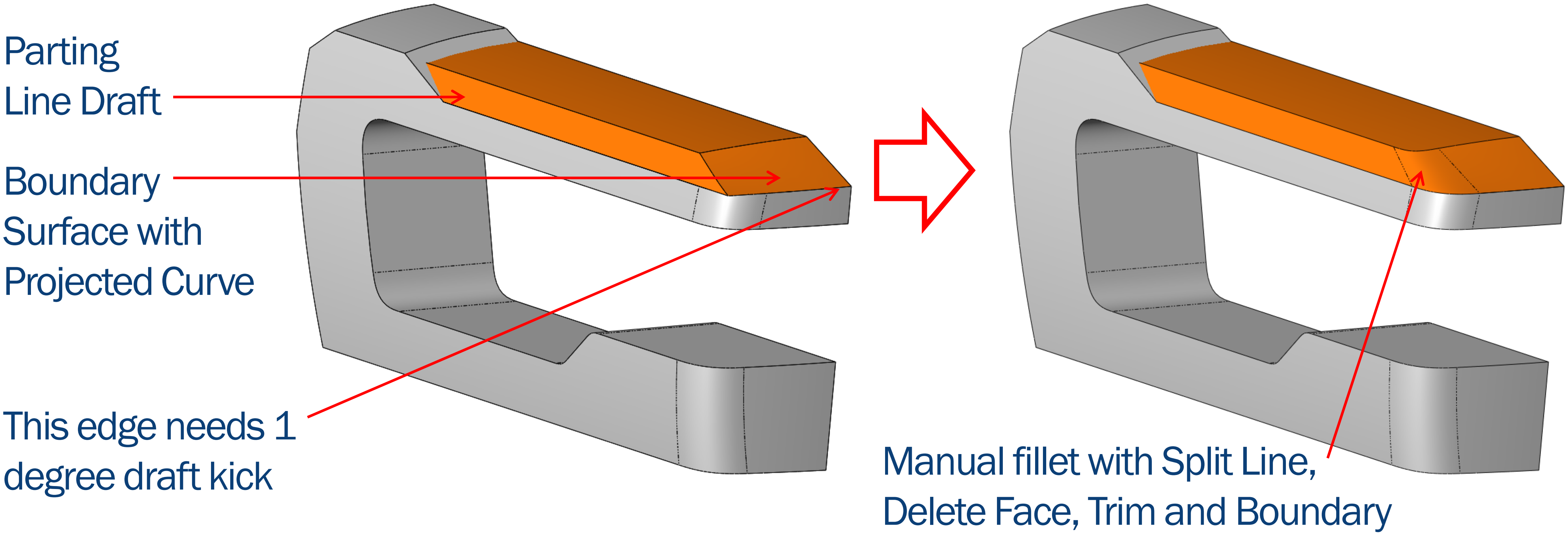
Detail The Top

The top of the flashlight is detailed in the same way as the base of the flashlight. The solid body is sculpted with surface features.



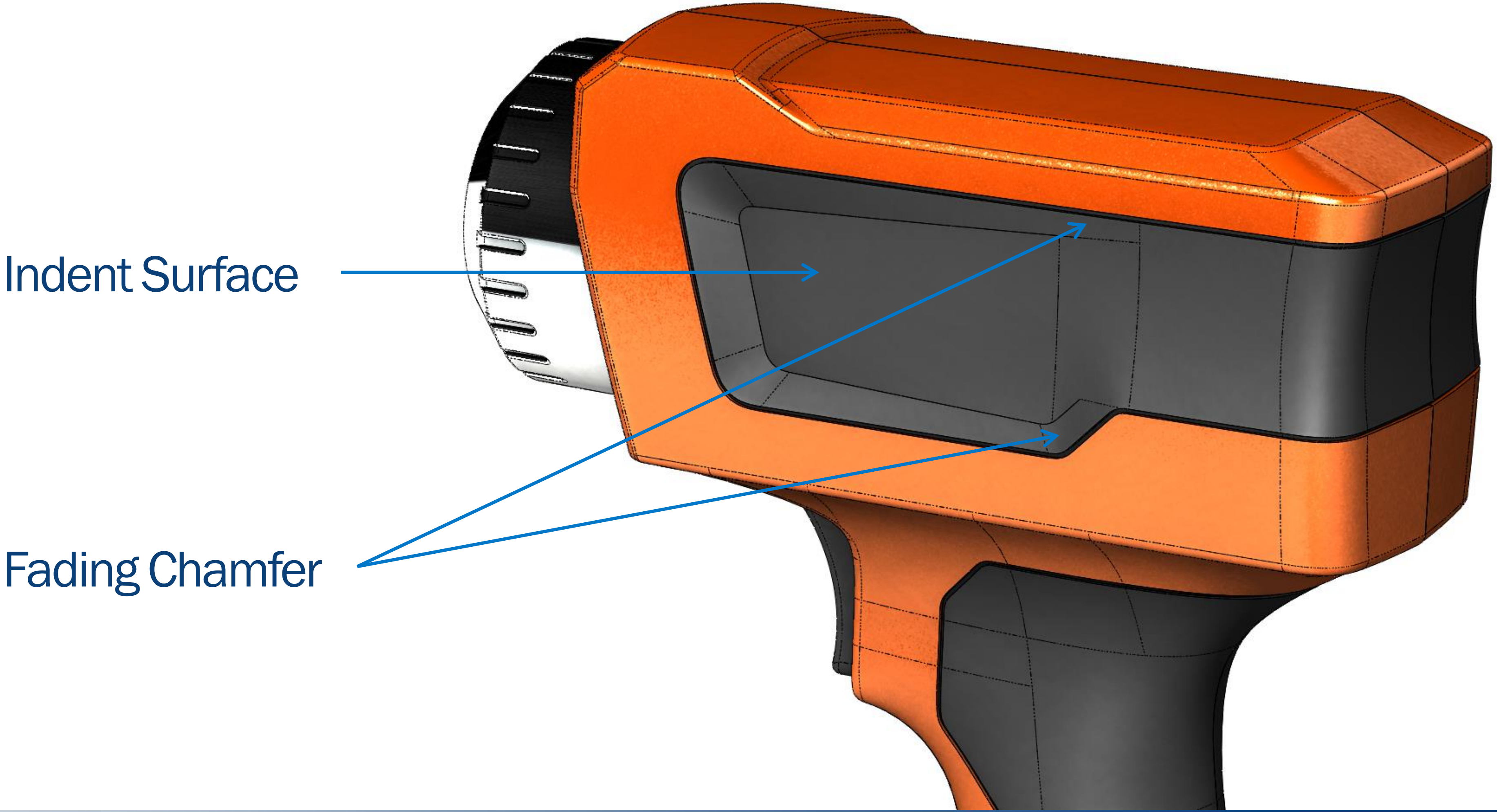
Detail The Top

The top shape is created as a separate body, just how the bottom area was split into two bodies. Similar modeling techniques are used to build a chamfer and the filleted corner.



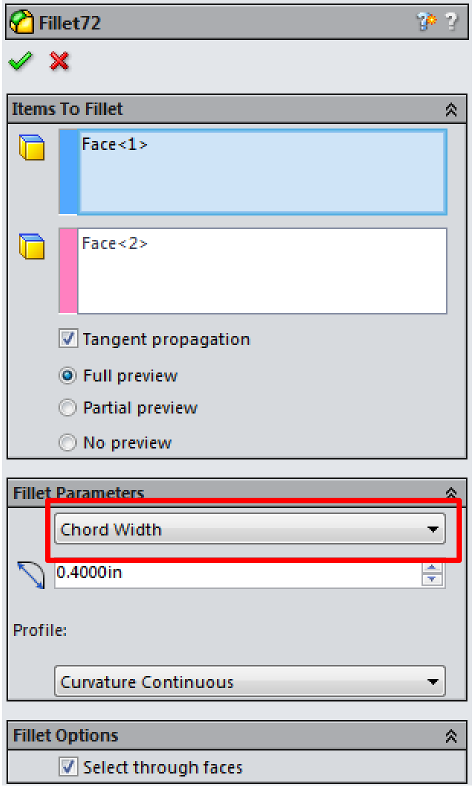
Indent and Fading Chamfer

Fading chamfers look difficult to model, but can easily be accomplished with the right technique.

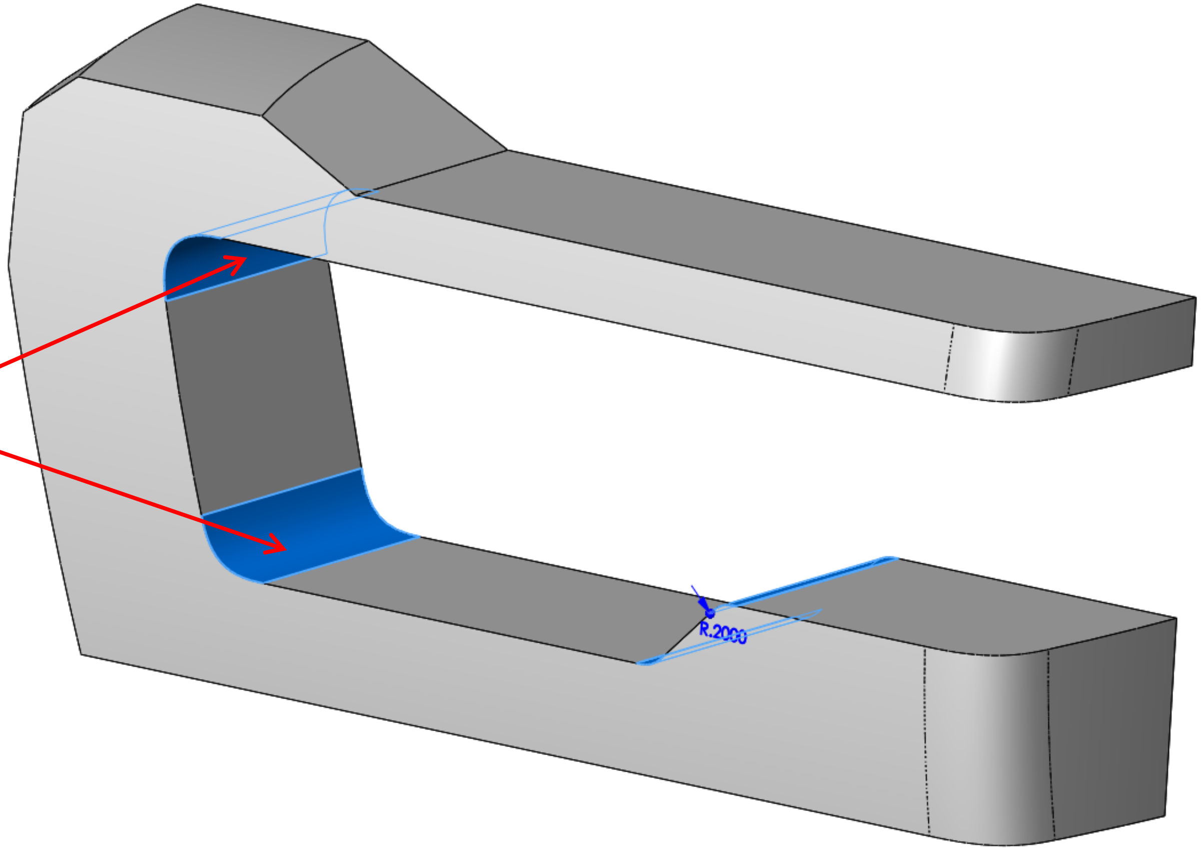


Prepare the Indent Area

Because the layout sketch was dimensioned to the virtual sharps, C2 face fillets are added to the sharp edges to complete e the indent profile. Note that the chord width option is enabled to ensure the fillets are the same size despite the different angles of the adjacent surfaces.



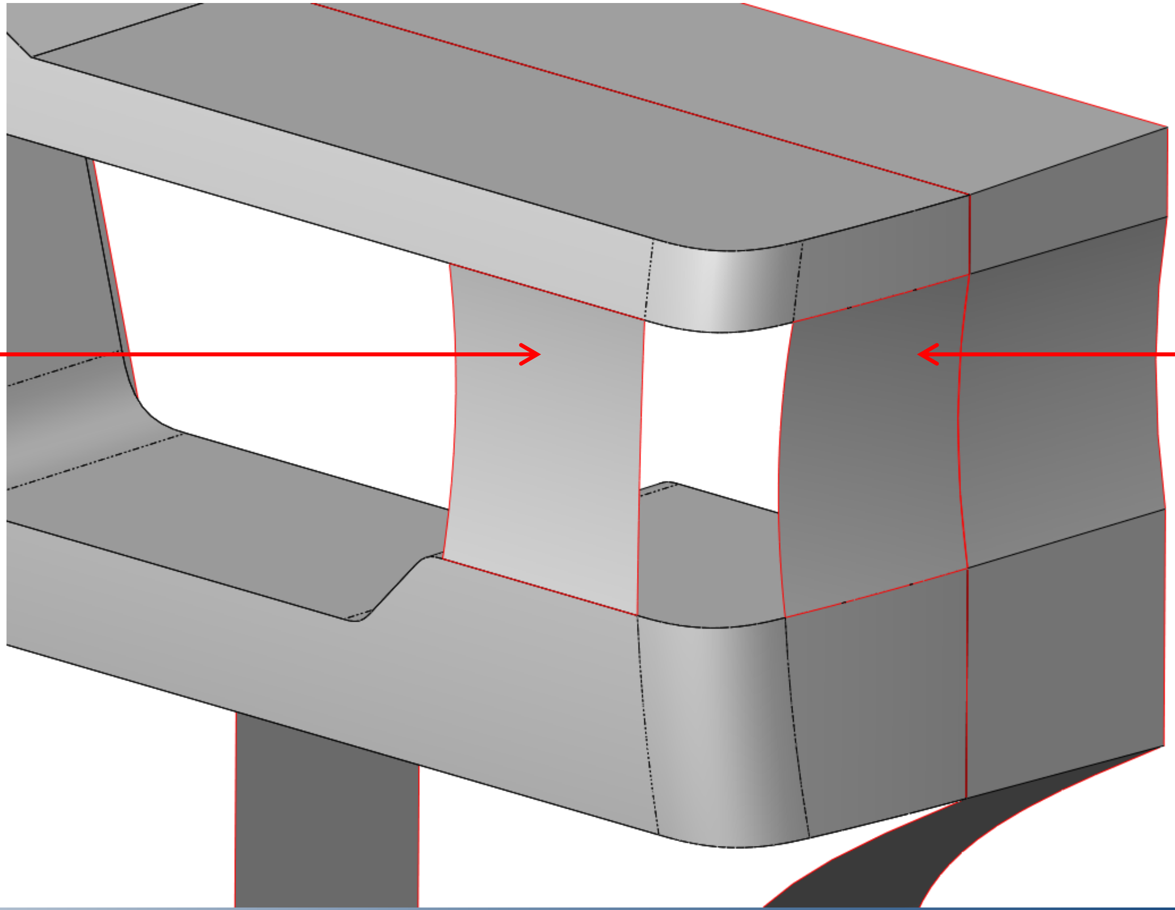
Chord width ensures these fillets appear to be the same size.



Start the Indent

The indent area is concave, vs the convex profile of the rest of the flashlight. This area can easily be modeled with surface features. The largest surfaces are modeled first.

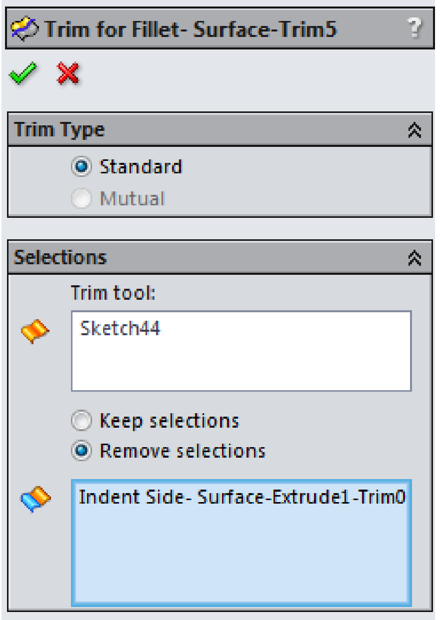
This face can be created with Surface Extrude

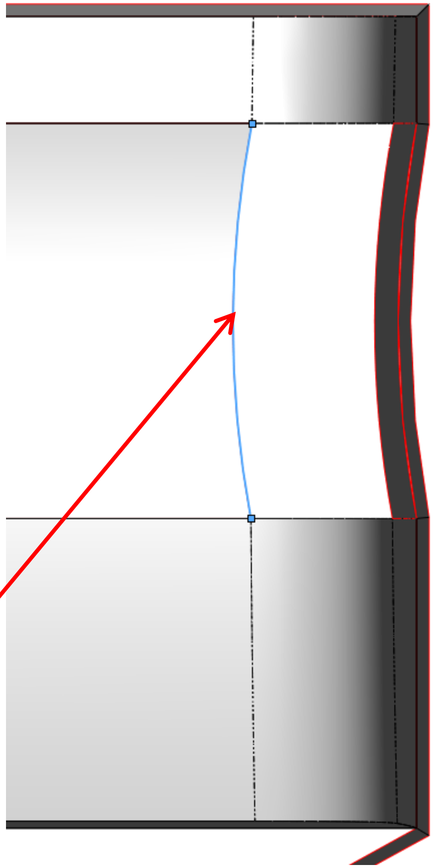


Boundary Surface using Draft Reference in D1, Existing solid body edges in D2.

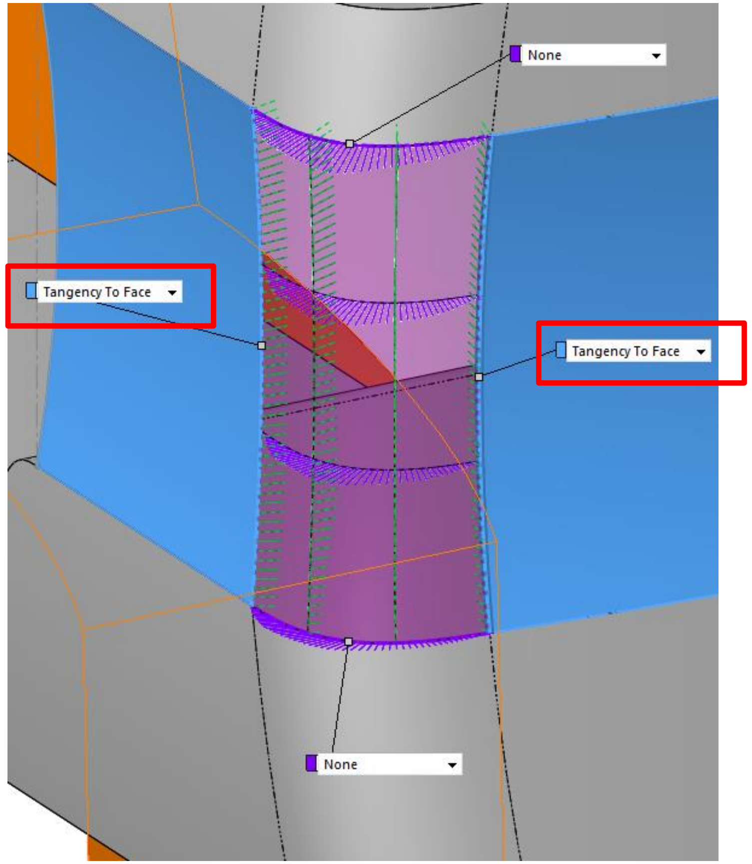
Complete the Fillet

The fillet will be built manually. The surface extrude needs to be trimmed back for the correct fillet flow. The fillet itself is built with a Boundary Surface.





Surface is trimmed back.



NOTE:

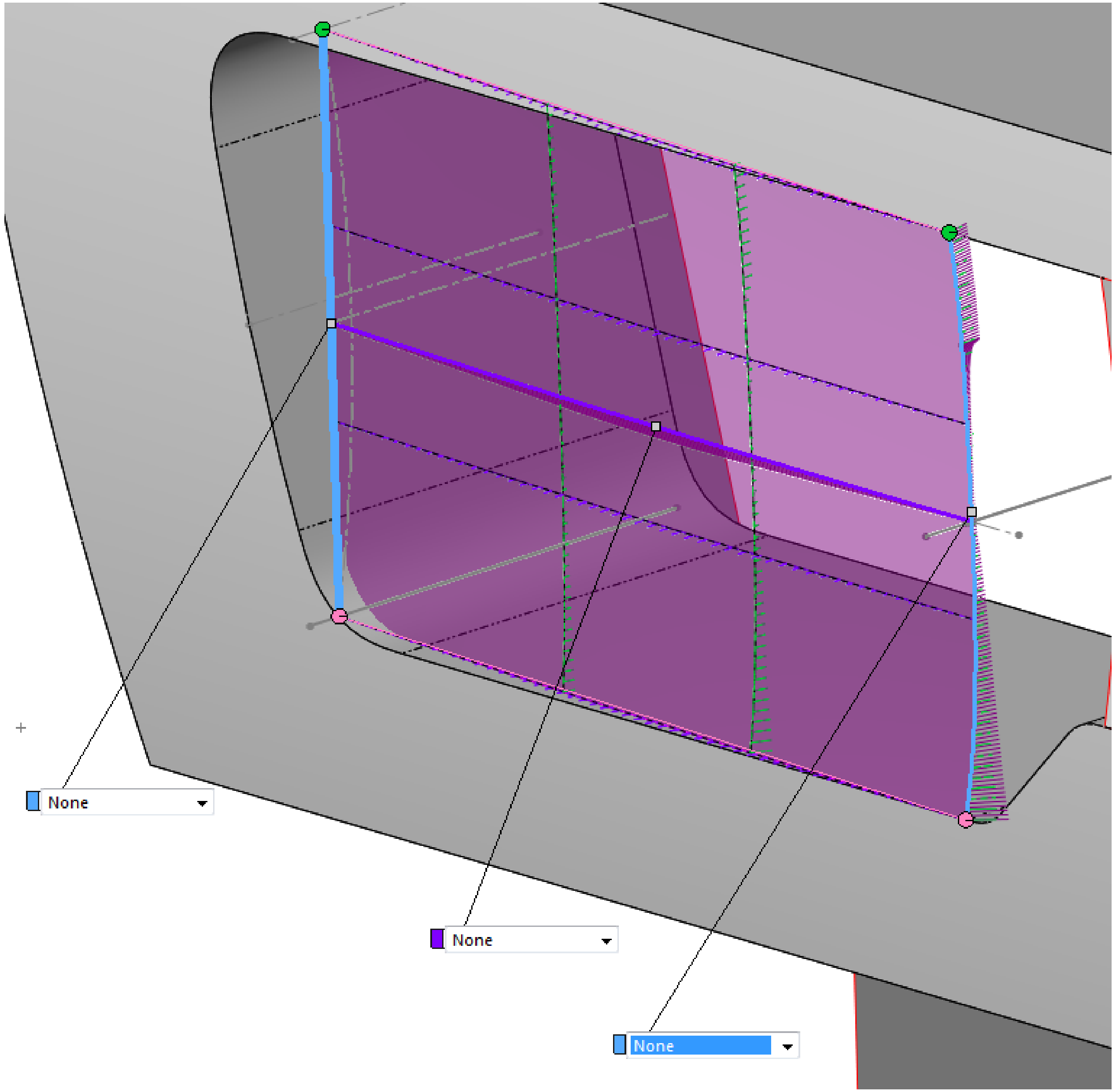
Even though the desired fillet should be C2 to adjacent surfaces, only tangency to face is employed. The surfaces curvature combs show that C2 is already achieved.

Build the Diving Surface

The fading chamfer requires a surface "deeper" than surround geometry. A "diving" surface is created allowing for the chamfer to fade out.

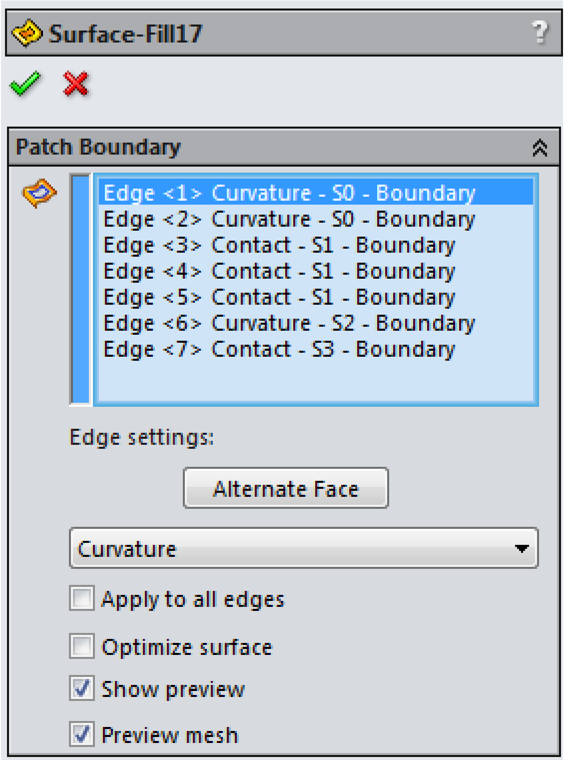
The diving surface is created from two profiles in D1 and sketched spline in D2.

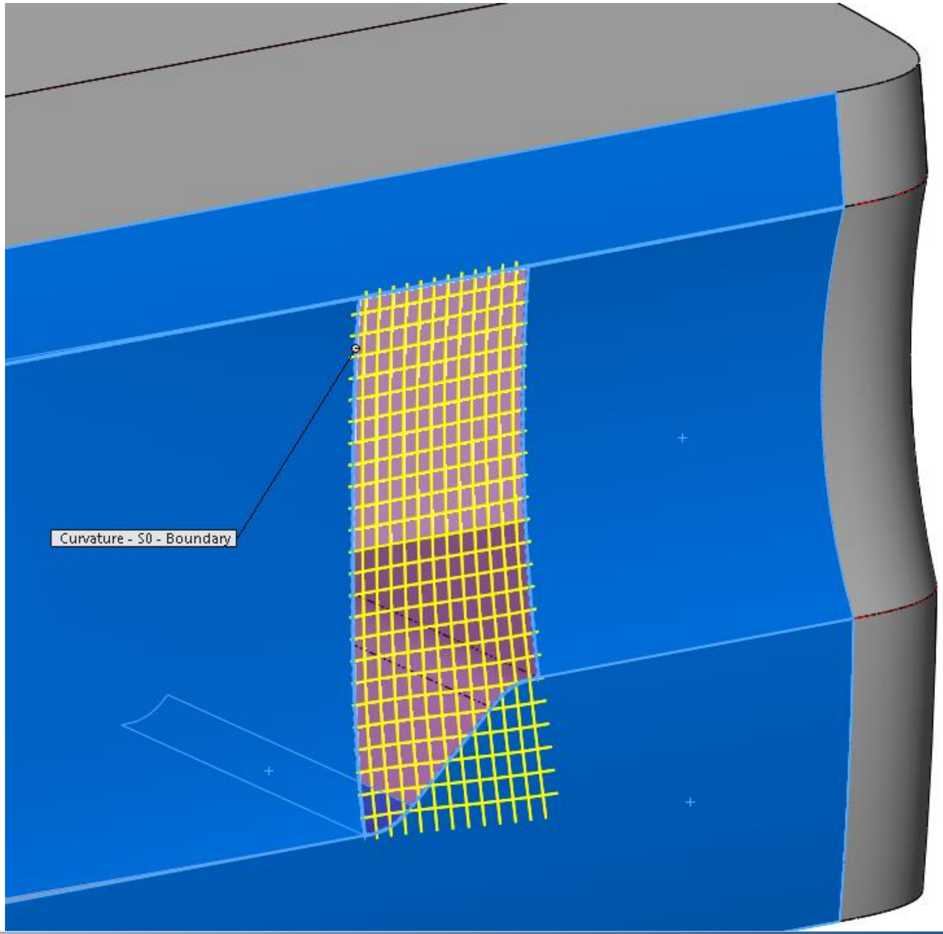
The surface does not need to contact the exiting model edges as it will be trimmed back to create the chamfer region.



Complete the Indent

The remaining open area of the indent has an N number of open edges. Surface Fill is used to complete the patch due to its ability to close complicated openings.



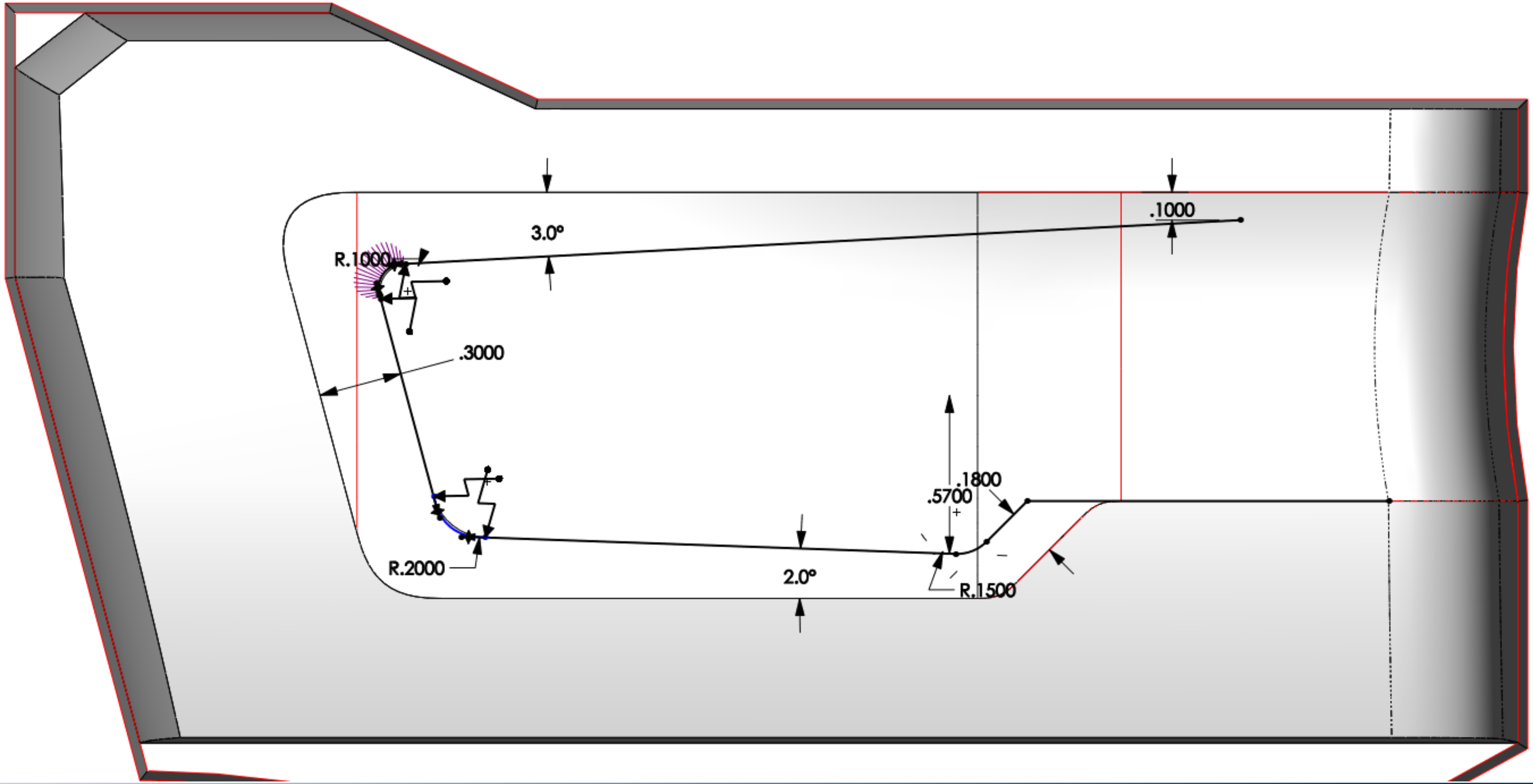


<u>NOTE:</u>

While Boundary Surface could be used on this patch, the ability of Surface Fill to overbuild and trim back the surface produces superior results

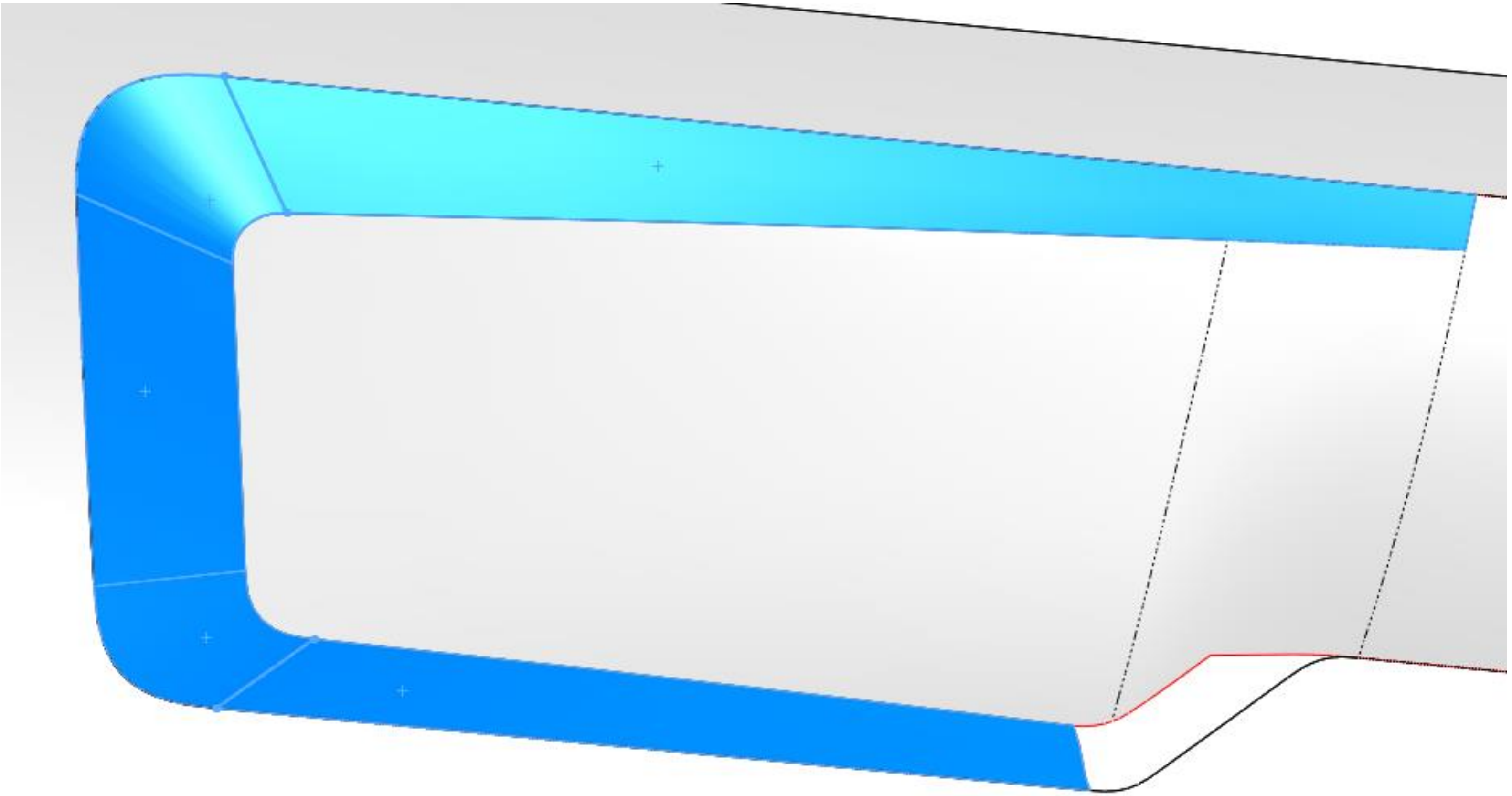
Trim the Chamfer Profile

The faces of the diving surfaces are knit together. The chamfer profile is then trimmed away from the diving surface.



Build the Chamfer

Construct each face of the chamfer with a boundary surface.



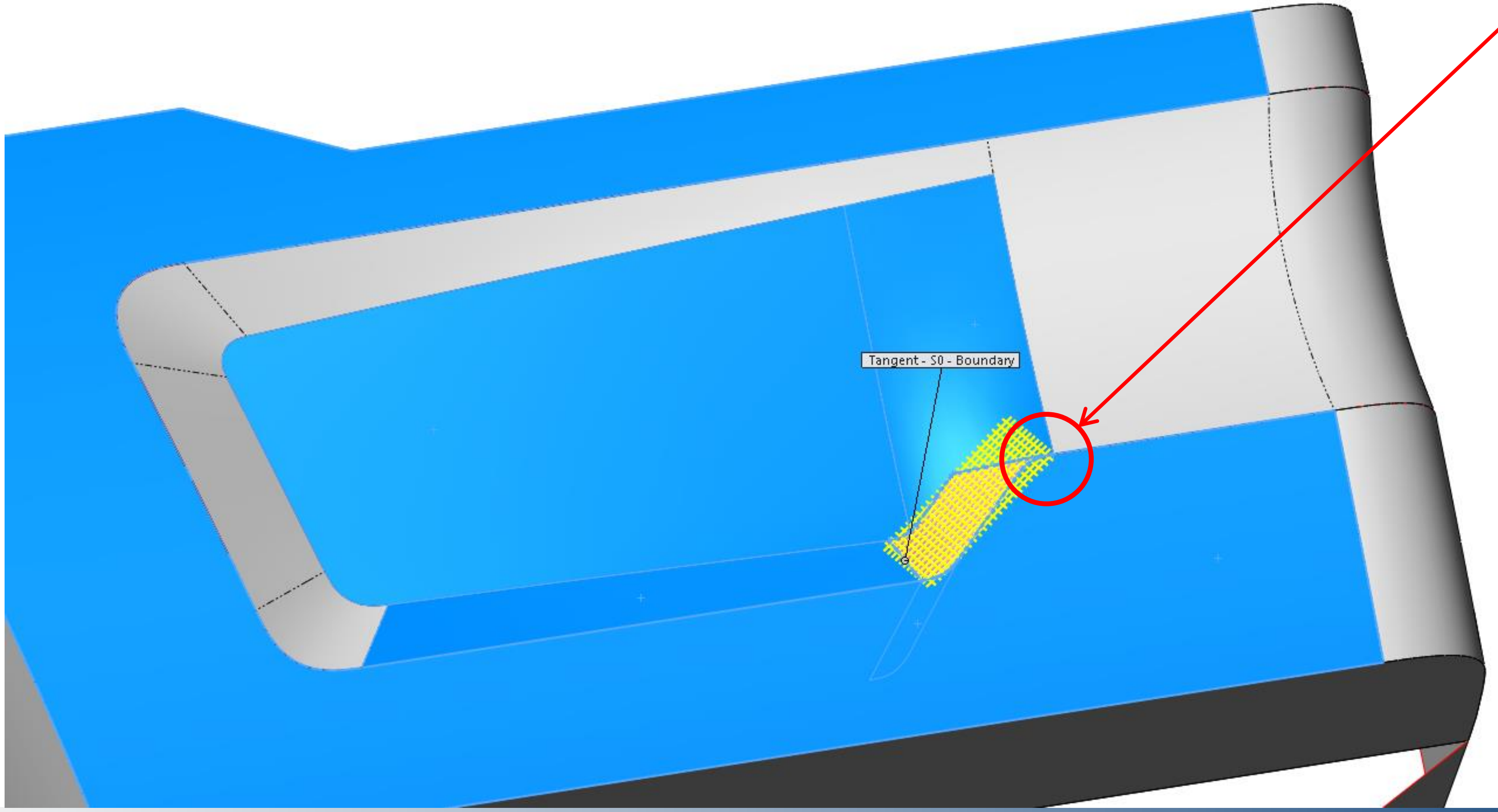
NOTE:

Tangency or Curvature is not used on profiles. The surfaces are automatically curvature continuous based on the profiles used.

Adding an end condition can introduce ripples to the surface.

Complete the Chamfer

The final patch of the fading chamfer is an N-sided patch. Surface Fill is used to complete the transition. While it may be 4 sided, one of the corners comes to a single point.



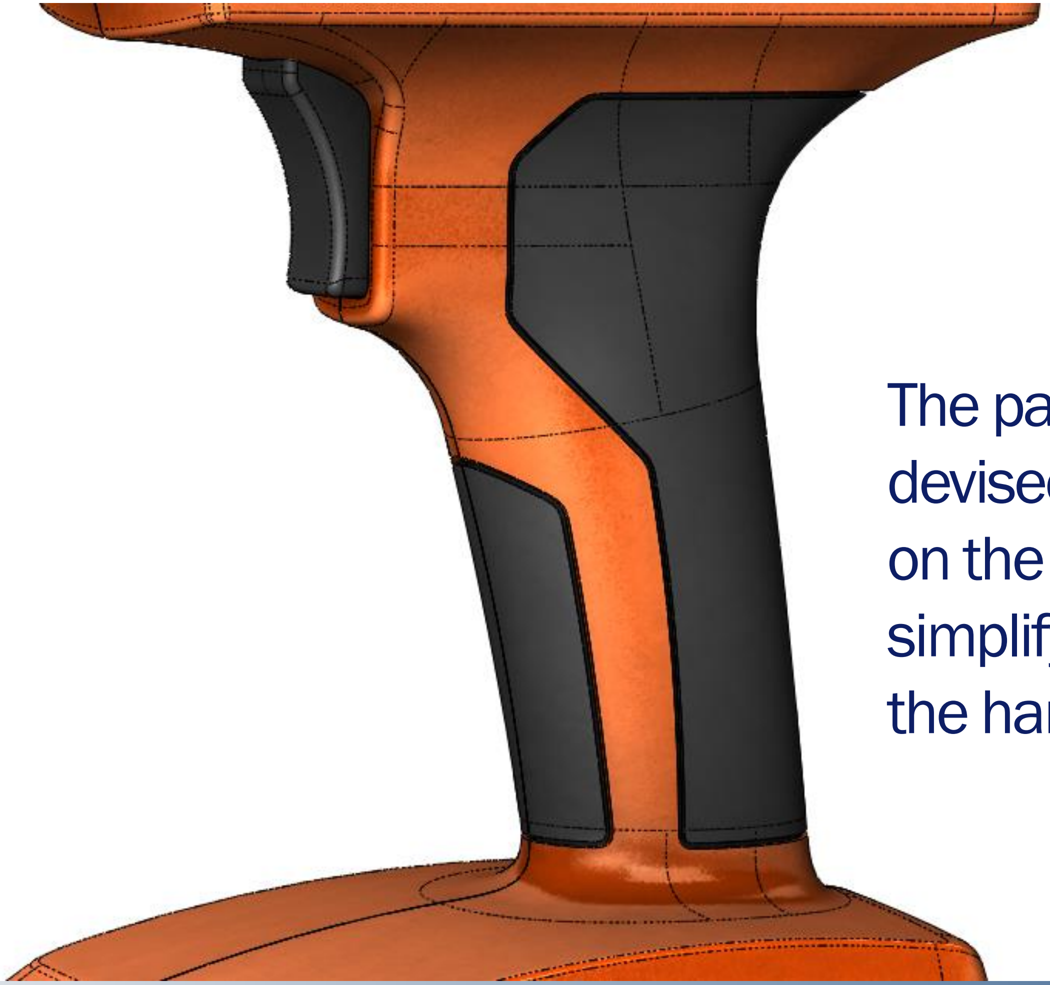
NOTE:

Curvature is not applied to edges as it can introduce defects into the surface fill and is not required.

Surface Fill is able to match adjacent surfaces slightly better than tangent, but not quite C2.

Construct The Handle

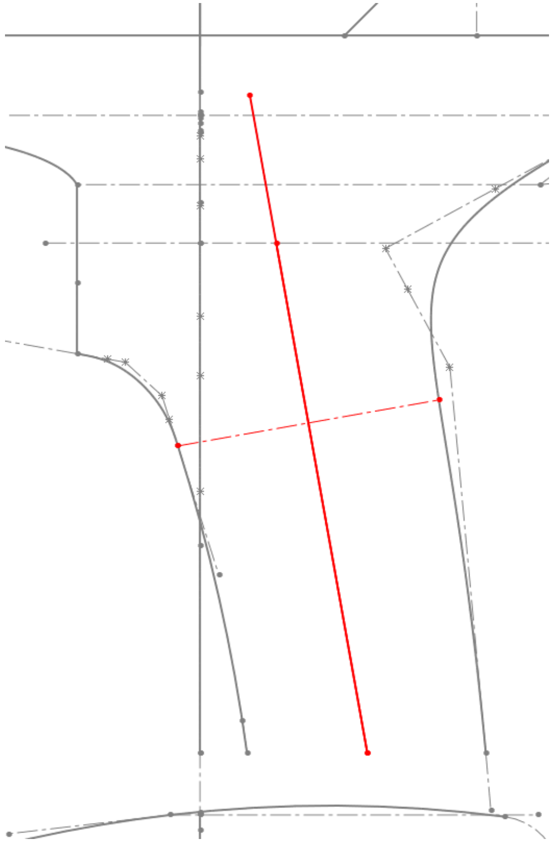
The smooth, organically shaped handle will be constructed entirely with surface features.



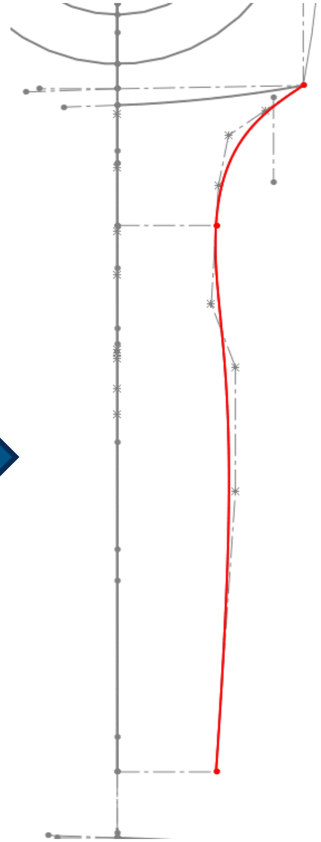
The patch layout devised before starting on the handle will help simplify the modeling of the handle.

Begin Laying Out Curves and Sketches

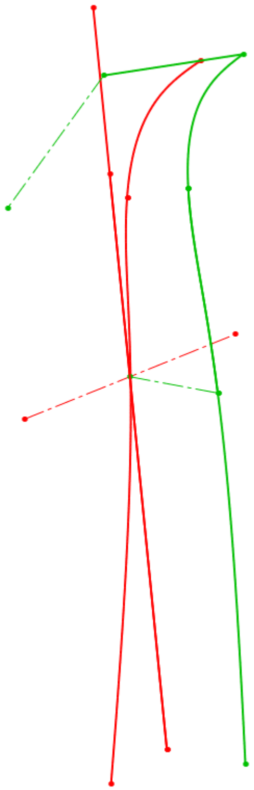
The handle surfaces will be built from curves and sketches. The front profile of the handle is projected onto a sketch on the right plane creating the main profile of the handle.



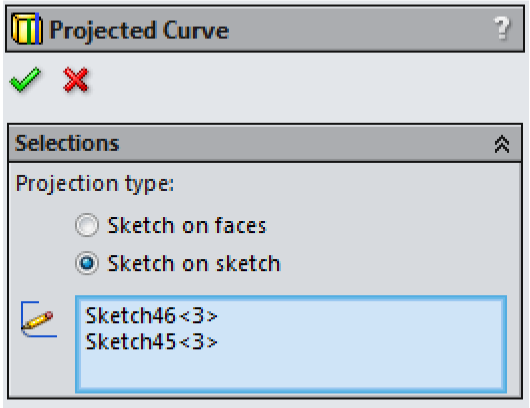
2D Sketch Right Plane



2D Sketch Front Plane



3D Projected Curve

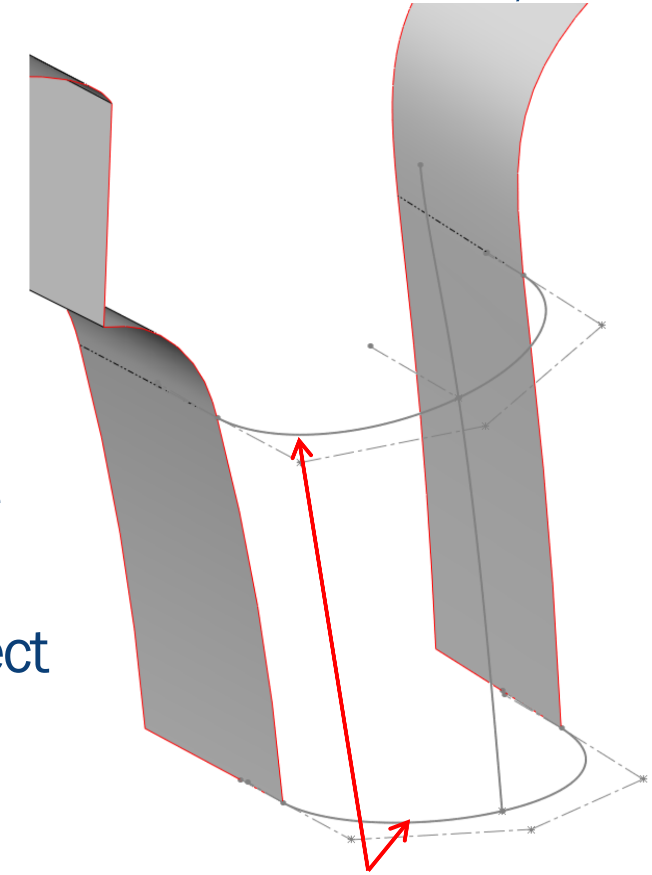


TIP: Convert the Projected Curve into a 3D sketch.

Create Surfaces from the Curves

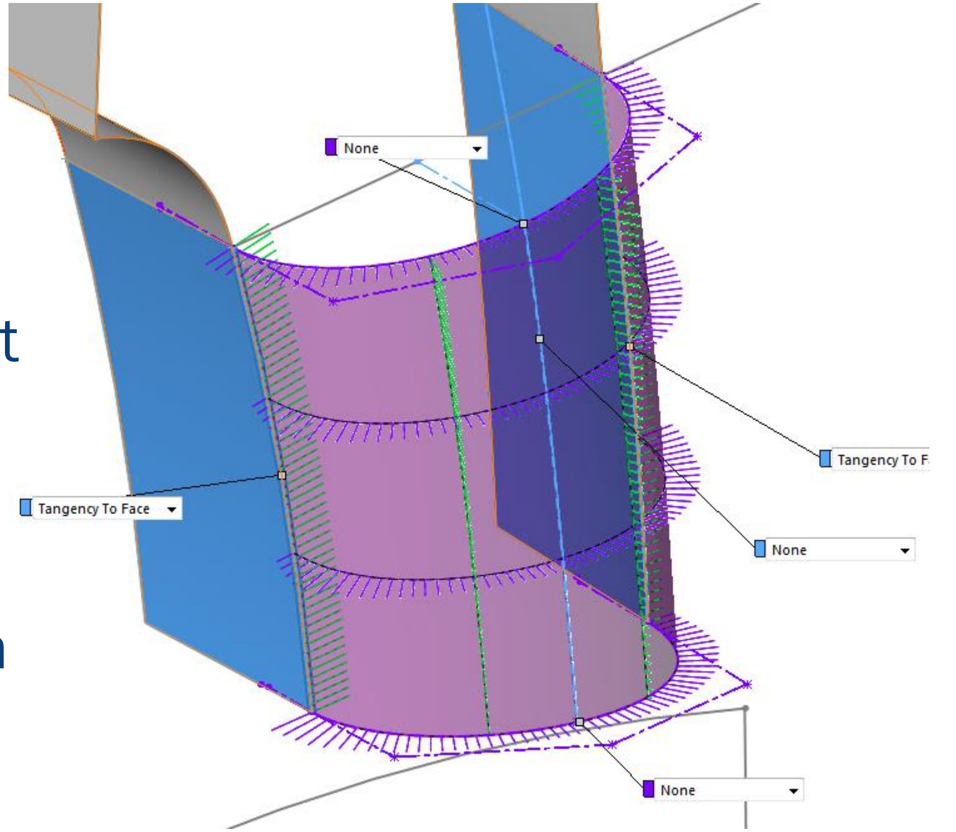
Style Splines are sketched on a series of planes to further define the shape of the handle. Once a perimeter is established, a Boundary Surface can be created.

Use the tangent face relation to ensure correct draft at parting line.



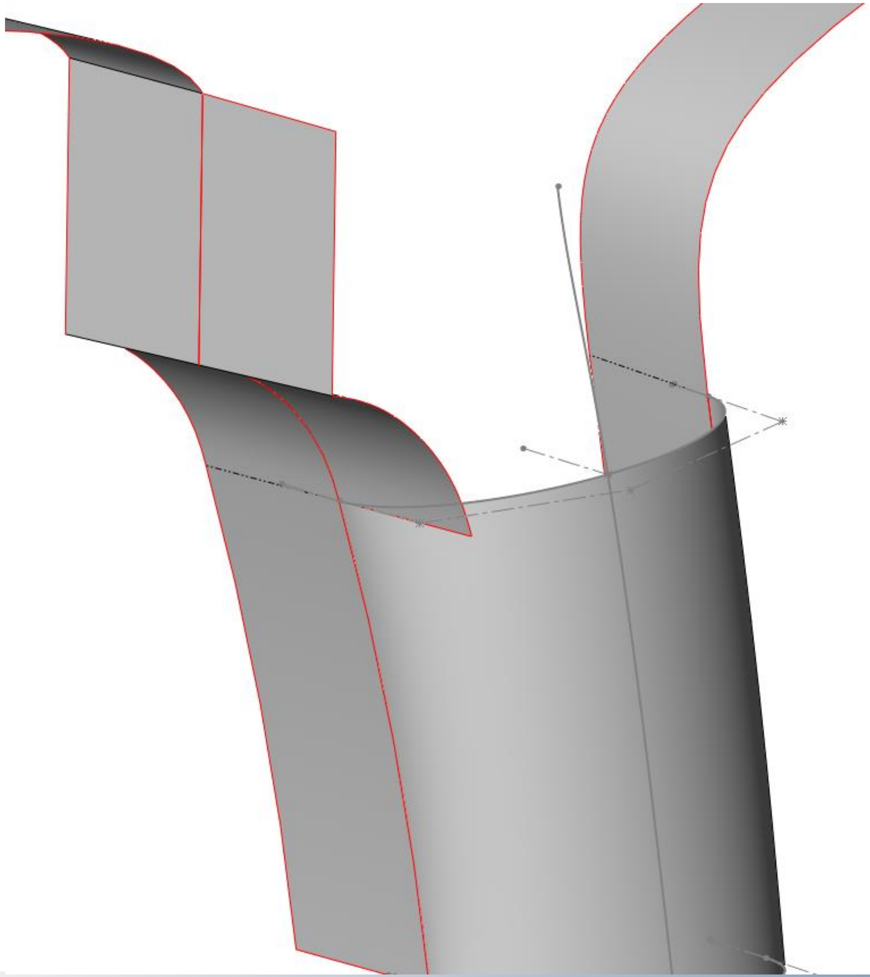
Style Splines

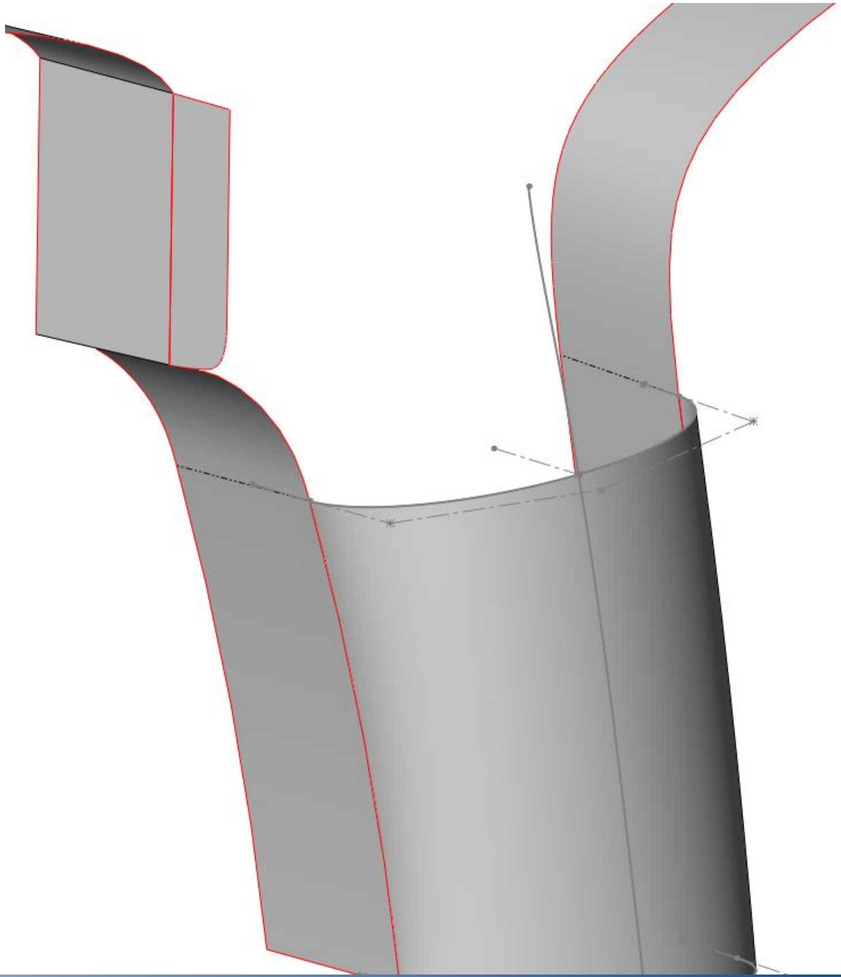
TIP: Evaluate the curvature combs of the surface before changing the tangent influence. In this example, changing from 0% only introduces a ripple in D2.



Create The Trigger Flat

A Surface Extrude with draft and a Trim complete the flat trigger area. Fillets will be added later.





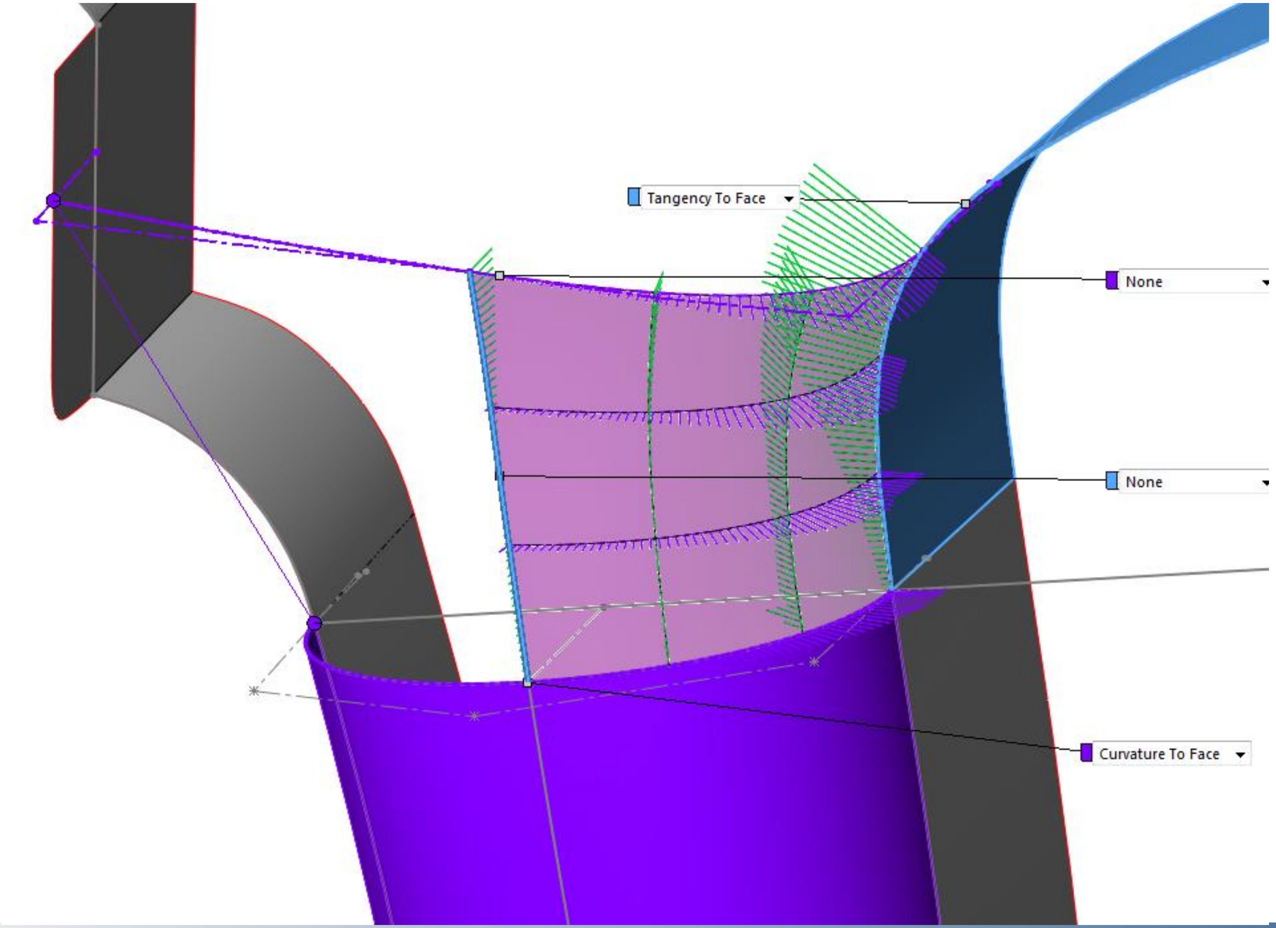
TIP:

Overbuild surfaces to virtual sharps and add fillets later.

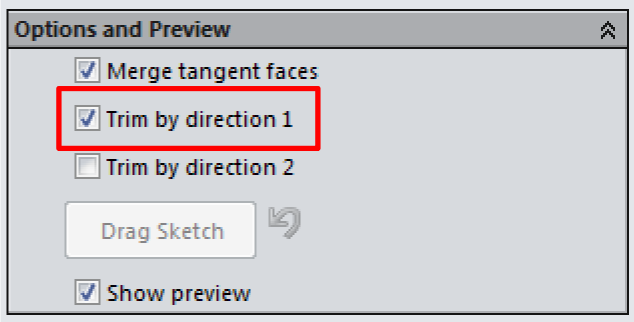
Trying to build fillet features into surfaces can be unwieldy.

Continue Building 4 Sided Surfaces

Additional Boundary Surfaces are created between existing model edges and 2D style splines sketched on planes.

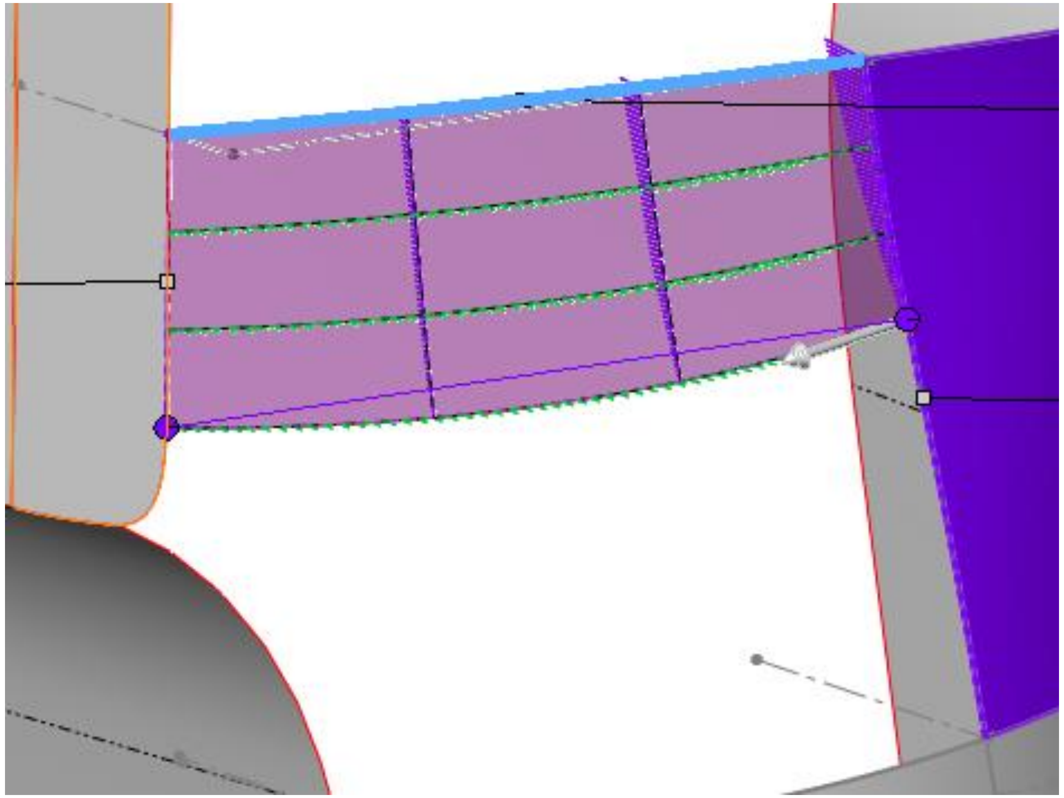


Note that the profiles in D2 extend past the first profile in D1. Enabling "Trim by Direction 1" causes the surfaces to be built only between the two profiles in D1.

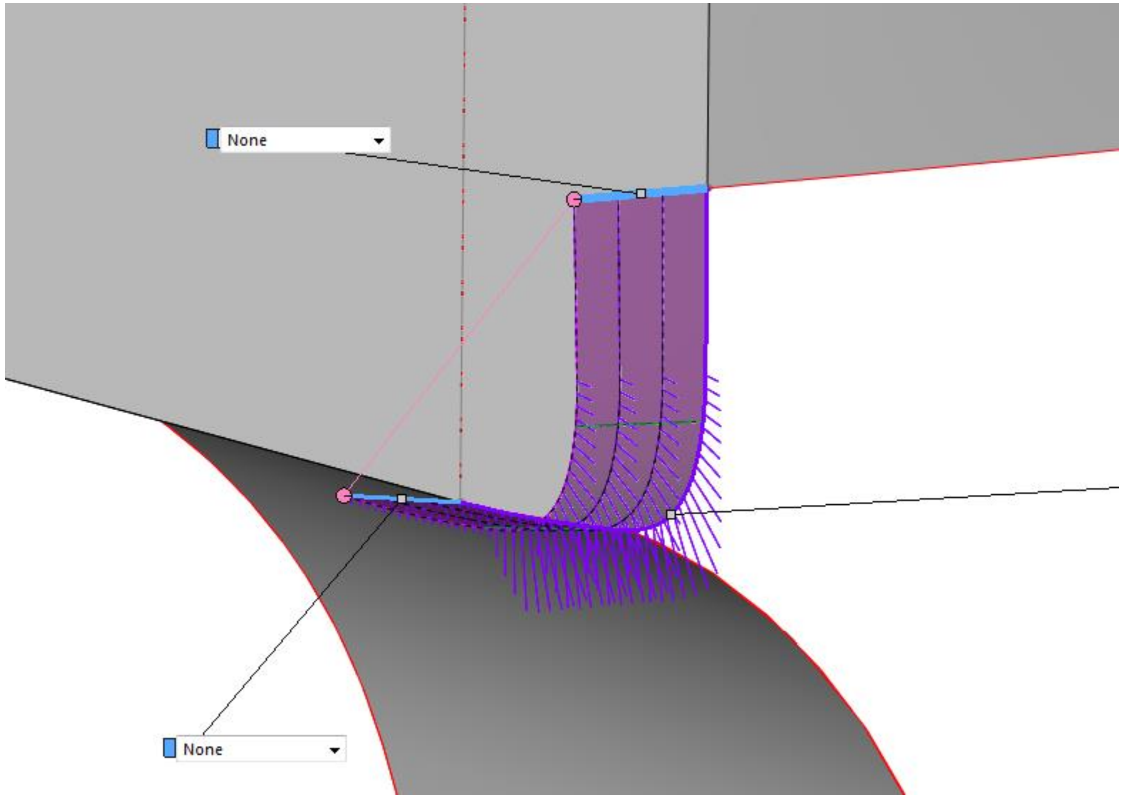


Create Guides for Surface Fill

Surface Fill works best when it can be shaped with the tangent constraint to adjacent faces.



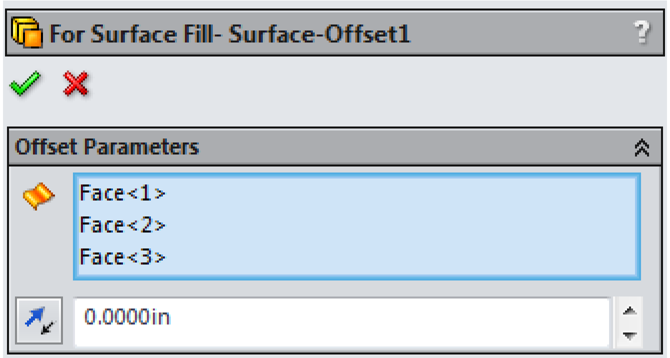
Boundary Surface is overbuilt and then trimmed back for clean curvature.

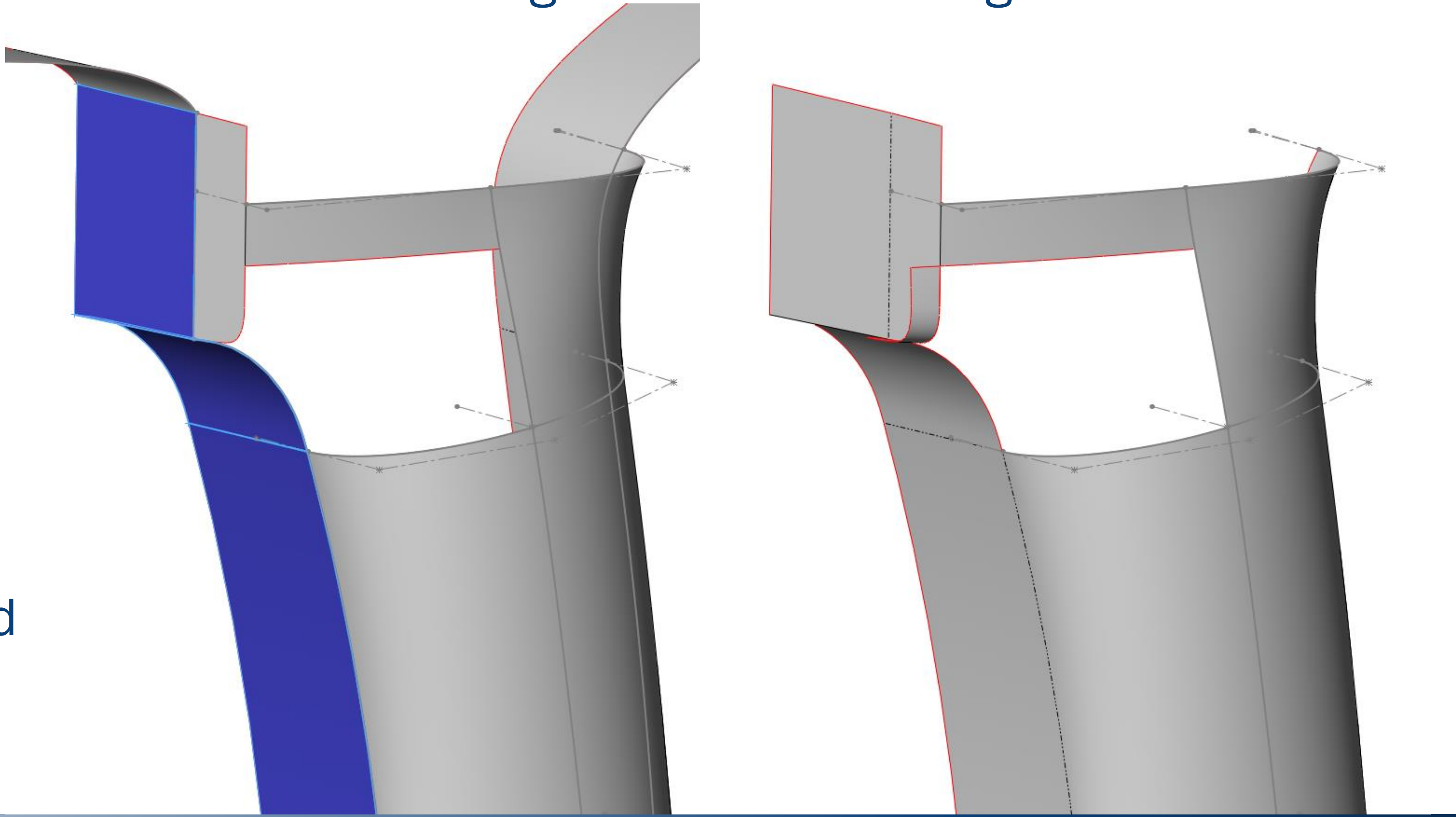


A helper surface is built from two line segments made tangent to geometry in a 3D sketch.

Create A Closed Perimeter

Surface Fill works best when it has a closed set of edges. While it will work with sets of open edges, tangency and curvature constraints can sometimes be ignored on non-knit edges.





Draft Reference surfaces are copied with a 0 distance surface offset, then knit into the model. They will be deleted after the Surface Fill.

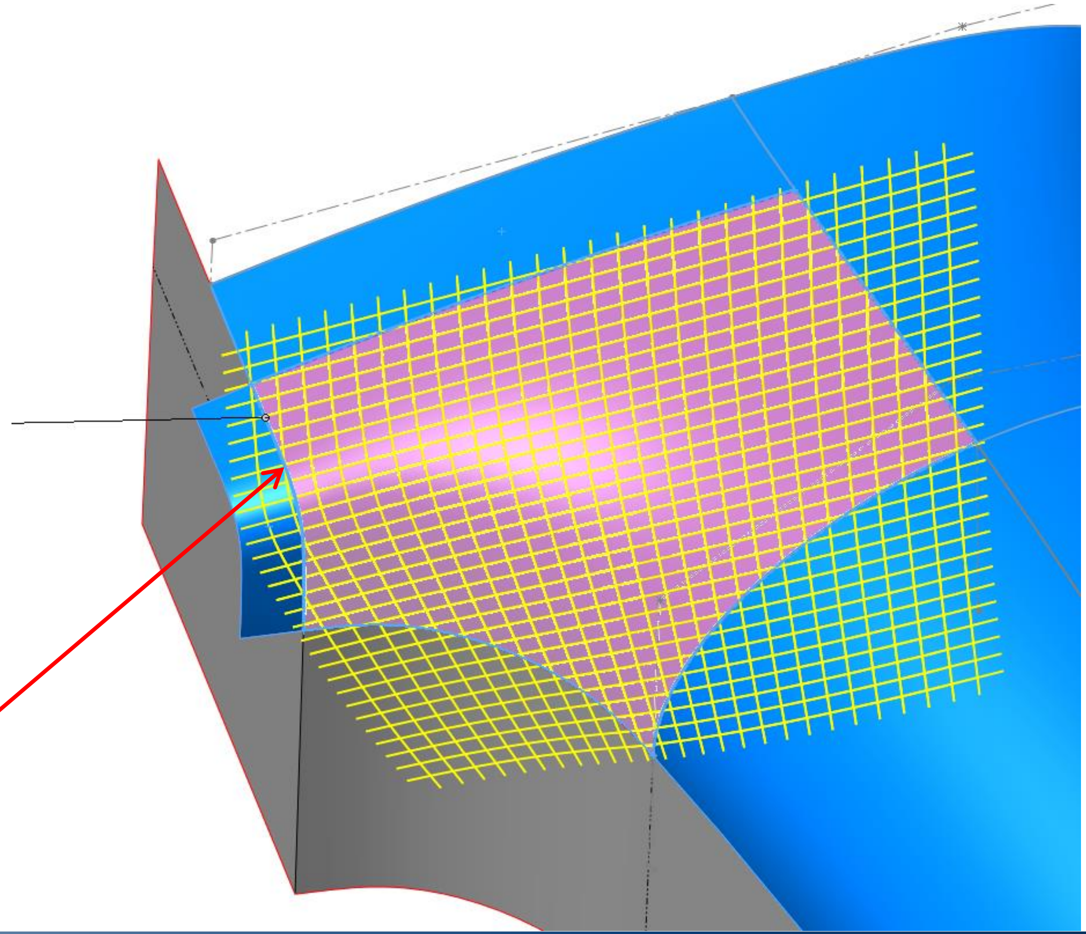
Create the Surface Fill

This 5 sided patch would be difficult to close with Boundary Surfaces. However, this area was specifically constructed to harness the power of Surface Fill.

TIP:

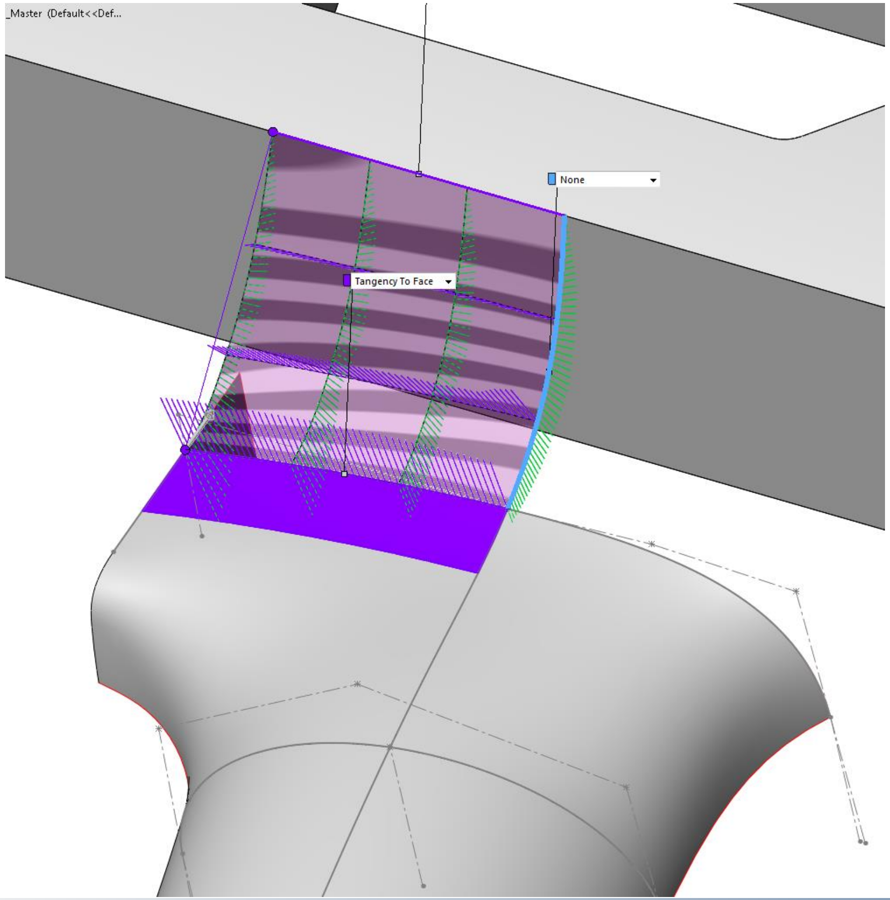
Start with tangency on all edges. Use zebra stripes to evaluate flow of surface. If required, add curvature. This example produces inferior results when curvature is applied to the 3 main edges.

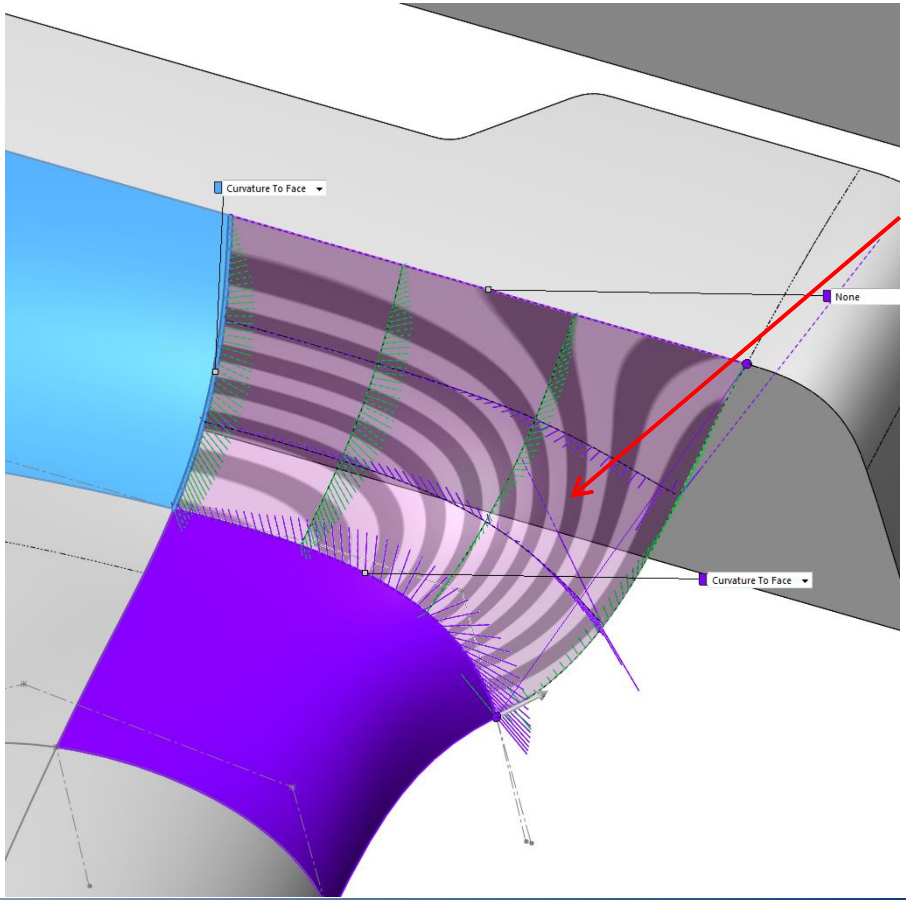
Helper surface guides the shape of the Surface fill. The fill uses tangency on all edges. Contact on the front edge causes an undesirable dip.



Build the Handle Blend

Start constructing the handle blend from additional Boundary Surfaces in the same way the handle was built.

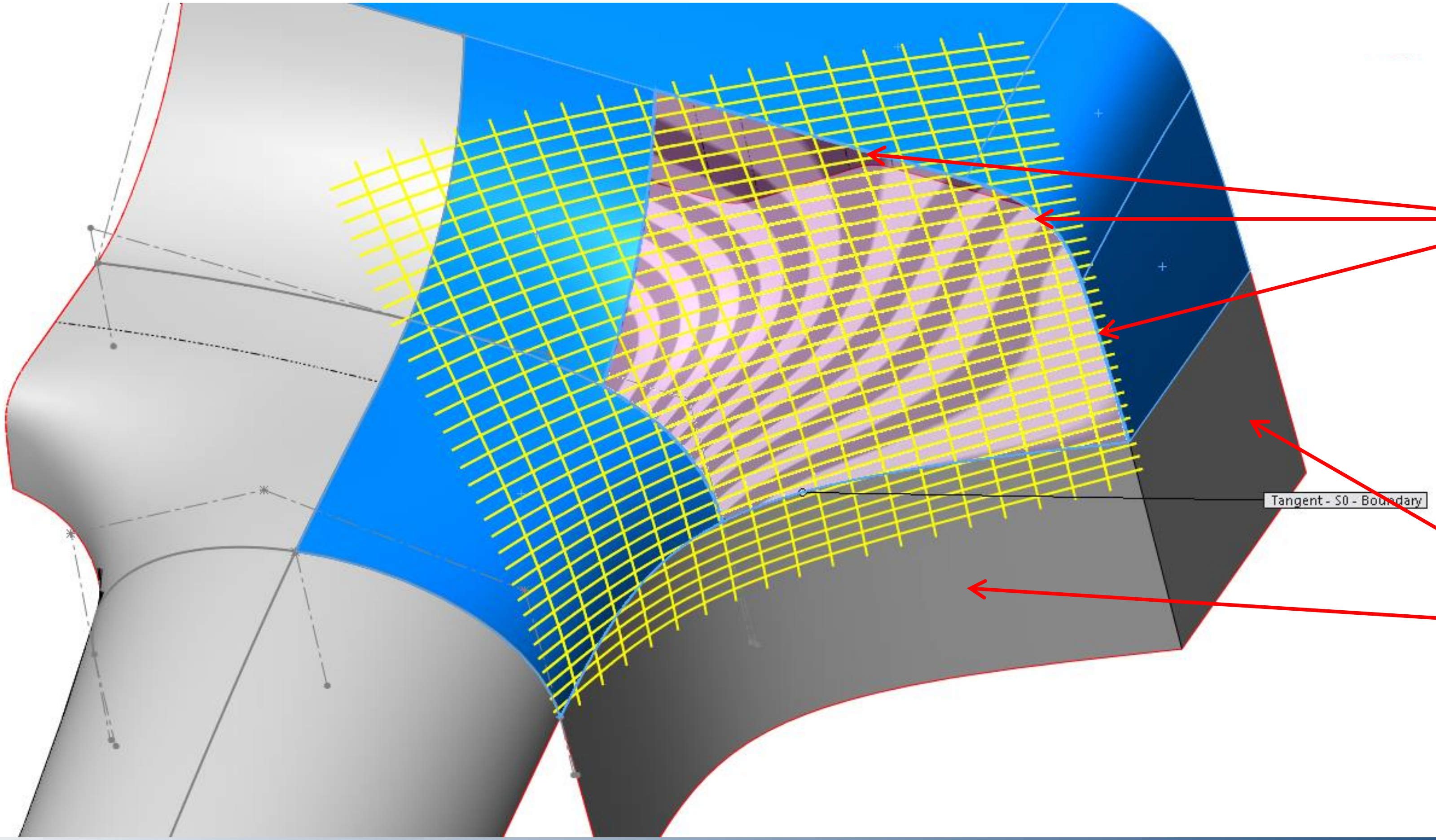




This surface is overbuilt and trimmed back. A Surface Fill will complete the remaining 5 sided patch.

Complete the Blend

A Surface Fill completes the blend. Surface Fill excels at getting out of tough modeling situations. I intentionally try to create 5 sided areas in tricky modeling situations just to use Surface Fill.

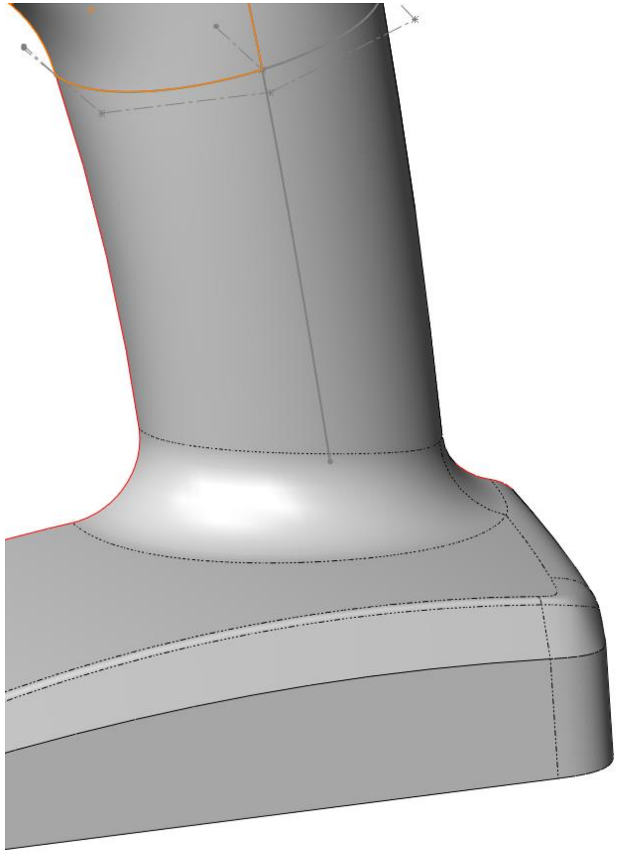


These edges are contact.

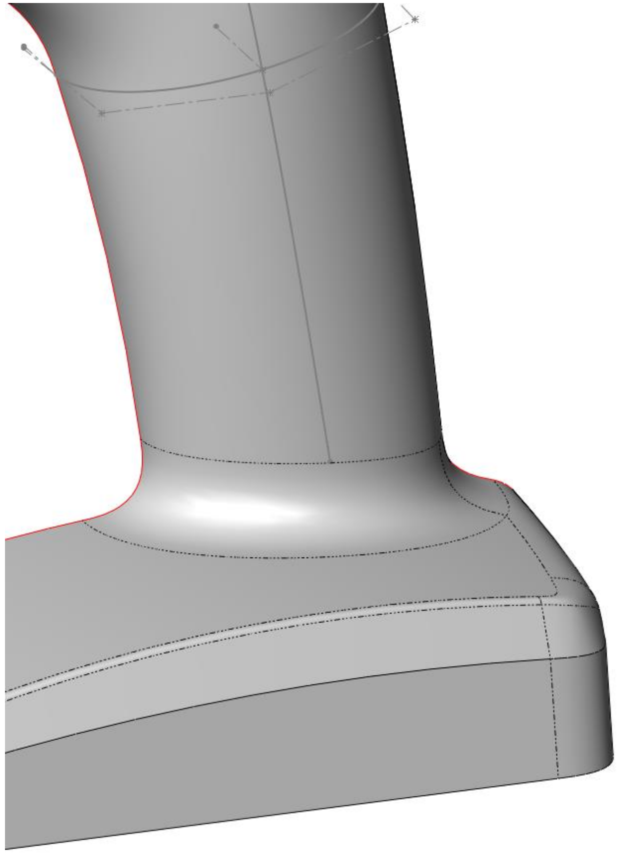
0 offset draft reference surfaces knitted into the model.

Blend Instead of Fillet

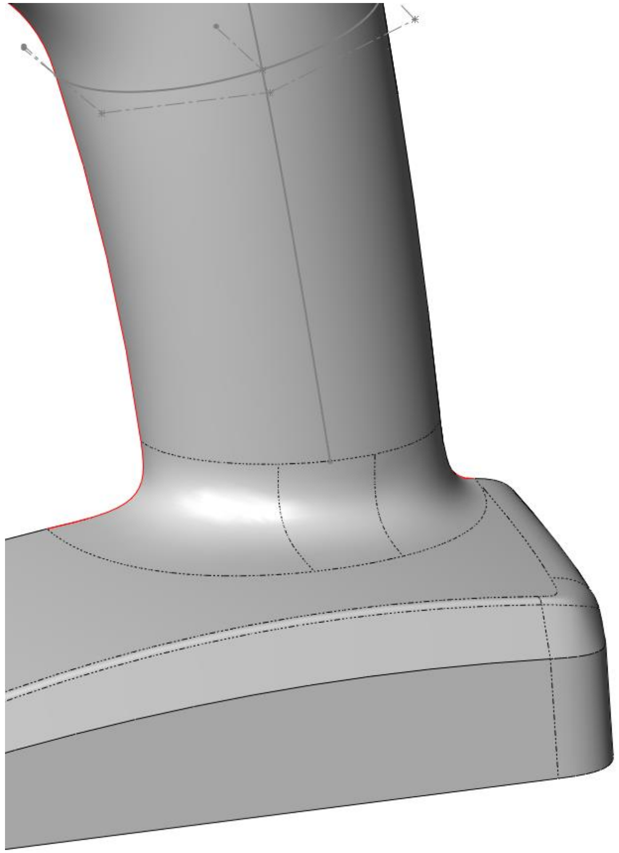
The fillet tool can sometimes yield satisfactory results. Building a blend manually always produces excellent results.



Constant Radius



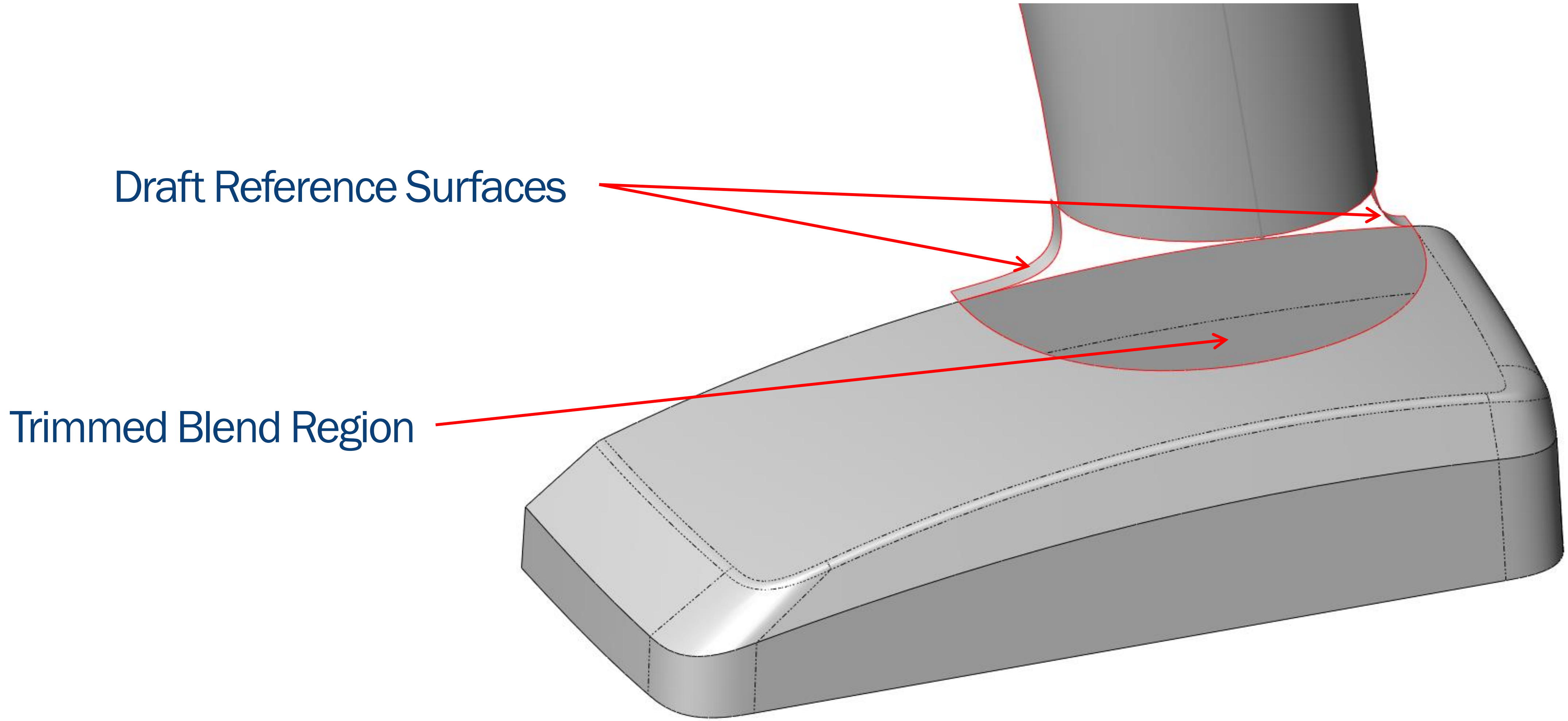
C2 Chord Width



Surfaces Blend

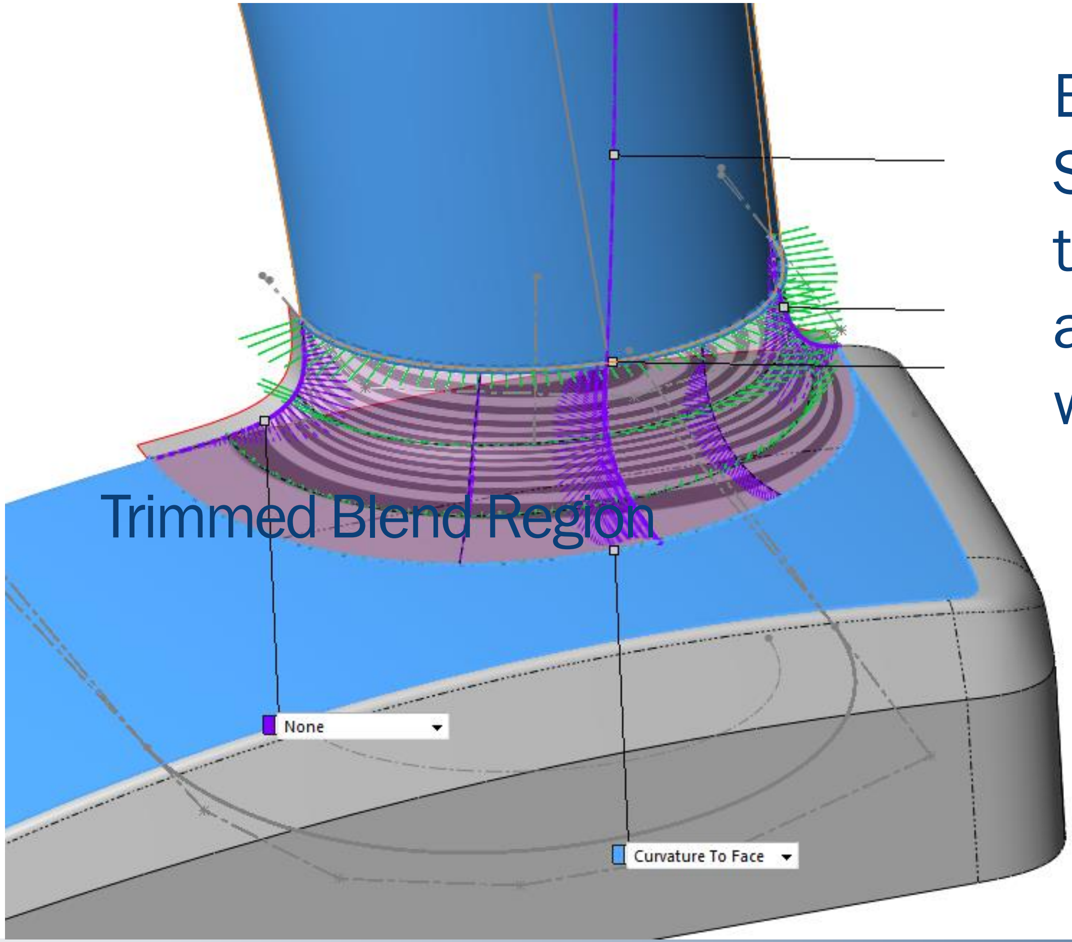
Trim Out the Blend Region

Create an area for the blend using the trim tool, or a split line and delete face.

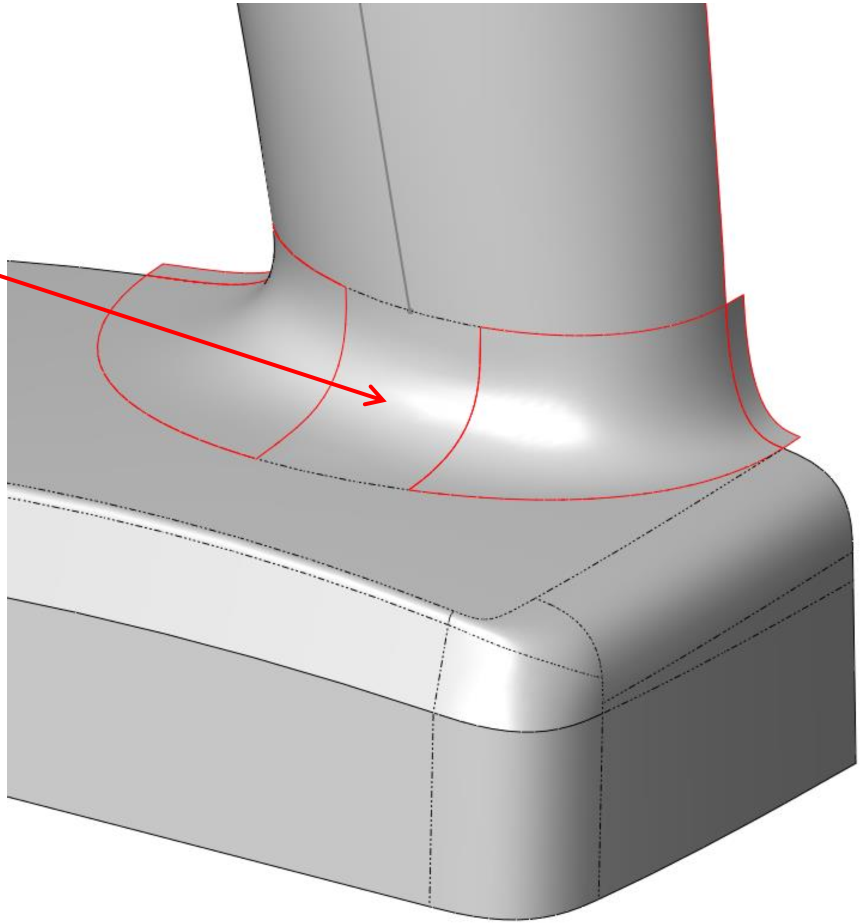


Complete the Blend

Complete the blend with a Boundary Surface. If the results aren't satisfactory, trim out portions of the Boundary Surface and Replace them with Surface Fills.

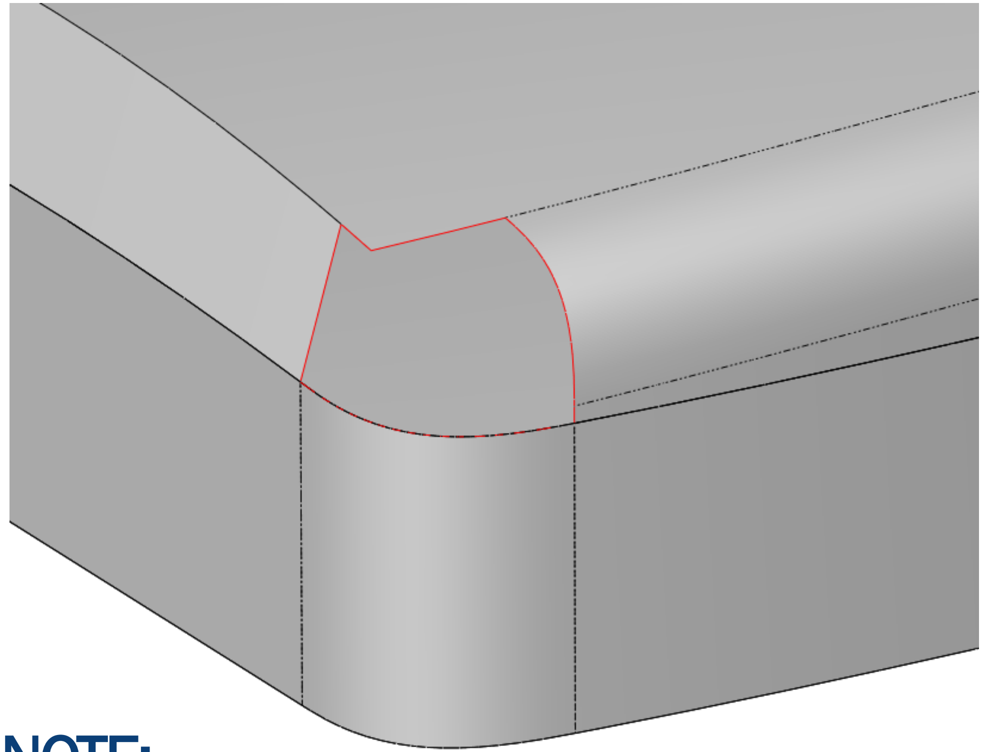


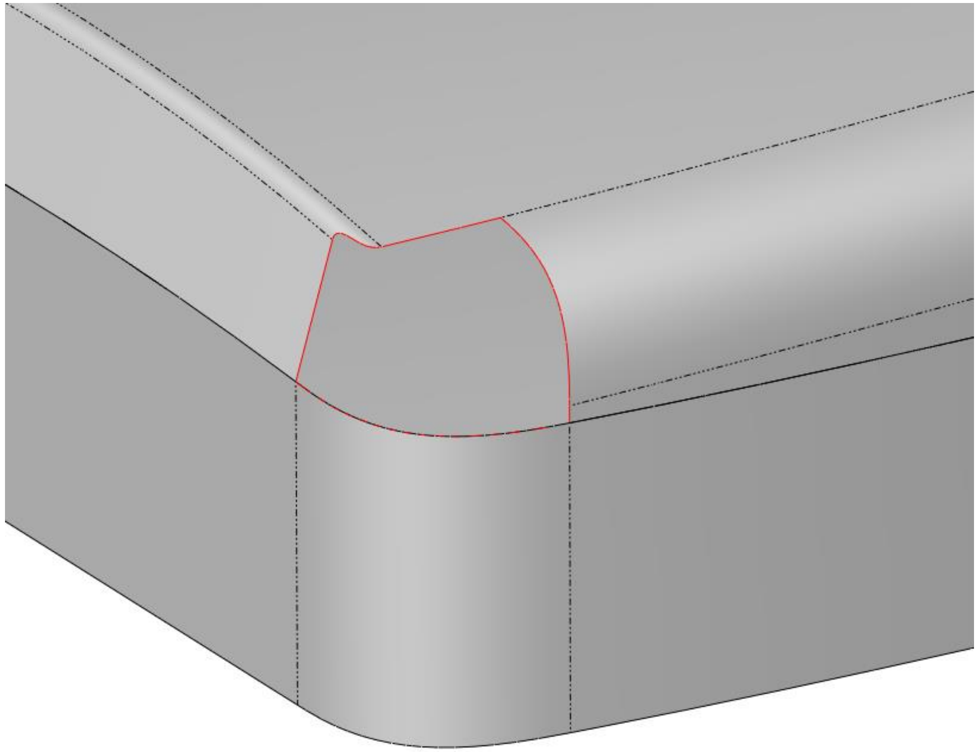
Boundary Surface is trimmed back and replaced with Surface Fills



Fillet Washout

The corner is trimmed out of the model. This allows a chord width face fillet to complete.



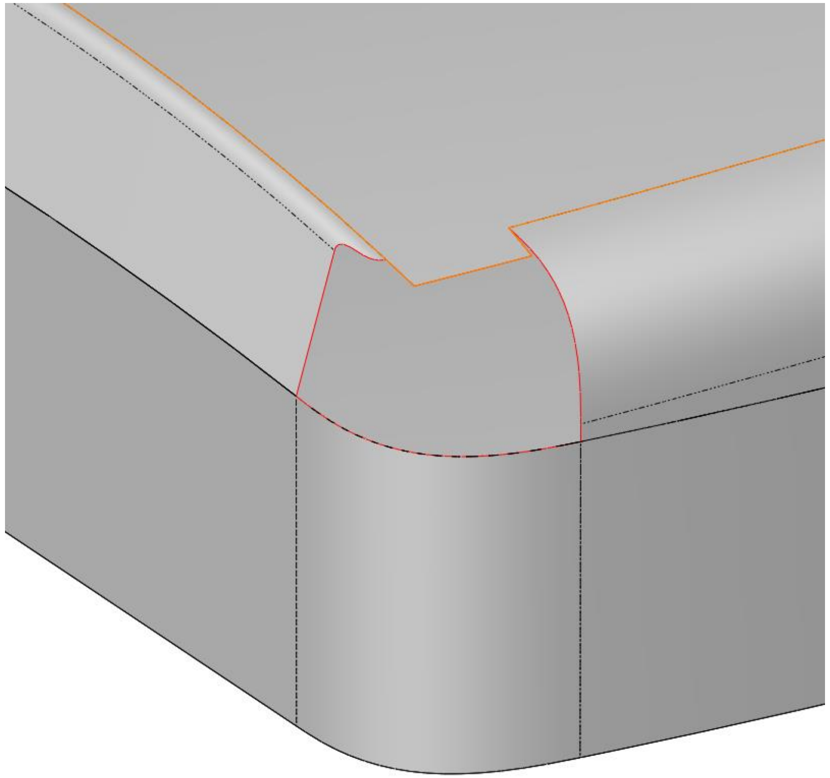


NOTE:

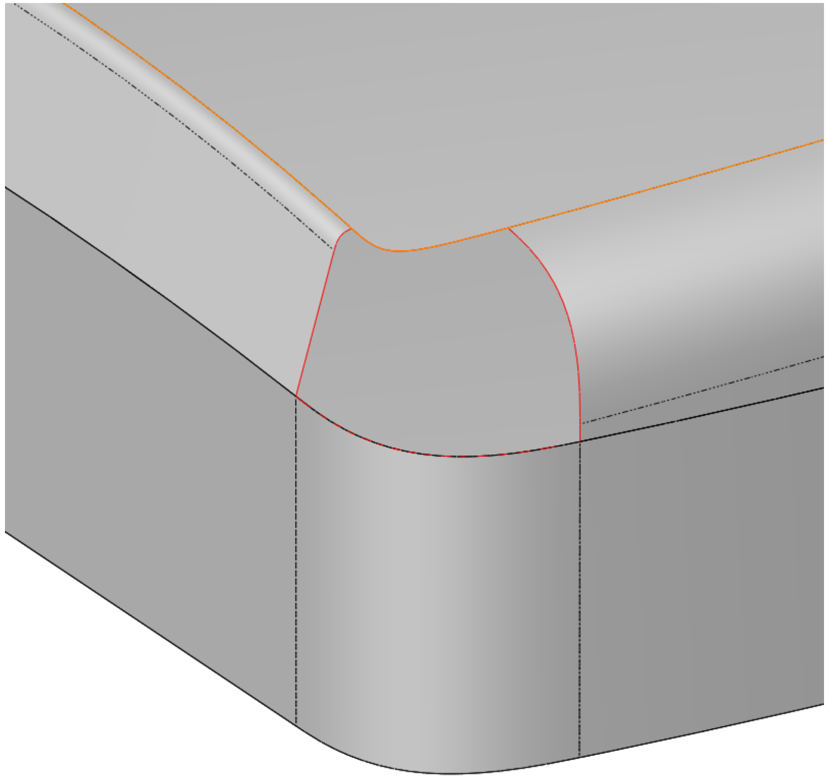
Chord width fillets have trouble terminating to zero. Standard radius fillets usually can complete in this situation. Trimming or cutting away parts of model allows these fillets to complete.

Fillet Washout

The blend region is not large enough for the fillet washout. The surface edges are extended and then trimmed back to the desired blend region.

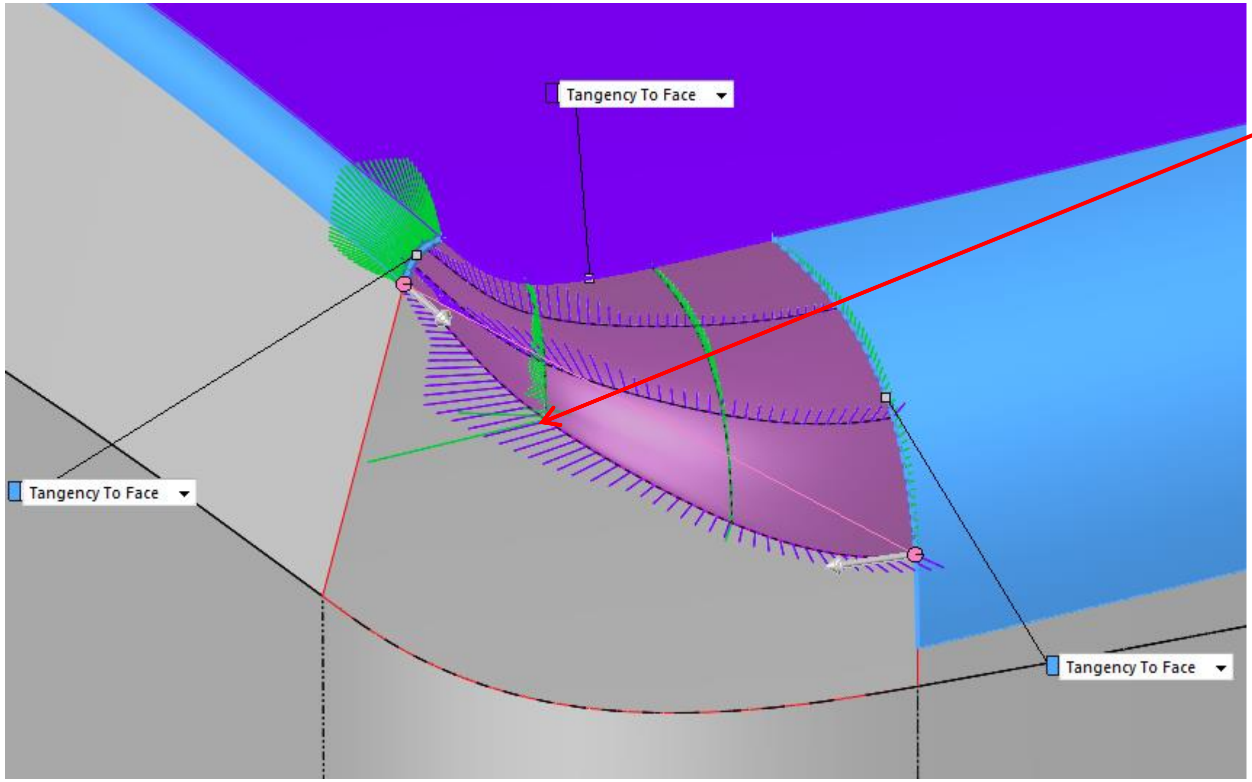


Edges are extended and then trimmed back to complete the blend region.



Fillet Washout

With the blend region created, a new surface can be created flaring the small fillet out into the larger fillet.



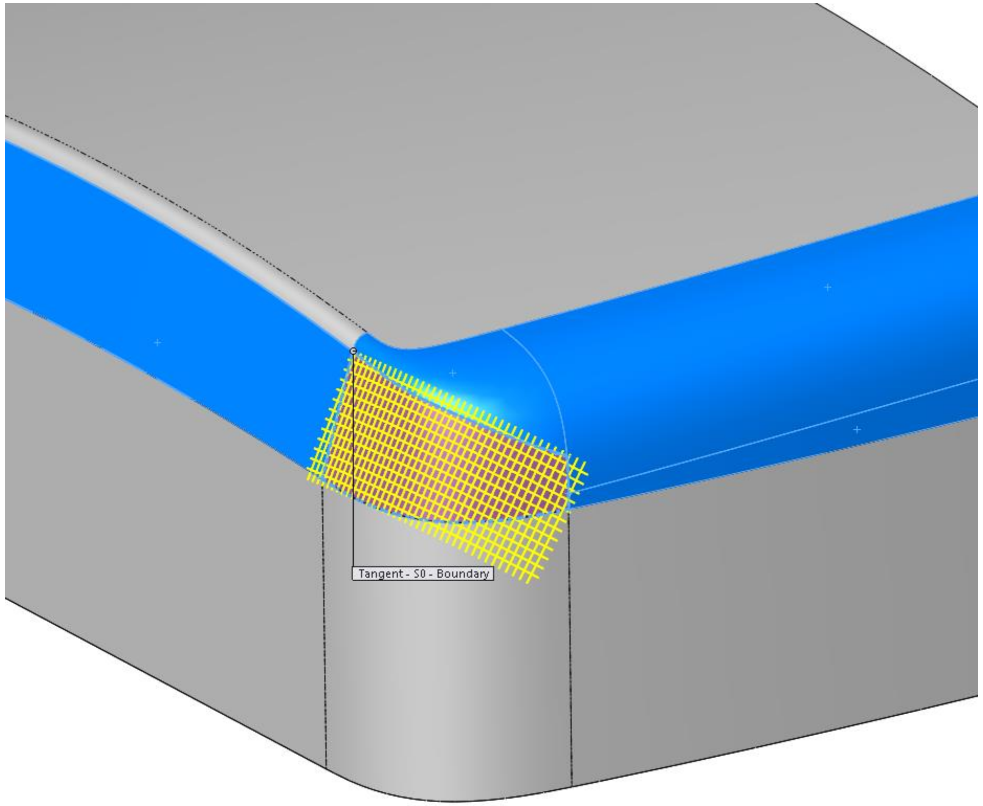
NOTE:

Boundary Surfaces with two profiles in one direction and one profile in the other are susceptible to having poor curvature along the unsupported edge.

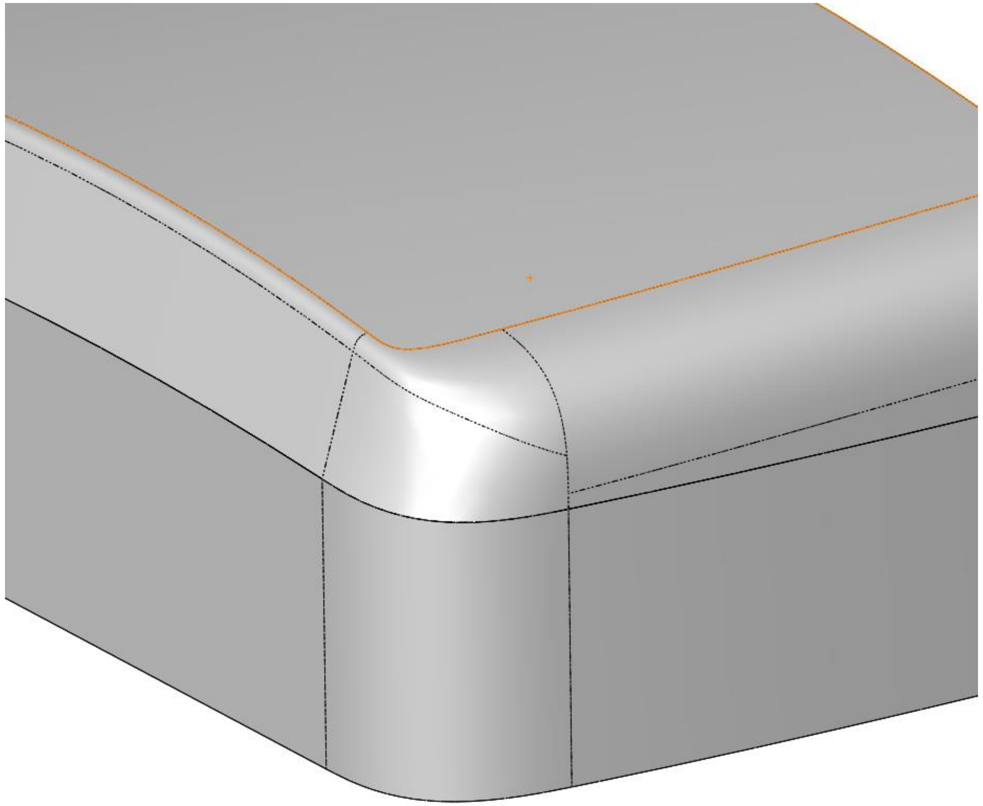
This can be fixed by trimming away a portion of the surface.

Fillet Washout

Now that the small fillet has been blended into the large fillet, a Surface Fill can be used to complete the transition.



Surface Fill



Finished Transition

Overmold

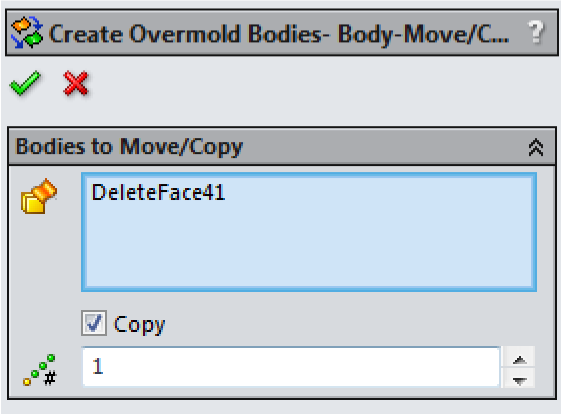
Regions of the flashlight will be overmolded with a TPE elastomer. Overmold on the handle will be used for increased grip and ergonomics. Overmolding on the main housing provides opportunity for a color break.

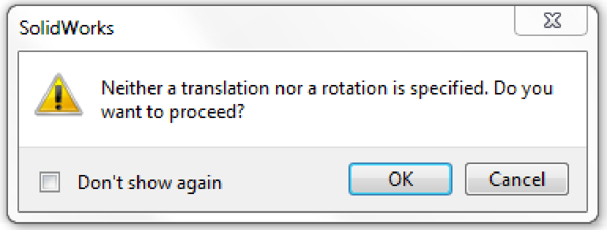
The overmolding process involves first molding the substrate (the rigid housing). The part is then placed into another tool where additional resin will be shot. The substrate requires a recess to account for the depth of the overmold.

A “gutter” is added to the overmold to help the tool shut off against the part and to limit flash.

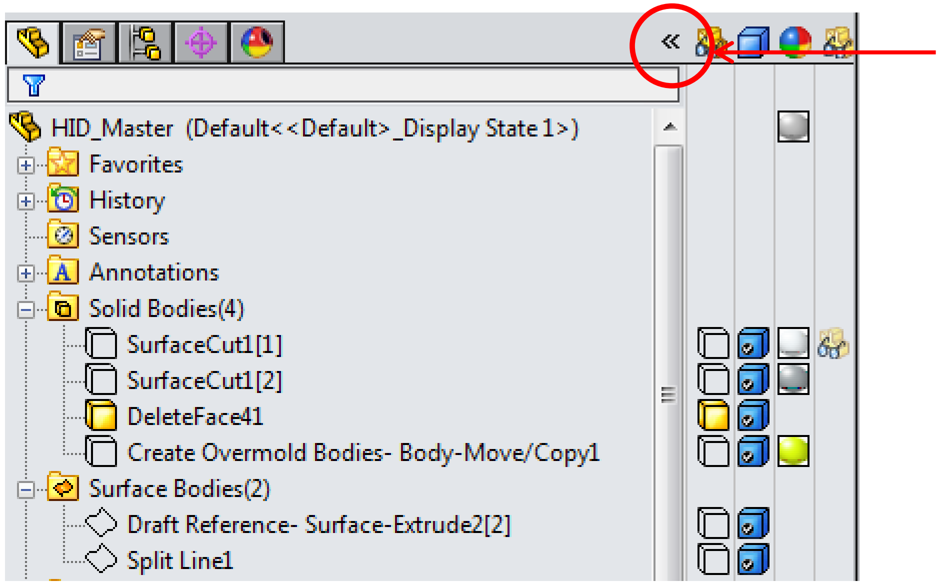
Move/ Copy Bodies

The overmold will be molded with the use of tool bodies (bodies used to subtract or add material to the main body). The first tool body is created with Move/ Copy Bodies





This error message can be ignored.

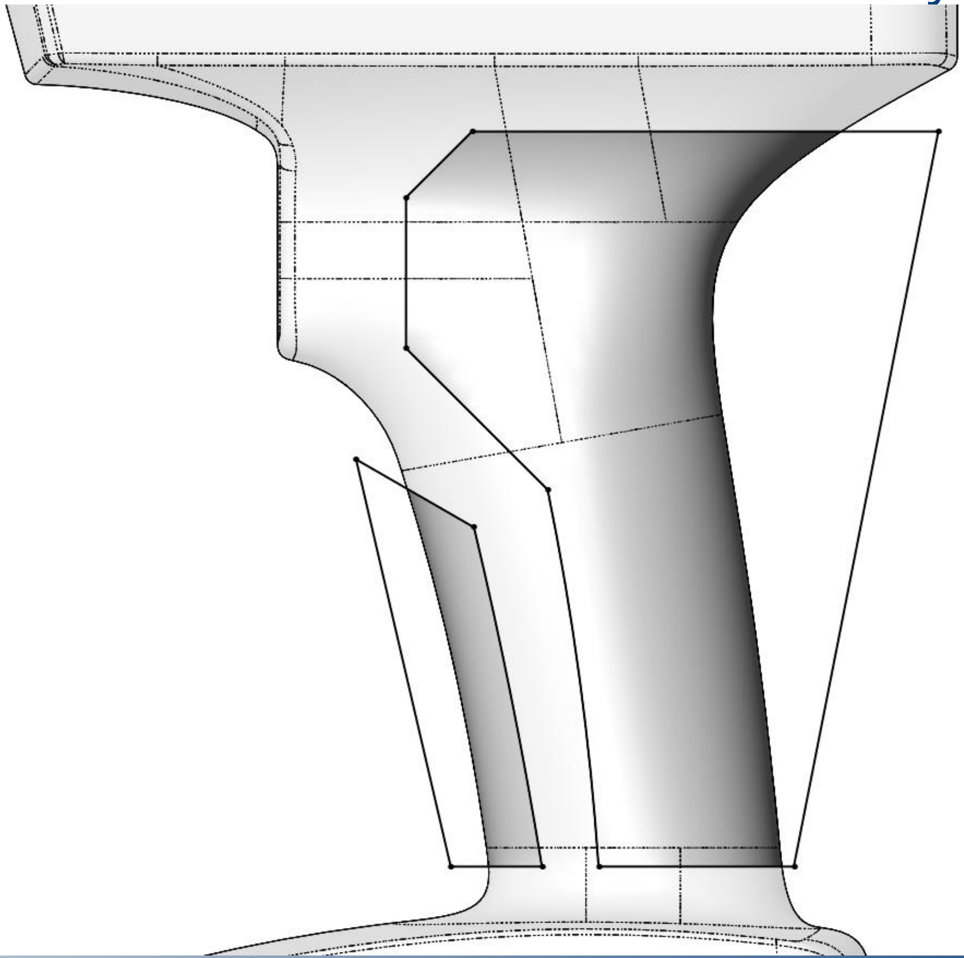


Show the Display Pane flyout by clicking the arrow.

TIP: When creating overmolds, numerous bodies will be used. Using the display pane makes hiding and showing the various bodies easier. Always use the copied body as the tool body.

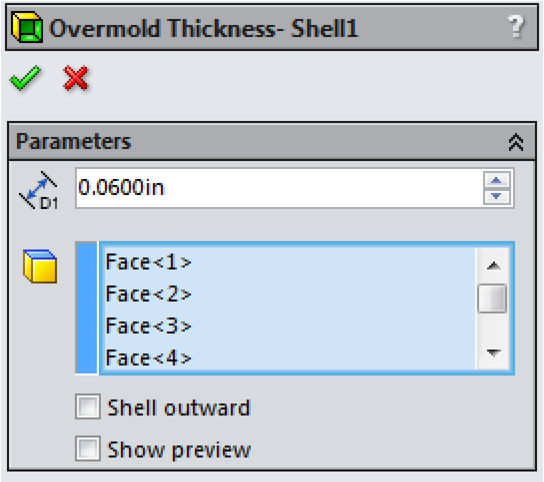
Define the Overmold Region

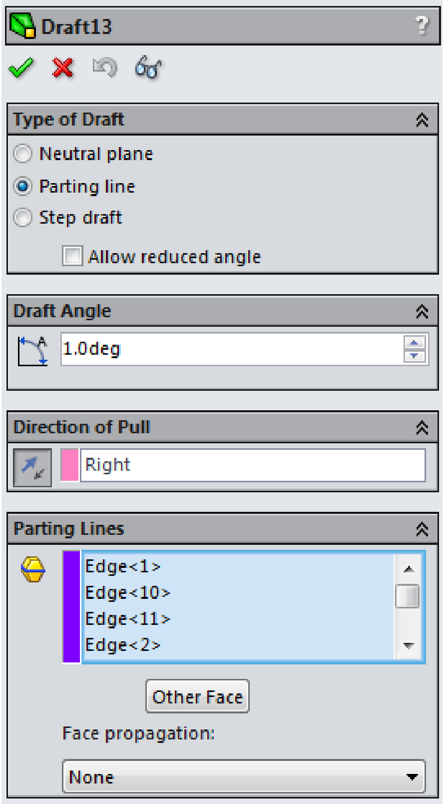
The parts of the model that will not be overmolded are cut away from the model using a cut-extrude. The sketch profile is converted from the overmold layout sketch.

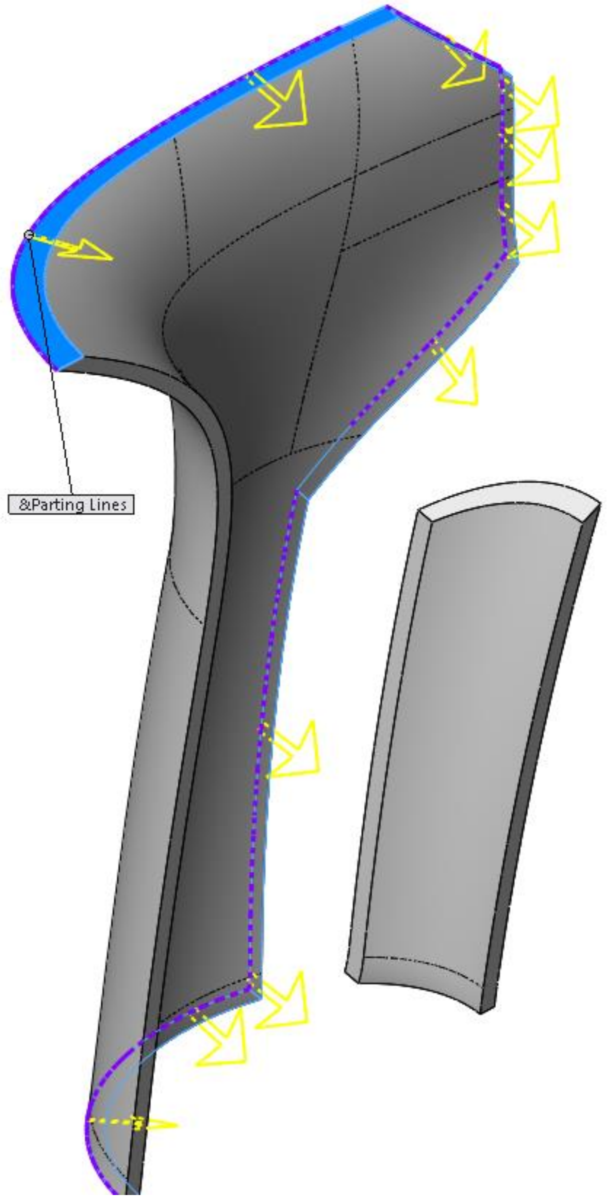


Define the Overmold Depth and Draft

Regions of overmold are now shelled to the desired overmold depth. The edges of the overmold are drafted. Remember that we are actually creating a pocket in the housing;add draft accordingly.

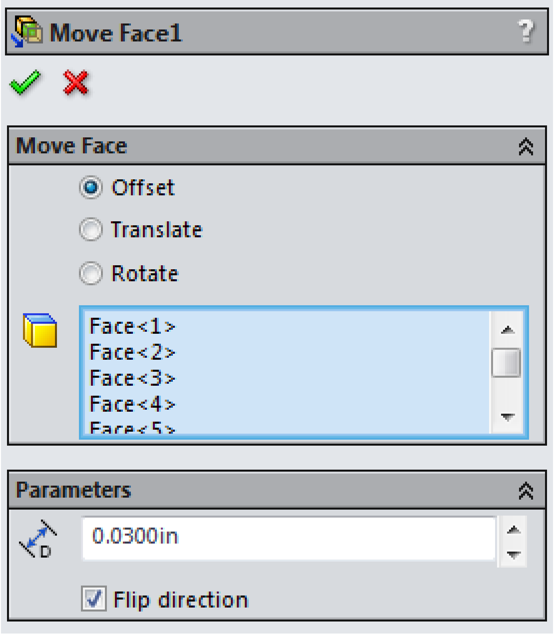


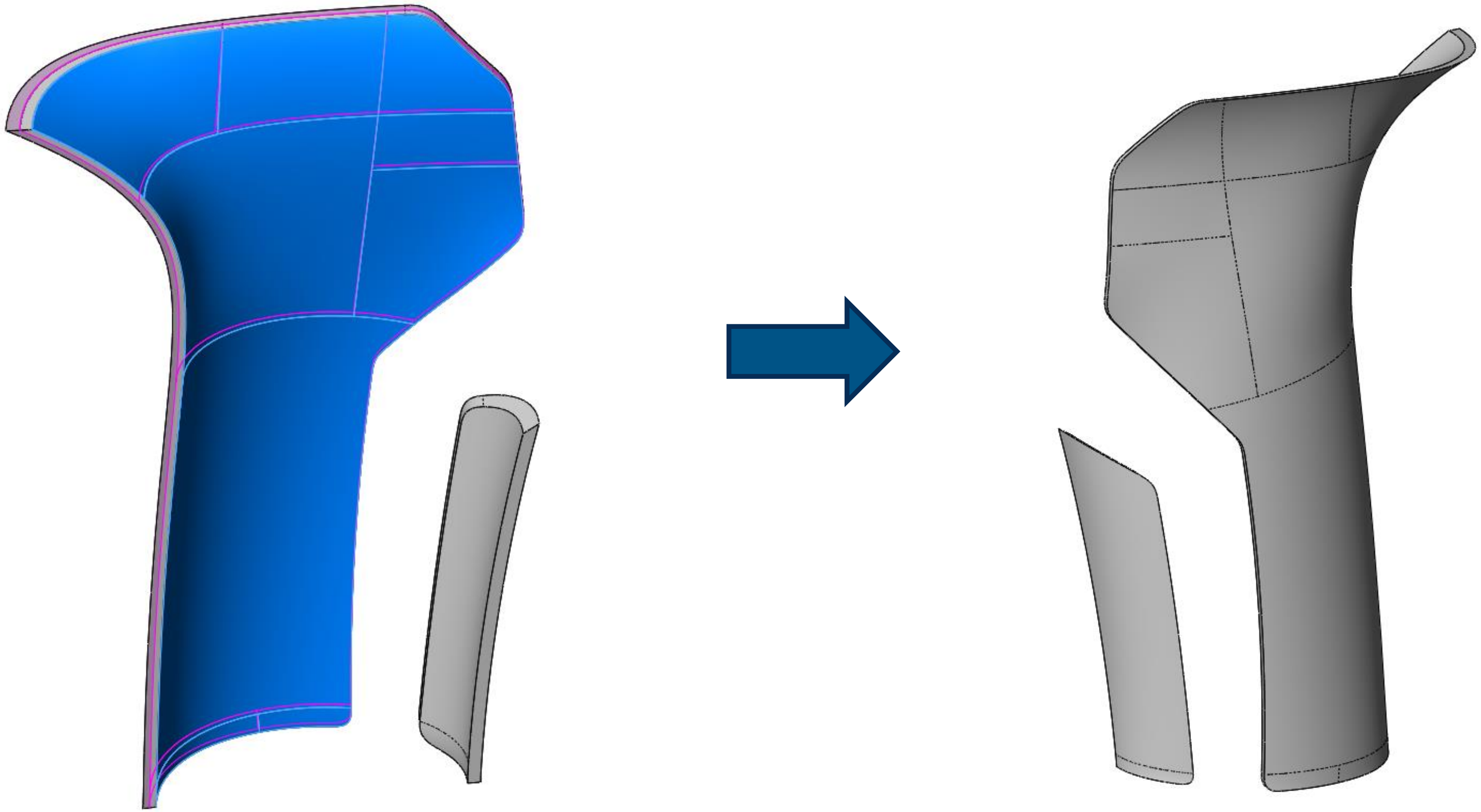




Create the Gutter Tool Bodies

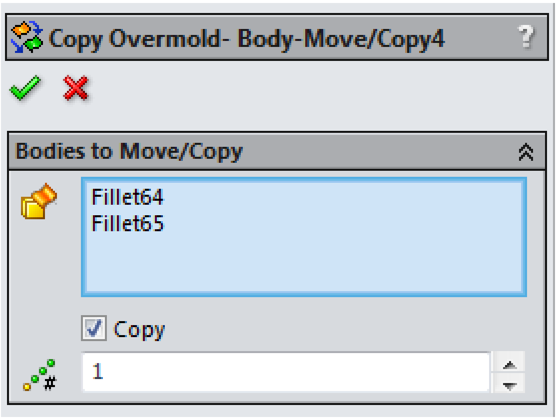
The overmold bodies are copied again. Move face is used to reduce the wall thickness from 0.06” to 0.03”. A cut extrude from an offset sketch creates the gutter tool body.

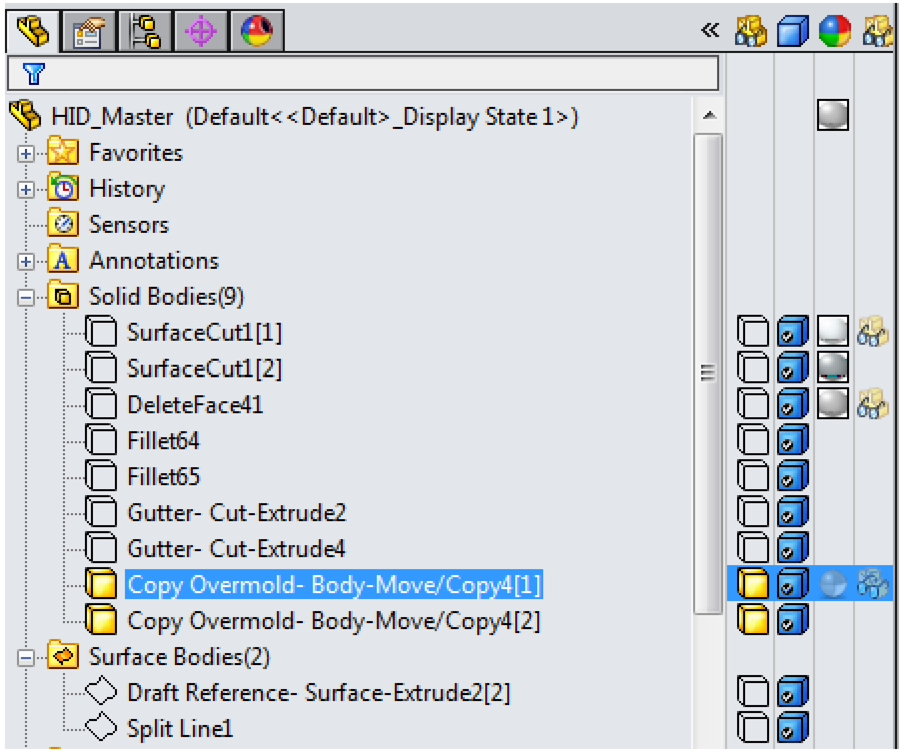




Create the Overmold Recess Tool Bodies

The overmold bodies before creating the gutters are copied. These will be used to create the recesses in the main part (substrate).



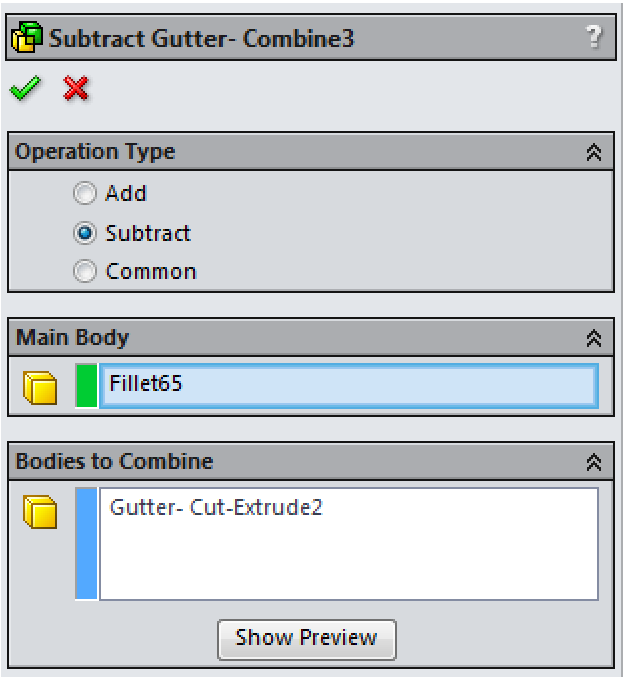


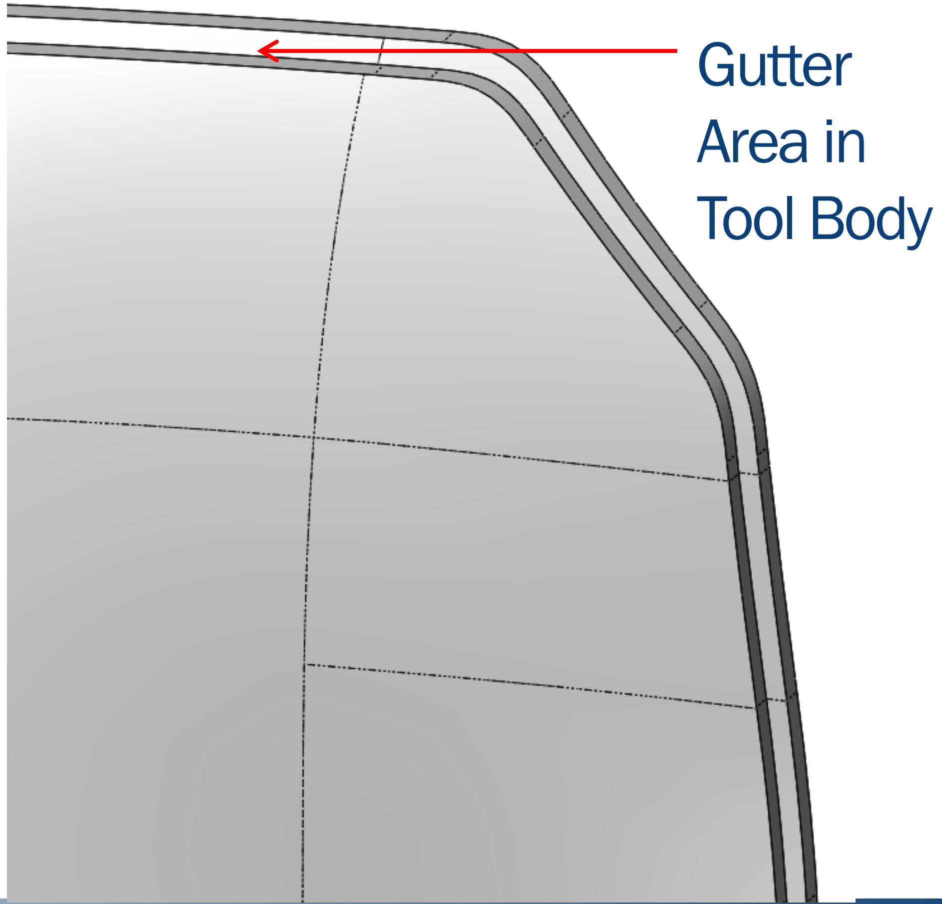
TIP:

The usefulness of the display pane becomes apparent when working with multiple bodies required for creating overmolds.

Create the Overmold Tool Body

The gutter tool body is combined with the overmold tool body.



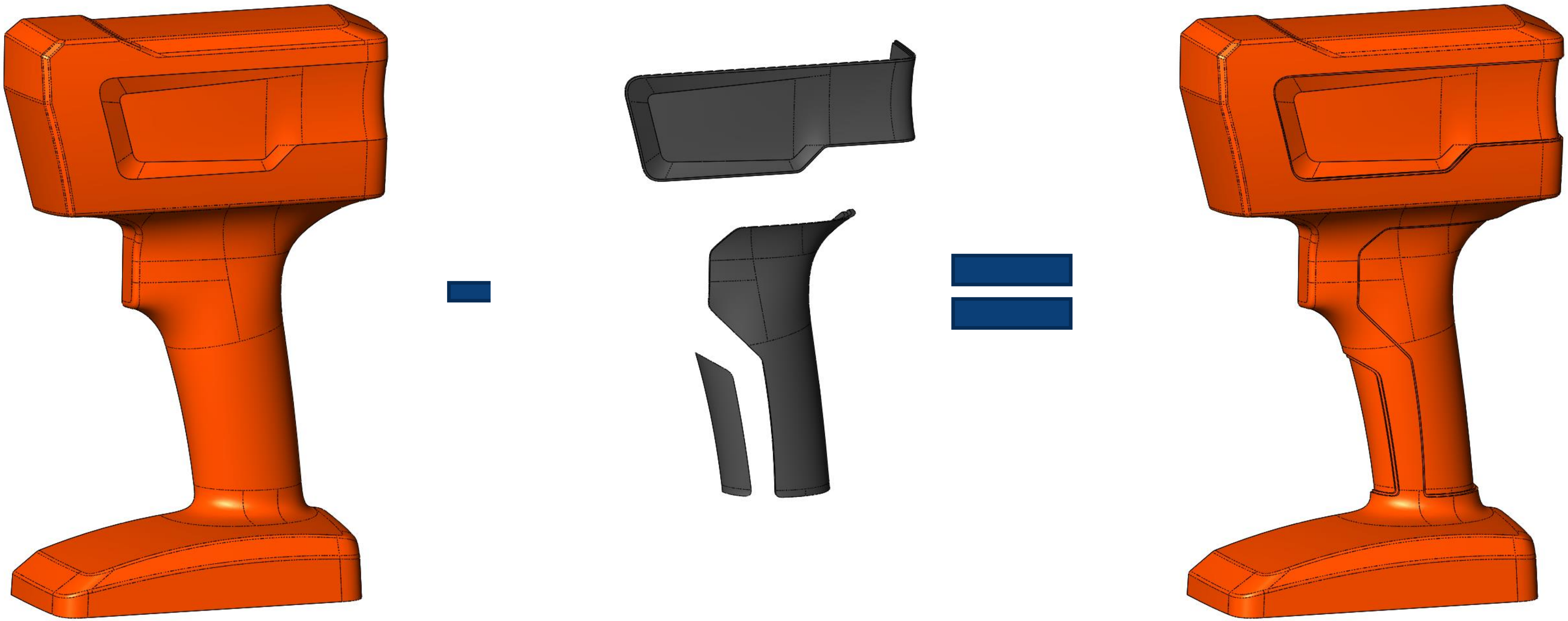


TIP:

Working with tool bodies can simplify complicated modeling situations. Model the material to be removed, instead of trying to model the cut.

Subtract the Overmold

The previously copied overmold bodies are subtracted from the main housing (substrate) creating the overmold recess.



The Finished Flashlight

With knowledge of advanced modeling techniques and strategies, complicated designs can be reduced to their component parts. You don't have to keep banging your head!

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QUESTIONS?

PRESENTATION AND SAMPLE FILES WILL BE AVAILABLE ON OUR WEBSITE SOON.

Andrew Lowe
Industrial Designer
DiMonte Group Inc

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