SolidWorks Learning Notes [Sketch]

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1/10/2018

I don't claim to have made everything here (but **mostly** it was me). Because this is for personal use only and **not** for sale, and to save time, I didn't reference any original author, when/if applicable. Also, this is a random collection of personal notes I take when I learn something FEA/Solidworks-related, so do not expect to understand what is here. It is written in very colloquial language.

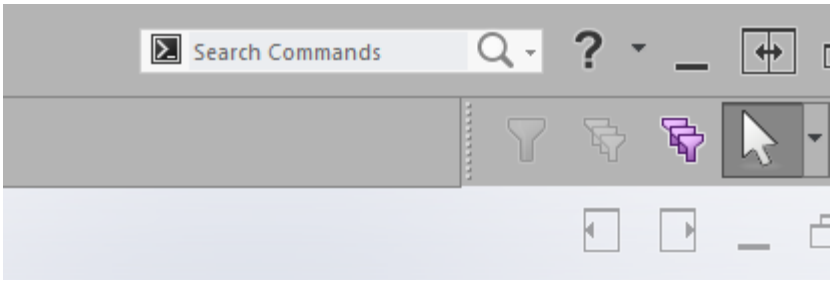
General Tips

Being faster

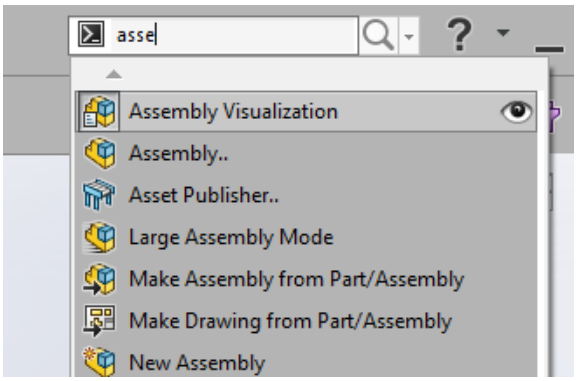
Watch some Starcraft 1 or 2 pro players and get inspired to be as fast for Solidworks (whenever speed is the bottleneck).

Quick Go to commands

Put a single key to go to the help bar and switch to commands, like this:



This eliminates the need to memorize where everything is, and mouse movement.



Selection

Select things by left dragging the mouse (just like pro RTS players do to select units/buildings) instead of clicking.

Action

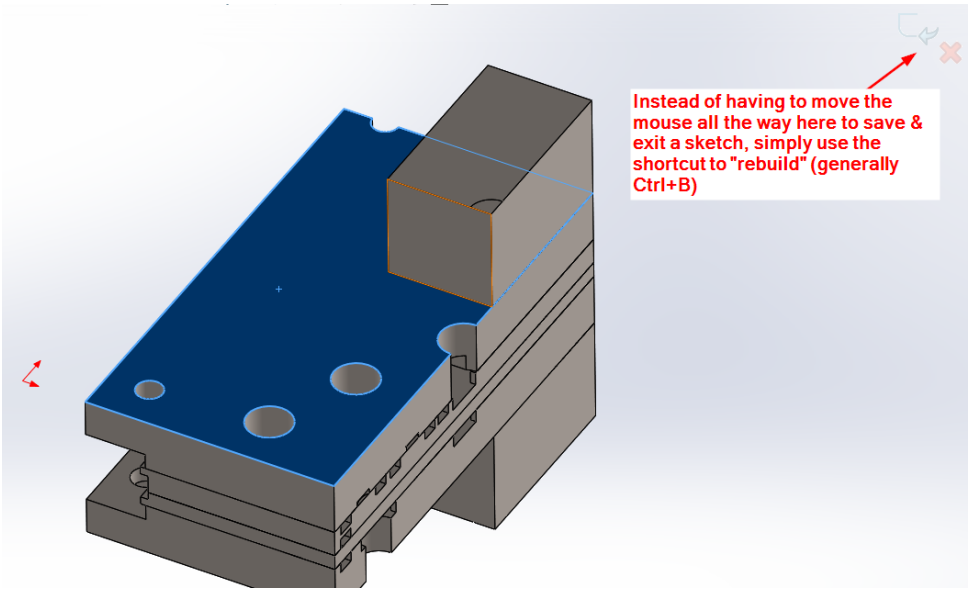
Suggestion for mouse gestures

Left-side: add "real" things
Right-side: remove things (trim) or add imaginary (centreline) or ideal things (circle)

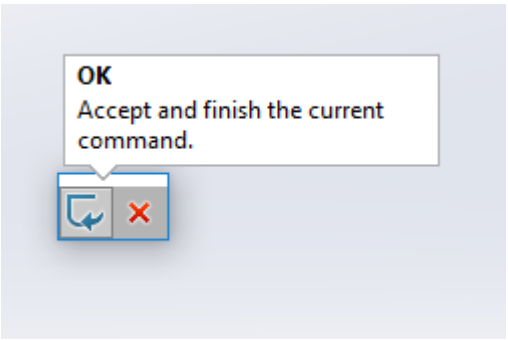
Quick feature

Click on feature first (even before making the sketch). This reduces the number of clicks (no need to click to make a sketch).

Quick sketch exiting



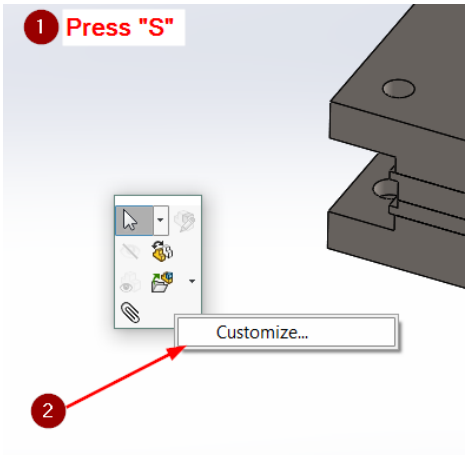
Or press D and click OK to save and exit the sketch.



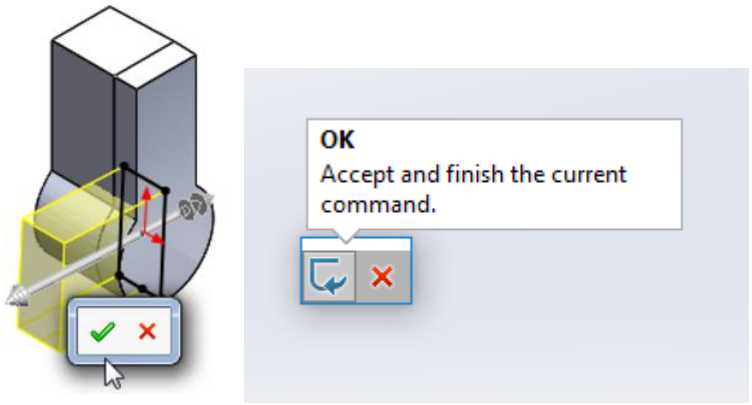
Must Know Shortcut Keys

"S" - This reduces the movement distance you need to make with your mouse, therefore increasing speed and actions per minute.

"W" - to go to the command bar and search for any other commands. Or use "S" for the same function.



“D” - to accept or cancel features, and to accept sketches



“Spacebar” – to show views

“Z” – as zoom selection

“F” – as zoom to fit

“Ctrl + B” – Rebuild

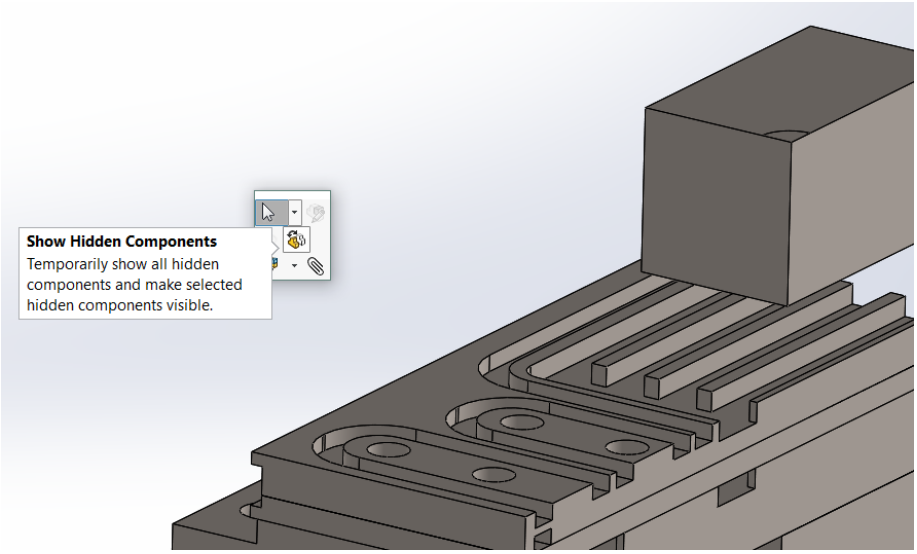
Quick show and hide

To hide:

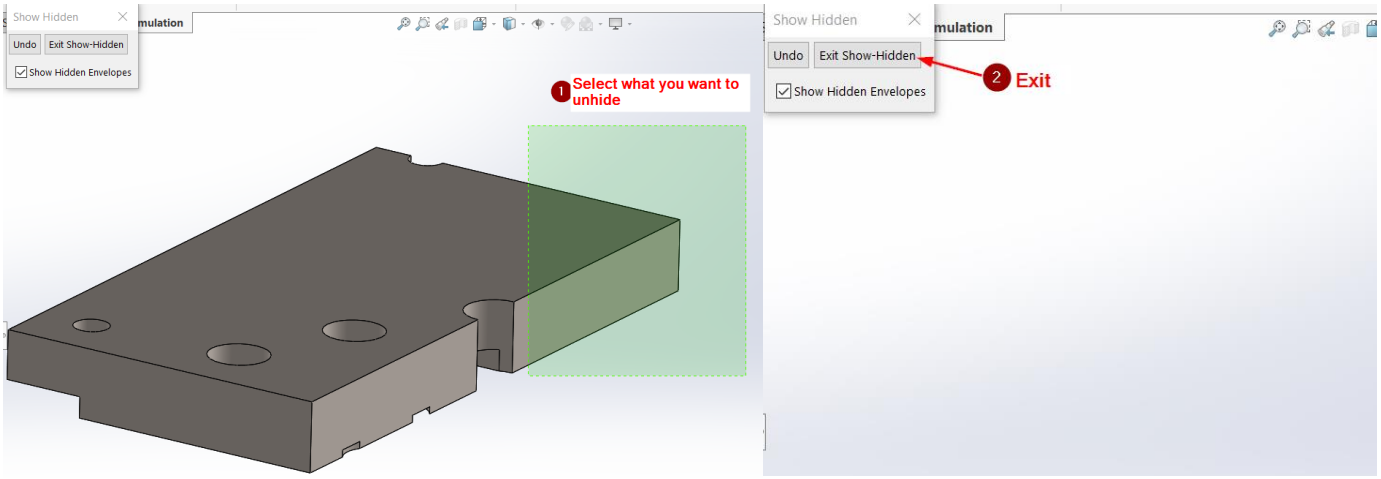
Put mouse on top of part to hide, then press “Tab” key

To unhide:

Press “S” key



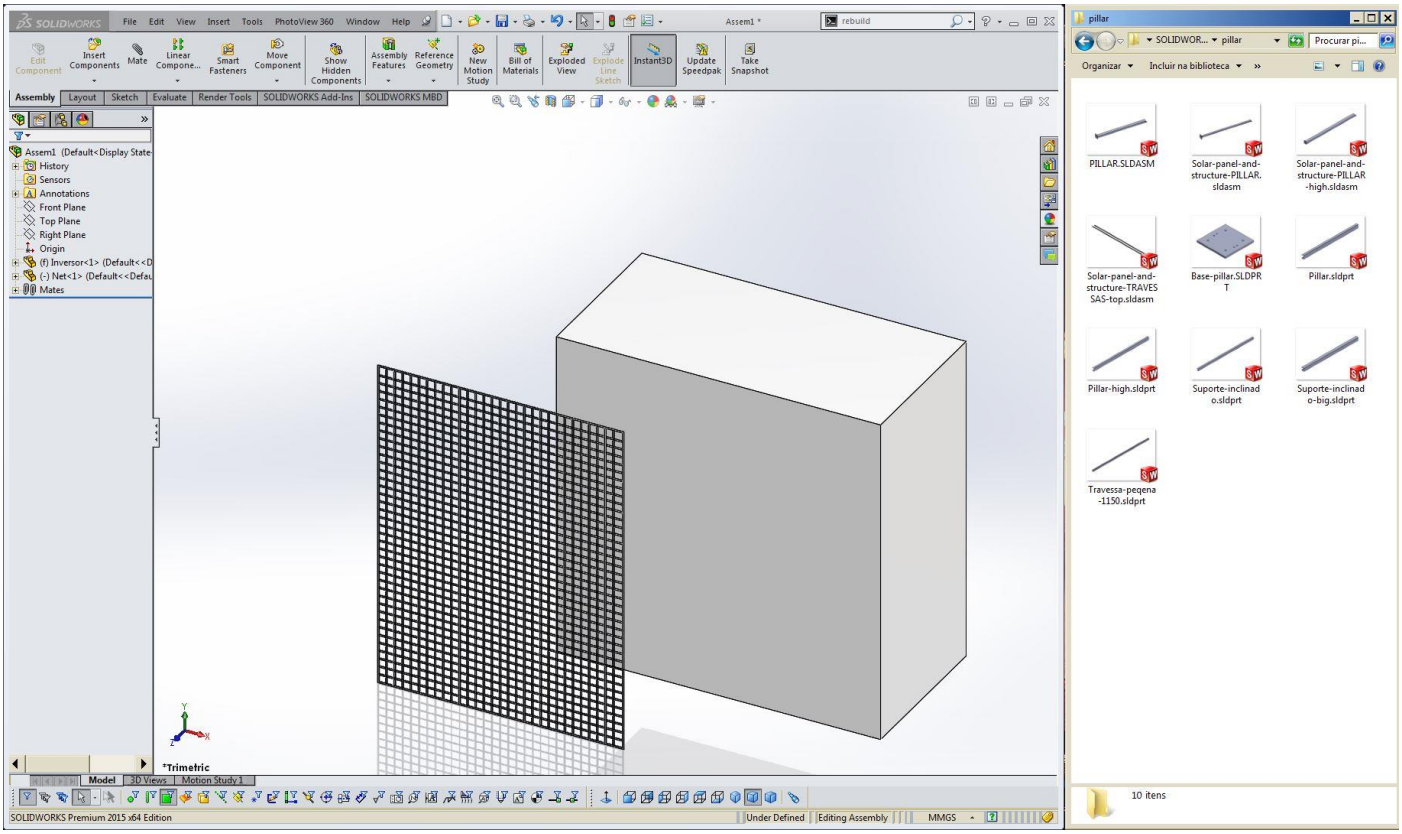
Then



Or press SHIFT+ TAB with the mouse over what to unhide.

Adding parts to assembly faster

To increase assembly efficiency setup your screen like this:



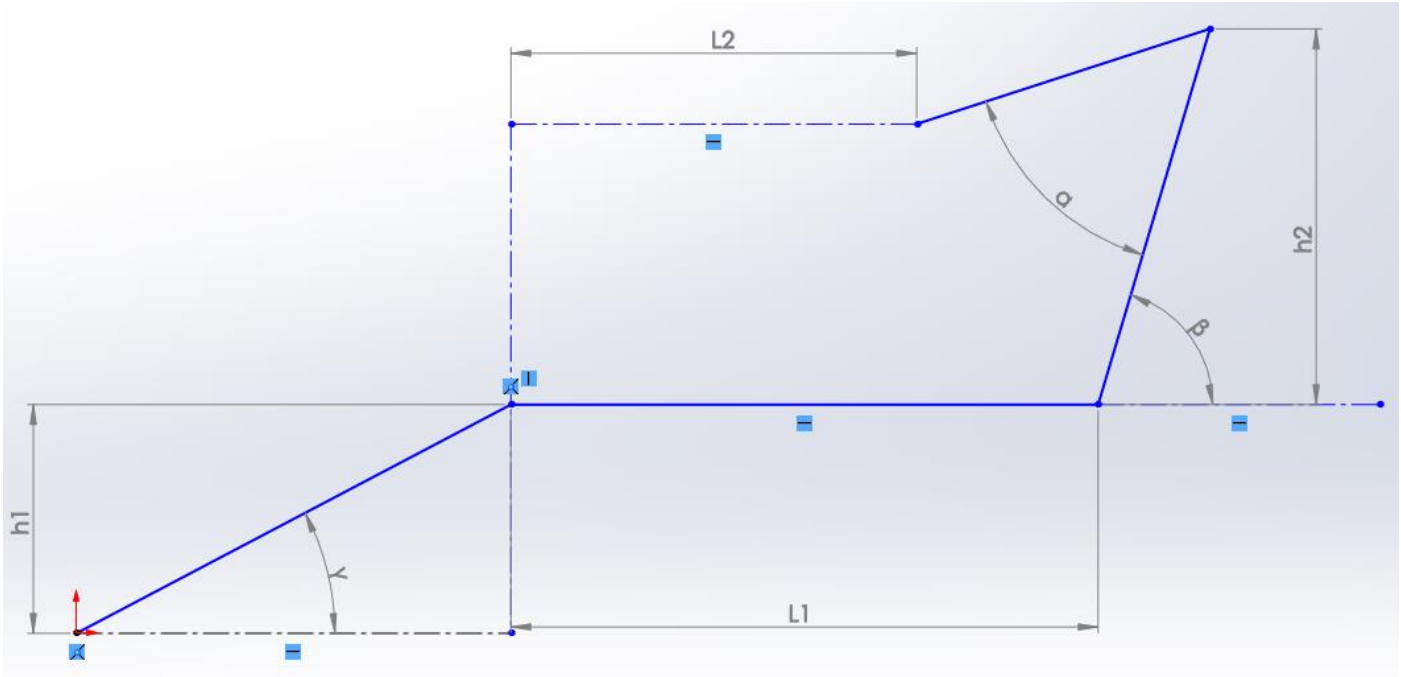
Left: SolidWorks

Right: parts and assemblies folder on explorer. If possible you should have a top level folder with only the sub-assemblies and single parts. These sub-assemblies then go get their parts from the respective folders. To insert a component into the assembly you just need to drag it into the window.

Use colors for different parts in assembly

It may be a bit uglier (depending on your taste) but it becomes much easier identifying different parts, and later on to caption those different parts.

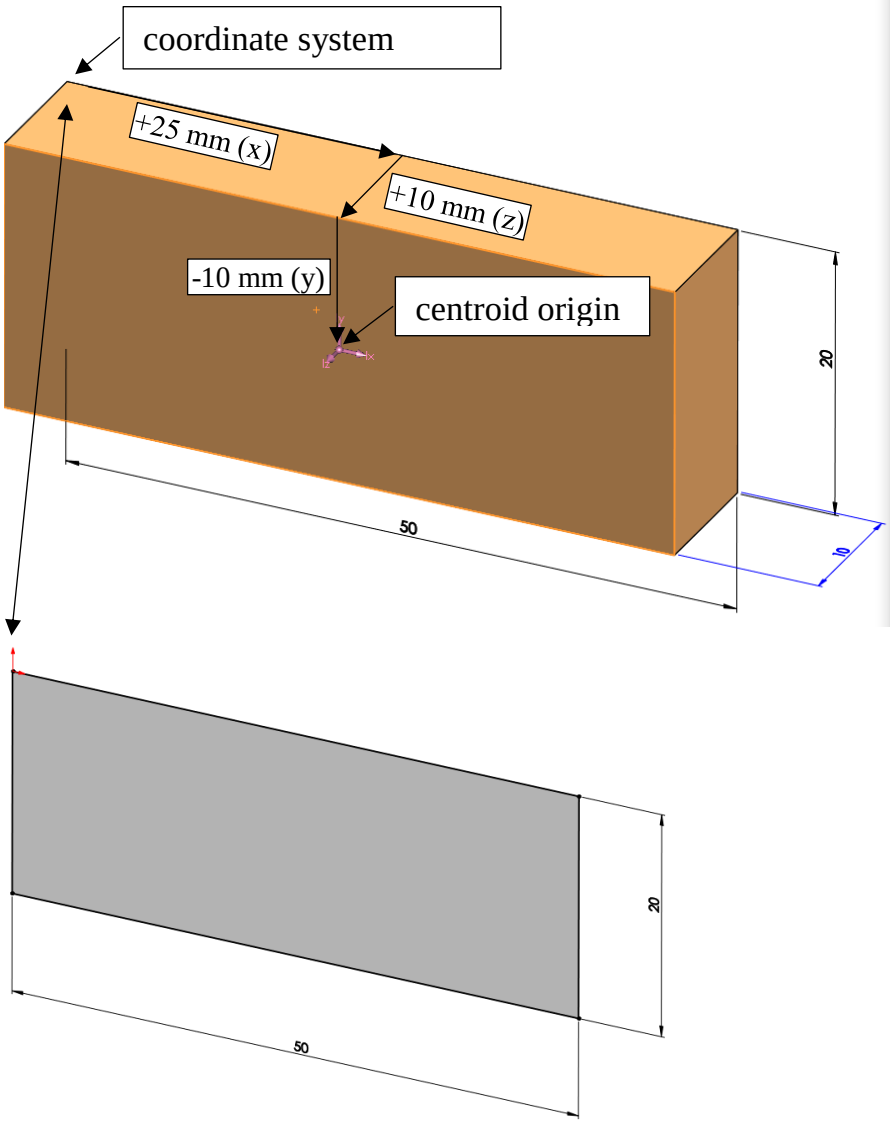
Use Solidworks as a geometry calculator

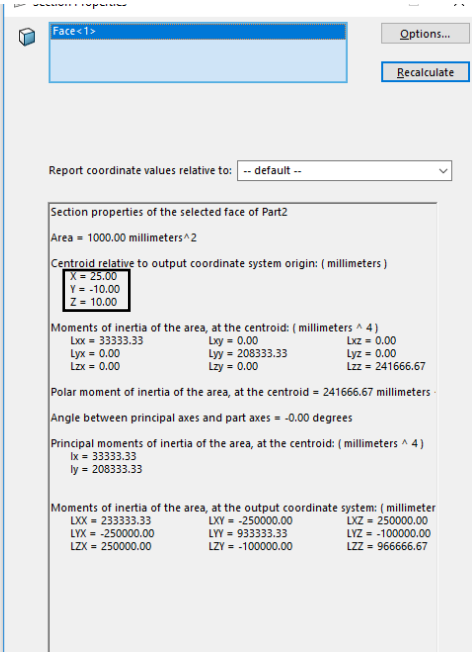


For angles, distances, moments of areas, inertias, centers of mass and gravity …you can use this to do free body diagrams. This is to avoid human error in calculations.

And with the help of Excel's solver add-in you can do a LOT.

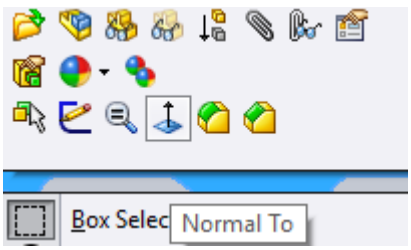
Understanding Section Properties





Normal to viewing

When you want to the normal view to a surface, but when you do that Solidworks flips the part, use this trick: First select the face to be normal to, then select a perpendicular face to be like the top plane. Then do Normal to (use keyboard shortcut, eg. Alt+1).



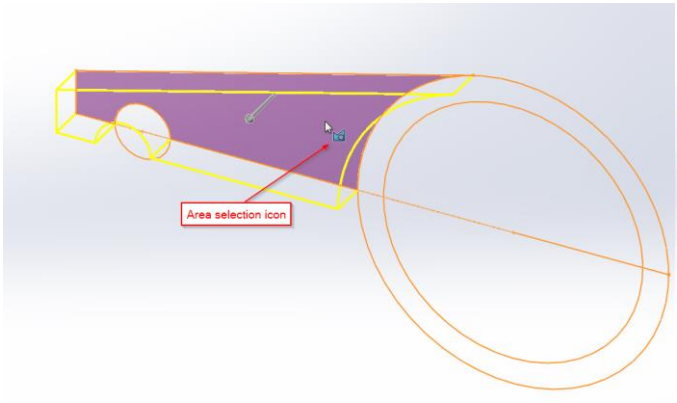
CAD, Modelling

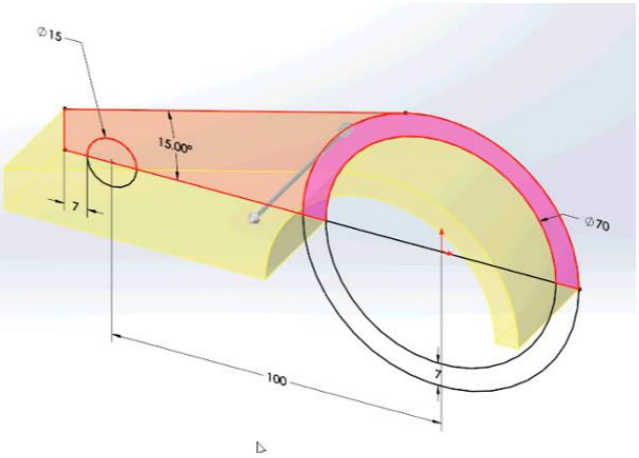
Sketching

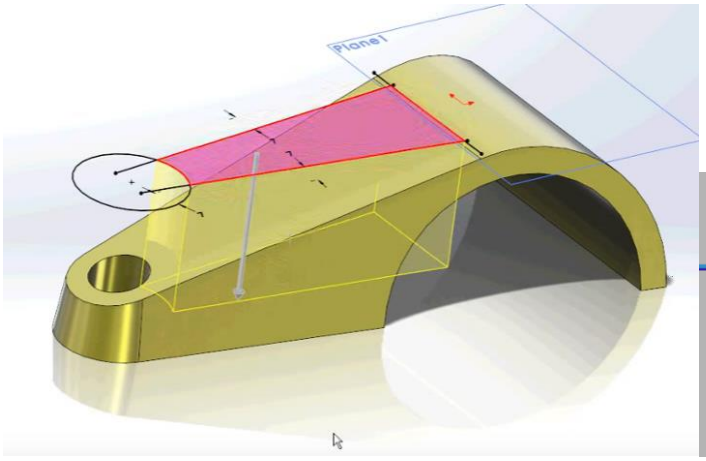
Another Way to Sketch

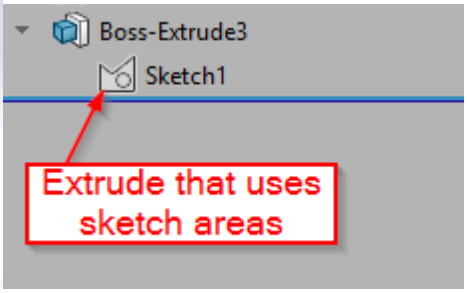
- It’s easier to change things we understand;
- It’s easier to understand things that we know the reasoning behind
- It may be easier to understand a part if there are fewer sketches for the same features

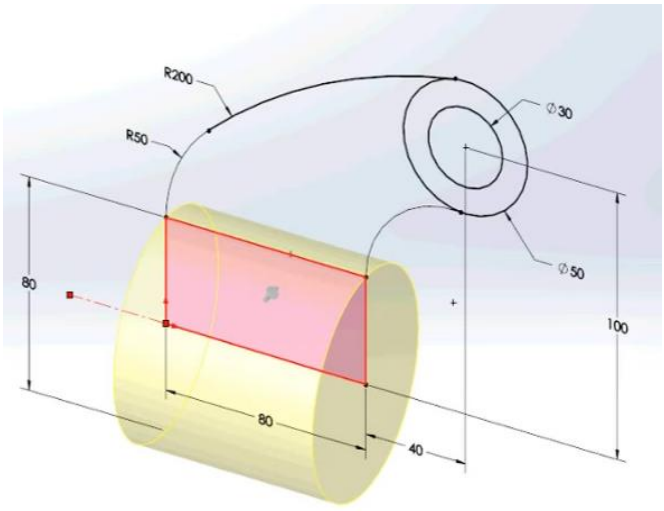
For these reasons

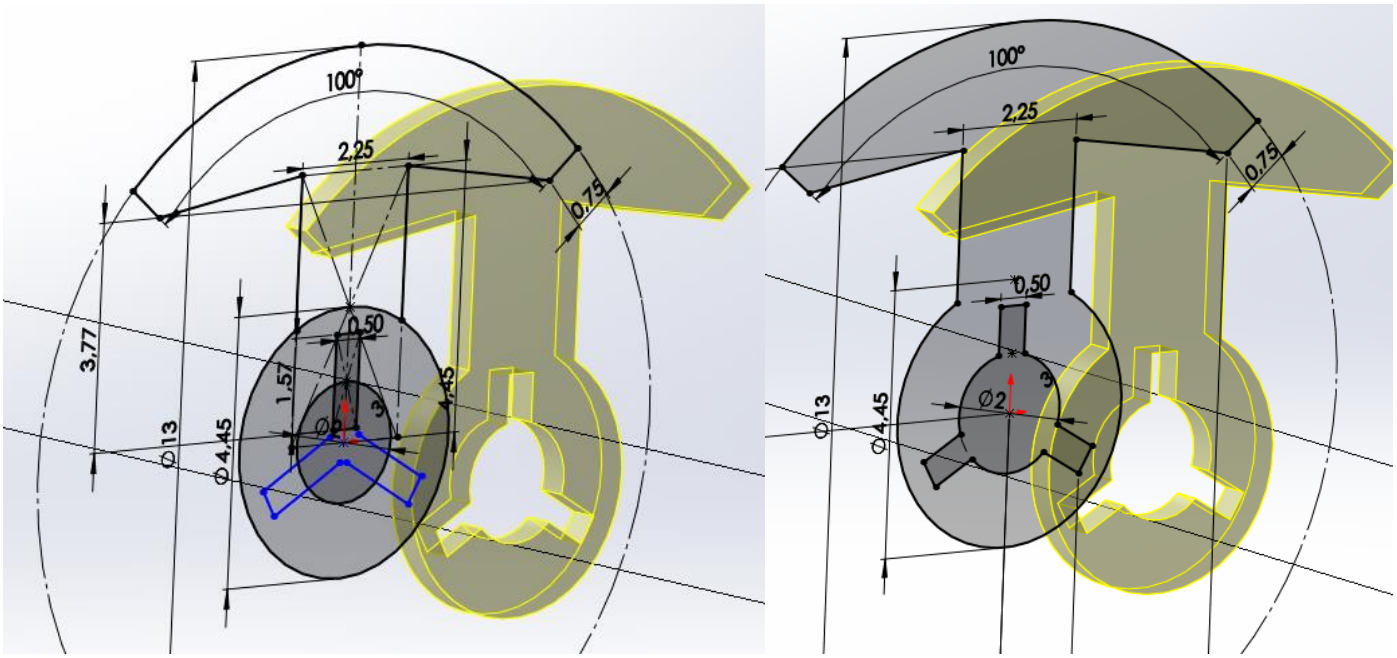








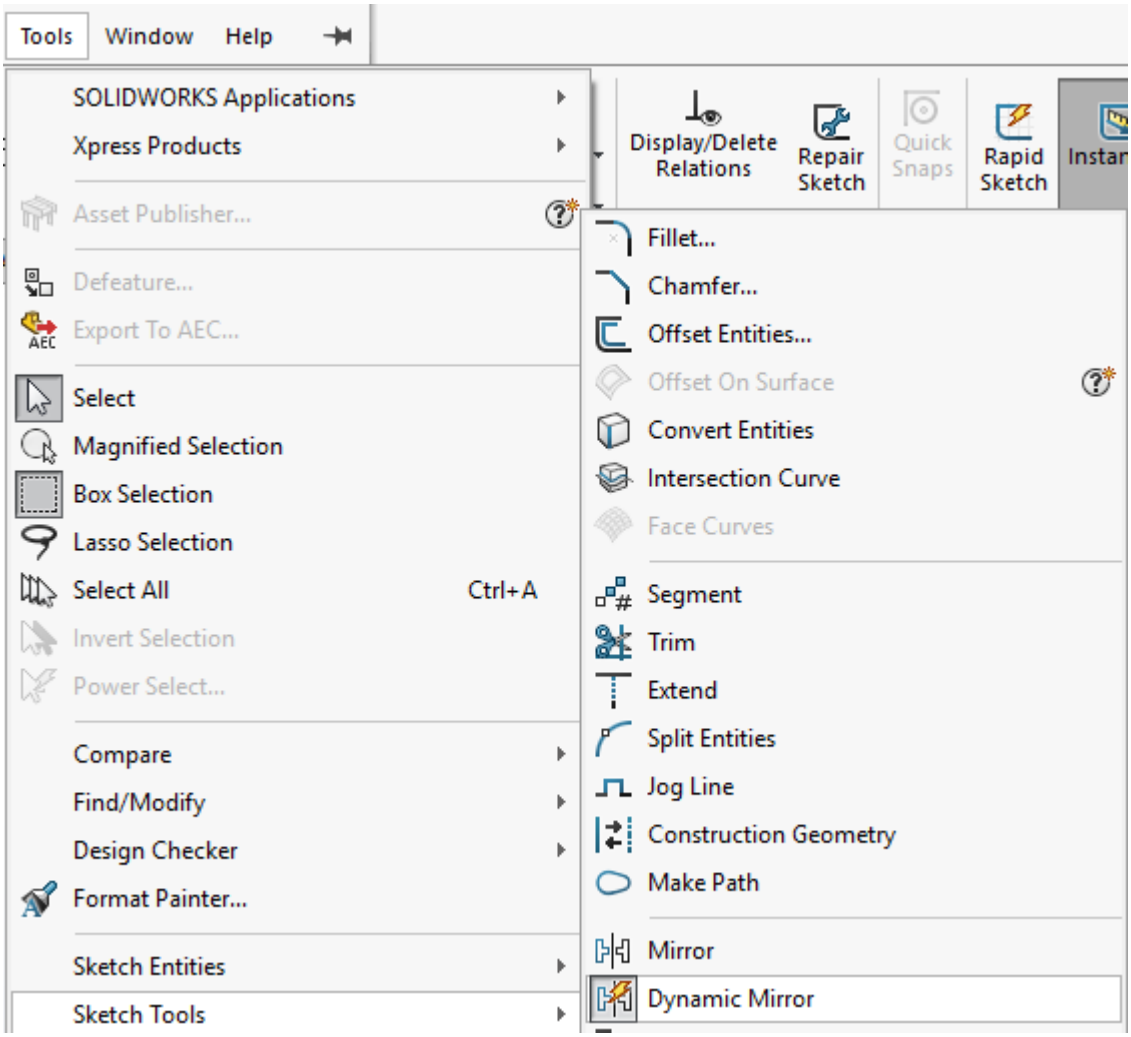




This information that was not used can be used later for reference to other features.

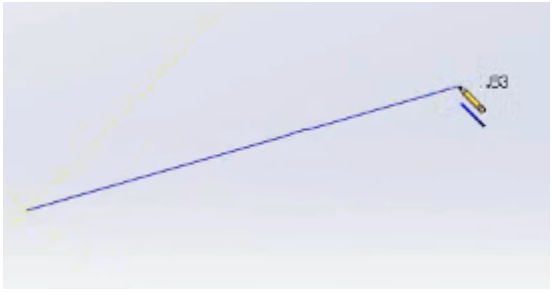
Clicking inside an area or on the contour has different behaviour.

Dynamic Mirror



Make a centreline then use dynamic mirror. As you sketch in one side of a centreline it automatically draws on the other side.

Polyline and Line



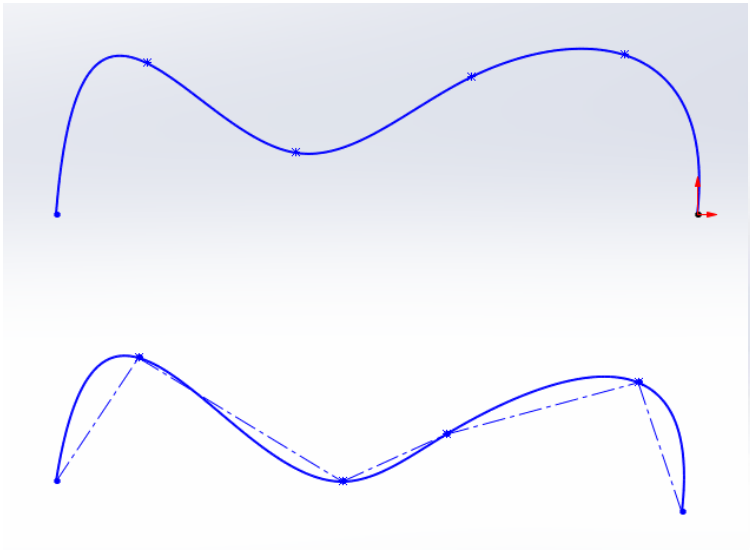
If you LM click and release you'll create a polyline (another one will start when you click to finish one line) (polyline = multiple line segments). If you click, hold and drag the mouse you'll create a single line.

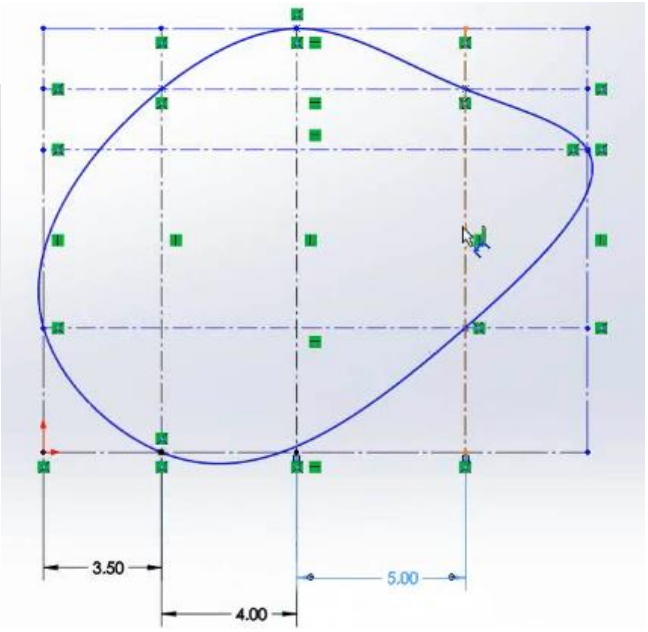
Copy sketch geometry

Ctrl + drag copies things in sketches just like in assembly.

Making splines with centerlines

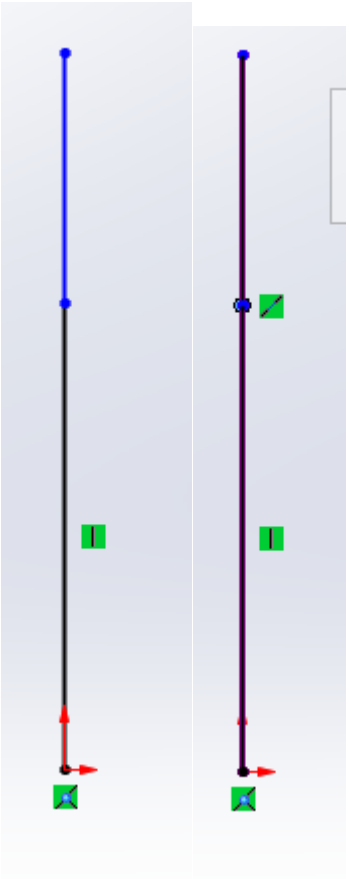
With and without centerlines. By using center lines it allows you to fine-tune certain parameters (see below).



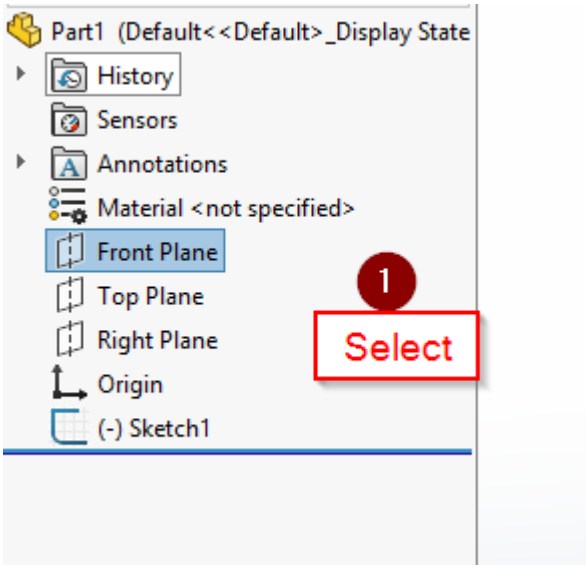


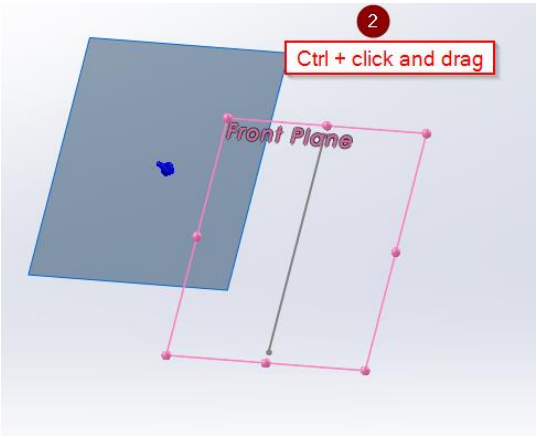
Merging line

To merge these 2 lines you must give them the relation of collinear then select the point and delete.



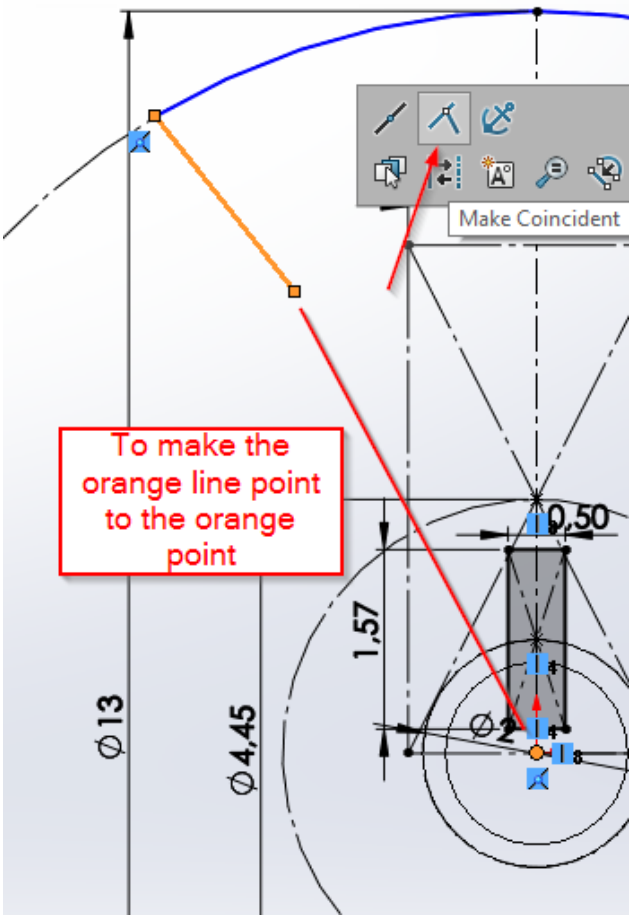
Quick create plane





Direction of line with a point

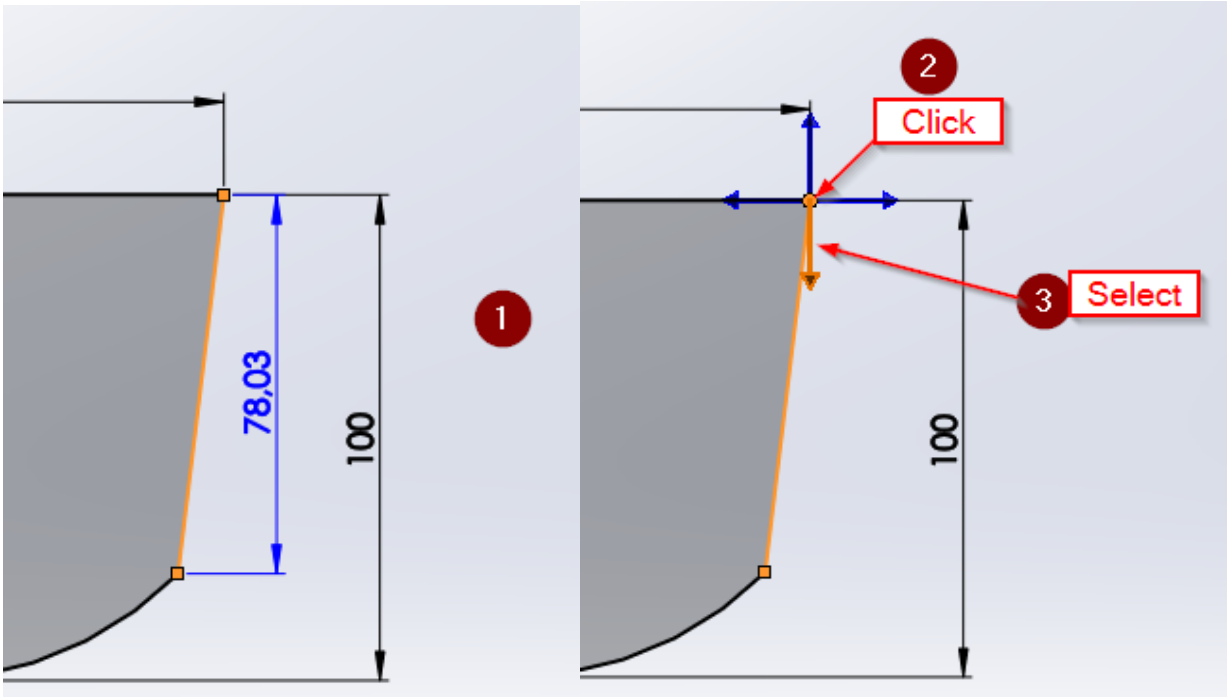
Solidworks is smart enough that it can imagine the continuation of a line and make that coincident with a line. This means you can use a point to establish the direction of a line.

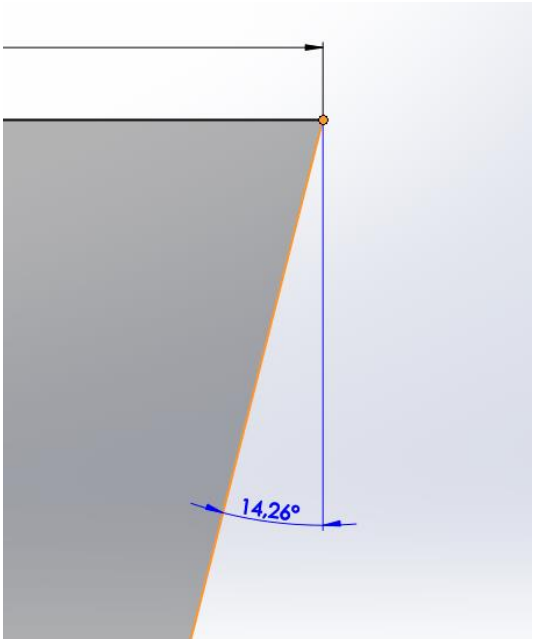


Pierce vs coincident

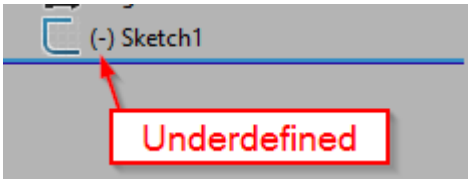
Basically, coincident = 2D (coincides on the sketch plane), pierce = 3D coincident.

Angle dimensioning





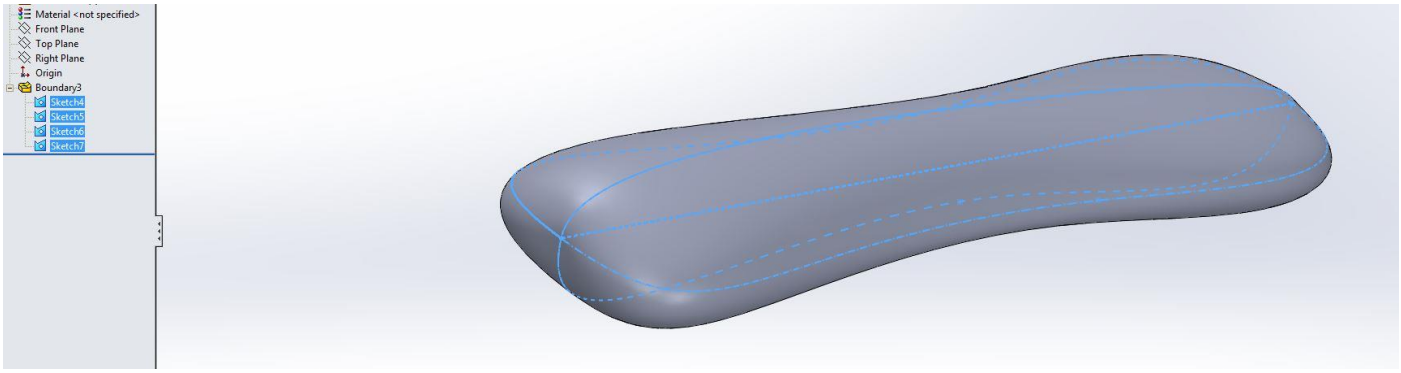
Other



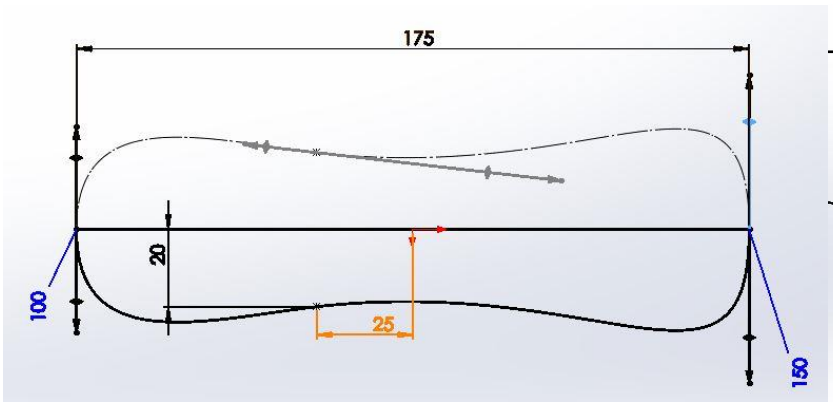
Strange surfaces

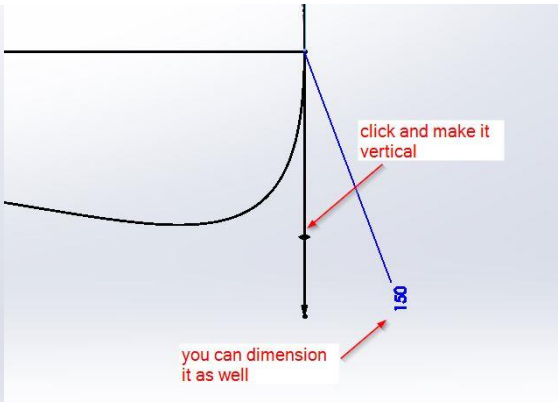
Remote Control

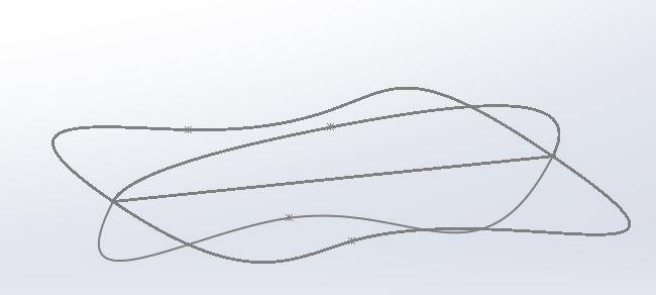
To achieve this result make 4 sketches, each with a 1/4 of the solid.



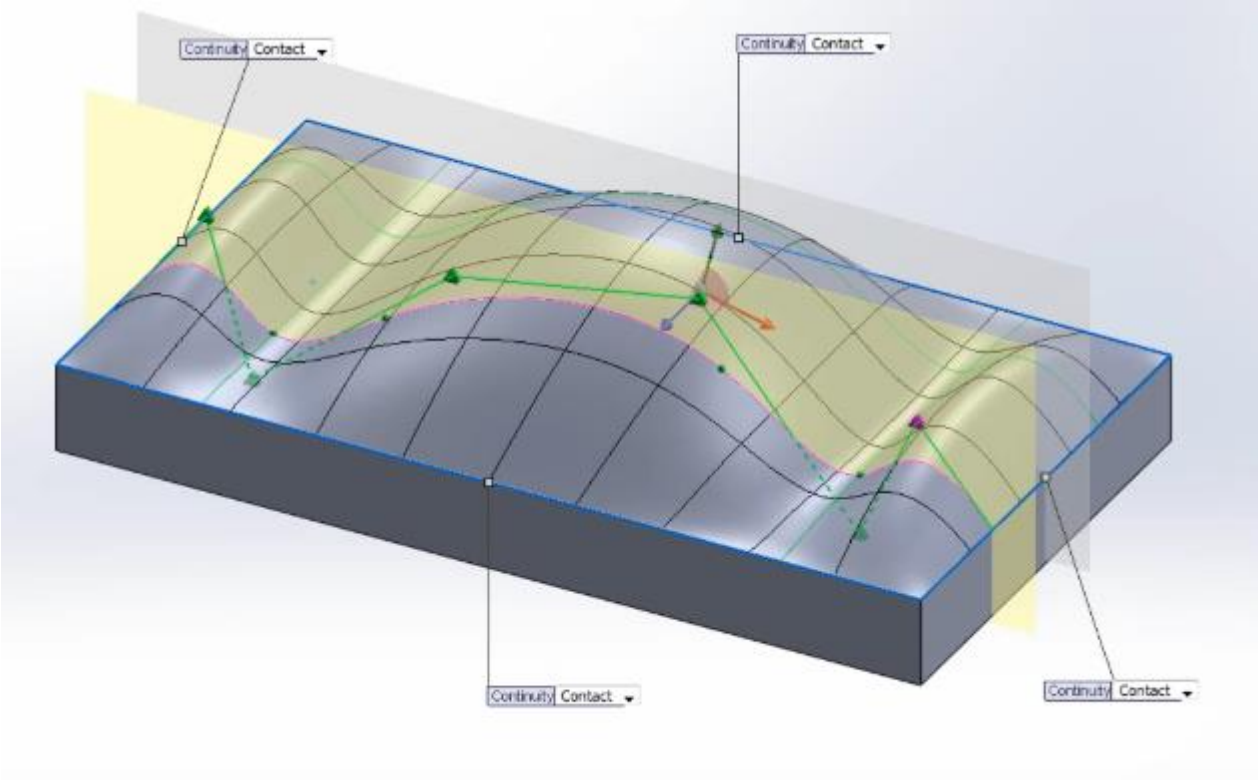
To make sure the sketches agree with each other make the lines at the ends vertical/horizontal like this:

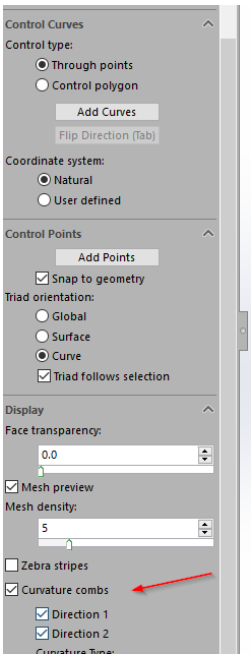


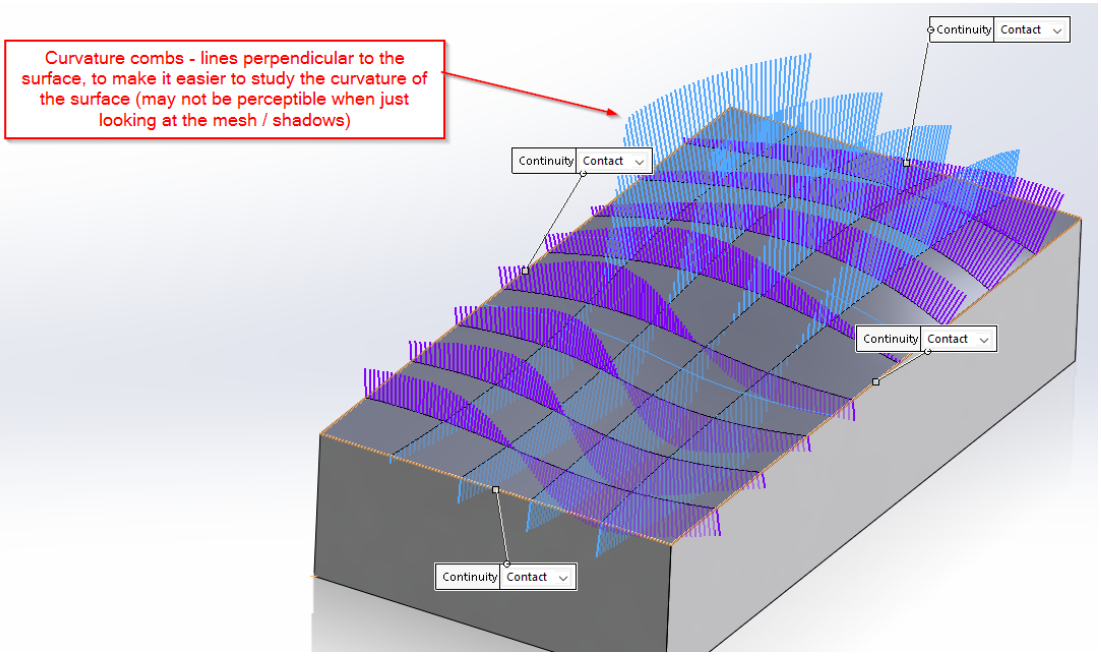




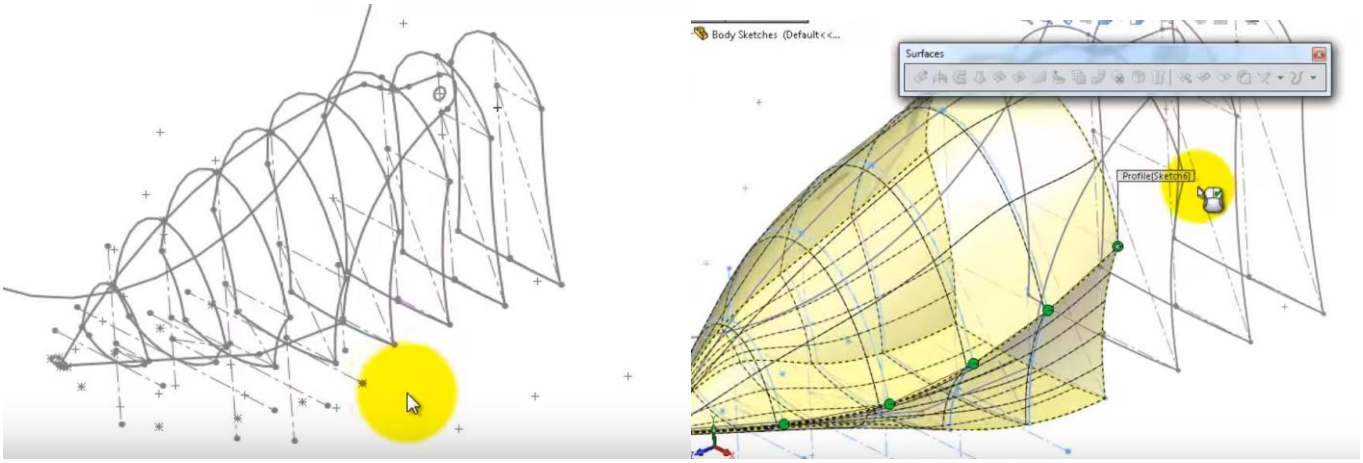
Using “Freeform”

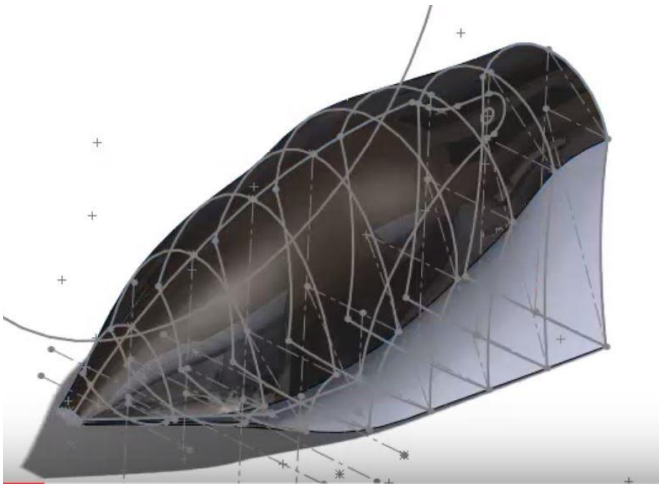






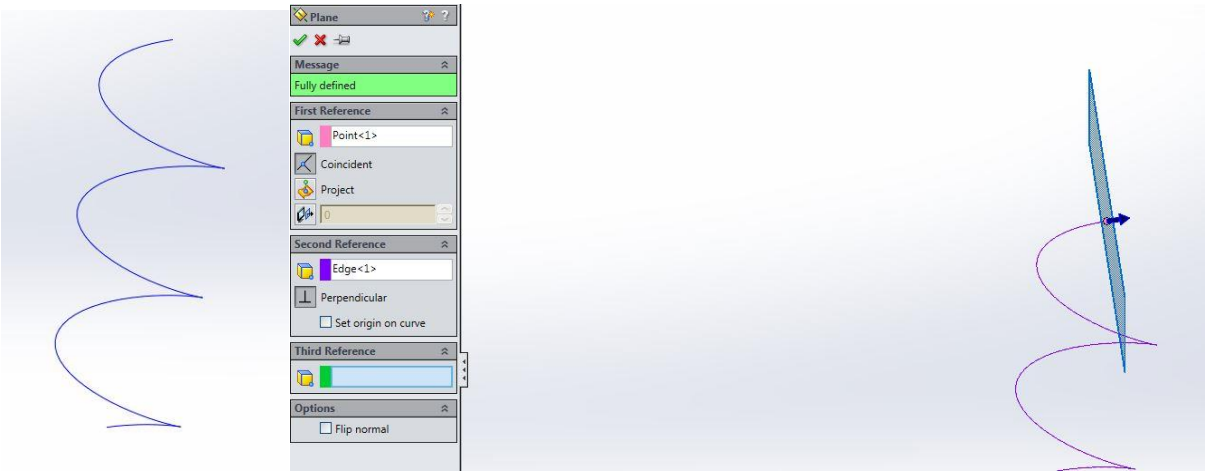
Helicopter Front





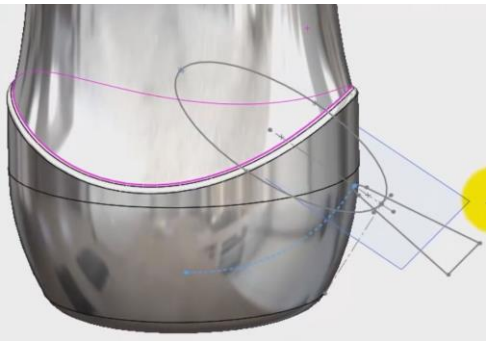
Create plane for face to sweep

How would you create a plane on the top of the helix to make a sweep?

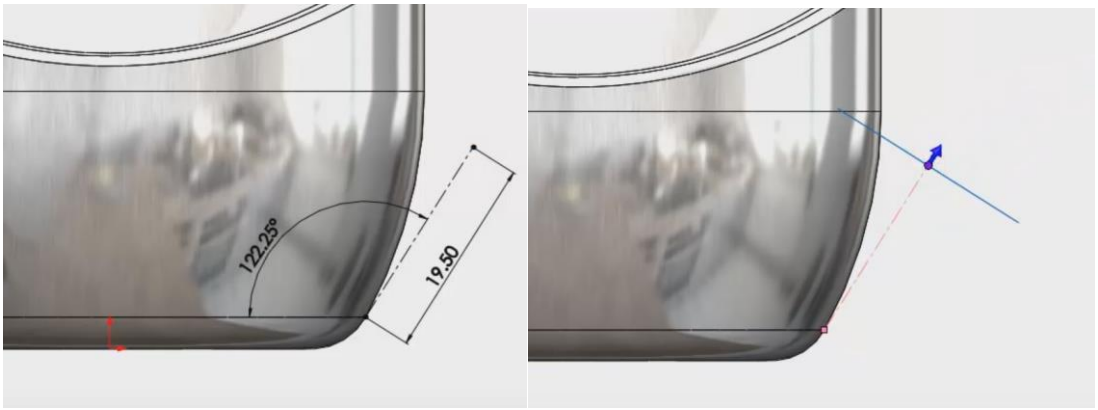


Select the point and select the edge. Remember that a plane can be defined by a line and a point (external to that line).

In this case the engineer is making a cut to give the bottom appearance of a coca-cola bottle. But to make the sketch that he will use to make the cut he needs an inclined plane.



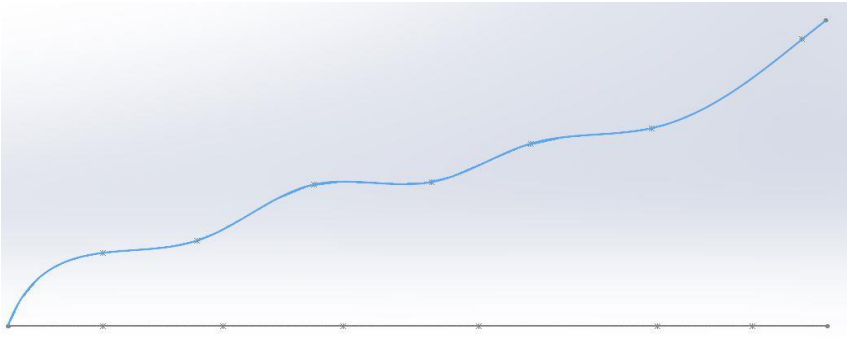
You do it the same way. Sketch a line, give it the desired angle and size, and then create a plane at the end of the line.



Projected Curve

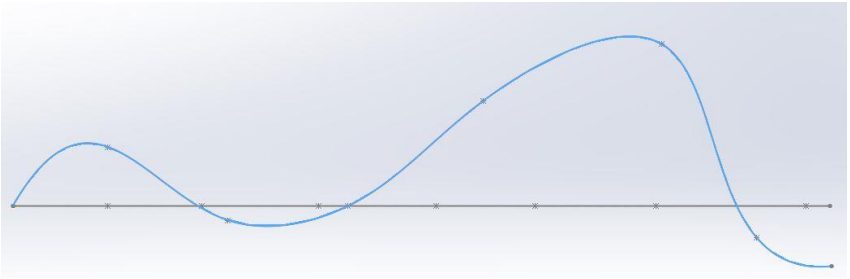
Imagine the road of a mountain. It climbs steadily when viewed from the front.

Front view

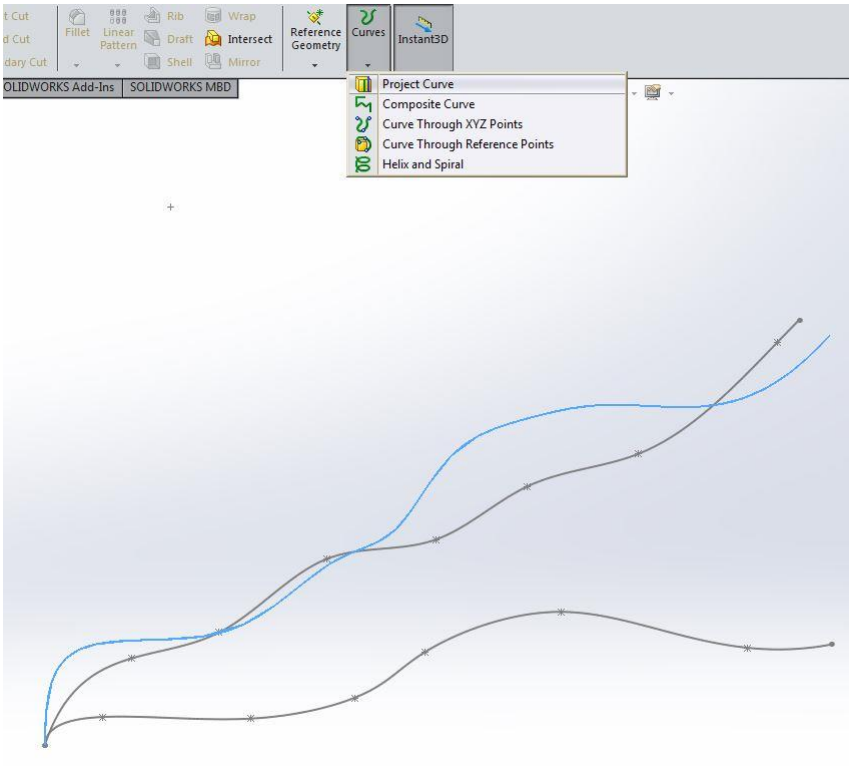


and curves a lot when seen from a map or from above.

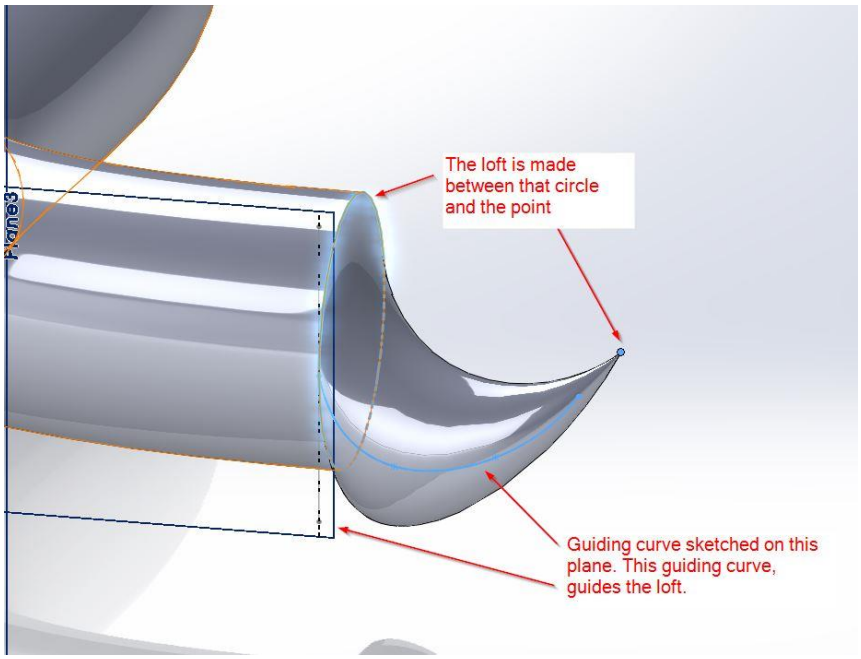
Top View



What if you wanted to get the 3D version of it? Then use the “project curve” which combines 2 sketches on different planes.

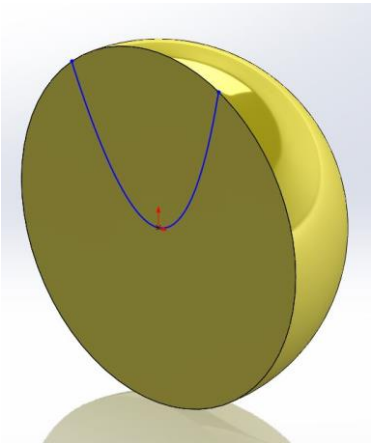


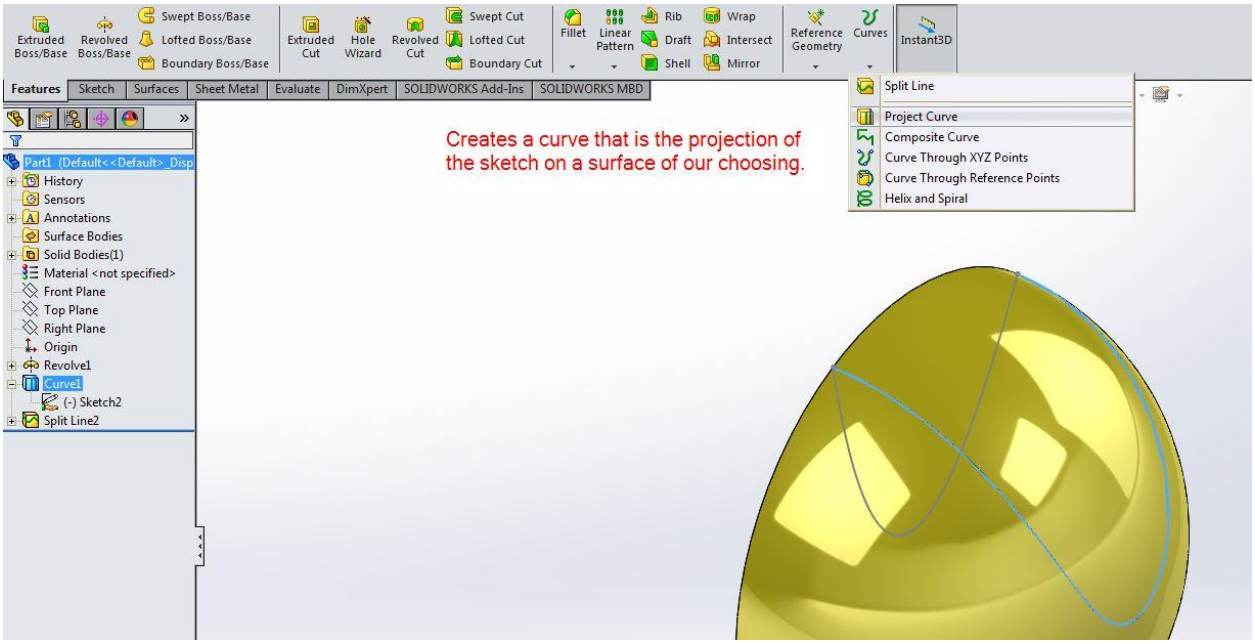
Loft Guiding Curves

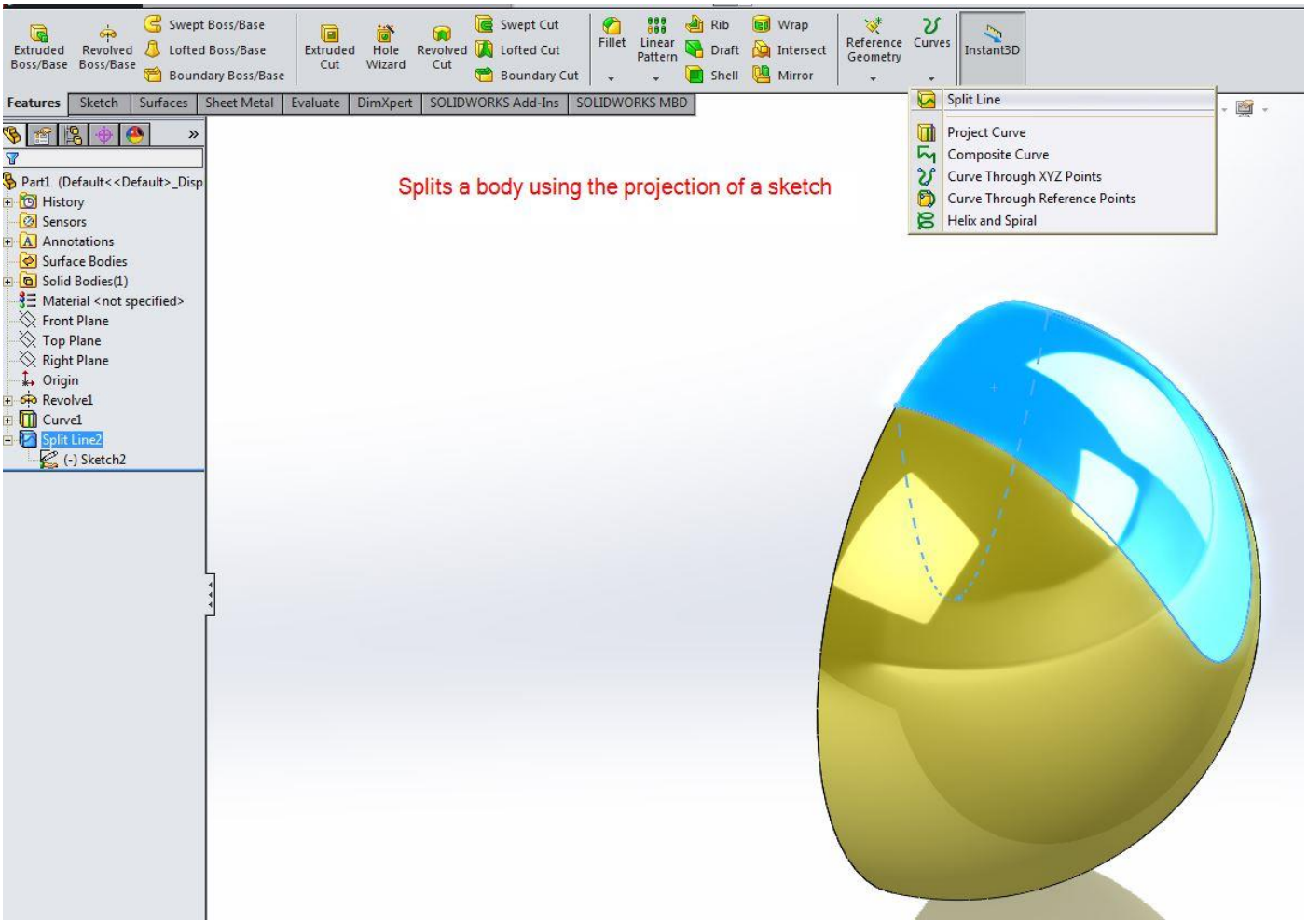


There can also be several guiding curves. In the example below the 4 sketches will guide the loft for this bottle from the top sketch to the bottom.

Split Line

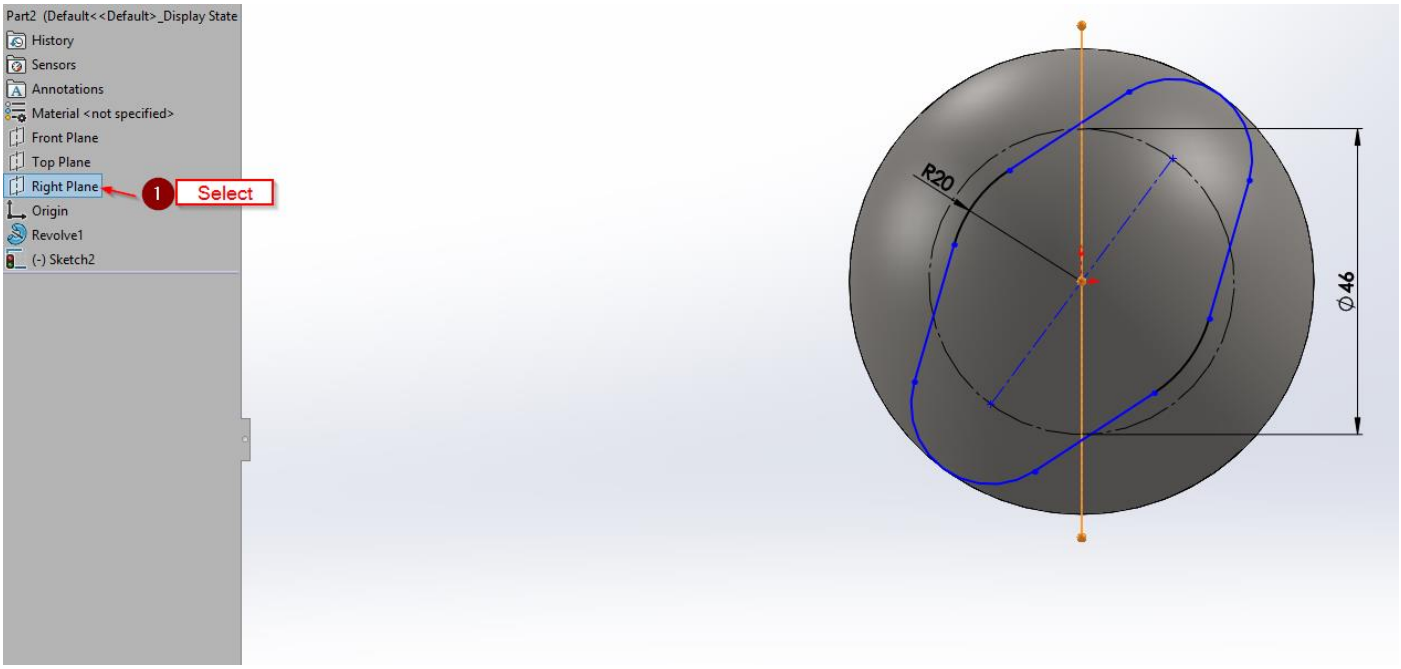



Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Fillet
Linear Pattern
Rib
Draft
Shell
Wrap
Intersect
Mirror
Reference Geometry
Curves
Instant3D
Features
Sketch
Surfaces
Sheet Metal
Evaluate
DimXpert
SOLIDWORKS Add-Ins
SOLIDWORKS MBD
Part1 (Default<<Default>_Disp
History
Sensors
Annotations
Surface Bodies
Solid Bodies(1)
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Revolve1
Curve1
(-) Sketch2
Split Line2
Split Line
Project Curve
Composite Curve
Curve Through XYZ Points
Curve Through Reference Points
Helix and Spiral
Creates a curve that is the projection of the sketch on a surface of our choosing.

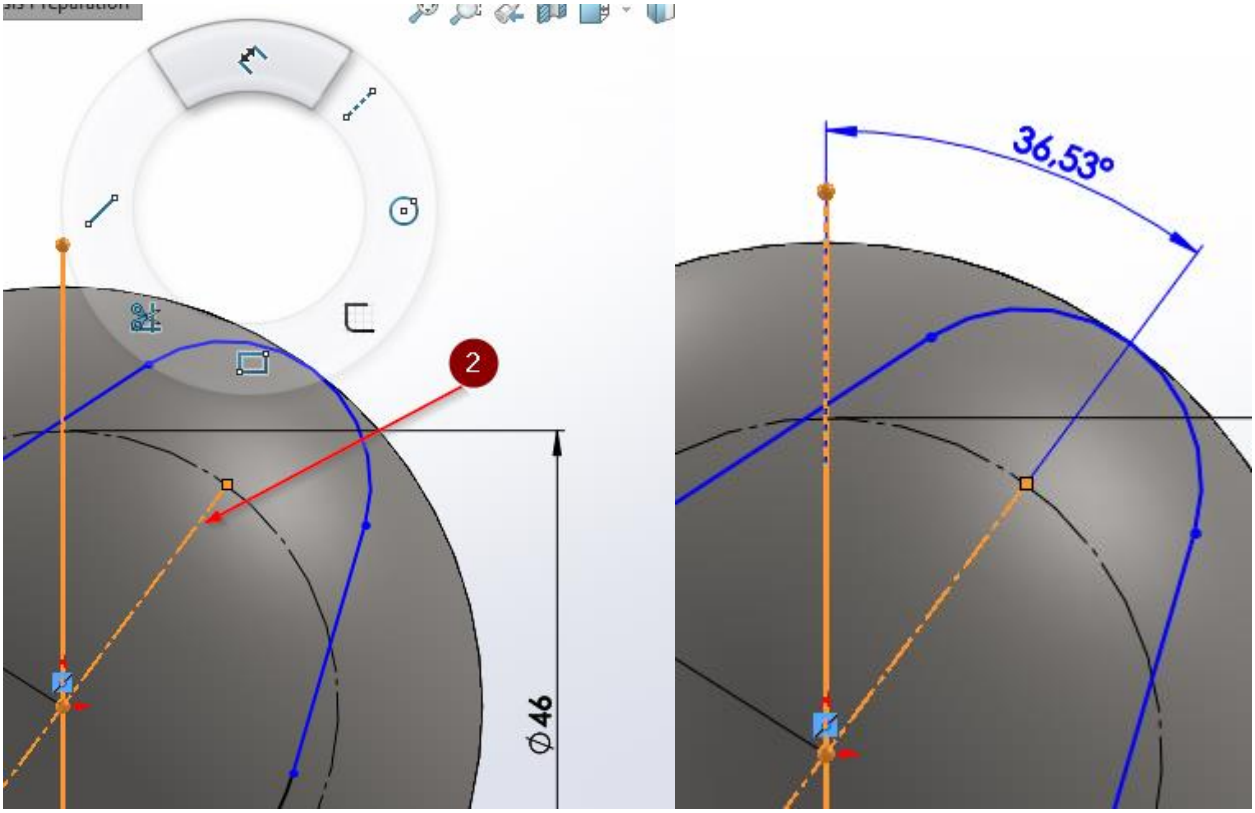


Dimension

To give relationship between plane and a line. First select the two things you want to distance between themselves. Then dimension.

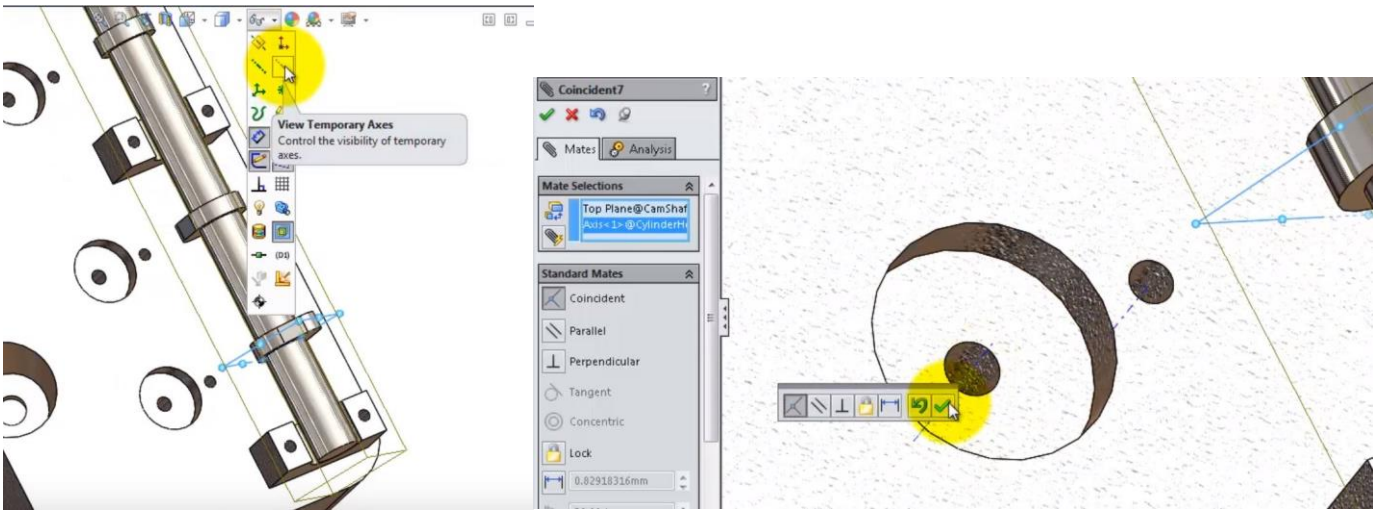


First we select the plane. Then the line. Then we give an angle between the two.



Assembly

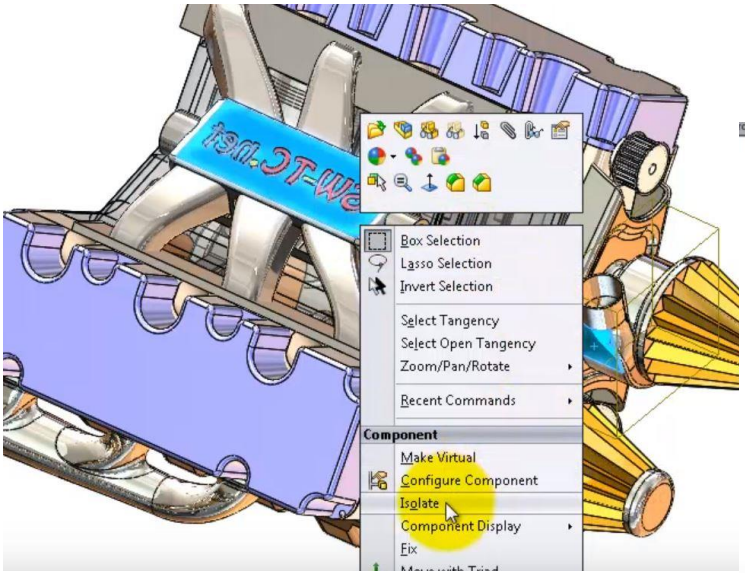
It’s possible to mate planes and axis and points of parts inside assemblies. In this case we’re mating the top plane with the axis of a feature.

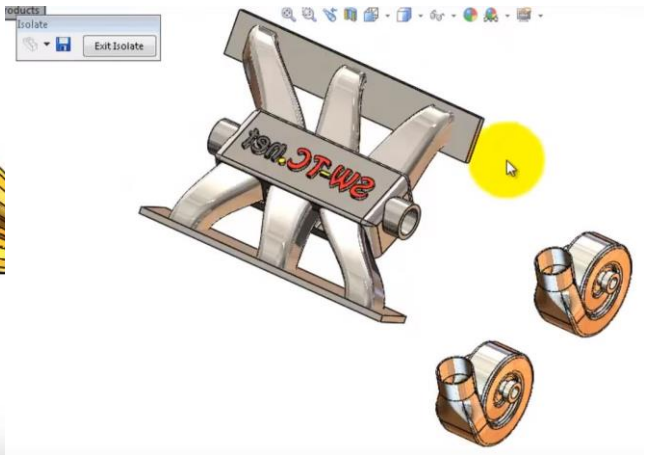


If you have an assembly which contains a sub assembly. Now this sub assembly has moving parts and you want them to move when inserted in the assembly. By default you won’t be able to because the sub assembly will be set as a rigid body. To fix this right click on the sub assembly, inside the assembly, go to “Component Properties” and put “Solve as flexible”.

Isolate

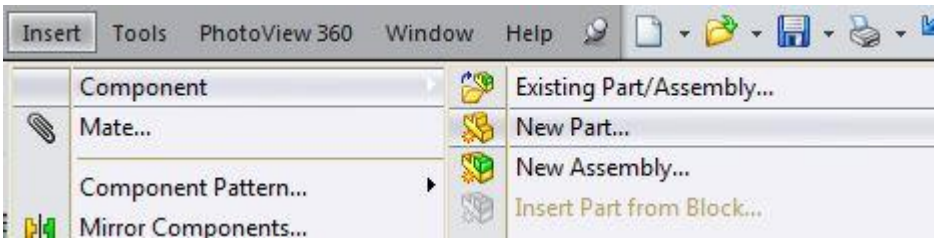
Use “Isolate” if you’re working with just a few parts of the assembly, for example to create a sketch of mates that only involve them

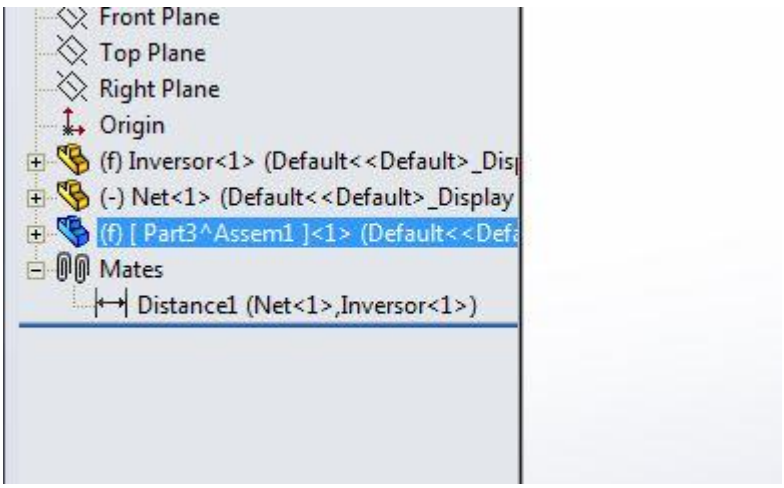




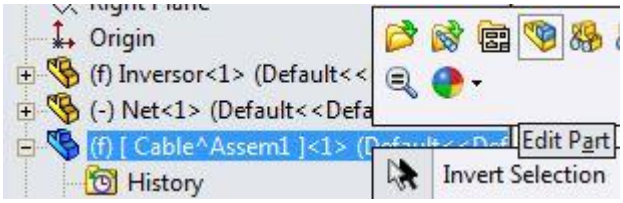
Creating New Part Inside Assembly

What if you want to create a new part that depends on the assembly? That is you need the assembly to create that part. In that case





Rename and Edit the part



And start sketching

Positioning Holes from Hole Wizard

1) Place the hole anywhere;
2) Close and edit the sketch that defines its position.

Fixing & Floating parts & sub-assemblies

You can fix the position of a component so that it cannot move with respect to the assembly origin. By default, the first part in an assembly is fixed; however, you can float it at any time.

It is recommended that at least one assembly component is either fixed, or mated to the assembly planes or origin. This gives a frame of reference for all other mates, and helps prevent unexpected movement of components when mates are added.

A fixed component has a (f) before its name in the FeatureManager design tree.

A floating, under defined component has a (-) before its name in the FeatureManager design tree.

A fully defined component does not have a prefix.

CAE, Simulation

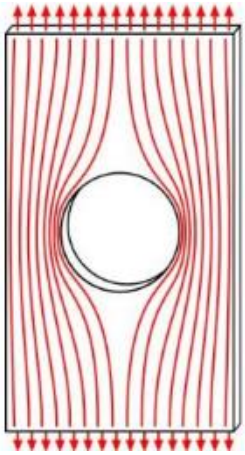
General

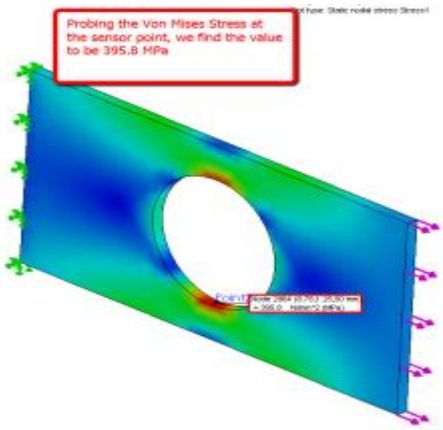
Imagine how stuff will deform and where the critical parts will be.

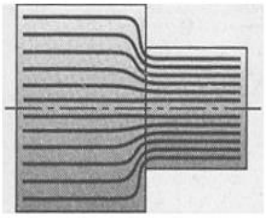
Pre-simulation

Pre-simulations are quick, draft simulations you do to get some results asap. The mesh is coarse and not much attention is paid to its quality. You do things approximately. The point is to get some quick results so we know the general direction of where the simulation may be going. This can also help acknowledge important considerations for further pre-simulations and simulations.

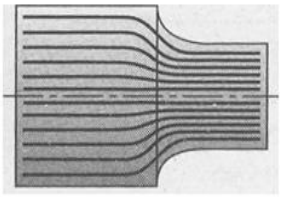
Stress concentrations and singularities





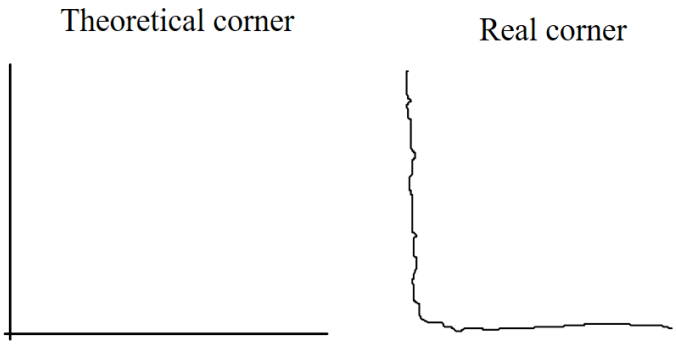


(a) Force flow around a sharp corner

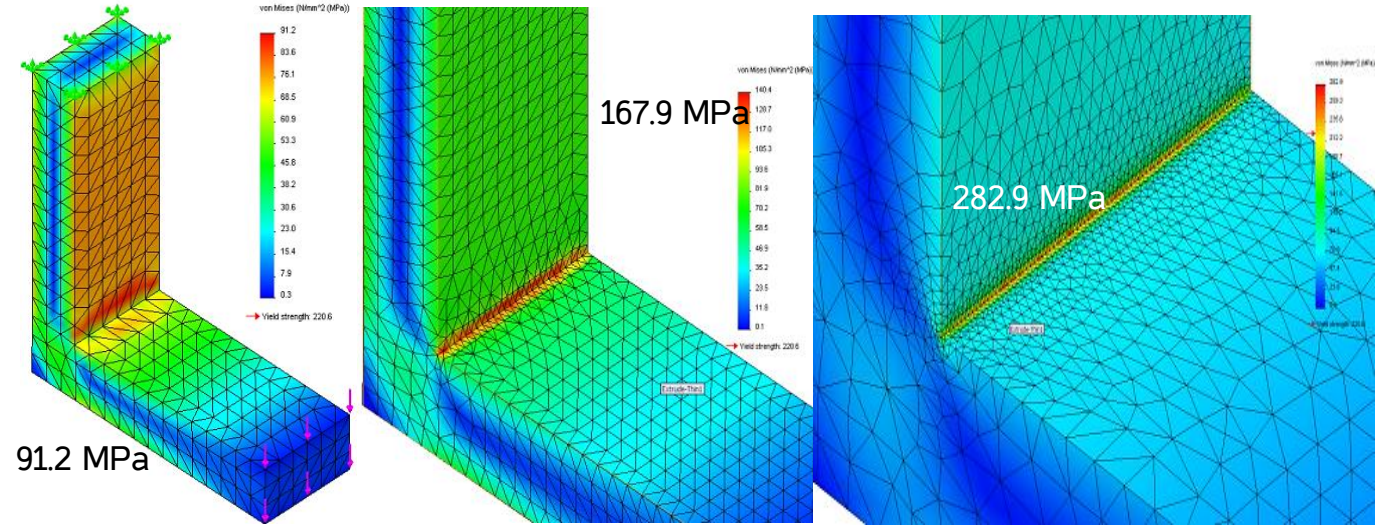


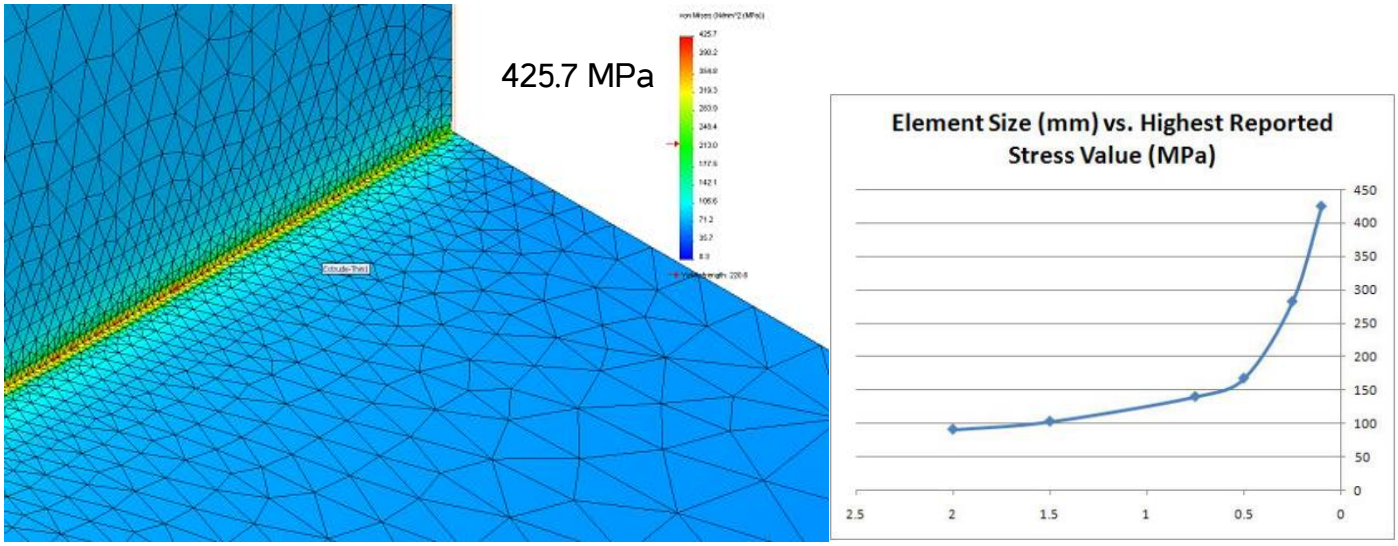
Force flow around a corner with fillet: Low stress concentration.

In cases where there are straight corners, there are often singularities. However these don't correspond to the reality because no corner in the real world is perfectly straight.



Applying mesh control to the edge in question, and locally reducing the element size, we see the stress progressively increases.

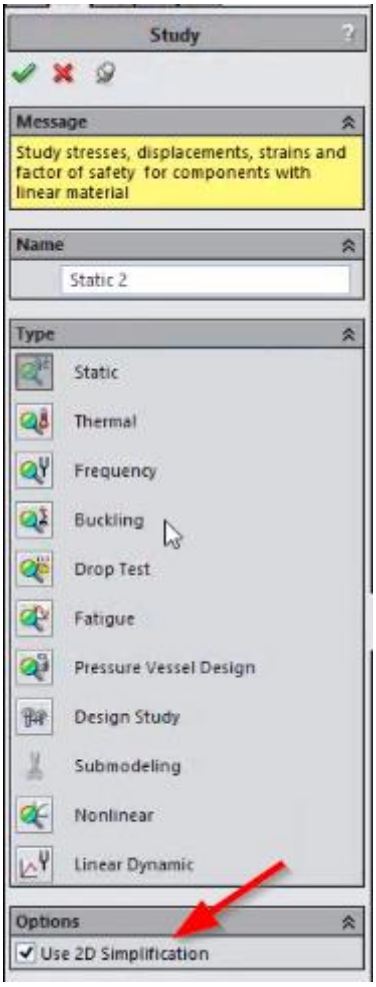




To fix this, add a small realistic fillet/chamfer that the part would have in reality.

2D, shells, beams and solids

Prefer 2D simulation over 3D as it is quicker to solve and provides better results. Also prefer to use shells and symmetry whenever possible for solving speed purposes.

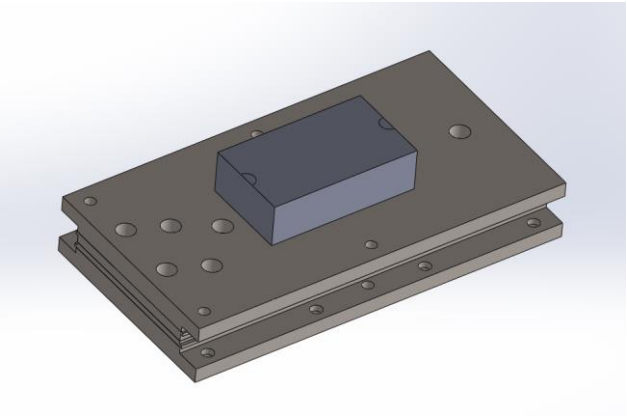


Apply force on area

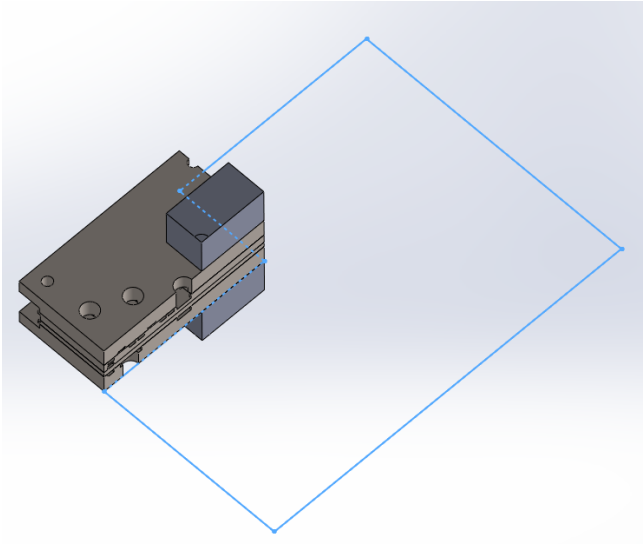
Create a split “line” for that. (sketch the area, then split line)

Symmetry

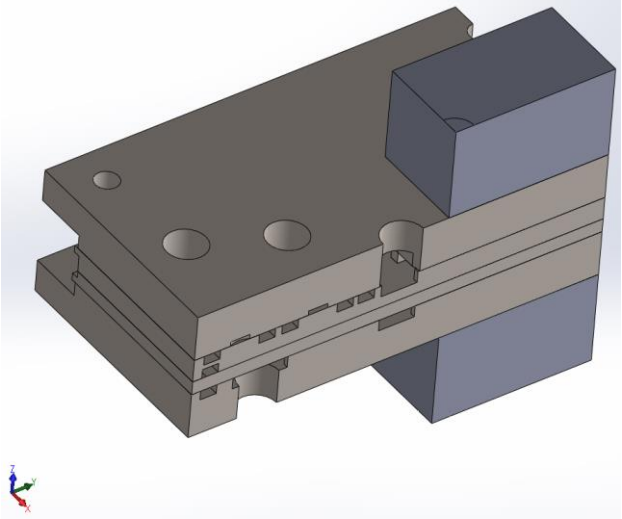
My assembly is roughly symmetric.



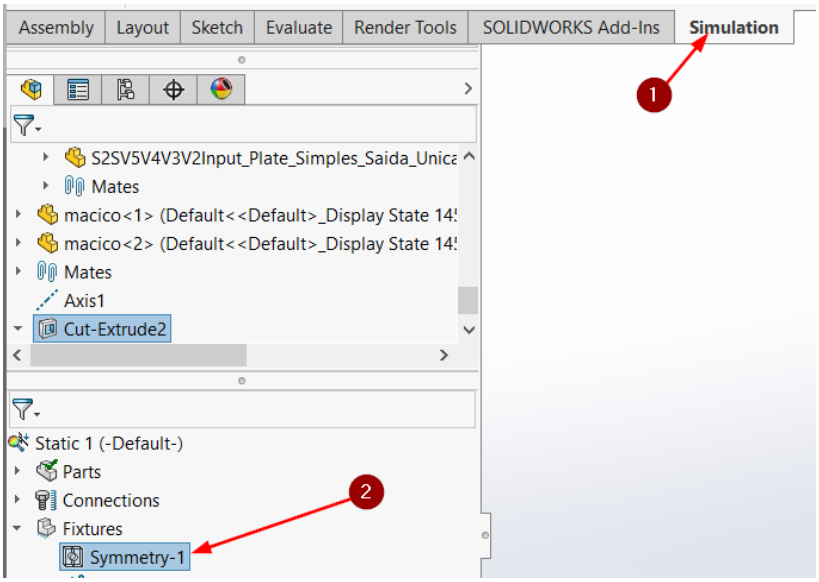
We create an assembly sketch which we will use to cut the rest of the assembly like this:



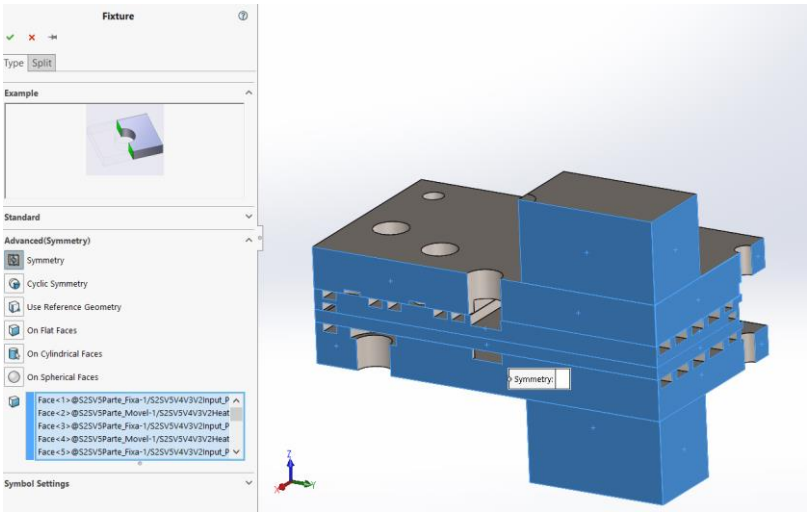
We can create the sketch in any of the planes as we will tell the extruded cut to cut both ways anyway.



After this, it’s time to define the symmetry in the simulation.



And select the cut areas



The force you put in an area will be also be put in the other invisible symmetric areas, so let’s say that if the total force is 4N over the whole assembly then you need to put 1N on the cut part.

Mesh

Running mesh convergence tests / mesh sensitivity tests

When possible, start with a draft coarse mesh, to get an idea where main stresses will occur, how it will deform, to confirm your imagination of what would happen (pre-simulation).

You just refine the mesh and compare the results. The stress will reach an asymptotic value. Compare the results for difference mesh densities to the asymptotic value to gauge convergence.

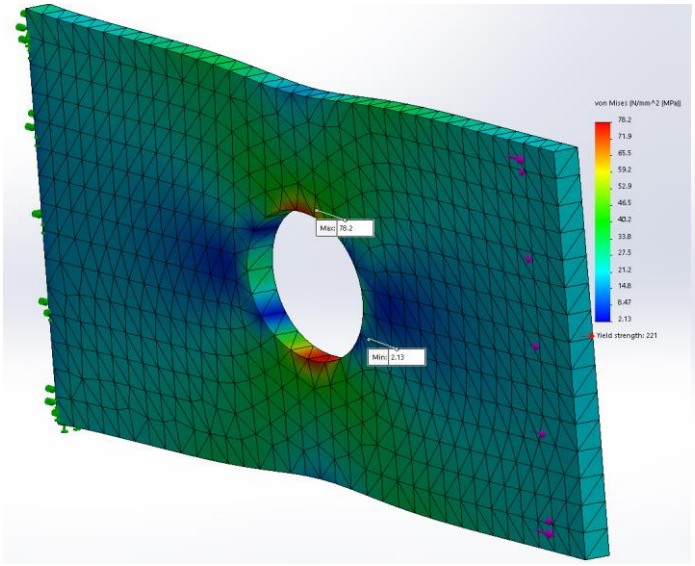
For general purposes, this works fine. However, note that stresses do not converge "smooth", but rather in a sawtooth-like fashion. This is in contrast to displacements, which do converge "smoothly". Just something to be aware of if the data doesn't show a nice smooth convergence curve.

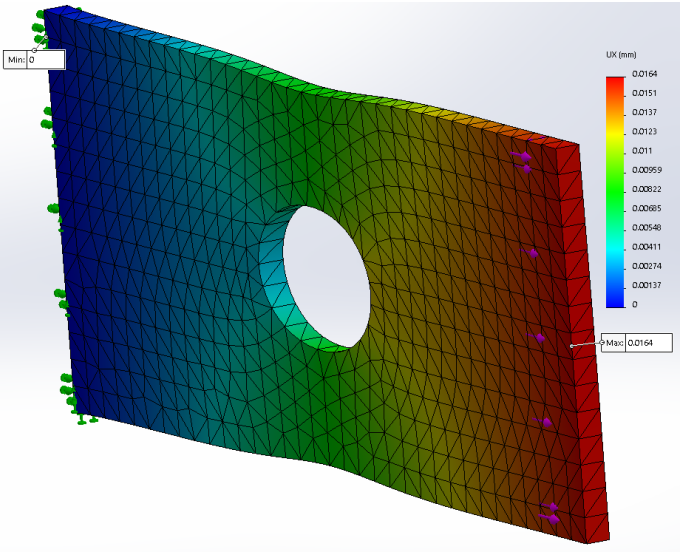
If you notice the stress/displacement doesn't change significantly when you change the mesh quality then it's very likely the problem is mesh **in**sensitive, which is awesome, because it means you can get accurate results with a coarse mesh.

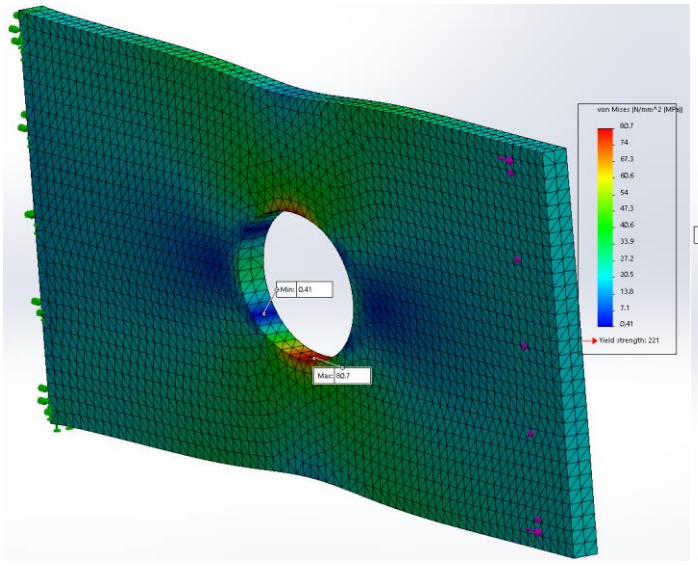
Elements through plate thickness

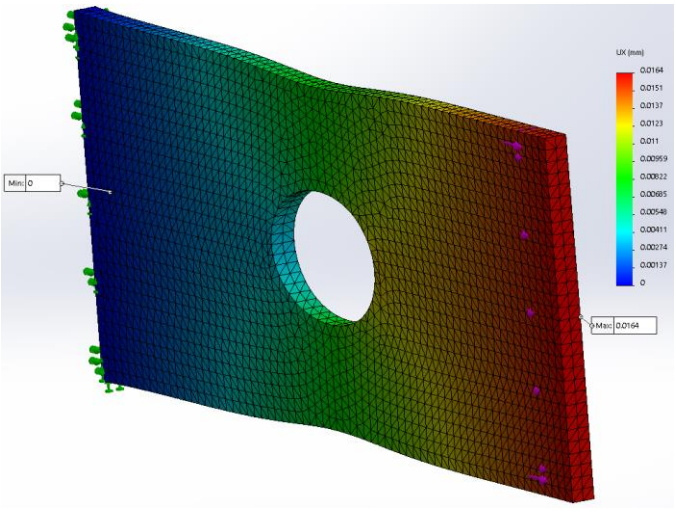
Typically, it is recommnended to have at least 3 elements through the thickness of a plate.

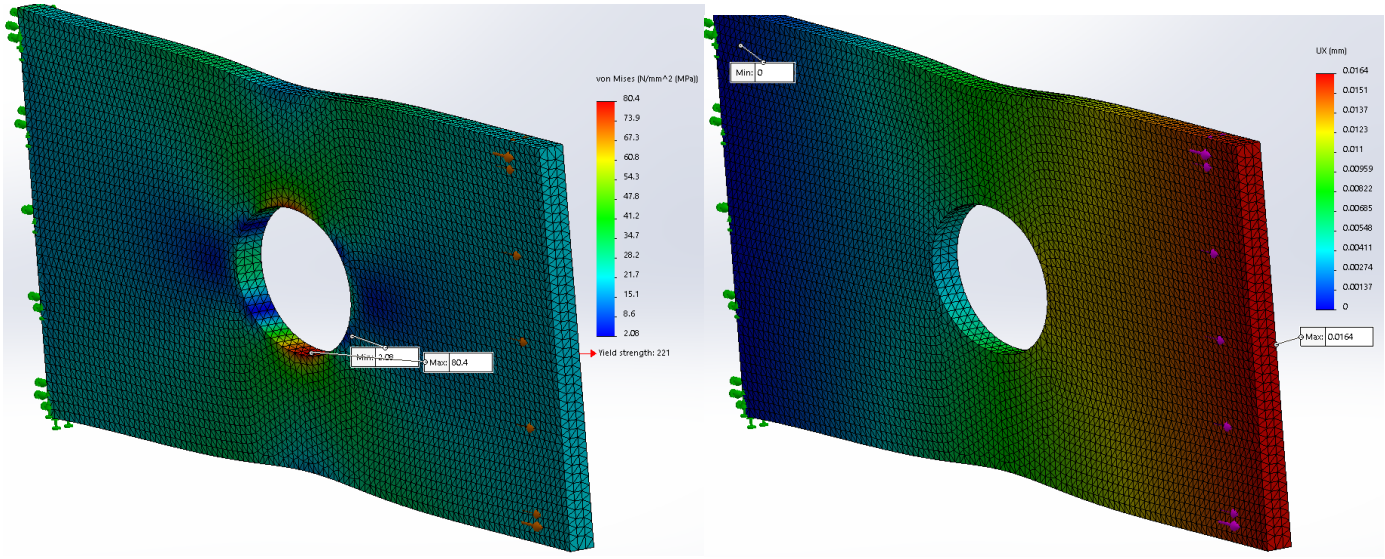
Plate thickness: 4 mm
Max element size: 4 mm (1 element across thickness), 2 mm (2 elements across thickness), 1.3333 mm (3 elements across thickness).



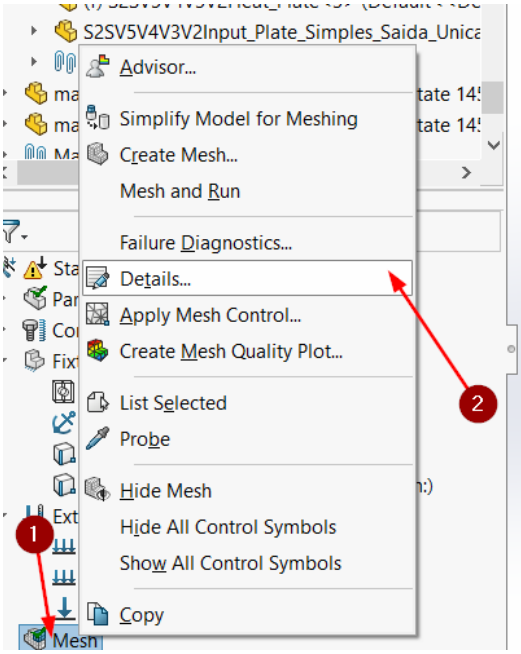








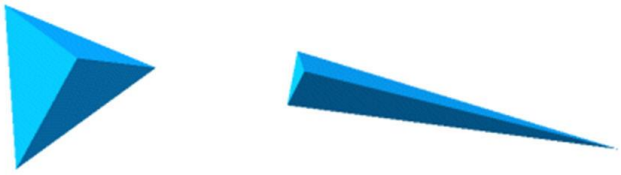
Elements' Aspect Ratio



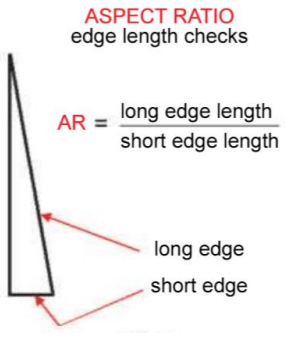
Mesh Details

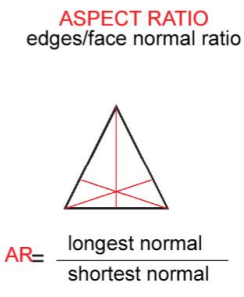
Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Blended curvature-based mesh
Jacobian points	4 points
Max Element Size	4 mm
Min Element Size	1.33332 mm
Mesh quality	High
Total nodes	37065
Total elements	21058
Maximum Aspect Ratio	11.232
Percentage of elements with Aspect Ratio < 3	95.5
Percentage of elements with Aspect Ratio > 10	0.0142
% of distorted elements (Jacobian)	0
Remesh failed parts with incompatible mesh	Off
Time to complete mesh(hh:mm:ss)	00:00:04
Computer name	ADFERREIRA

We want a low aspect ratio



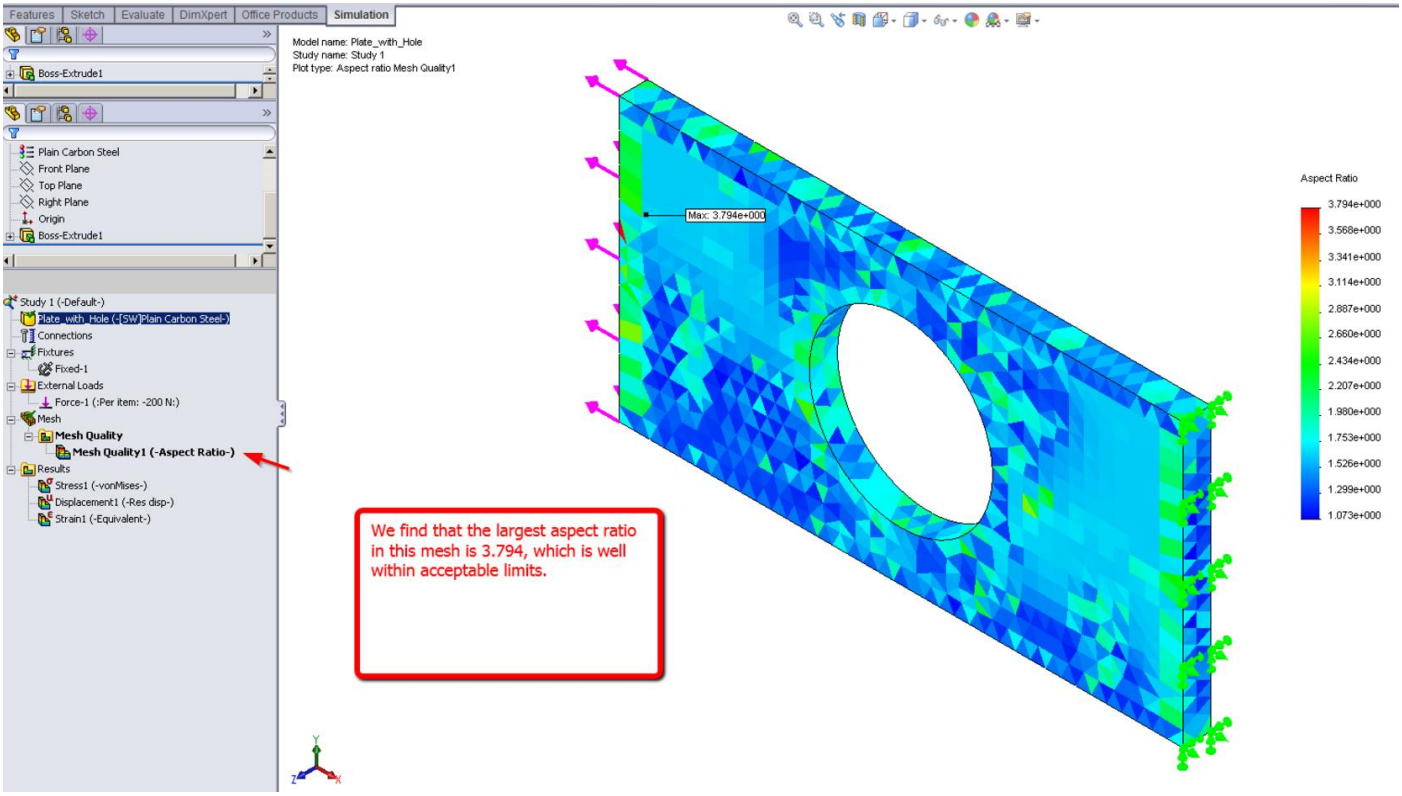
Element with aspect ratio close to 1.0 **Element with large aspect ratio**





If the aspect ratio of an element is 10 it means its largest edge is 10x bigger than the shortest. Rules of thumb:

- Keep the aspect ratio below 5 in areas where stress is of crucial importance (high stresses), because you need accurate results there.
- Don't have more than 10% of the elements with an aspect ratio higher than 10. A way to fix this can be to introduce mesh controls in the places where the AR exceeds the value you want. Extremely large values >> 40 should be closely examined to determine where they exist and whether the stress results in those areas are of interest or not

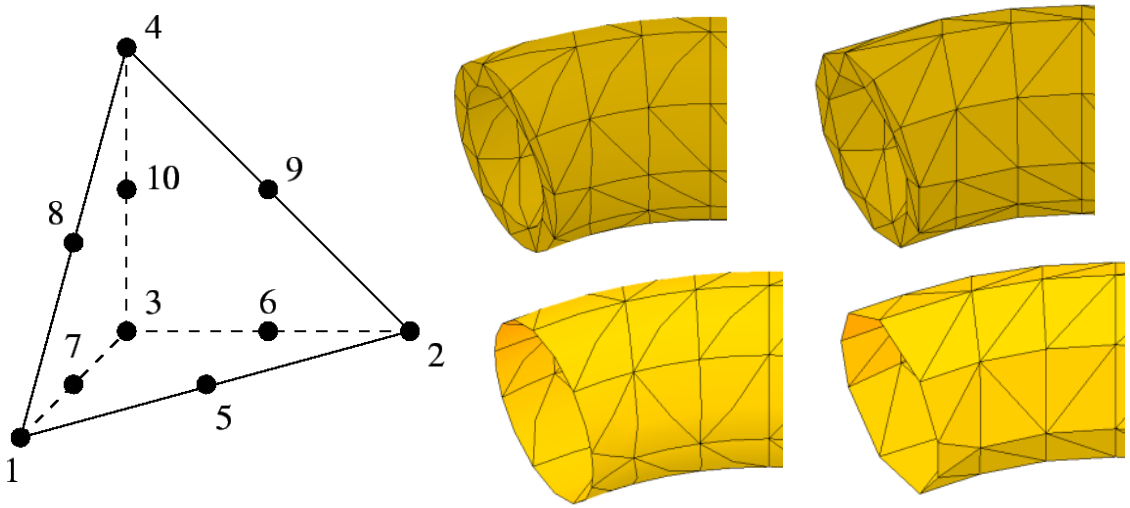


Draft and High Quality Tetrahedral elements

There are two types of solid tetrahedral elements in SolidWorks Simulation

- First order (Draft Quality) Tetrahedral: 4 nodes
- Second order (High Quality) Tetrahedral: 10 nodes

A similar thing happens with shell triangular elements.



Notice how the parts on the left side better represent the curvature of the piece due to the increased number of nodes in the existing elements.

The second metric used to determine mesh quality is the Jacobian Ratio. This method is only available for second order (High quality) mesh elements. Parabolic (second order) elements are able to map curvilinear geometry more accurately than the first order linear elements. The midside nodes are placed on the actual geometry of the model, and in extremely sharp or curved boundaries, the edges can cross over each other. This can result in a negative jacobian ratio which will cause the solver to fail.

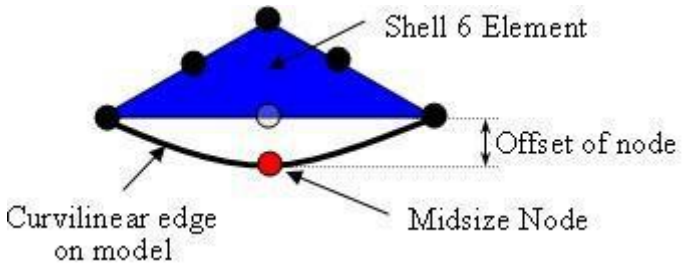


Figure: The Jacobian is a measure of the curvature of the edge and distortion at the mid-side node.

This is a 2-D representation of an element with a negative Jacobian Ratio. The curvature of the geometry has caused the edges of the element to cross over each other and resulted in an element that will cause the solver to fail.

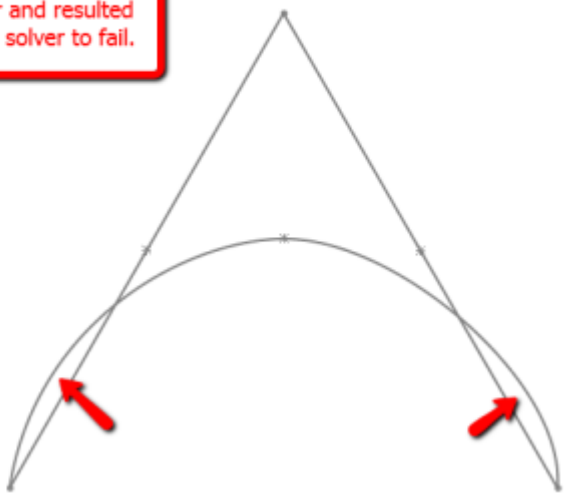


Figure: This is an example of a 2-D representation of an element with a Negative Jacobian ratio. The curvature of the geometry that the element was trying to map was too great for the size of the element, causing the edge to collapse in on itself creating a negative jacobain ratio. This will cause the solver to fail.

As with the first order elements and the aspect ratio, the jacobian ratio of a perfect tetrahedral element with linear edges is 1.0. The jacobian ratio of an element increases as the curvature of the edges increase and are calculated at the selected number of Gaussian Points for each tetrahedral element. In general, elements with a jacobian ratio less than 40 but not negative are acceptable.

You can also create a Mesh Check Plot similar to the Aspect ratio check in SolidWorks.

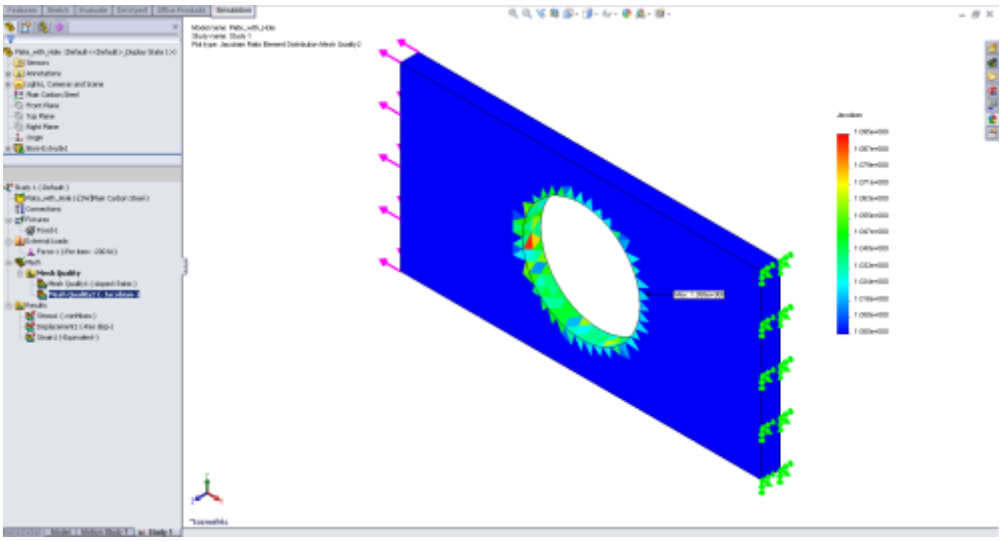
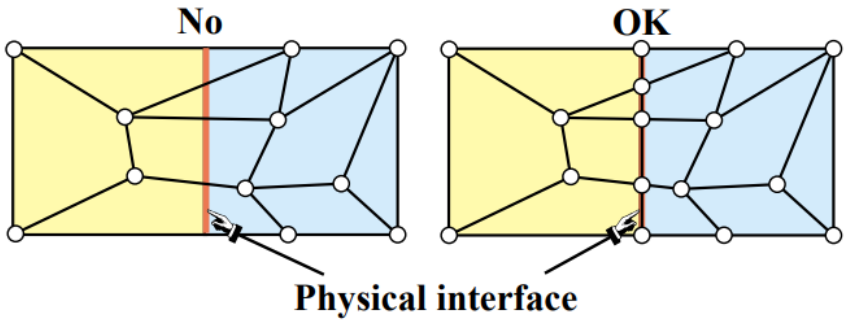


Figure: The Jacobian check plot shows that the only areas where there are elements with a non-unity Jacobian value are areas where there is actual curvature of the geometry. Here the maximum value of the Jacobian is 1.095, and there is no need to further refine the mesh.

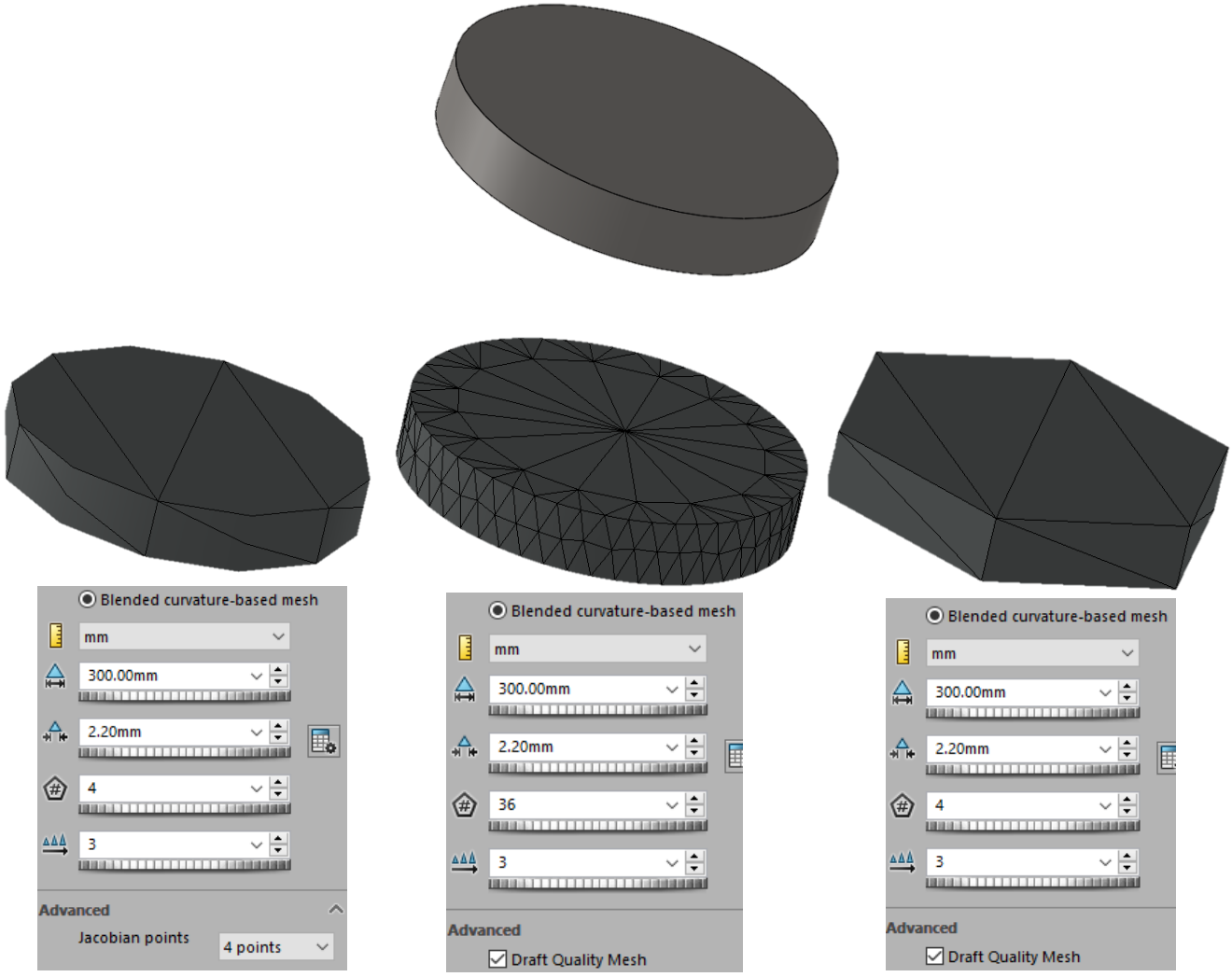
Elements Must Not Cross Interfaces

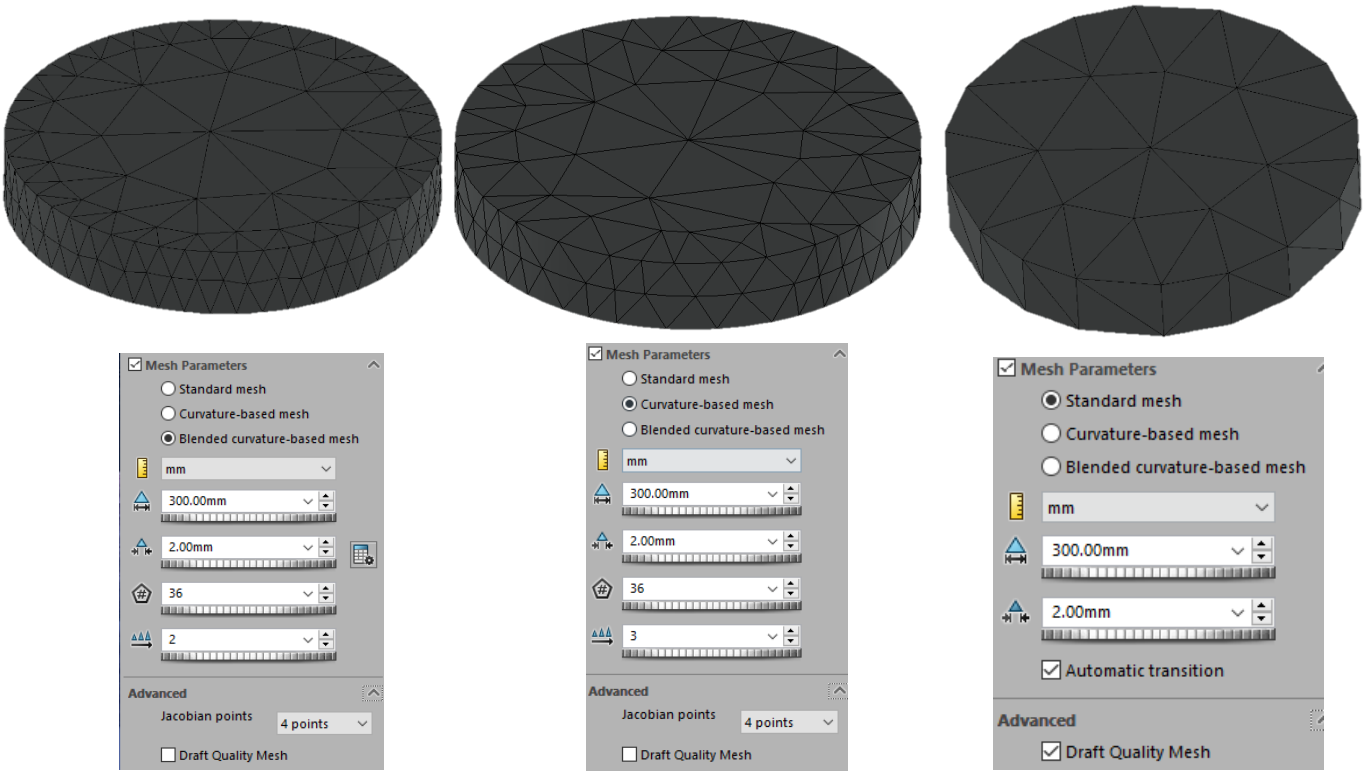


The first unknown in linear static FEA that is solved is the displacement of each node. Based on this, the stresses and strains are calculated. The stress values at nodes in an FEA study are calculated at Gauss, or Quadrature points in the element, and then averaged with the stress values from the surrounding elements. While the displacements are solved explicitly at the nodes, the

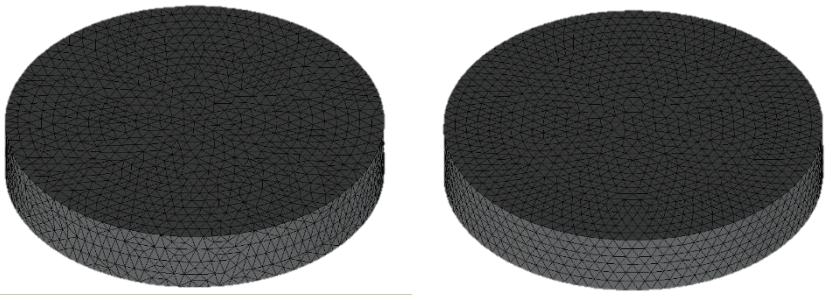
stresses are an averaged value and if there are insufficient stress values present in an area, the stress value averaged at the node can be inaccurate. Calculations are first made for the nodes and then for stresses, which are averaged between themselves. That makes sense within a part or material, but not between materials or parts.

Standard Mesh vs Curvature vs Blended Meshes





Automatic transition automatically applies mesh controls to small features, details, holes, and fillets. Uncheck Automatic transition before meshing large models with many small features and details to avoid generating a very large number of elements unnecessarily.

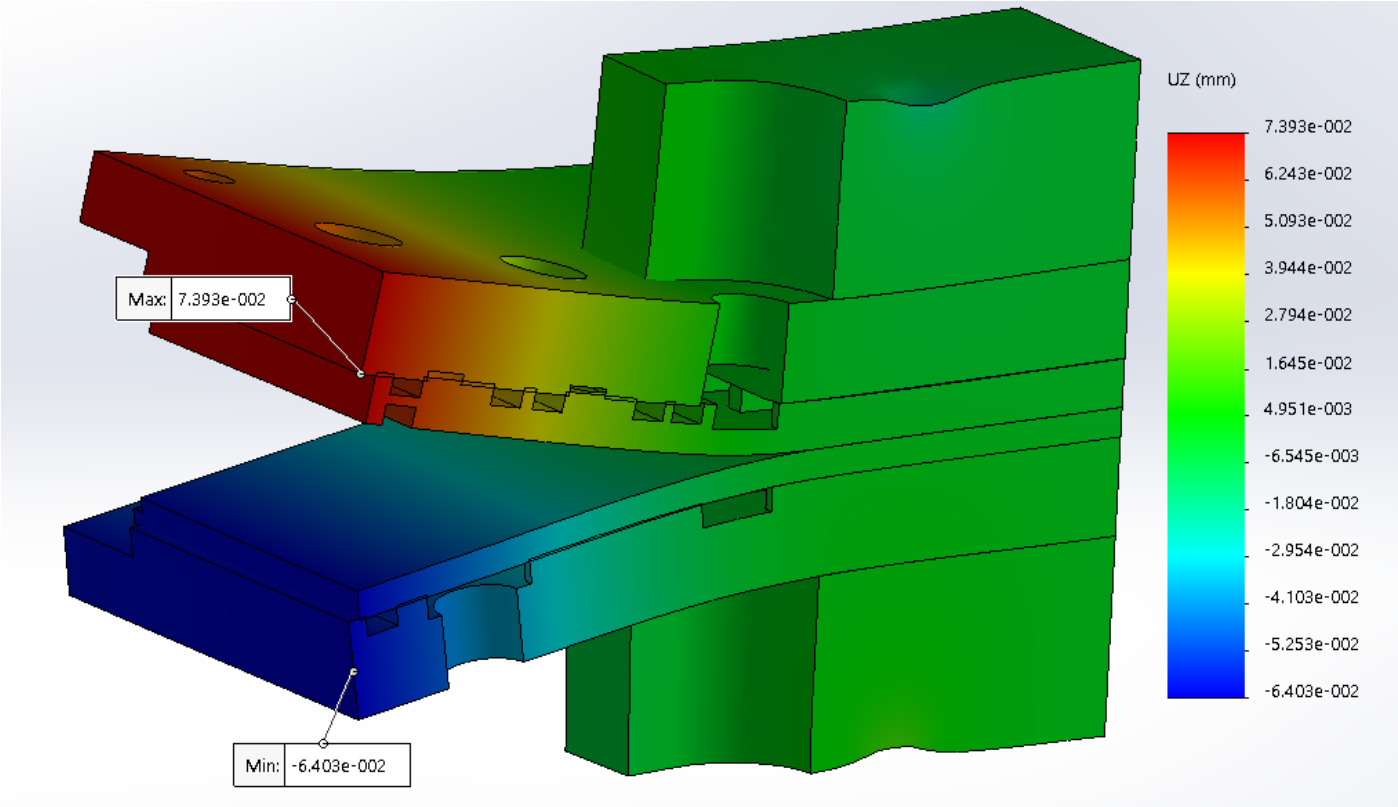


Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Curvature-based mesh
Jacobian points	4 points
Max Element Size	10 mm
Min Element Size	2 mm
Mesh quality	High
Total nodes	43014
Total elements	28911
Maximum Aspect Ratio	3.7223
Percentage of elements with Aspect Ratio < 3	99.8
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:02
Computer name	ADFERREIRA

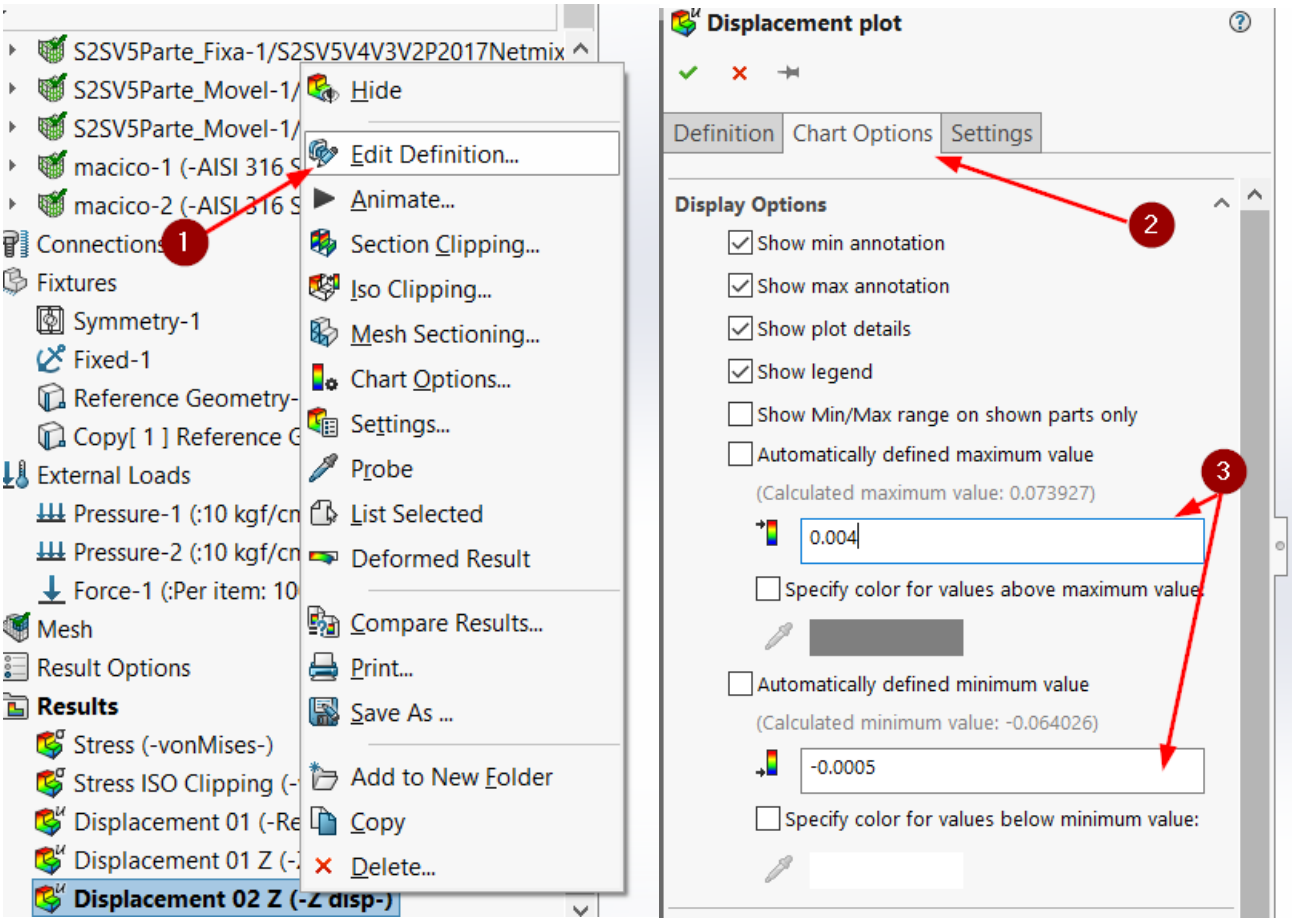
Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Blended curvature-based mesh
Jacobian points	4 points
Max Element Size	10 mm
Min Element Size	2 mm
Mesh quality	High
Total nodes	50461
Total elements	34341
Maximum Aspect Ratio	3.6787
Percentage of elements with Aspect Ratio < 3	99.9
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:02

Blended takes a bit longer, but results in a better mesh (elements are more regular, with lower aspect ratio) and lower number of elements.

A good mesh will put the elements in the regions of geometry change as they tend to be stress concentrators, and thus have higher stresses, and so that's where the part will tend to break first.



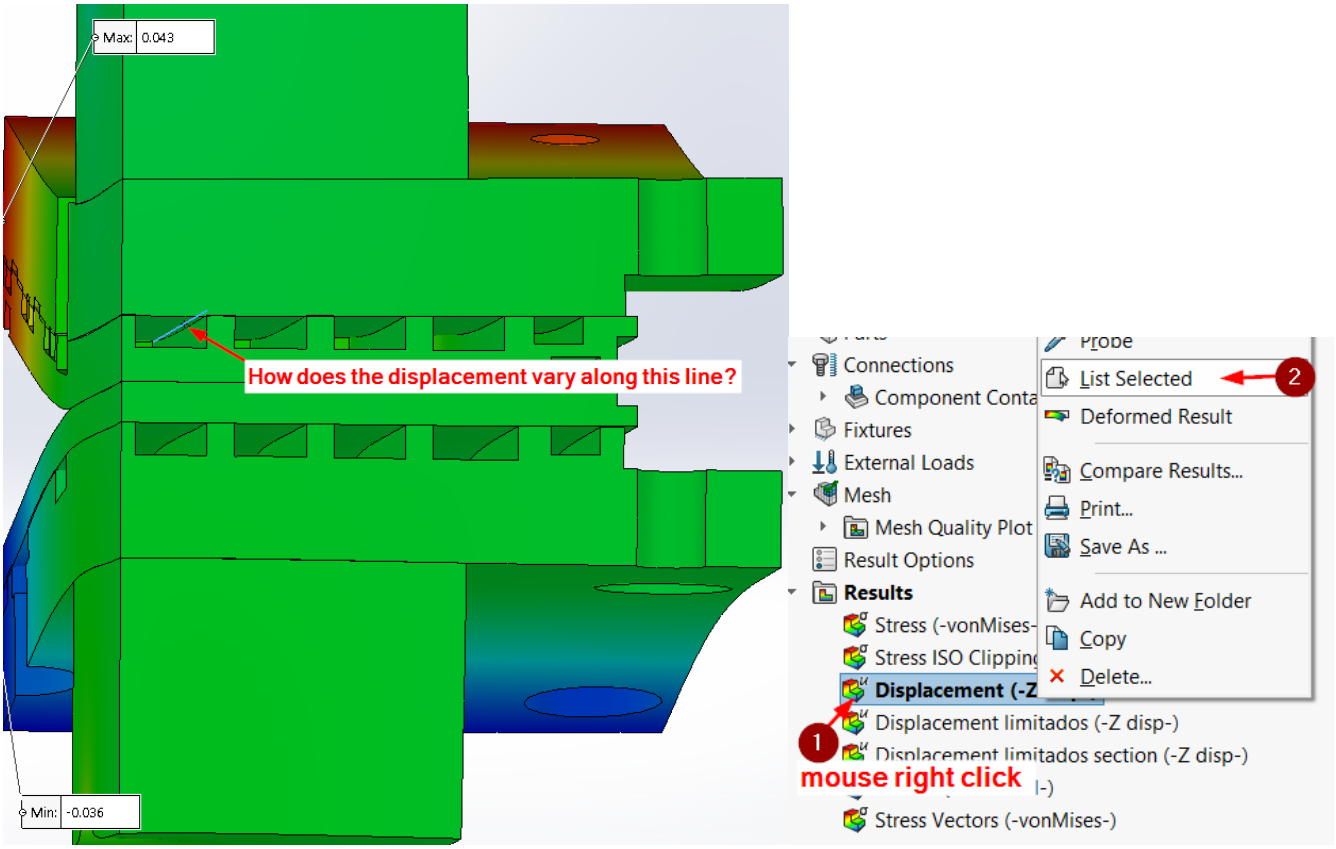
Imagine we see that the displacements in the "mouth" are ok and we wanted to focus on the displacements on the body (green area). We can set limits:

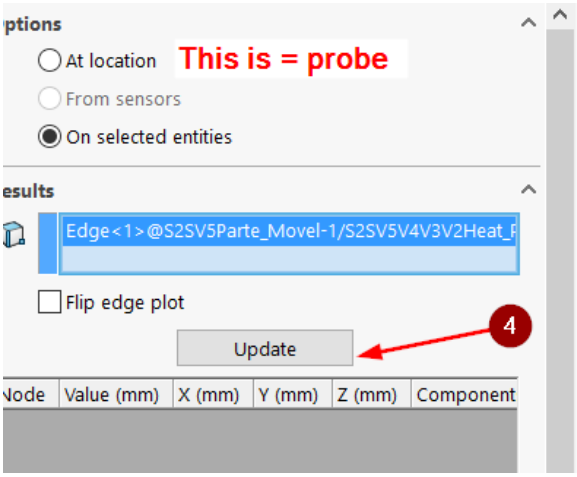


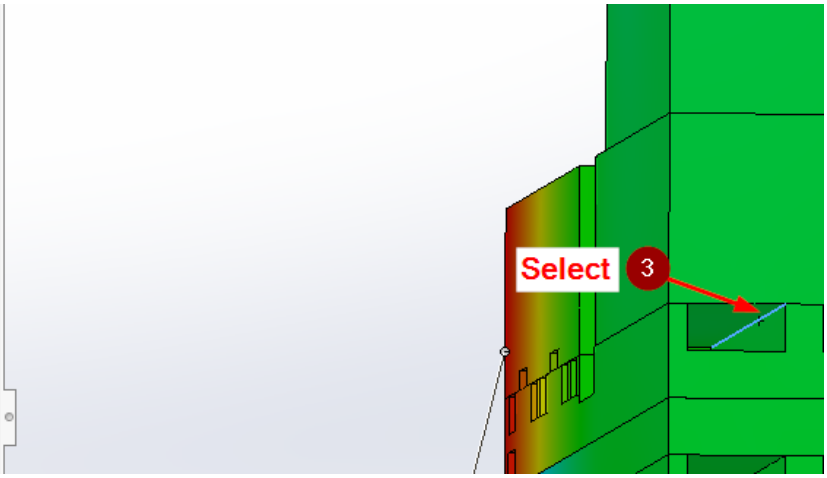
Everything above the maximum will be red.

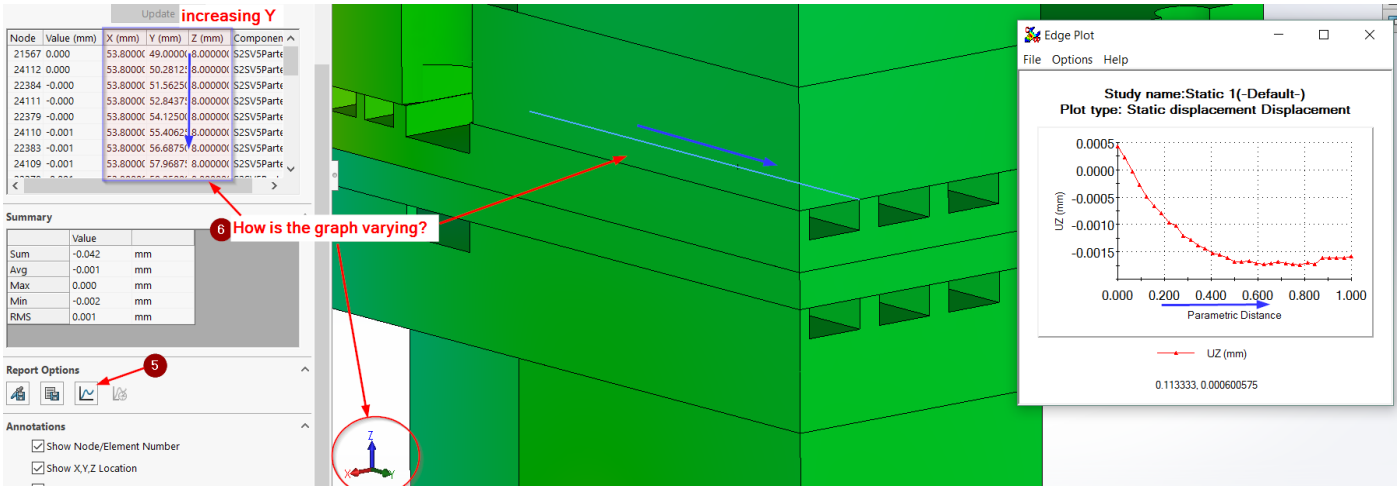
Pro-tip: Instead of right clicking and then going to “Edit Definition”, we can simply double click on the result and it takes us directly to the same page.

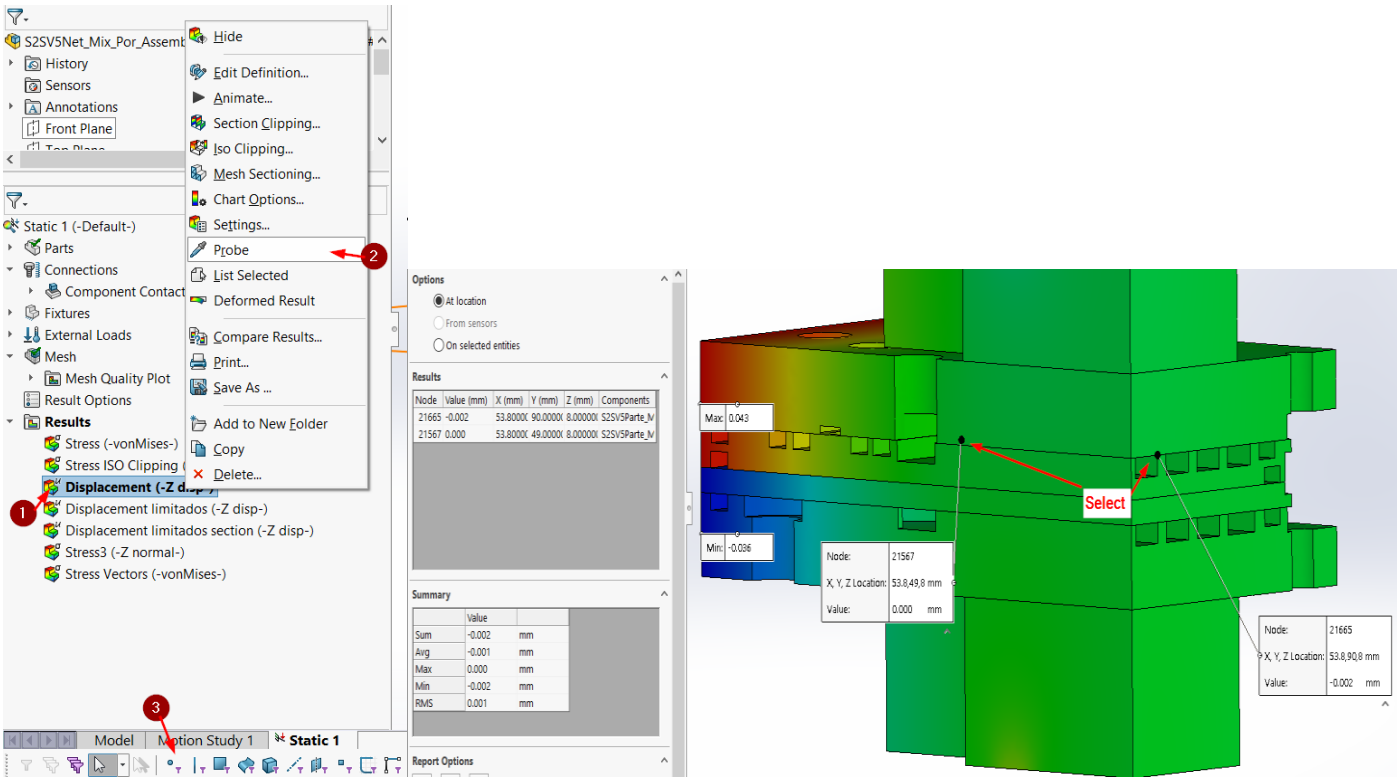
Variation along a line and values on a point



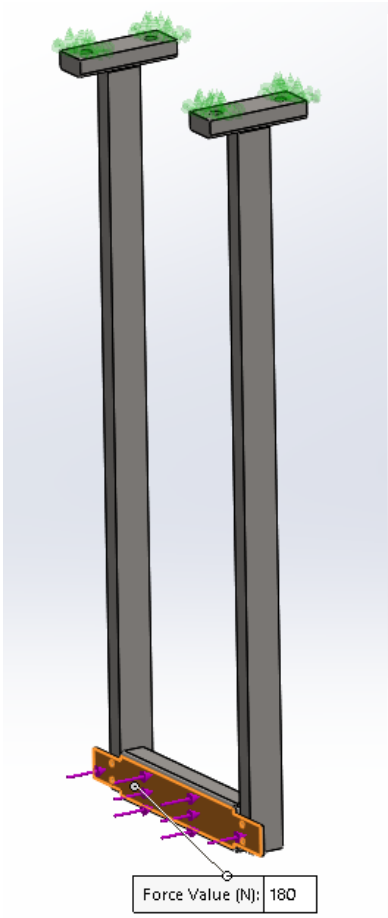


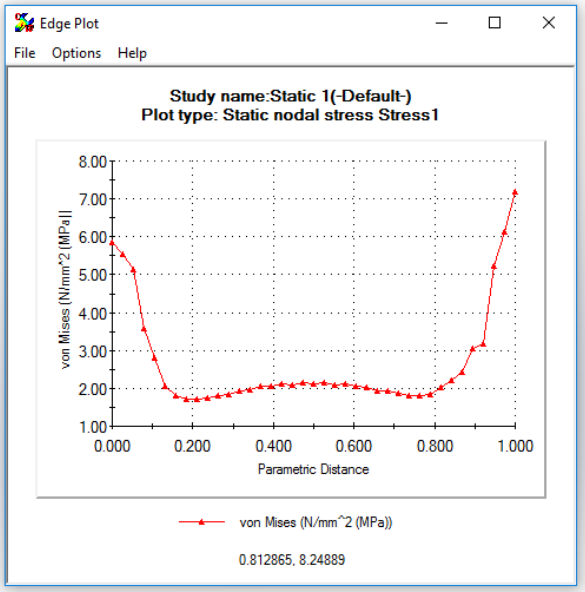


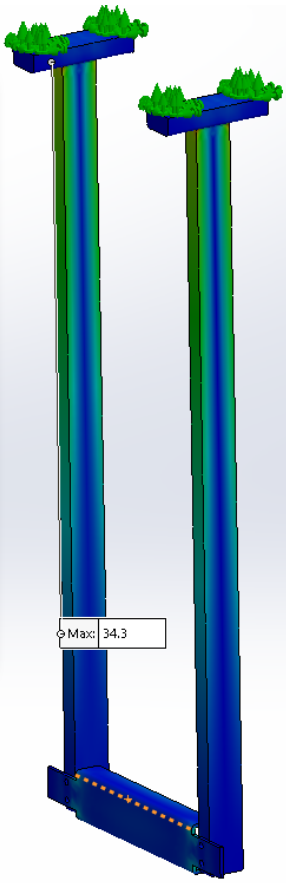


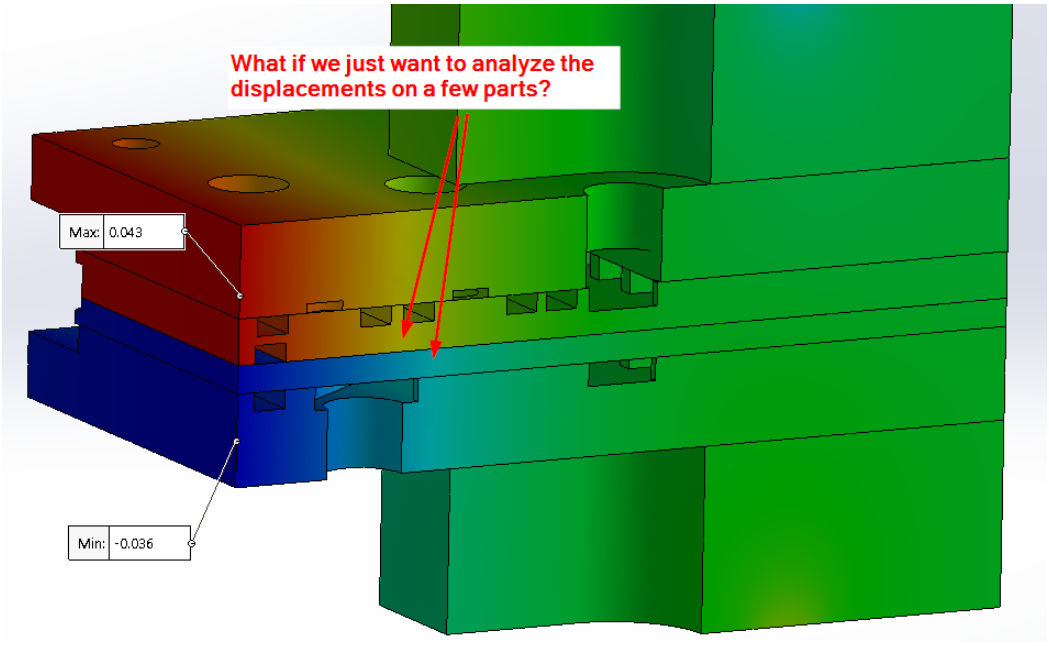


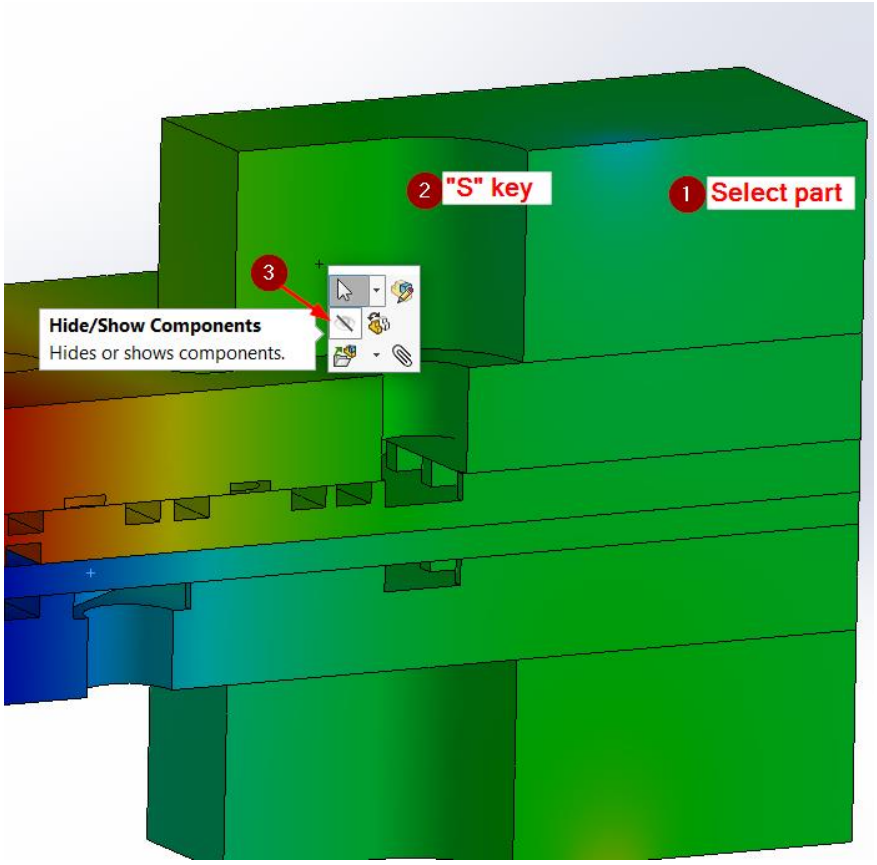
Another simpler example


Force Value (N): 180


Edge Plot
File Options Help
Study name:Static 1(-Default-)
Plot type: Static nodal stress Stress1
von Mises (N/mm^2 (MPa))
8.00
7.00
6.00
5.00
4.00
3.00
2.00
1.00
0.000
0.200
0.400
0.600
0.800
1.000
Parametric Distance
von Mises (N/mm^2 (MPa))
0.812865, 8.24889


Max: 34.3

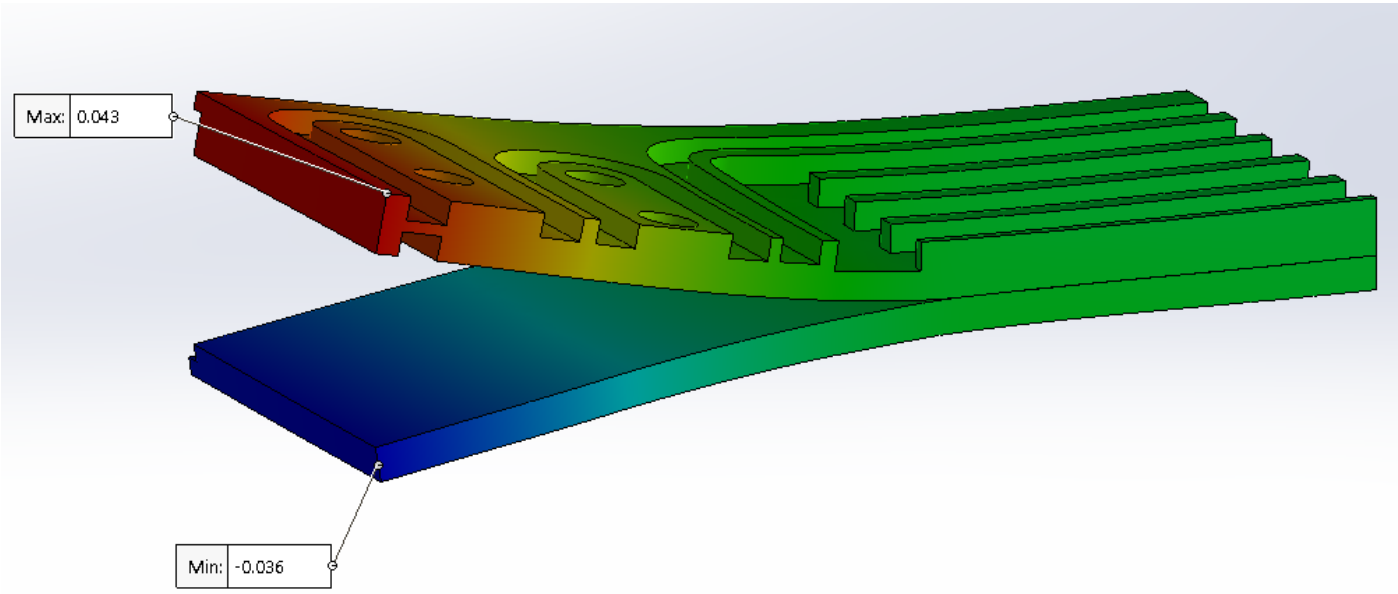

What if we just want to analyze the displacements on a few parts?
Max: 0.043
Min: -0.036



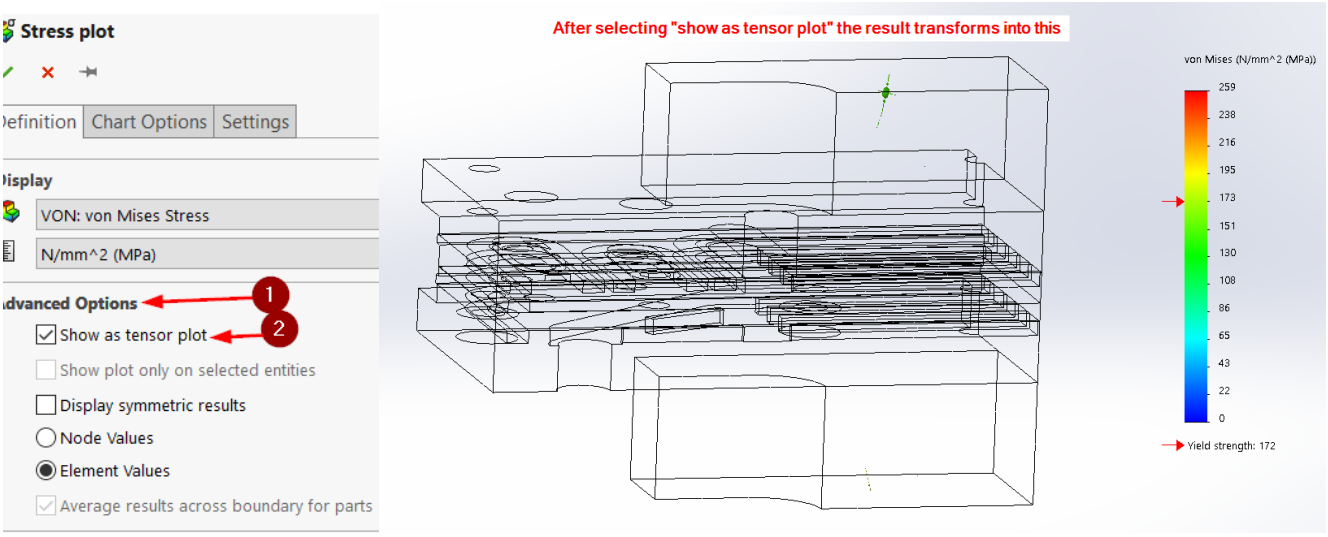
Sometimes this may get buggy, and you may need to

1) Select a point or line from the body to be able to select it
2) It may appear as if nothing is happening. In this case you'll need to reopen the result (double click on it).
3) To unhide

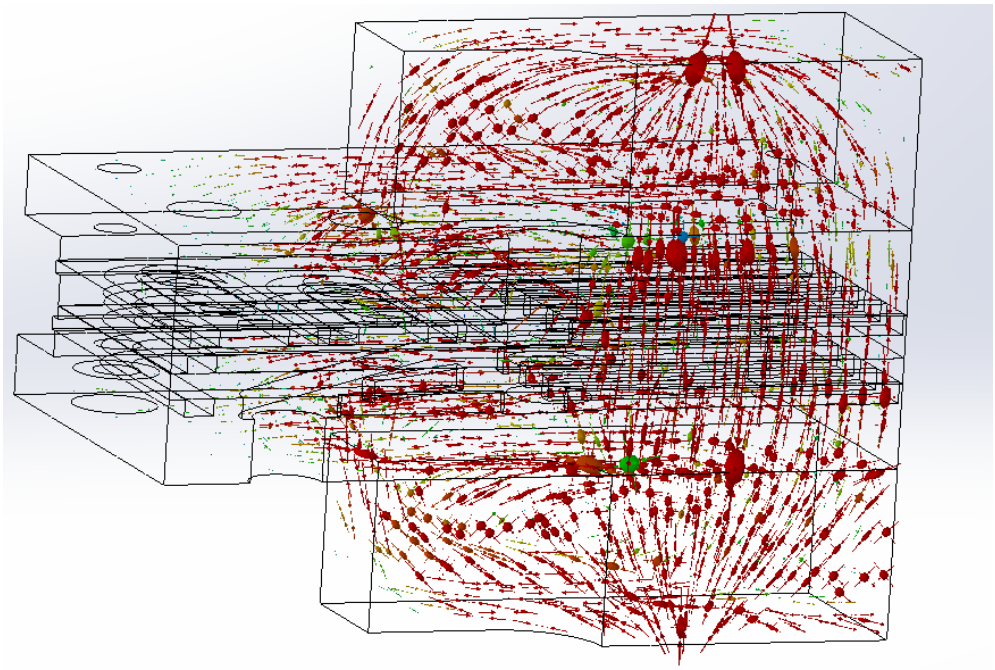
Result



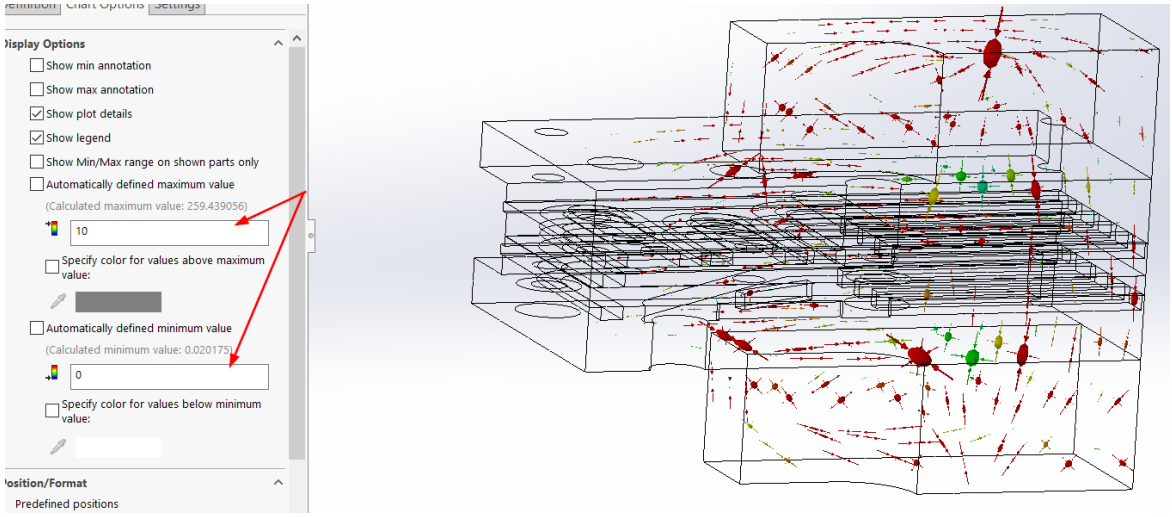
Vectorial View / Stress Flow Lines

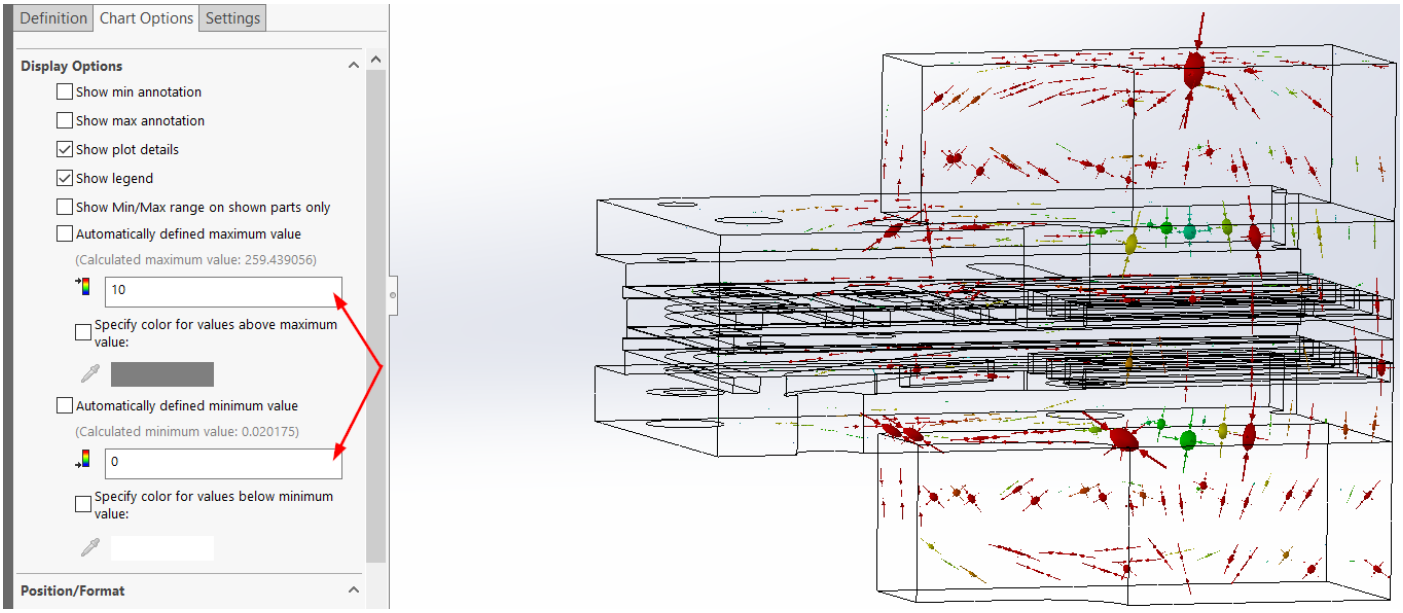


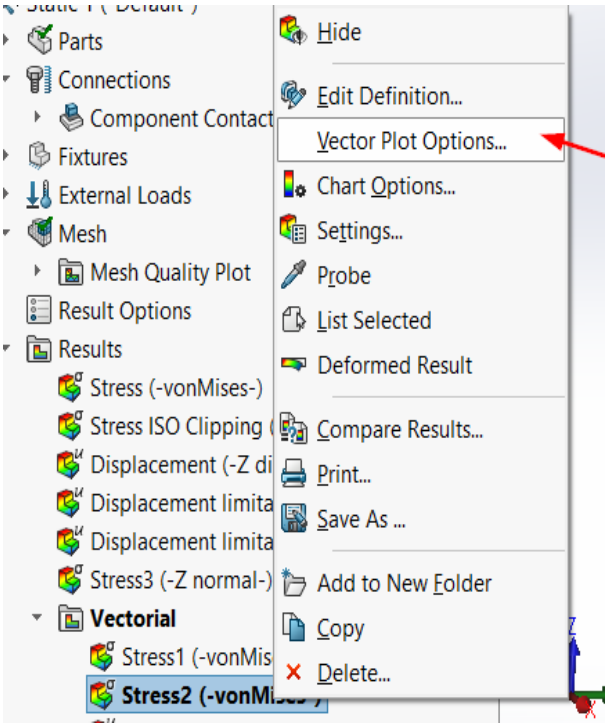
Let’s get that into this (stress flow lines)

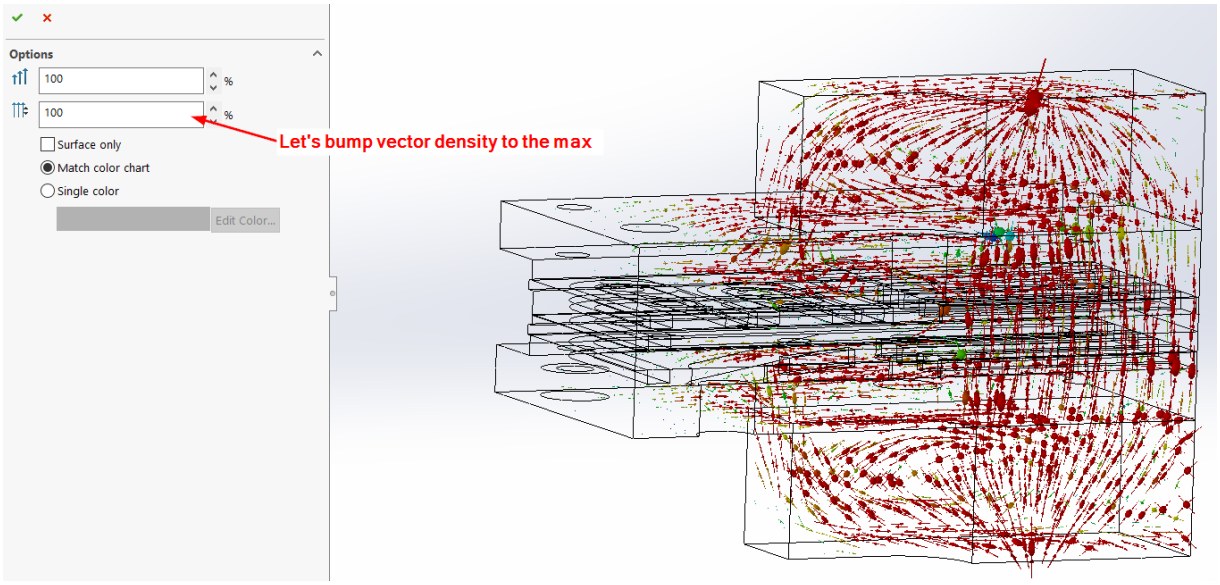


Let’s limit the stress range

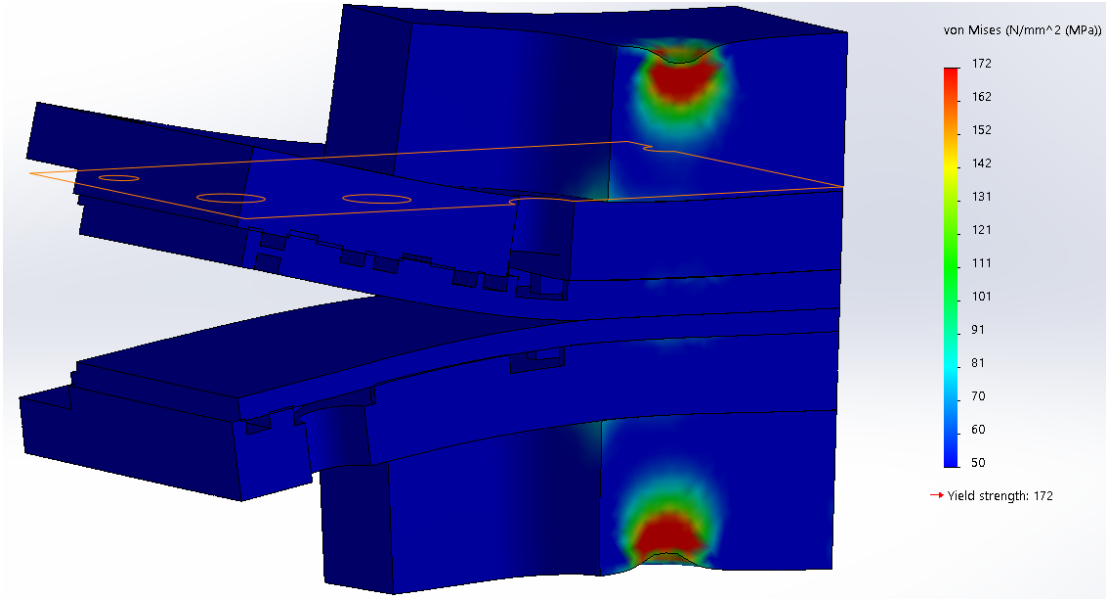








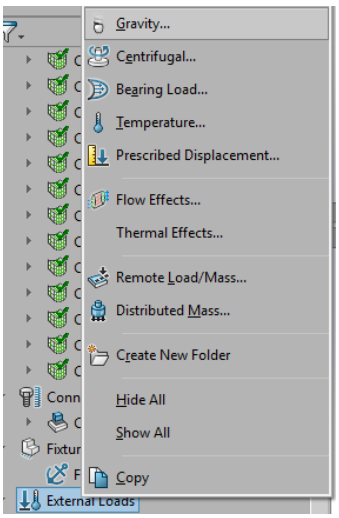
Let’s see what’s above the yield strength and that is above 50 MPa.

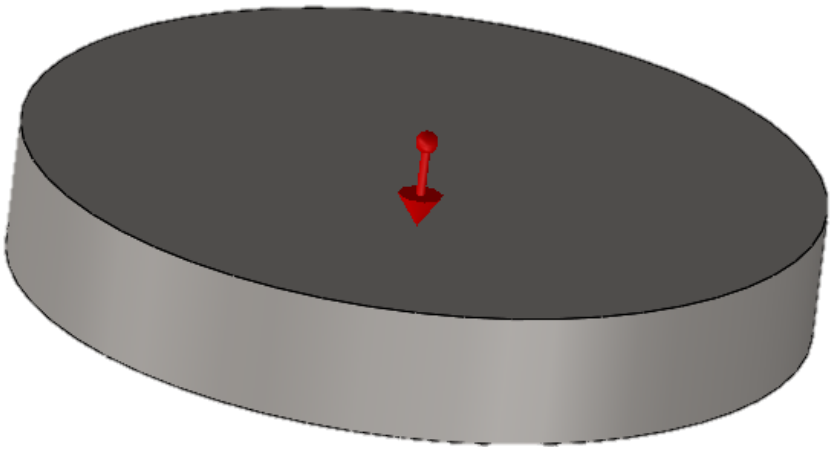


Load factor

Other

- Don’t forget to add gravity.

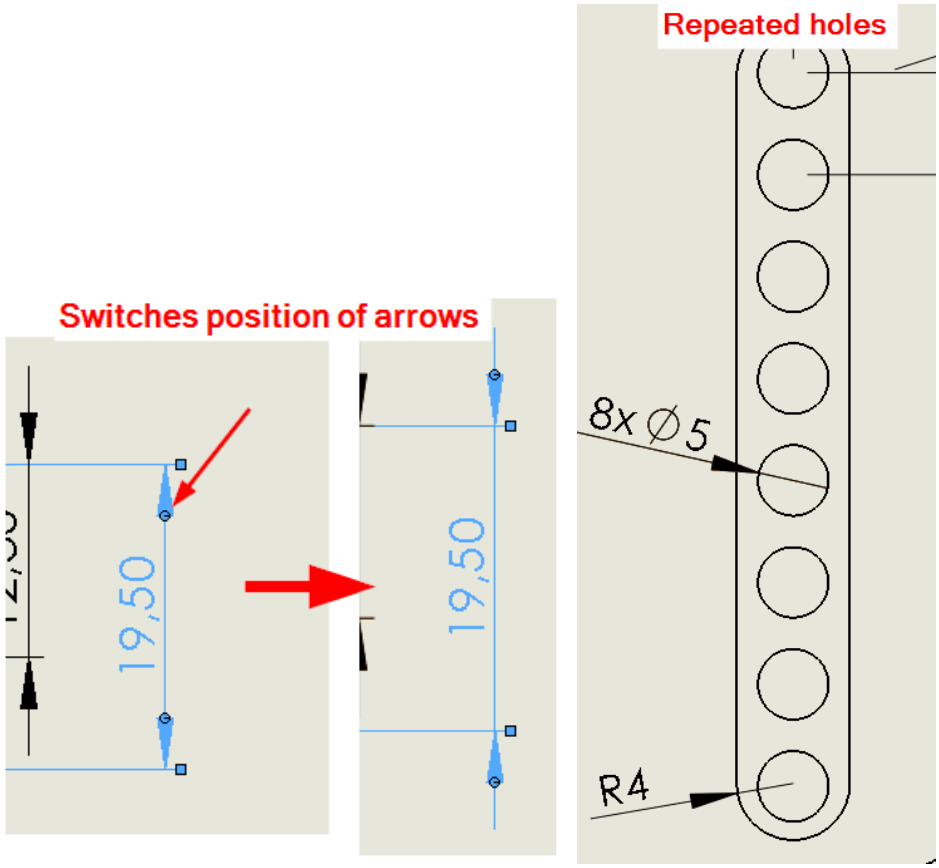




Drawings

Don’t put dimensions for all repeated things.

For holes: 16xR6 (16 holes of radius 6). Their dimensions are generally put from above (why?)



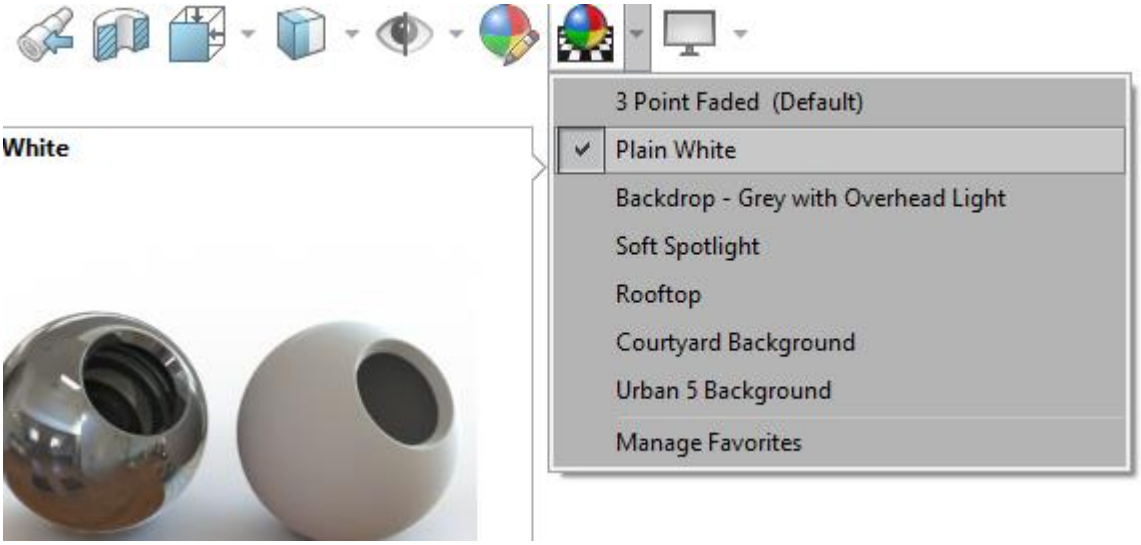
Mass PDF Exporting

To mass export drawings as PDFs: Open Solidworks Task Scheduler >> Export files >> Escolher pasta e tipos de ficheiro (origem e destino).

Appearance & Rendering

Fast Pretty Prints

Put white background (plain white).



FeatureWorks

FeatureWorks allows SolidWorks users to recognize and automatically create parametric features from imported geometry.

How to get more appearances for SolidWorks?

There's a folder that comes with your SW install. In the appearances point to this folder:

C:\Program Files\SolidWorks Corp\SolidWorks\data\Images\textures

Turning on RealView

1.) Open up the registry editor (Start Button -> Run... -> "regedit")
2.) Navigate to "HKEY_CURRENT_USER\Software\SolidWorks\SOLIDWORKS 2015\Performance\Graphics\Hardware\Current"
3.) On the right side doubleclick on "Renderer"
4.) copy (Strg+C) the Value
Since I have a GeForce Card mine is "GeForce GTX 970/PCIe/SSE2". Yours will be different, mentioning an AMD card. It is important to copy ALL what's in value.
5.) Navigate to "HKEY_CURRENT_USER\Software\SolidWorks\SOLIDWORKS 2015\Performance\Graphics\Hardware\Gl2Shaders\R420"
6.) On the left side (the treeview) rightclick on "R420" and choose "New / Key"
7.) Rename the new key with the copied value (Strg+V)
8.) Click another Key and back to your newly created to make sure you are editing the right values. The right side should be empty but an entry (Default) whose data is "value not set"
9.) rightclick in the right side and create "New / DWORD (32-bit) Value"
10.) Rename the DWORD to "Workarounds" and, after that, doublecklick it
11.) Leave the "Base" to "Hexadecimal" and enter 30008 into the field "Value data"
12.) Leave the Registryeditor open and start Solidworks. Create a new part / open an old one and try to enable RealView.

If RealView is still greyed out:

13.) Close Solidworks and go back to the registry editor. Doubleclick the DWORD "Workarounds" and change the "Value data" to 30408
14.) Fire up Solidworks and try again to activate Realview (Part or Assembly open)

Hi Nicolas, I had same problem and I solved it.
This is pretty old topic, but I write solution for this anyway if someone needs it in future.

I'm not sure how this work with AMD cards, but with all Nvidia Geforce cards this should work.

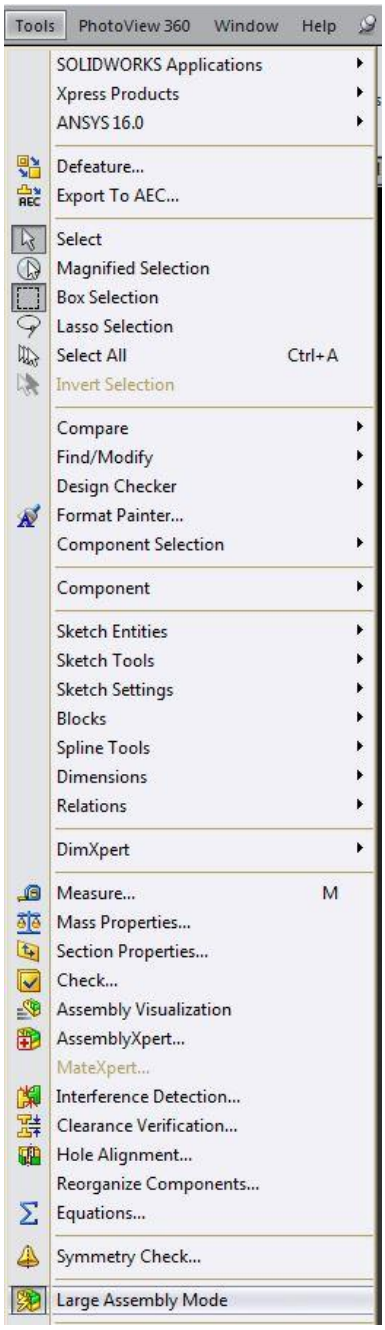
1.Open the registry (Run -> regedit)
2.Navigate to "HKEY_CURRENT_USER>SOFTWARE>Solidworks>SOLIDWORKS 2015>Performance>Graphics>Hardware>Current"
3. Double click on the "Renderer" key and copy the text under "Value Data". For example, for th GTX 770 mine was "GeForce GTX 770/PCIe/SSE2"
4.Navigate to "HKEY_CURRENT_USERSOFTWARE>Solidworks>SOLIDWORKS 2015>Performance>Graphics>Gl2Shaders>NV40"
5.Right-click on the NV40 folder on the left pane and select "New -> Key"
6.Rename the new key with what was copied in step #3
7.The only key in this folder should be a "(Default)" key
8.Right-click on the right pane and select "New -> DWORD (32-bit) Value"
9.Name this new key "Workarounds"
10.Double-click on the "Workarounds" key and enter the value "30408". You should be in Hexadecimal mode.
11.RealView should now be available. If it is not, try setting the value in step #10 to "31408" or "30008"

hope this works for you.

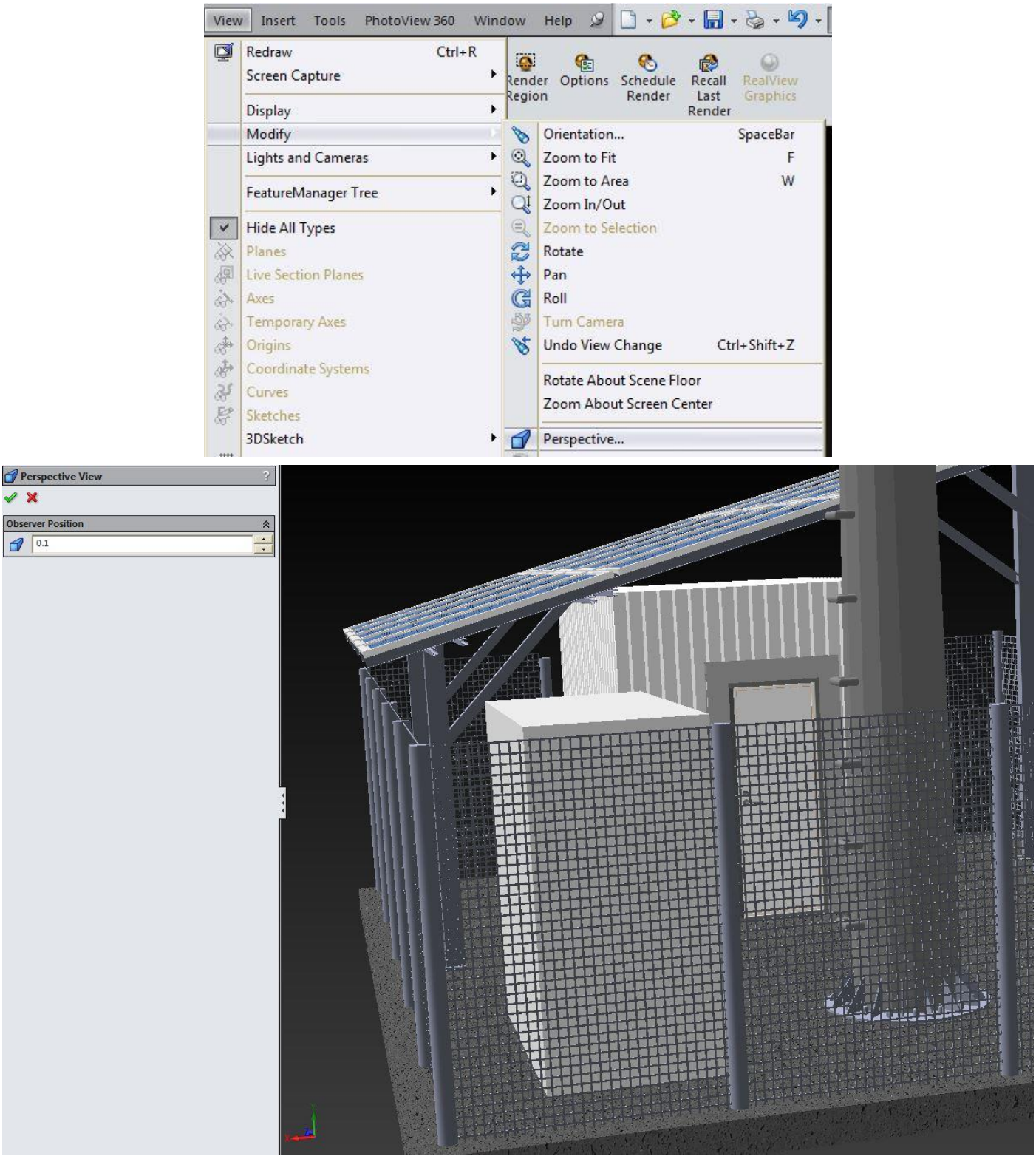
For anyone looking into doing this, be sure to add the new key (in steps 5 and 6 of Leucetius' answer) to the correct folder for your graphics card. You will probably need to find the equivalent workstation card to your graphics card, then find that card's folder under GL2Shaders, then add your new key to that folder. For example, I have an Nvidia Geforce GTX 560, which uses the Fermi micro-architecture. So I Google'd the Quadros, and found that the Quadro 600, 5000, and 6000 are also based on Fermi, and those three Quadros are under the NV40 folder in the registry. So, for steps 5 and 6, I used "NV40" instead of "R420".

In short, "R420" is specific to the original poster's card, you will need to match your card to the proper folder.

You may also need to disable Large Assembly Mode



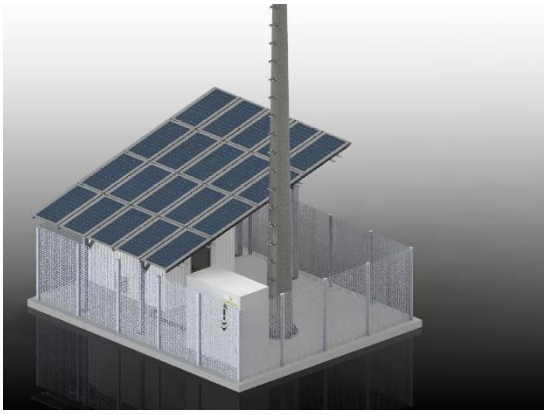
Improving perspective

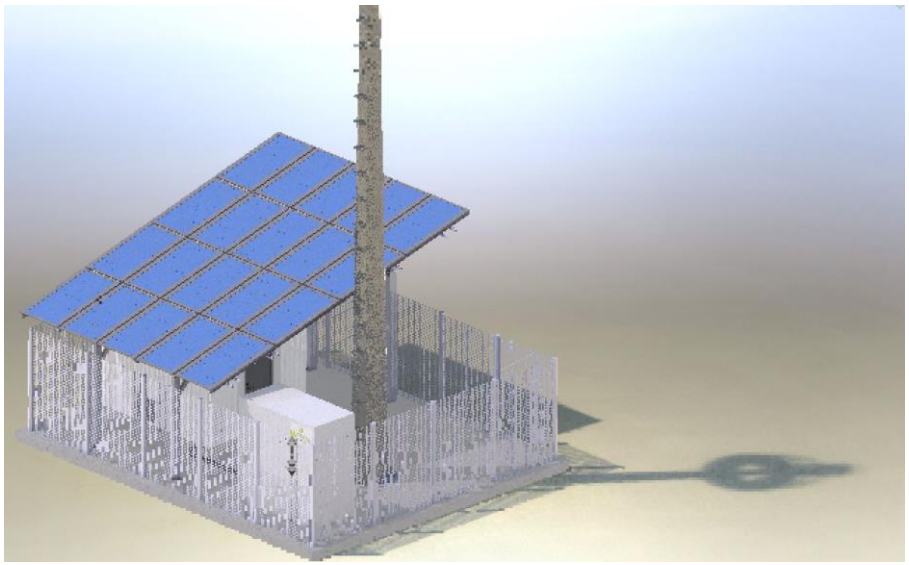


Observer position means that we're viewing the thing x times the approximate size of the thing. In this case this station has about ~6m so we're viewing it from approx. 0.6m of distance.

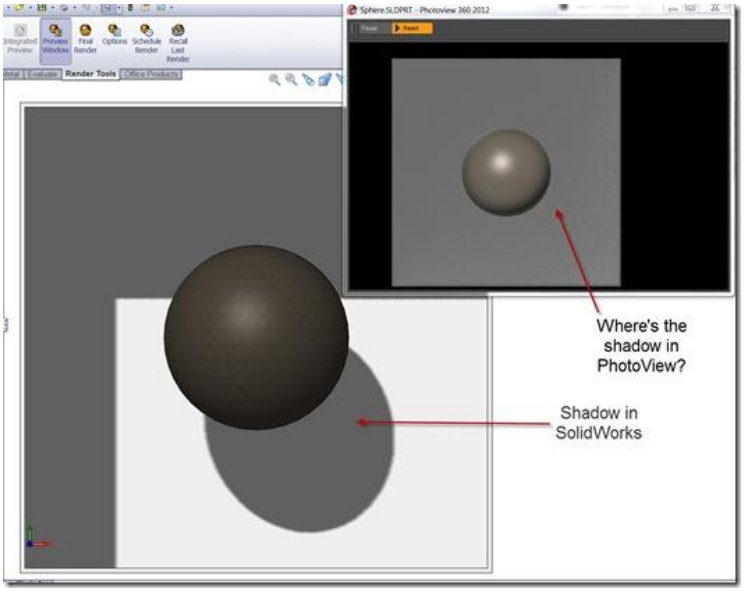
Rendering things

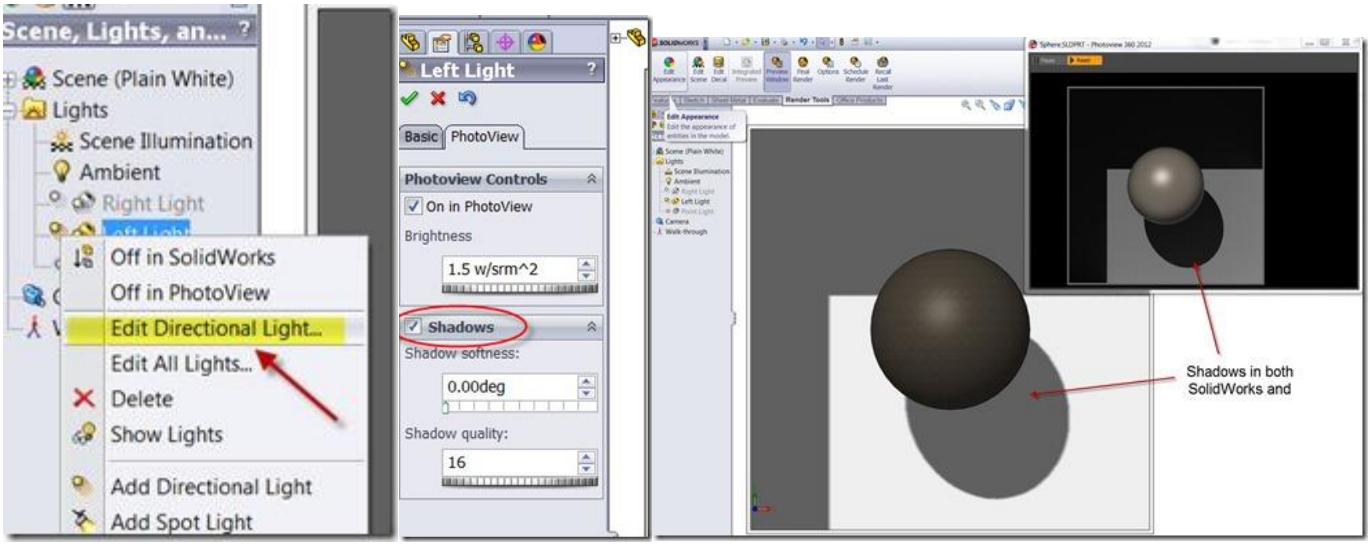
Black & White gradient background





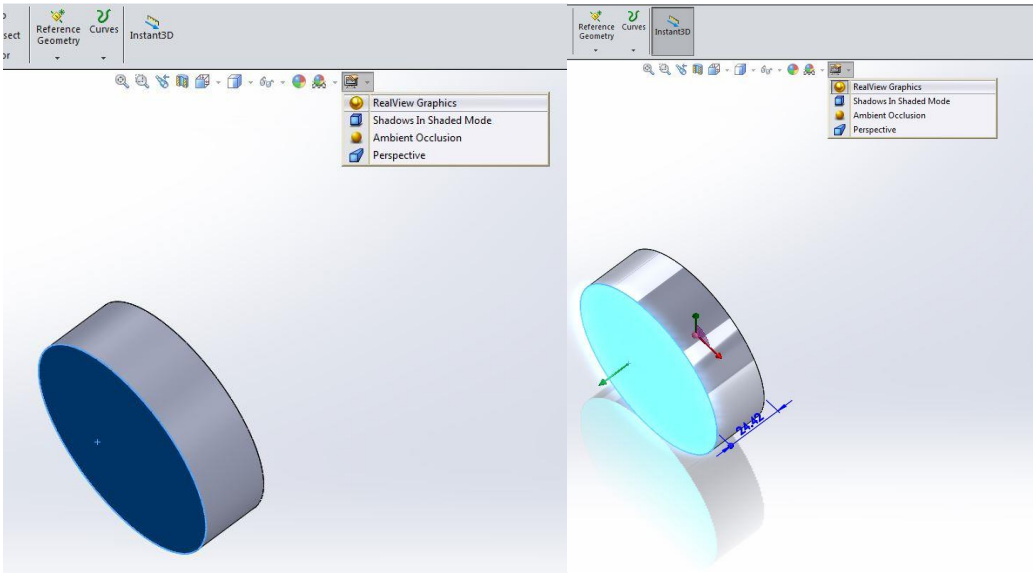
Shadows that don’t appear in Photoview



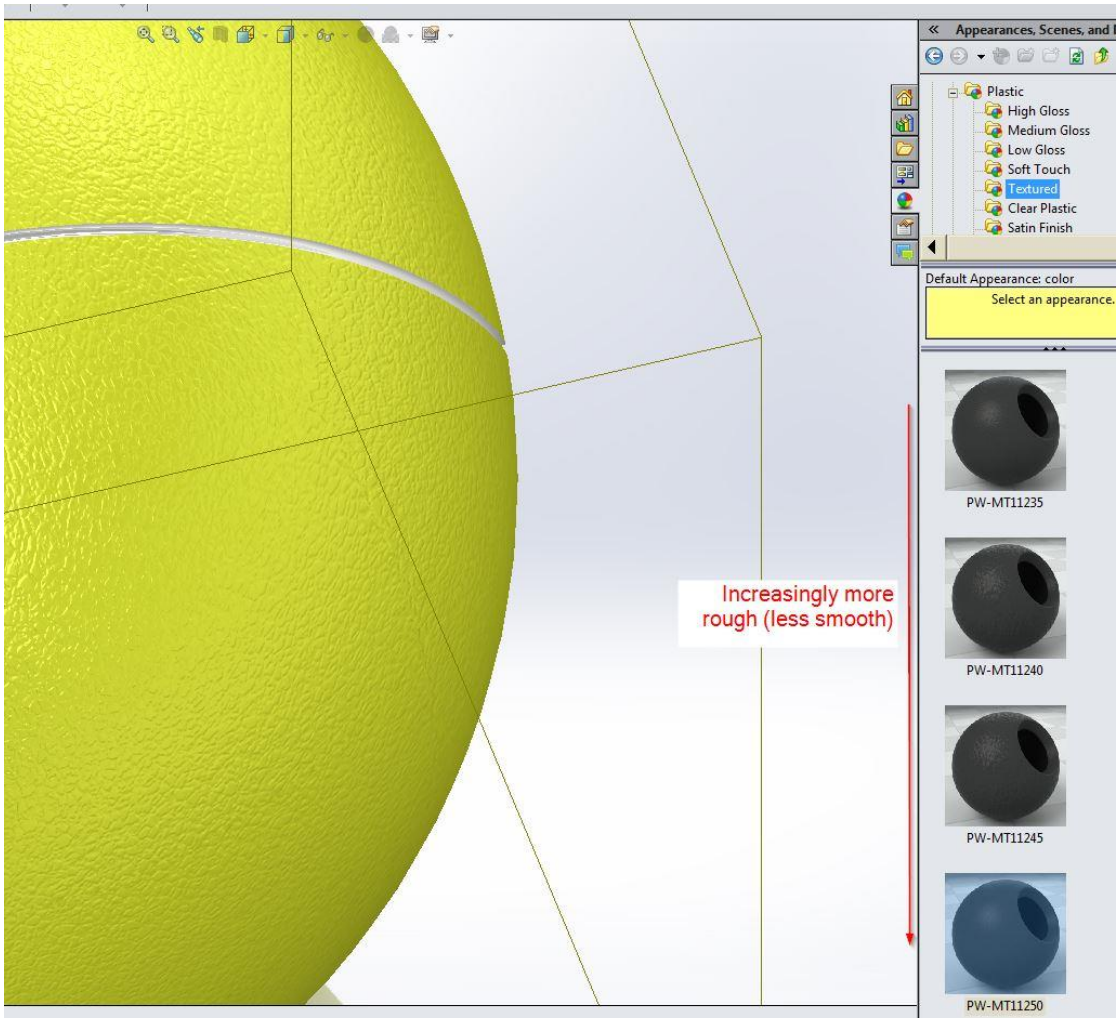


Instant 3D and RealView

With Instant 3D and RealView off and then on. RealView gives the pieces a more realistic appearance, and that light blue glow. Instant 3D enables dragging and changing dimensions in 3D.



Textured Surface



Other

Making stuff lighter

Start perhaps with the heavier parts.

Assembly Visualization

File Name	Quantity	Total Weight
C10 Patins	2	20504.8
Teto C4 Chapa sup flange	1	16241.9
C10 Dente direito	8	12060.1
80x80LBx1300.00	2	11099.3
C10 sup batente	4	7154.6
C10 chapa dente suporte	2	6801.3
C10 chapa dente suporte comp...	2	6801.3
C10 Perfil Maytech 43L sup verti...	4	5986.1
C10-pisa-carga-3-cil	2	5969.4
Flange lig robo	1	5413.8
C10 guia top	8	1570.7
C10 transmissao forca	2	1540.9
C10 batente	2	1314.8
C10 apoio cil	2	1221.6
1463250 DSBC-32-350-PPSA_(Z_0)	2	744.7
C10 guia dentes	8	642.6
C10 cilindro40	2	568.5
C10 patim pisa carga	8	386.3
C10 guia pisa carga	4	288.0
C10 transf forca	2	268.0
80x80LBx1300 capa	4	184.8
Perfil Maytech 43L 200	2	130.5
1463250 DSBC-32-350_(K)	2	117.7
DIN-439-B - M10x1_25(F)	2	14.6

For each part, for the simulations that you got, remember that material that is “blue” (low stresses) is working little, and thus dimensions could be reduced to as to increase stress, (but balance that with the increase in displacements though).

Zero thickness error

An error about zero thickness was appearing and it wouldn’t let me extrude. I was having some trouble figuring out where the problem was. To fix that, extrude but do **not** merge the bodies. That way you can more easily (in 3D) see where the problem is and fix the sketch.