ANSYS Learning Notes [Sketch]

by André Duarte B. L. Ferreira
10/07/2016
ANSYS v.16

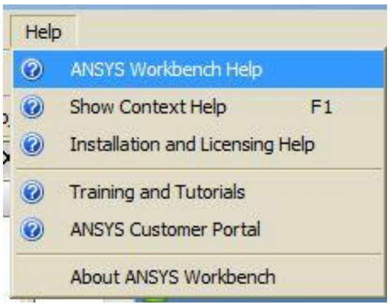
Courses: https://courses.edx.org

Mini-course: https://confluence.cornell.edu/display/SIMULATION/ANSYS+12+-+Beam

If you liked this, get the book: H.H. Lee - Finite Element Simulations with ANSYS Workbench, 2012, where these notes are based on.

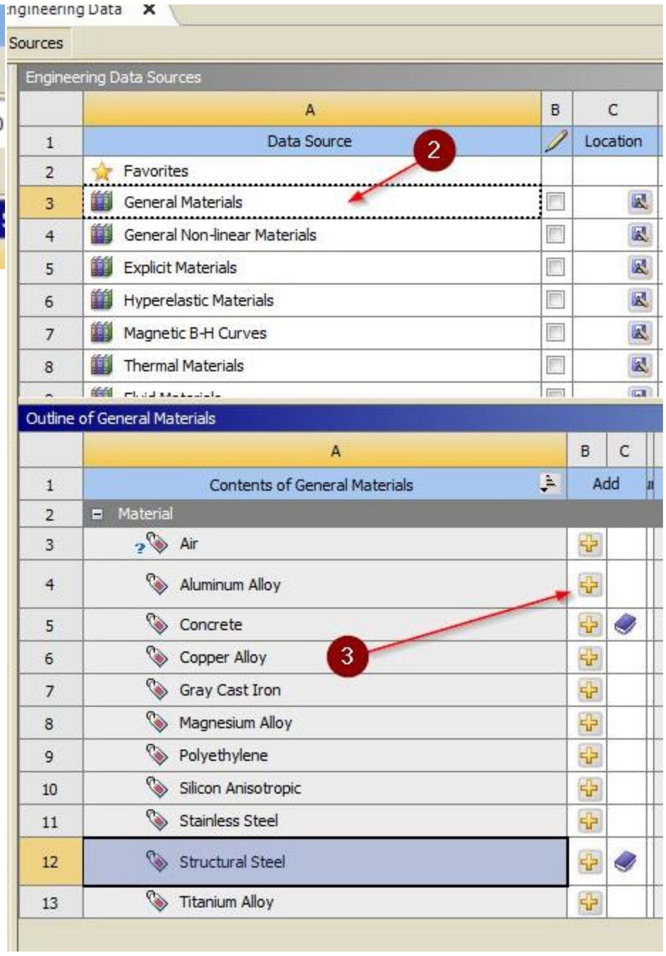
Though most images were done by me, there are a few I didn't do, but since this isn't for sale and only for studying purposes only, (and unless people find this online by chance) for personal purposes, I chose not to make any references besides the book above, where these notes are, so far, based on.

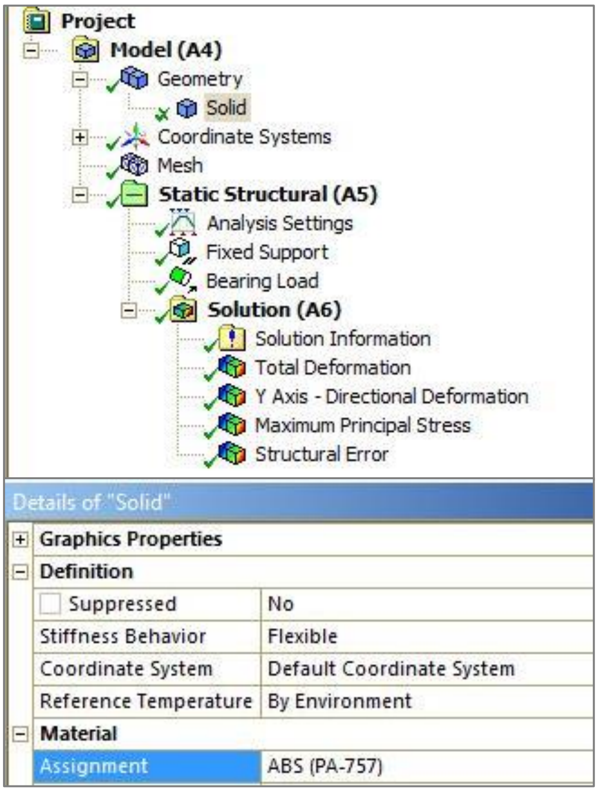
For now this is mostly focused on statics, later I'll complete it with dynamics.



Materials

Adding a material & Assigning it to Part





Models

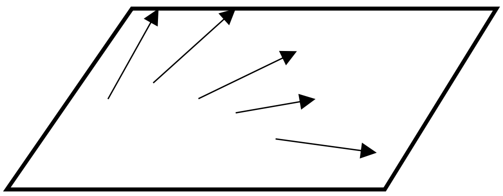
- 2D Models include
    - Line bodies
    - 2D-surfaces
- 3D models include
    - Line bodies
    - 2D and 3D-surfaces
    - Solid bodies

Each simulation can only have 1 type of model — there can't be, e.g., shells and solid elements at the same time.

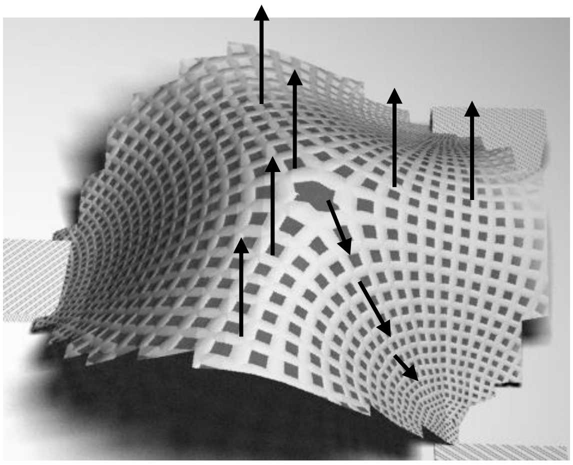
2D Modelling

You should use a 2D model whenever possible (as well as symmetry) as it saves on processing power and improves accuracy. For the same processing time we can have more nodes/elements, and the more elements, the better the results. Remember that the difference between a 2D model and a surface, is that a 2D model doesn't have out-of-plane bending/stresses/...

2D Model:
1) 2D surface;
2) Only in-plane stresses/deformations



3D Model:
1) 3D surface;
2) Out-of-plane stresses/deformations



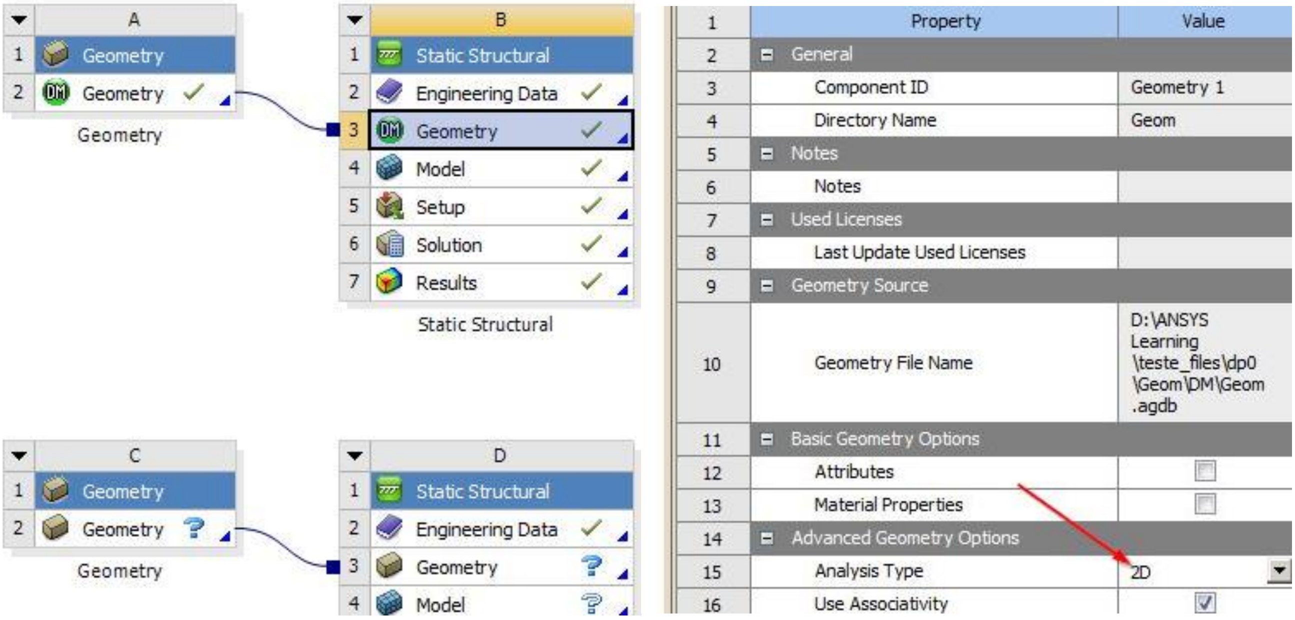
When a real world body is thin and subject to bending, it is generally a good candidate for a 3D surface body. ANSYS will mesh surface bodies with shell elements. There are many advantages of using surface models.

1) Creating surface models is usually easier than creating solid models.
2) The problem becomes simpler to solve for ANSYS compared to using solid models, due to the efficiency of shell elements. This results in
    a. a much faster computation of the result
    b. more accurate results

Consider using surface models over solid models whenever possible. In the old days surface models were visually weird because they had 0 thickness. Now ANSYS can render thickness in order to the engineer to visually better imagine what they're working with.

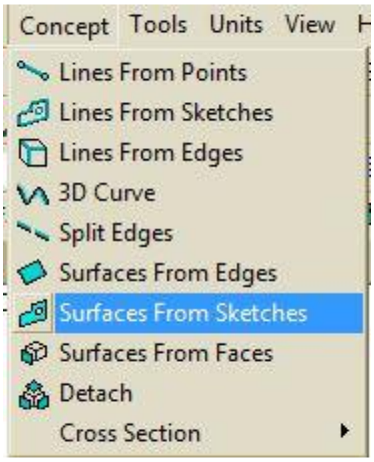
A 2D analysis can only have 2D bodies, but a 3D analysis can have both 3D and 2D bodies (surfaces).

Choosing 2D analysis:

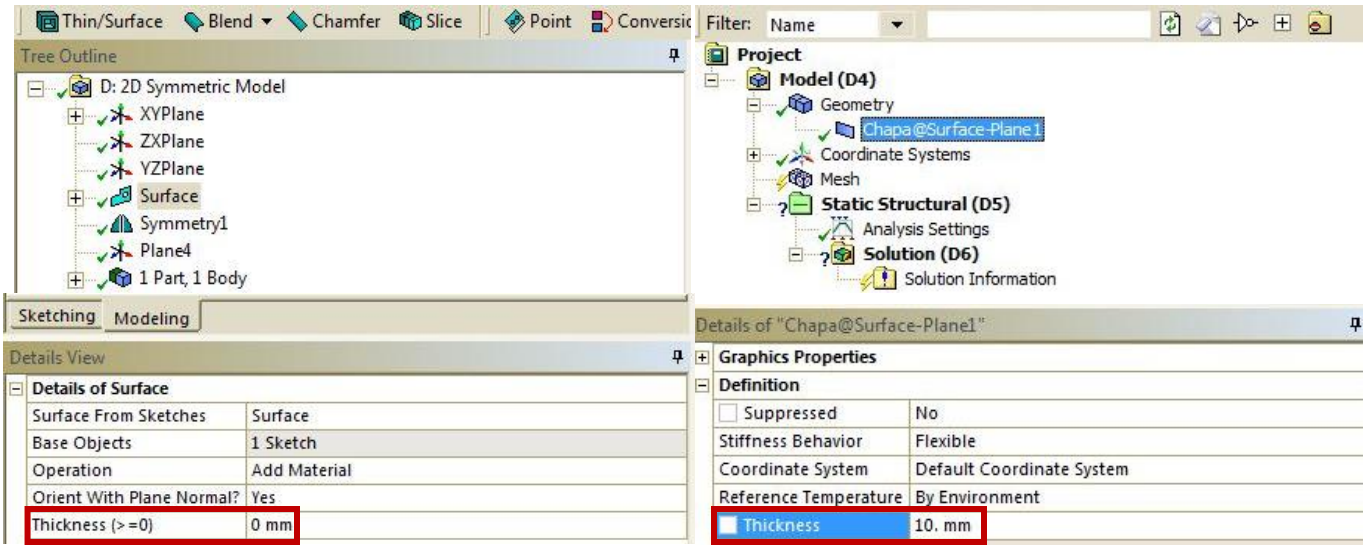


Creating 2D Surface

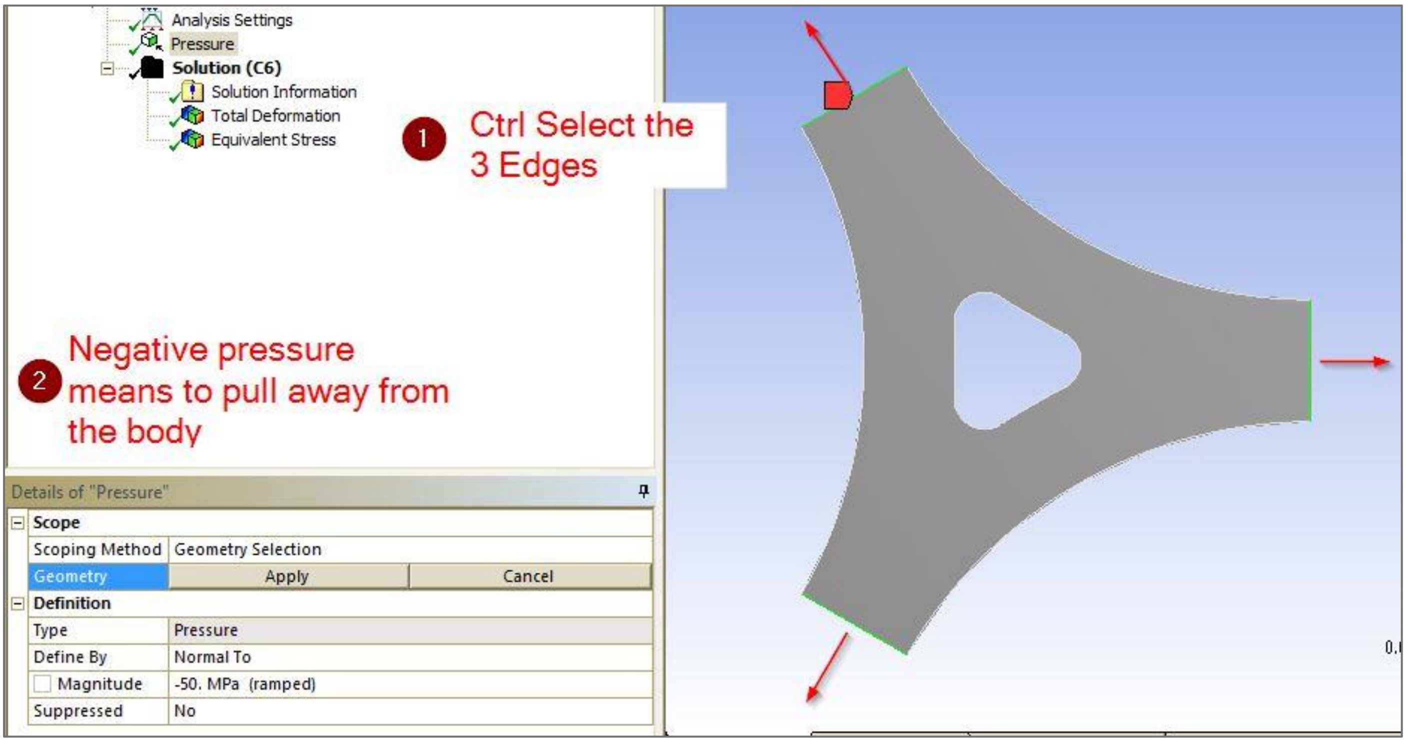
After the Sketch is done, and you selected 2D as above, create a surface:



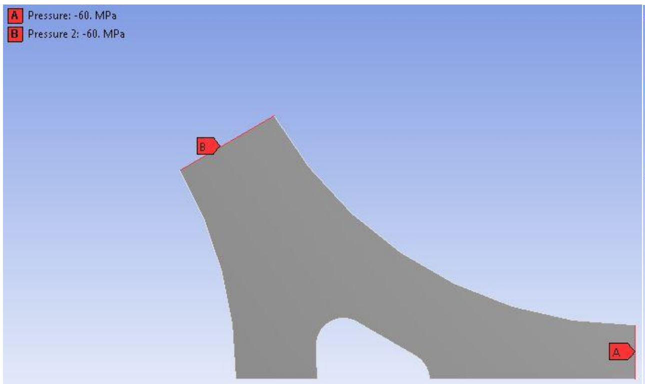
The thickness can be specified in DM or in Mechanical. Because we chose 2D analysis there's no difference between specifying the thickness in DM or Mechanical. But if it was a 3D analysis, if you specify the thickness of a surface in the DM, then when meshing the elements will be solid elements. If you specify the thickness in Mechanical, then the elements will be shells.

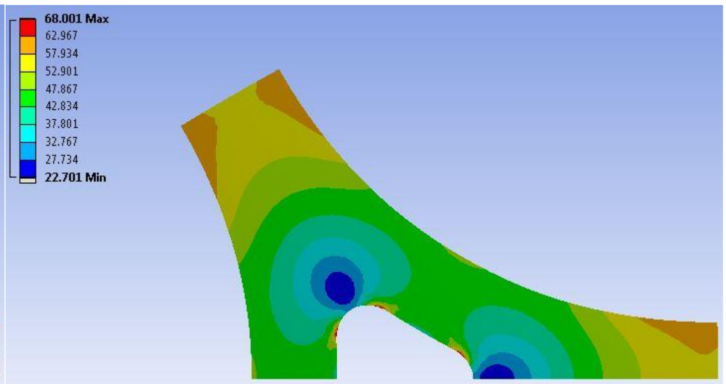


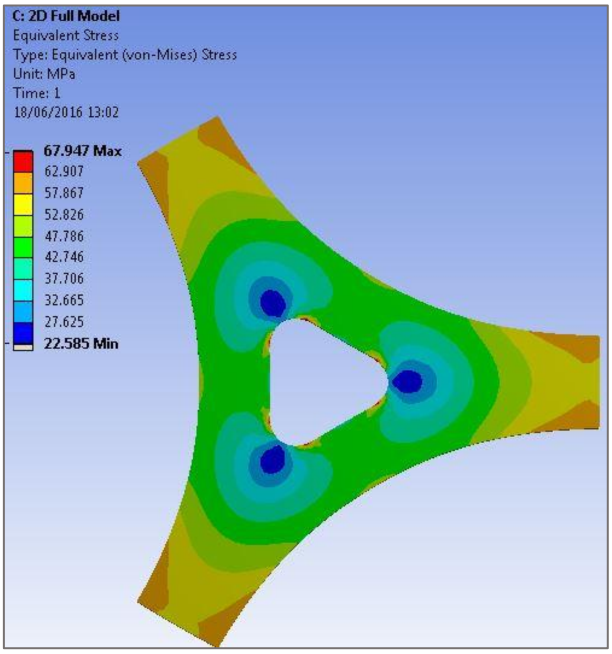
Applying Pressure 2D Model



If you are using symmetry (from DM) ANSYS will know that the other part will have the same pressure applied on the opposite side. As for the right side, it will propagate that pressure value.







The stress in the symmetrical model is slightly higher because we were able to use more elements for the same computing time, thus achieving greater accuracy. ANSYS overestimates the stiffness matrix so it underestimates the stress and deformation values. As we know both this and that the symmetrical result was calculated with a higher element density, the higher value is the more accurate.

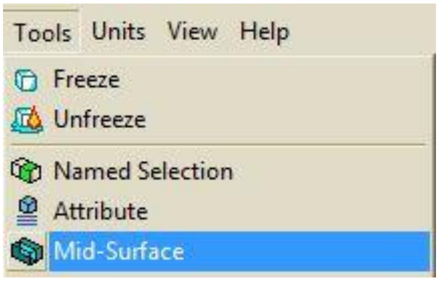
3D Modelling

3D Surface Models

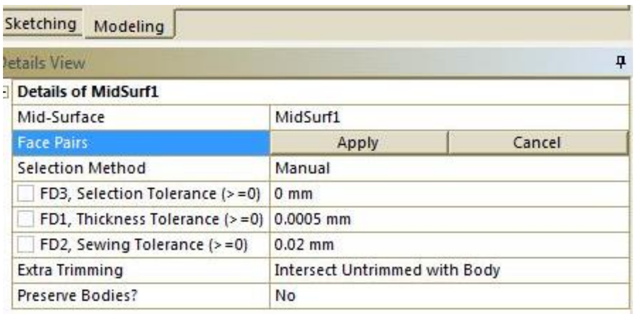
Creating a Surface From a Solid

In Section 4.1, we created a 3D solid model for the beam bracket; the model was used for a simulation in Section 5.1. Since the seat plate (flange) and the web plate are relatively thin and have uniform thicknesses, it raises a curiosity that if it is possible to model the beam bracket as a surface body and obtain a comparable result.

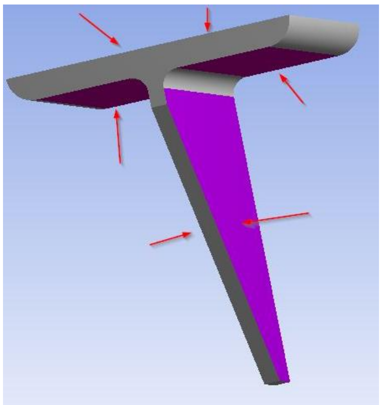
To create a surface model for the beam bracket, we don't have to start from scratch; we can use a tool in <DesignModeler>, called <Mid-Surface>. In fact, surface models are often created in this way. CAD models are usually created as 3D solids, since they are created for many purposes, and simulation is only one of them. When a surface model is needed for simulation, <Mid-Surface> is a powerful tool to extract a surface model from a solid model.

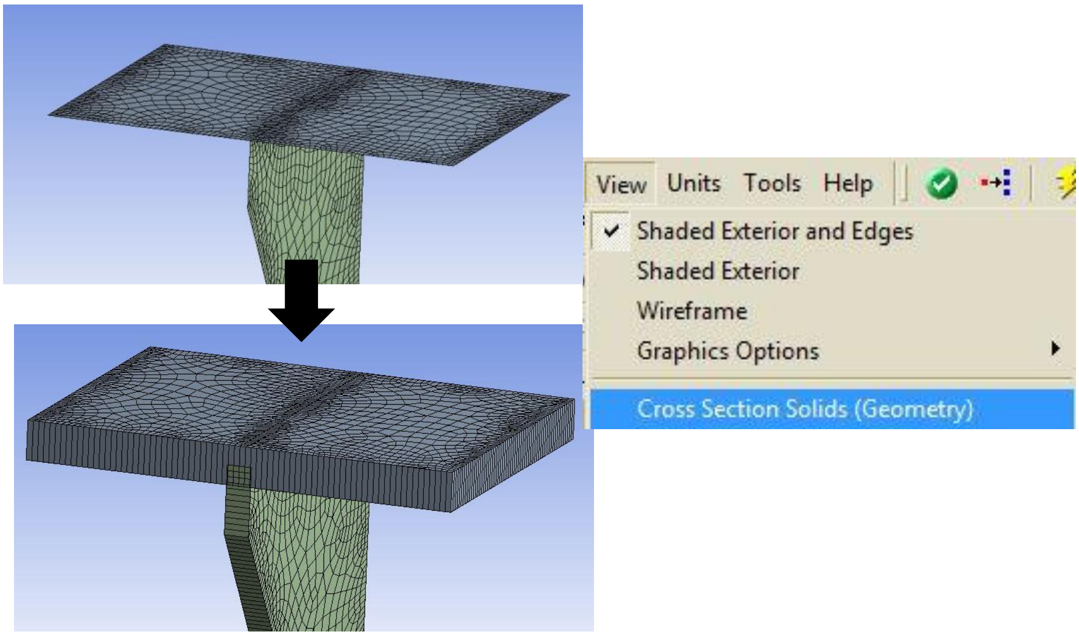


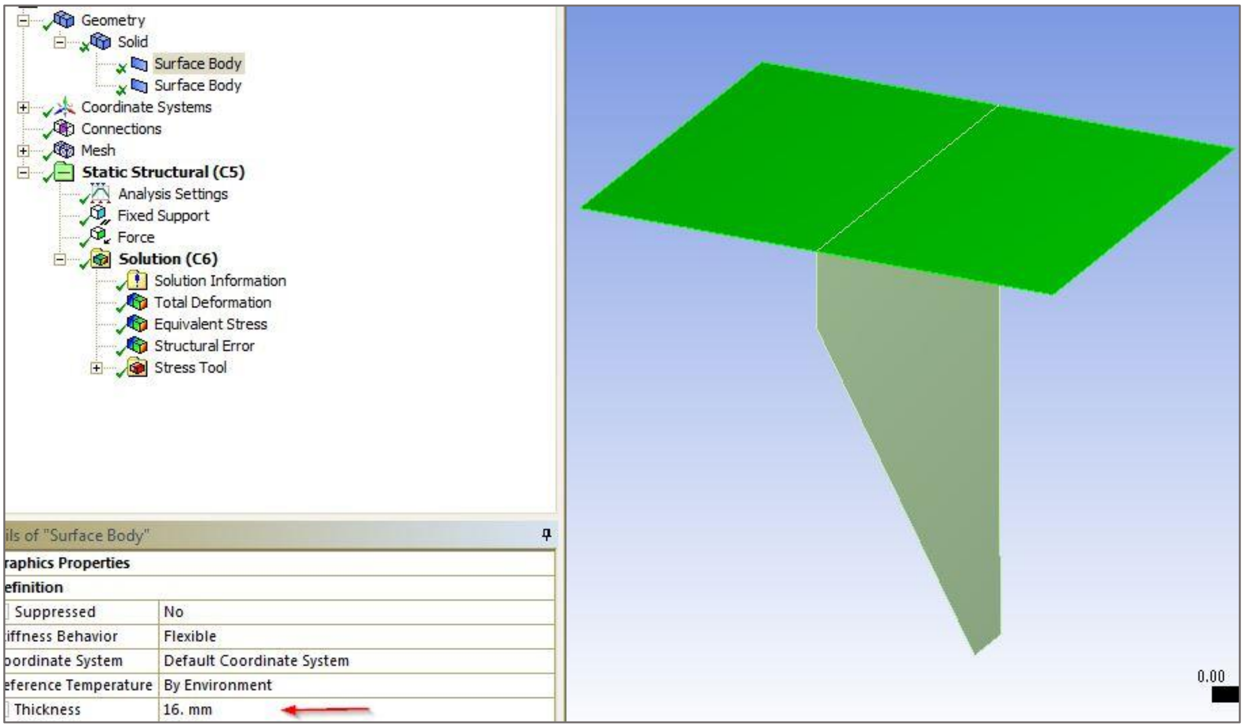
-100 to 100



Details of MidSurf1	
Mid-Surface	MidSurf1
Face Pairs	Apply / Cancel
Selection Method	Manual
FD3, Selection Tolerance (>=0)	0 mm
FD1, Thickness Tolerance (>=0)	0.0005 mm
FD2, Sewing Tolerance (>=0)	0.02 mm
Extra Trimming	Intersect Untrimmed with Body
Preserve Bodies?	No



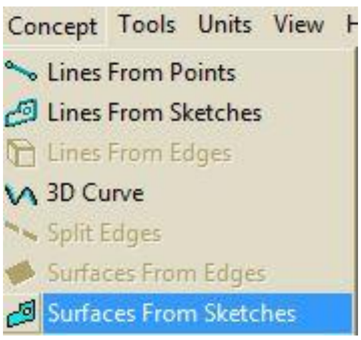




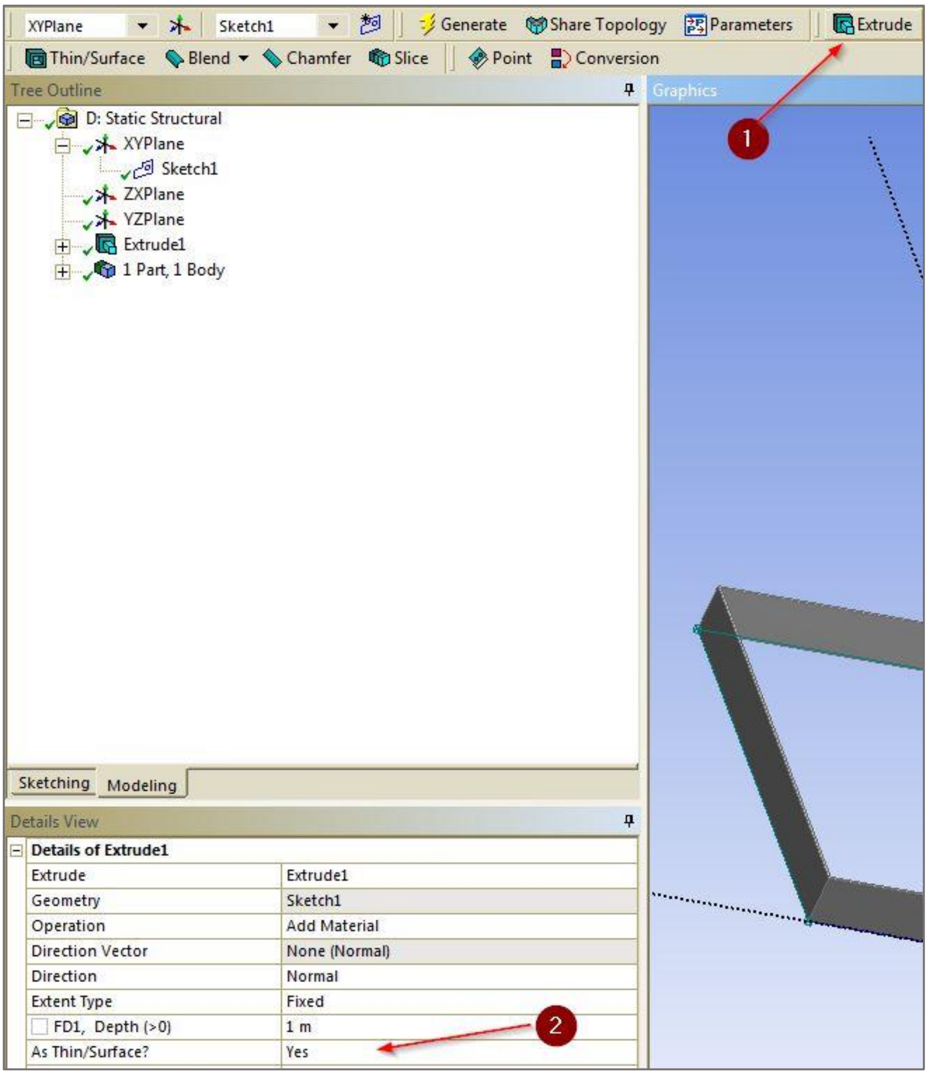
By using the surface model instead of the solid model we were able to achieve very similar results. However the surface model only consisted of 1000 nodes while the solid model consisted of 6500 nodes. Before using a solid model, consider the possibility of using a surface/line model first.

Methods for Creating Surface Bodies

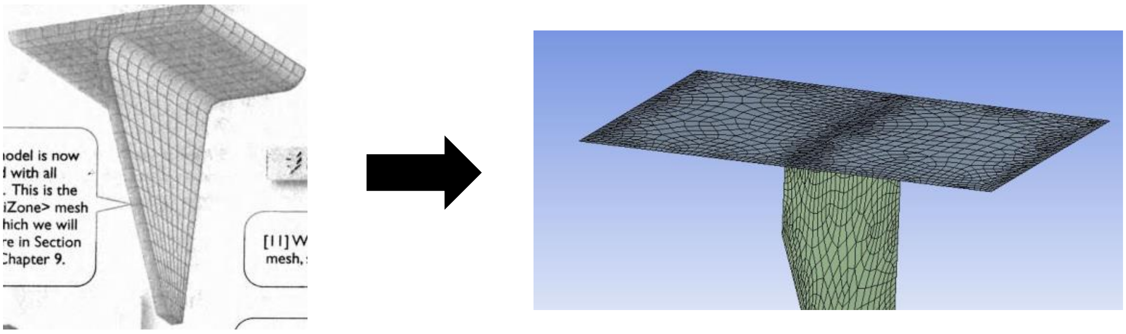
a) Surfaces from Sketches in DM



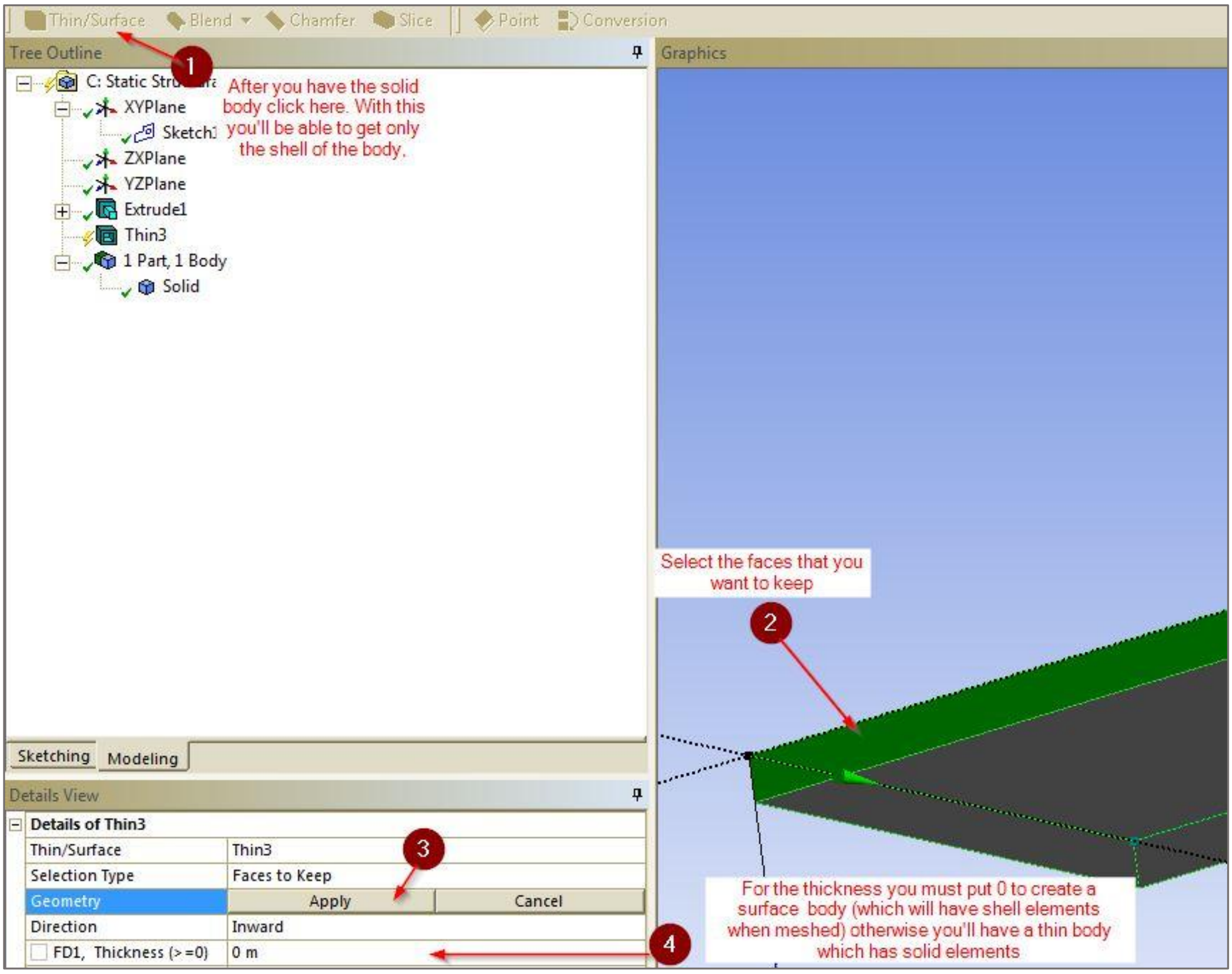
b) Extruding, revolving, sweeping or lofting lines

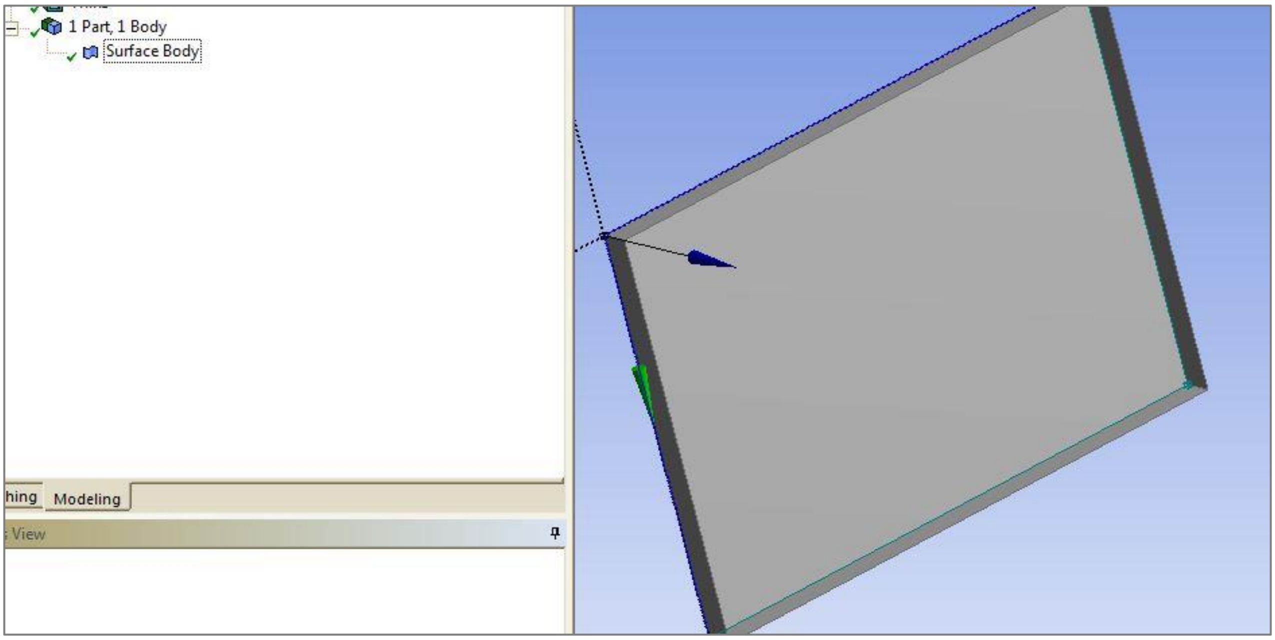


c) Using Mid-Surfaces

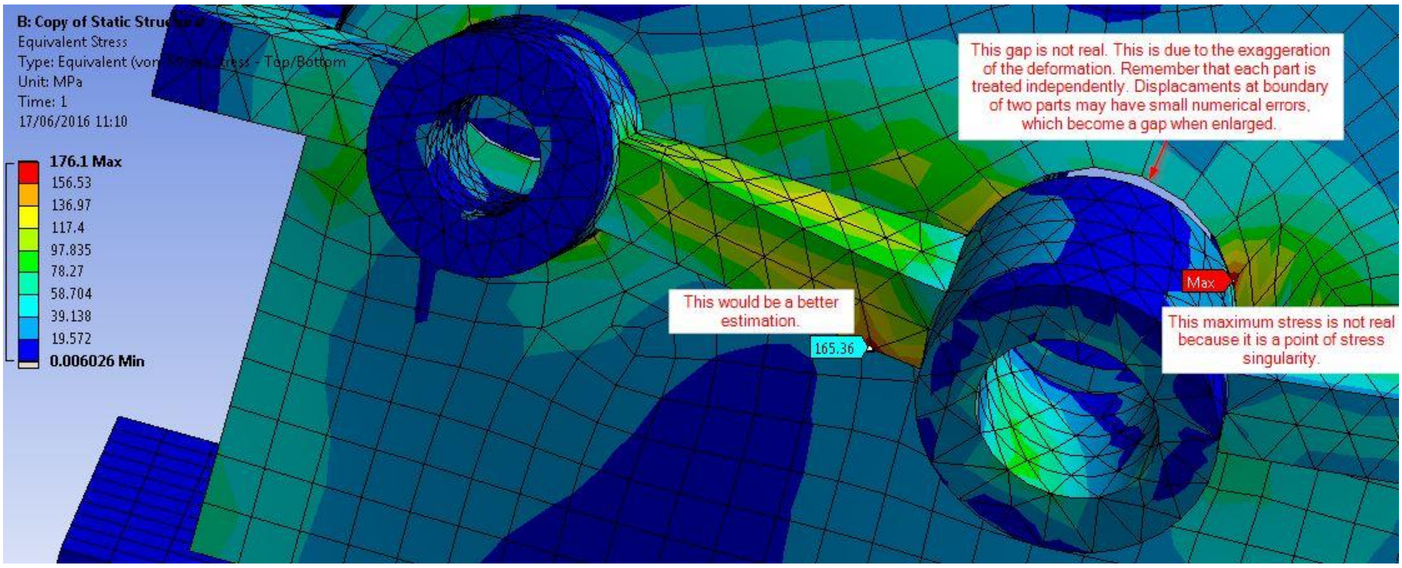


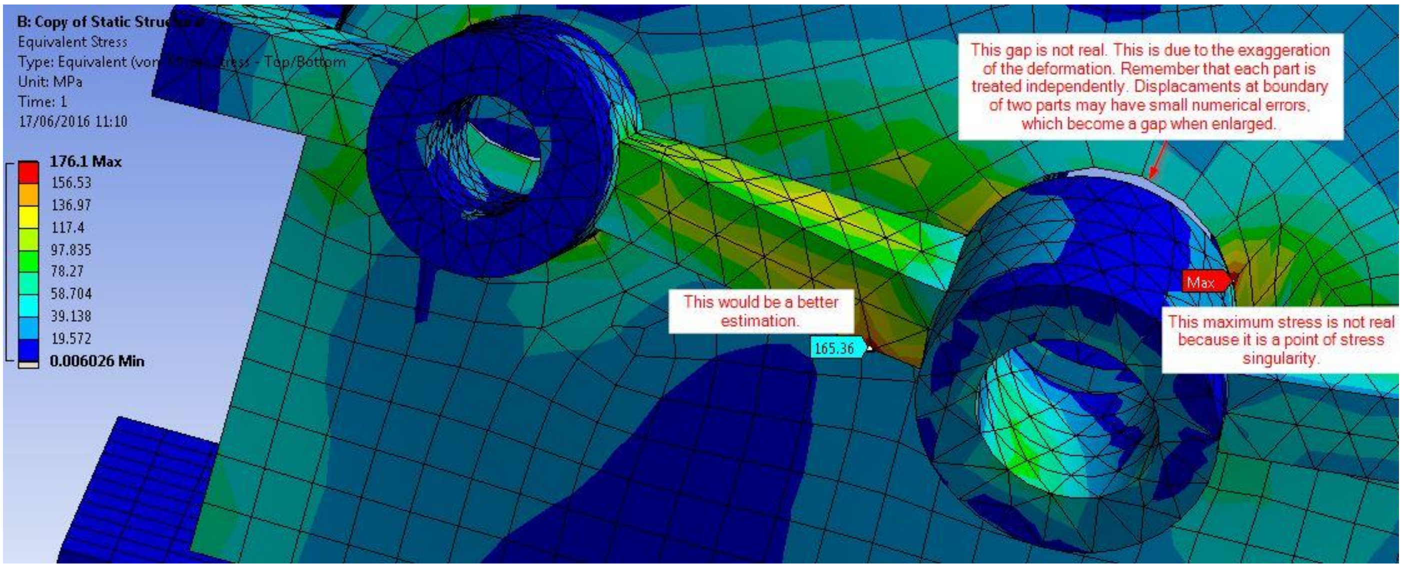
d) Using Thin/Surfaces





This piece below was obtained with the Thin/surface method.





Shell Elements

Shell Elements

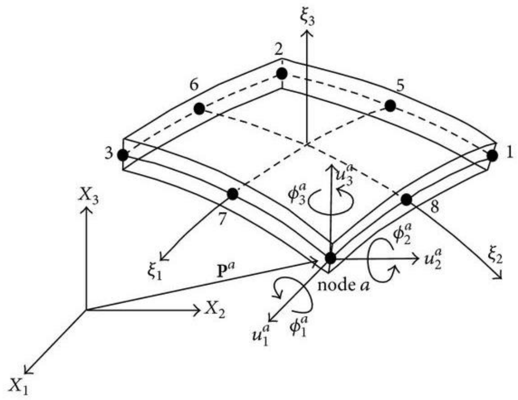
A shell element is a planar (2D) element that can be arranged in the 3D space. It is used to mesh a body when one of its dimensions is much smaller than the other two dimensions. Each node has 6 degrees of freedom: 3 translational and 3 rotational. Due to the presence of rotational degrees of freedom, it is very efficient to model the problems dominated by the out-of-plane bending modes, contrasting to a solid element, which does not have rotational degrees of freedom.

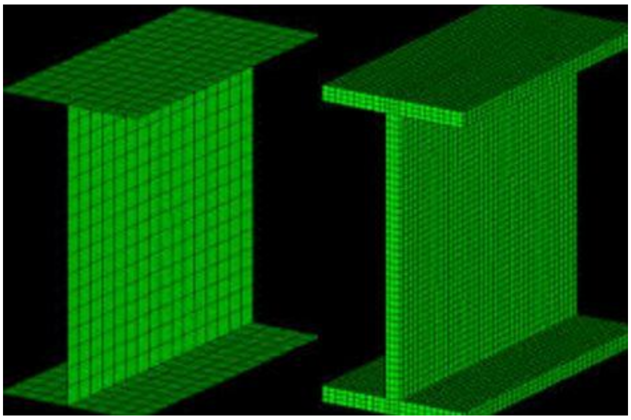
Top/Bottom of Surface Body

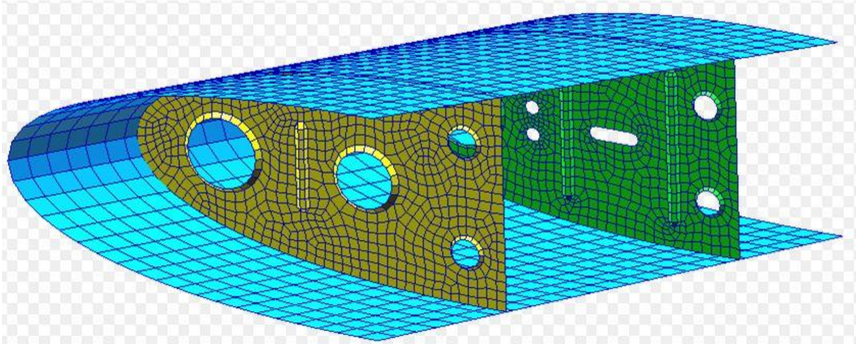
Each surface body has a top side and a bottom side. When you select a surface body, only the top side is highlighted. Loads are applied on the top side. By default, results are reported on both sides.

Auto-Detection of Contacts

When a geometry attaches to <Mechanical>, it automatically detects and establishes possible contacts between parts, wherever the gaps between parts are less than a tolerance. The contact type is <Bonded> by default. The result of auto-detection may not be accurate and needs to be manually modified.



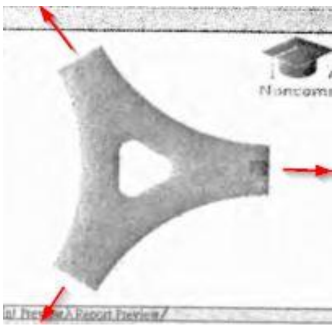




Triangular Plate

Q: Can we perform a simulation with a 3D surface model for the triangular plate in Sections 3.1? If positive, is there any advantages of doing that over the 2D solid model?

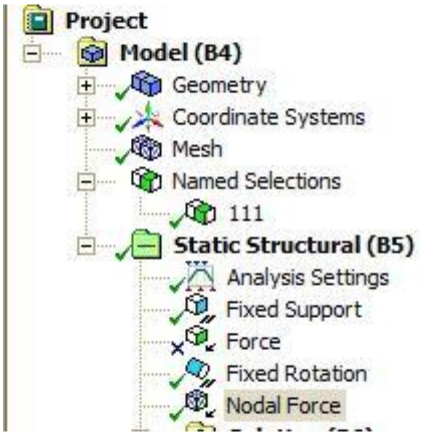
A: Yes, we could, but there is no advantages of doing that. Remember that 3D surface bodies are meshed with shell elements. The essential difference between shell elements and 2D solid elements is that shell elements can have out-of-plane deformation (warpage) while 2D solid elements cannot. In the case of triangular plate, there is no out-of-plane deformation. There is no needs to use shell elements. The above discussions apply also on the spur gear of Section 3.4 and the filleted bar of Section 3.5.

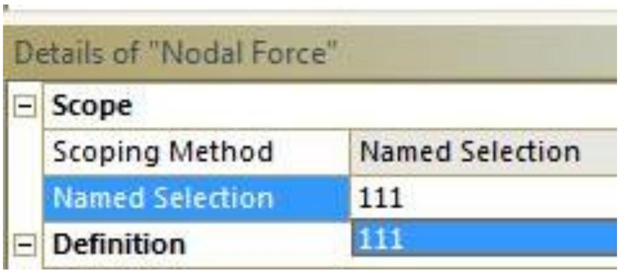


Applying a Force Somewhere in a Surface

Method #1 — by applying the force on nodes

1) Create named selection with the nodes you want to apply the force on. See chapter at the end.
2) Create nodal force, and choose that selection



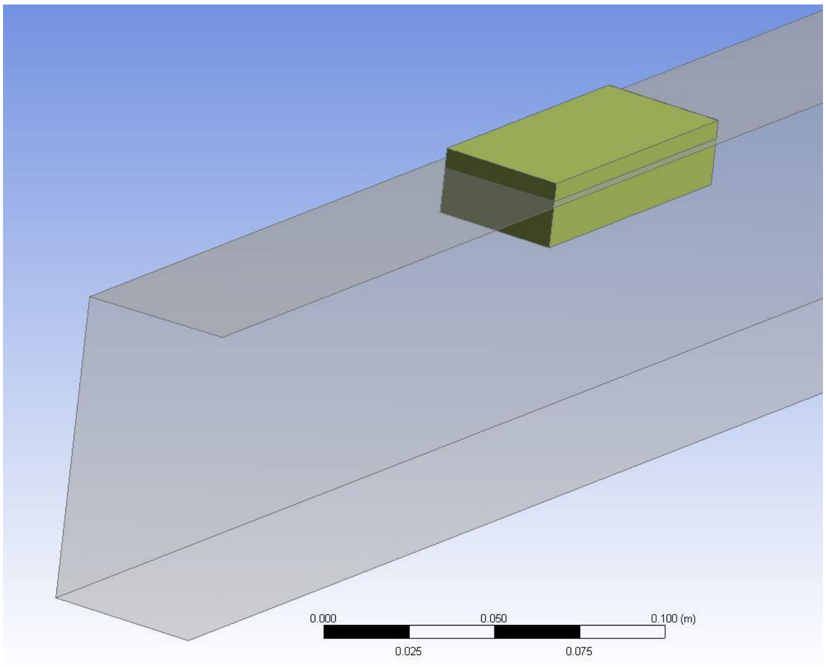


Method #2 — by applying the force on an area

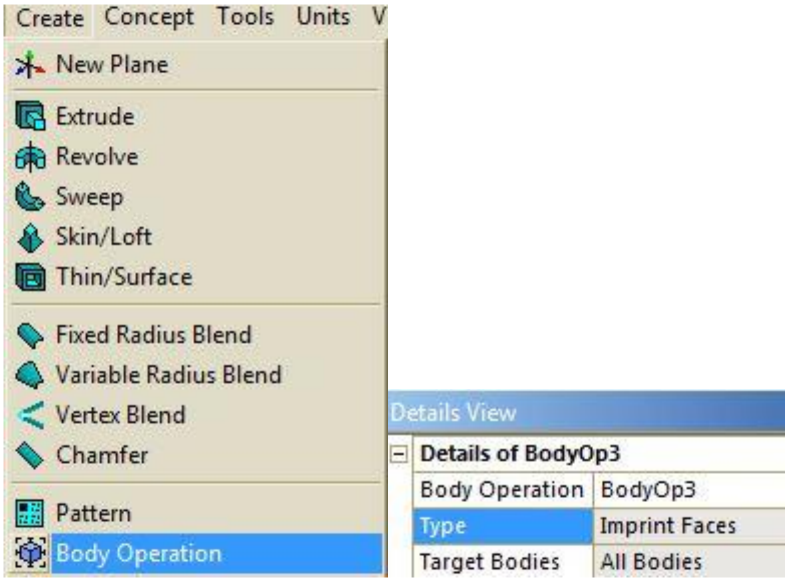
Method #2.1 — getting that area from an imprinted face

1) Go to 

2) Create this. A body that intersects the area you want to apply the force (create plane, sketch, extrude, add frozen)



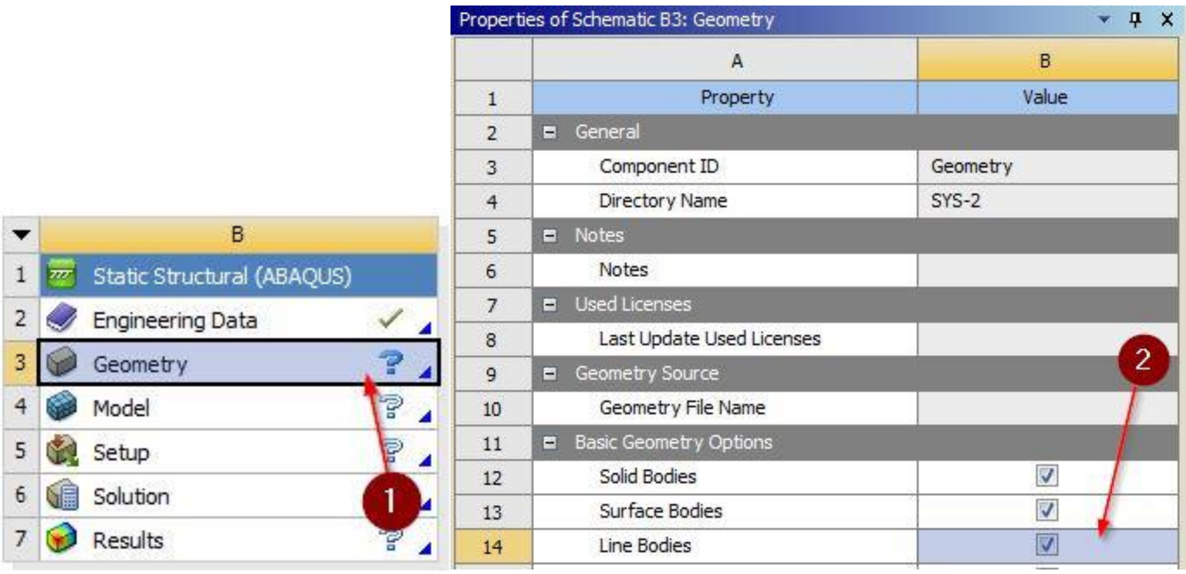
3)



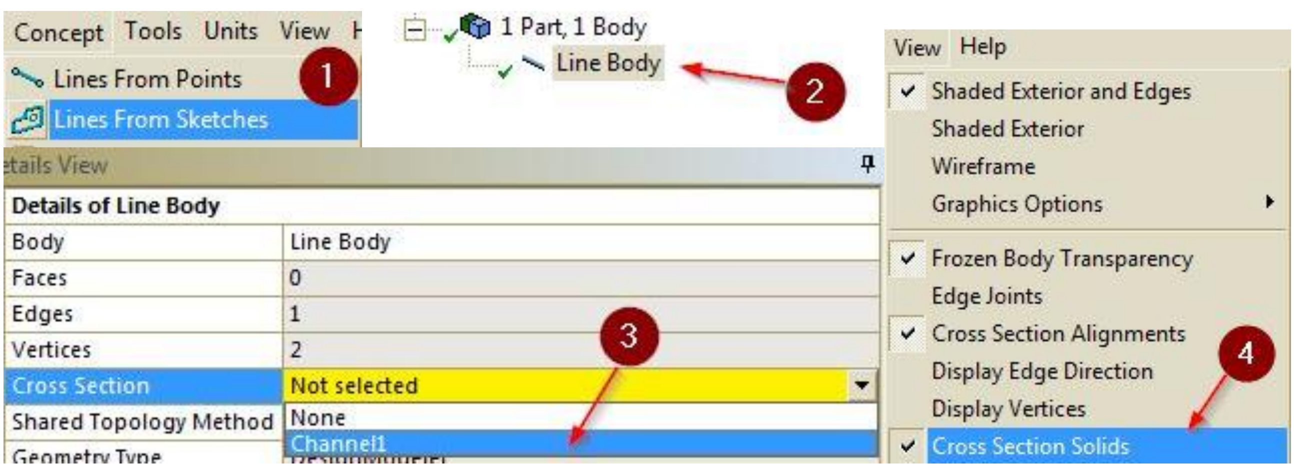
Done. Area created.

Line Models

Enable line models before anything else. Then proceed.



After you created the line sketch and the section for it, create the line body (1) then assign that section to the line body (3). Then render it visually (and only visually) select (4).

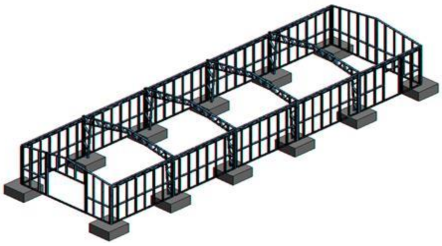


There are many real world objects that can be modeled as line bodies. As long a structure, or part of it, has small lateral dimensions and uniform cross section, it is a good candidate to being modeled as a line body.

Trusses

Frames

Beams

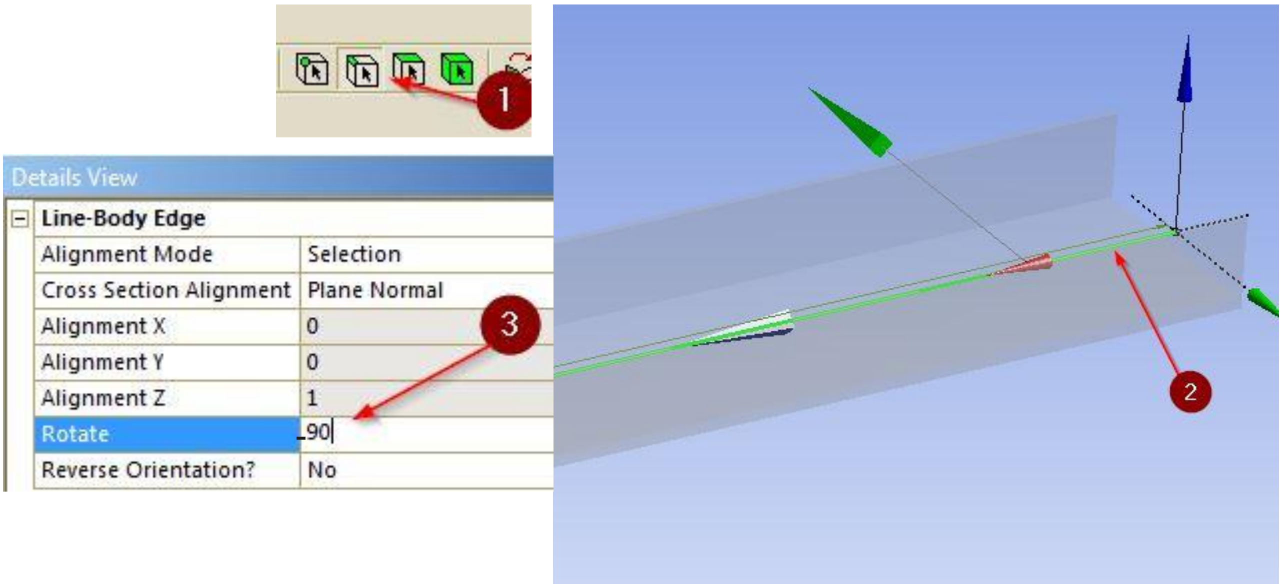


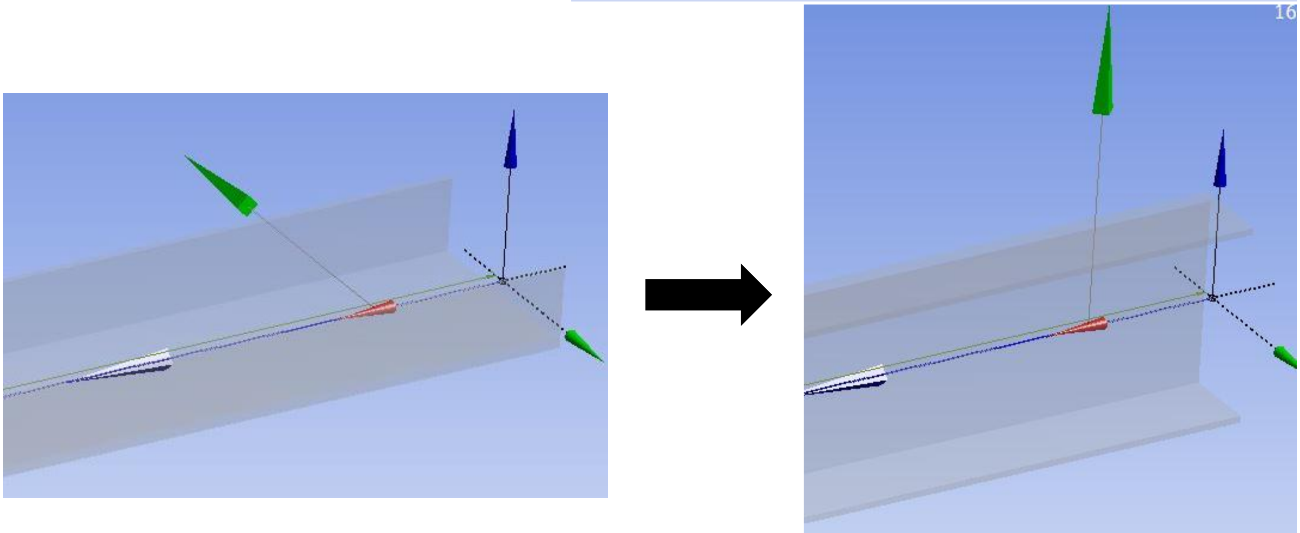
Workbench meshes a line body with beam elements. The advantages of using line bodies are similar to those of using surface bodies, but even greater. Engineers should consider using them whenever possible over solid models, and even over surface models.

Multiple Cross Sections in a Single Model

You'll need another body for that, since each line body can only have one section. So: make another plane, sketch there, create the line body, create another section, and assign it to the new line body.

Adjusting Cross Section Rotation





Applying Loads at Some Point in a Line

Method #1 – Edge Split

Split the line bodies.

Method #2 – Creating a Multi Body Part

Create several line bodies.

Trusses

Traditionally, a truss is defined as a structure consisting of two-force members. By two-force member, we mean that the members are pin-jointed at the ends and the loads apply on the joints so that the members are either stretched or compressed but not bent. Two members connected by a pin-joint can rotate about the joint independently. In reality, structural members are rarely connected each other by pin-joints. Modern structures are constructed using either welds or multiple bolt-and-nuts; the members are rigid jointed, not pin-jointed. Even in the old days, pin-jointed structures are not common. Main reason of pin-joint assumption is to ease the computational difficulty, in the days when computers were not widespread, if existing. Note that, due to the neglect of joint rigidity, pin-joint assumption leads to a conservative design: safer, but over-designed.

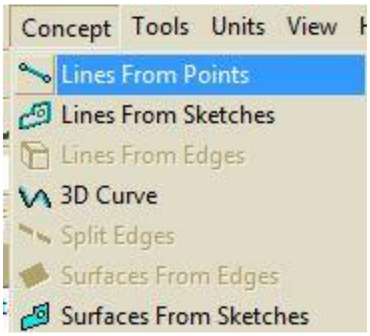
How much is the error caused by the pin-joint assumption? This is a good exercise problem for the engineering students (Section 7.4-2). Let me state the problem more precisely. Given a rigid-jointed structure, we perform twice of simulations: once with rigid joints (the reality), and the other with pin-joints assumption. What would the difference between these two models be? The amount of error depends on the slenderness of the structural member. If the members are slender enough, there is no essential difference between two models. On the other hand, if the members are not slender enough, then the pin-joints assumption may induce substantial errors.

Currently, the beam element (BEAM188[1]) is the only element supported in <Mechanical> to mesh the line bodies. The "truss elements" (such as LINK180[2]) are not directly supported. To model a pin-jointed structure, you need to explicitly specify revolute joints between the structural members, or insert APDL commands.

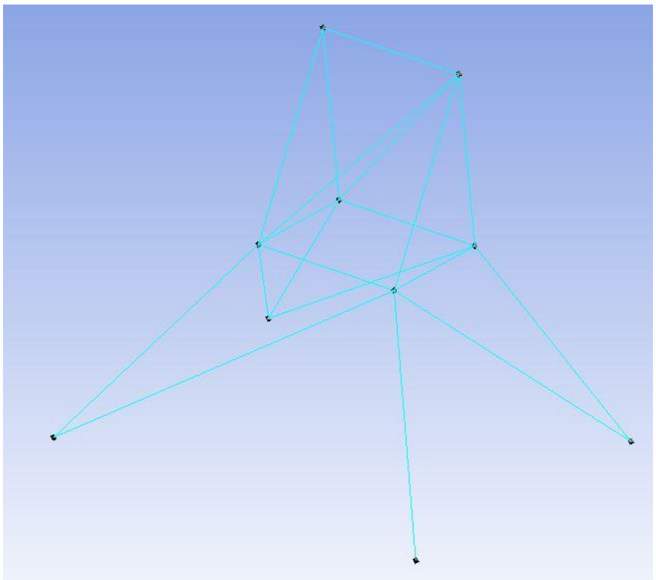
Create text file with the coordinates, like below.

Group no. (unique for part?)	ID no. (unique for each point)	X-coord.	Y-coord.	Z-coord.
1	1	-37.5	0	200
1	2	37.5	0	200
1	3	-37.5	37.5	100
1	4	37.5	37.5	100
1	5	37.5	-37.5	100
1	6	-37.5	-37.5	100
1	7	-100	100	0
1	8	100	100	0
1	9	100	-100	0
1	10	-100	-100	0

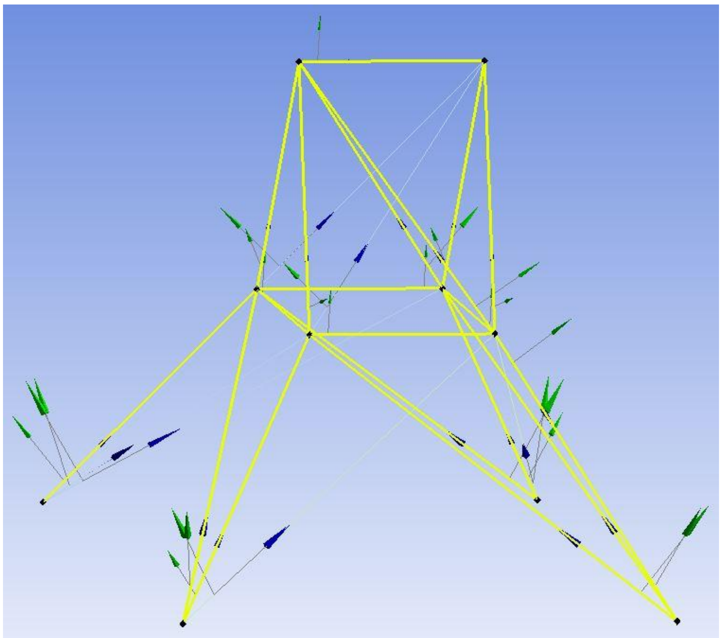
Then

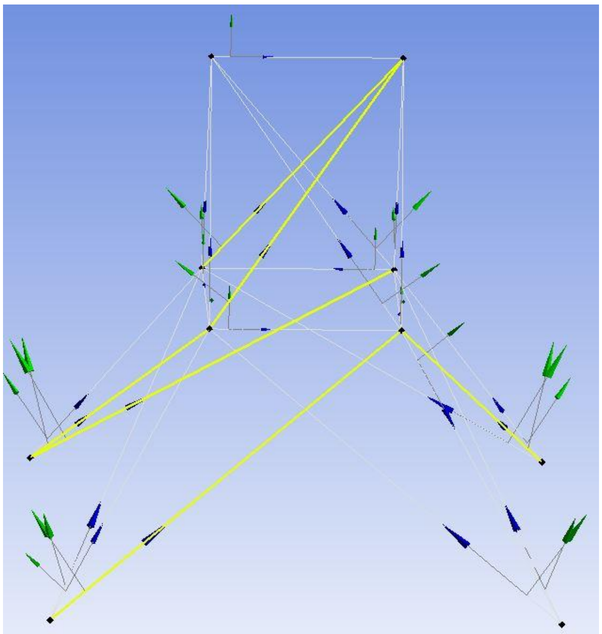


Then click on 1 point, then Ctrl click on next to make a line. To delete a line you made by mistake it's the same.



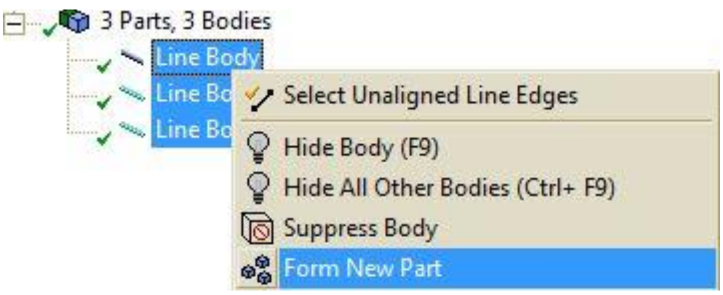
Don't cross lines. Then you'll do the same thing but with the option "add frozen" and that's when you'll do the lines that cross.



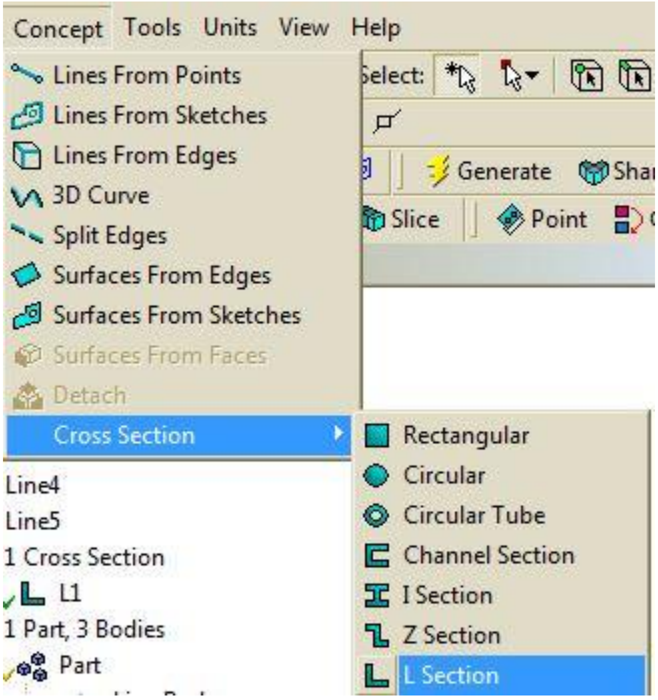


Details of Line5	
Lines From Points	Line5
Point Segments	6
Operation	Add Frozen

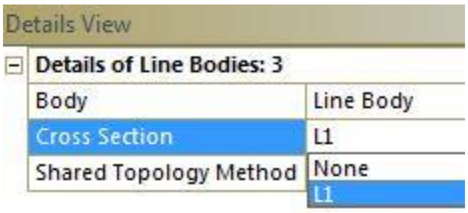
Then



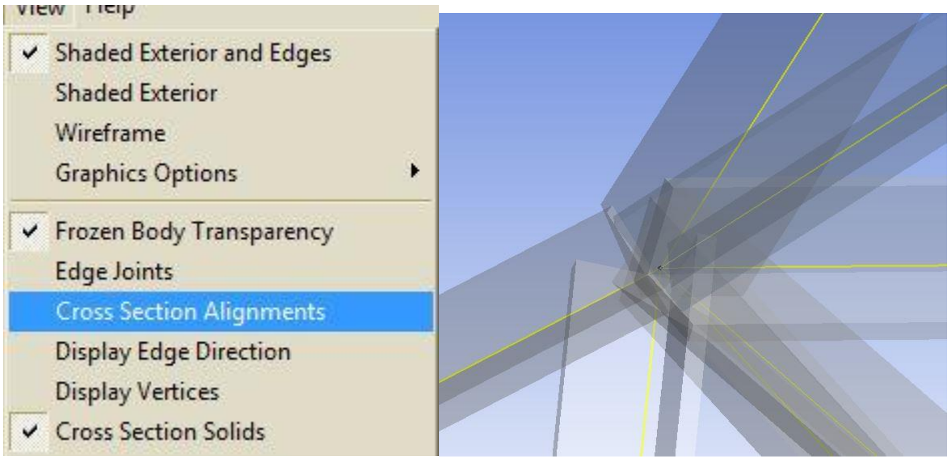
The model doesn't change appearance. This is only so that they are bonded together. Also since ANSYS tries to make meshes continuous within each part, this also has that point.



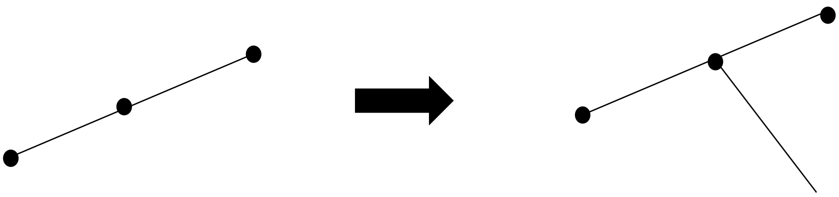
Give the dimensions you want then select the line bodies and



Put view like this to view the cross sections



Note: Whenever you want to connect a line to another line, they need to be connected by a point. That means.



Using Surface/Line Models Whenever Possible

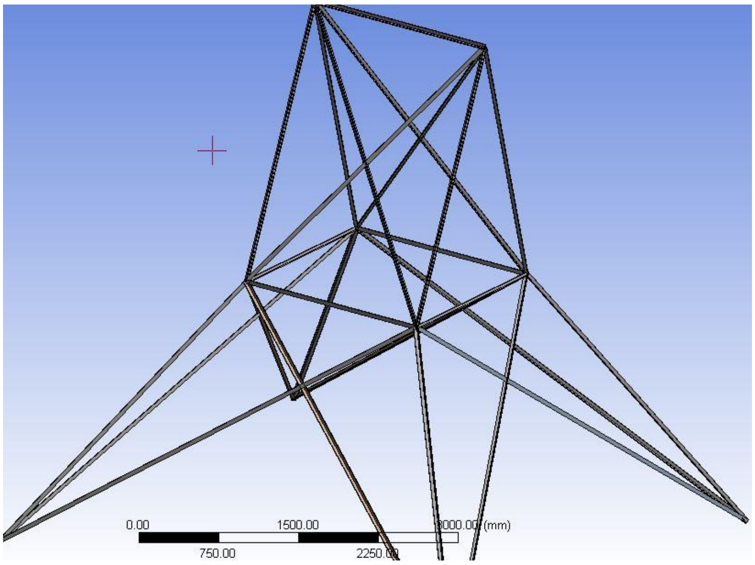
Since the solution of a model which is meshed with beam elements or shell elements converges very fast (i.e., very accurate solution can be obtained with only a few elements), we should consider a line model or surface model whenever possible. This is particularly true for those problems requiring many number of iterations or substeps, such as nonlinear problems, dynamic problems, optimization problems, etc.

When using Line Bodies, mesh with a very large number so that you get only an element per beam allowing you to get a solution equal to theoretical values.

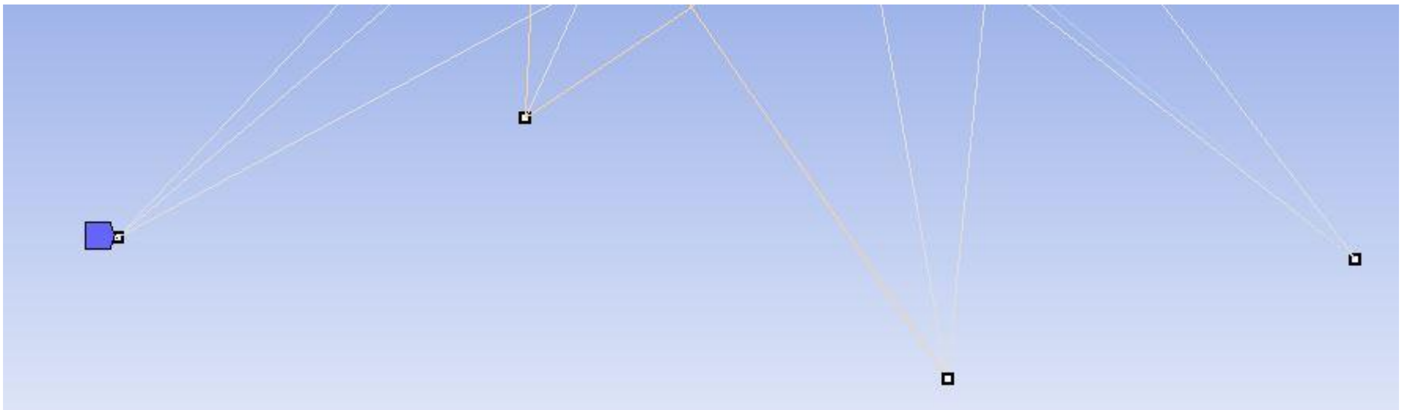
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	99999 mm

Statistics	
Nodes	35
Elements	25

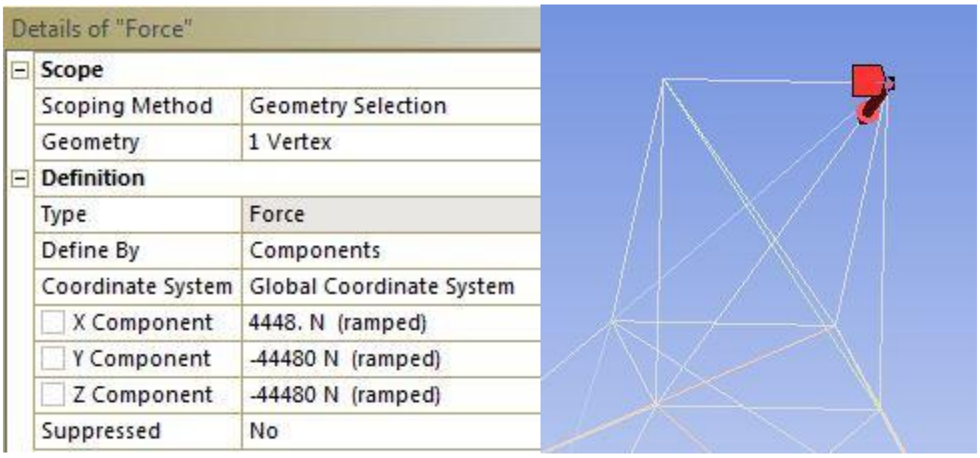
The reason why there’s 10 more nodes than elements is because each beam element as a node in the middle.

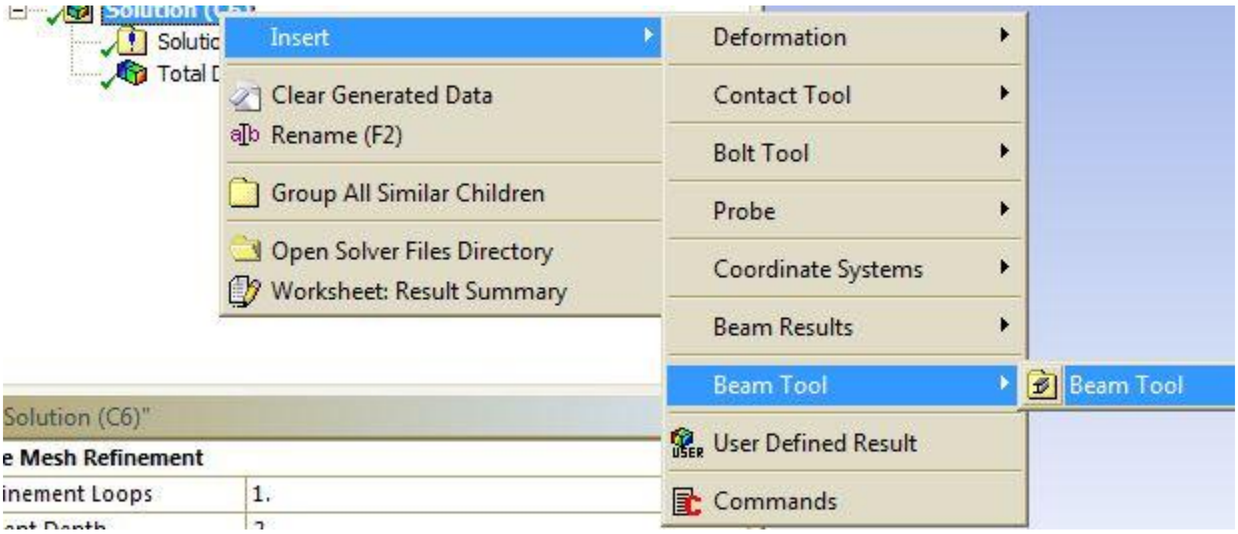


Fixed Supports there (base vertices)

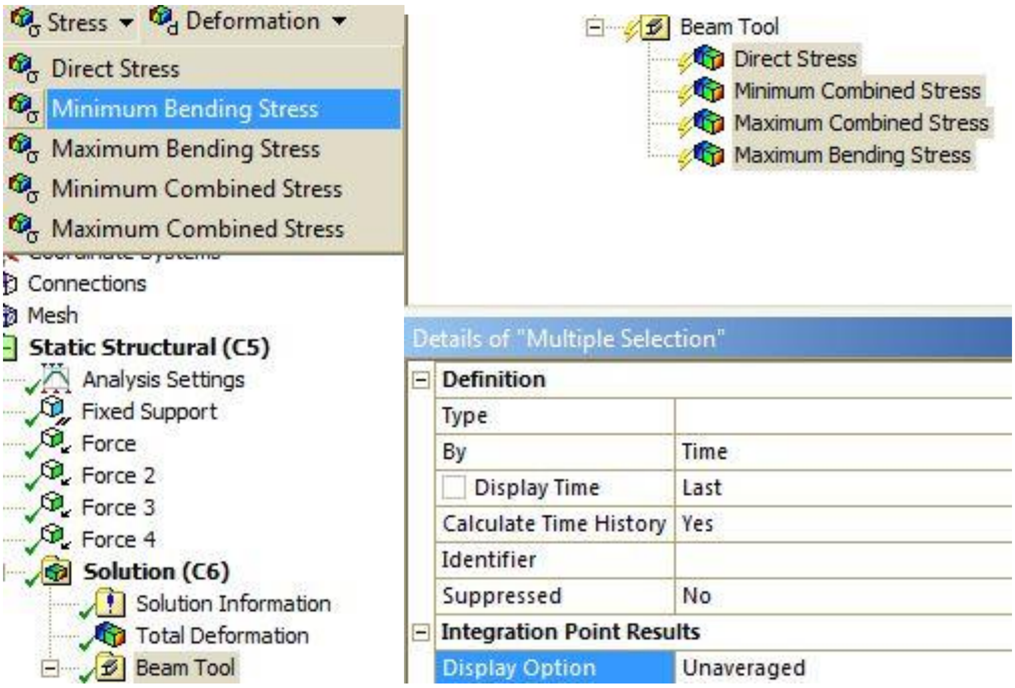


You can apply forces by components like below (or vectors (default option))





You can add a few more. The combined stress = normal stress + bending. To see how much is from which we should had the bending stress.

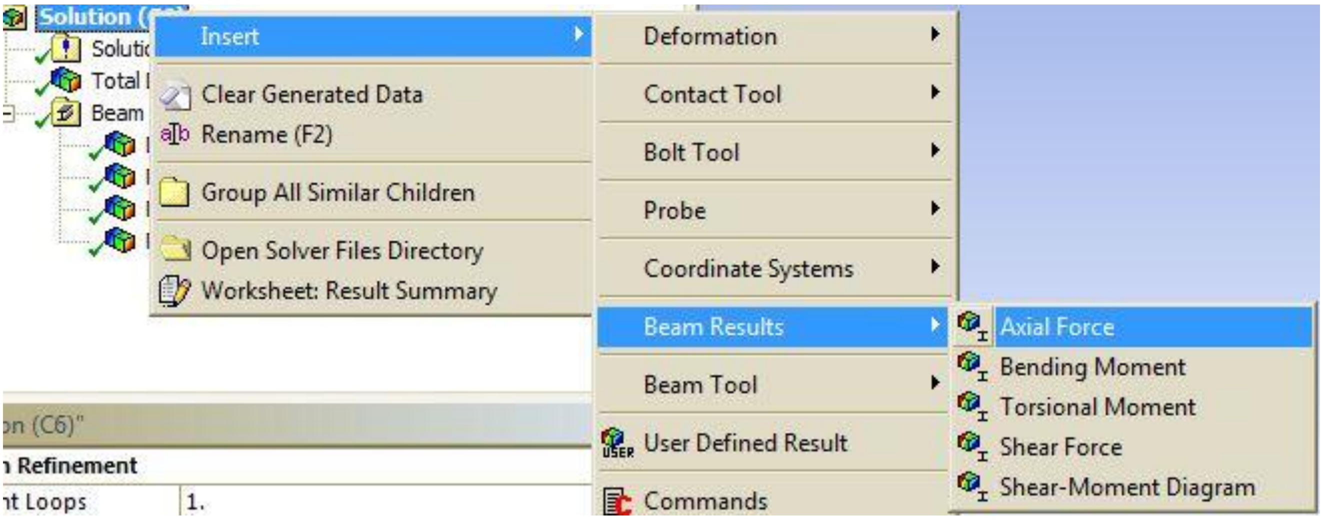


As usual $\sigma > 0$ is traction and $\sigma < 0$ is compression.

Why Turn Off <Use Average>?

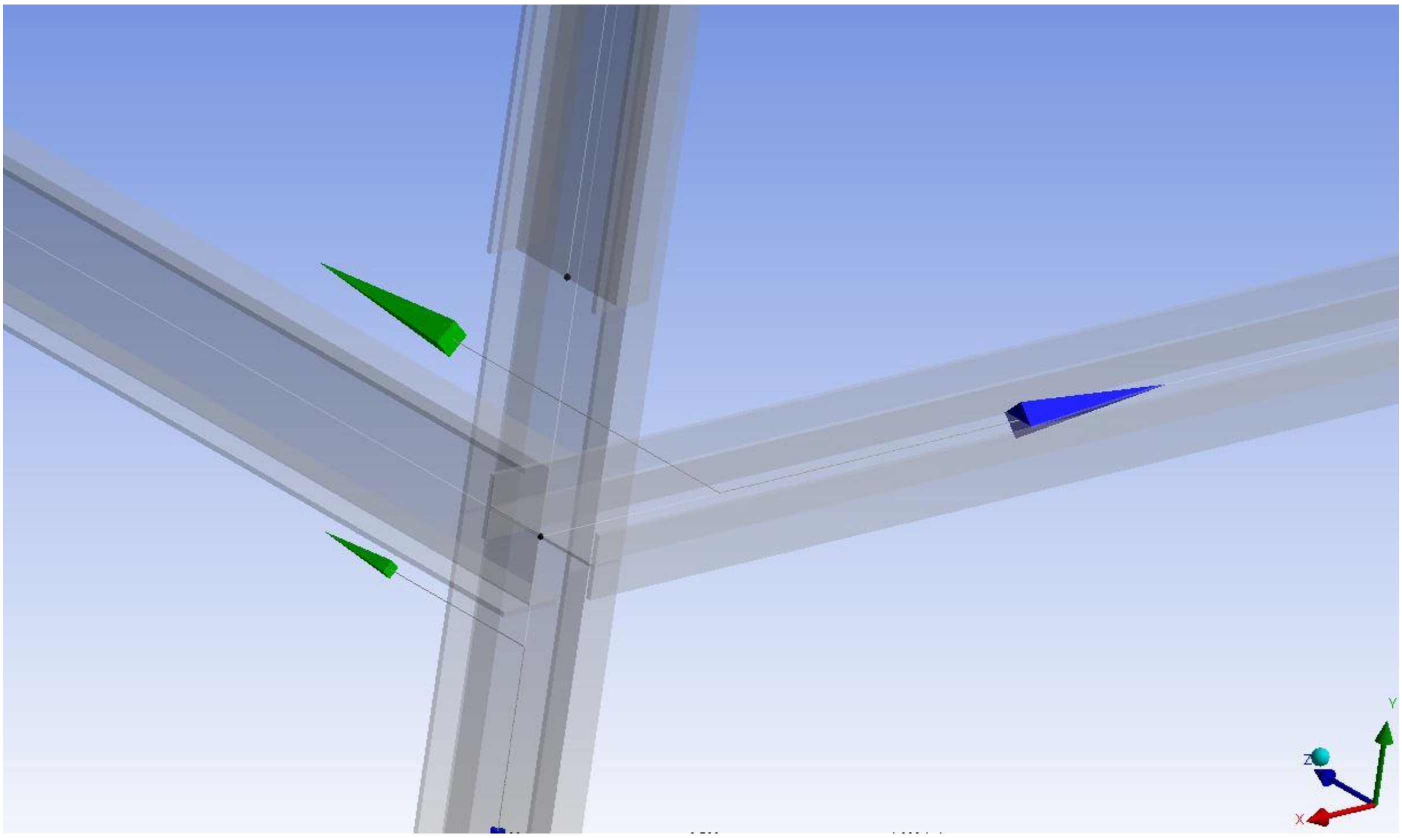
We've introduced the notion of averaged and unaveraged stresses (Section 3.5-6). Unaveraged stresses are also called *element stresses*, since they are calculated at points (usually geometric center or integration points) inside elements. On the other hand, averaged stresses are also called *nodal stresses*, since they are calculated at nodes, which are located at element boundaries. Since we mesh each member with a single beam element, and if we didn't turn off <Use Average>, every two adjacent members' stresses would have been averaged and reported. The averaged stresses would not have any meaning.

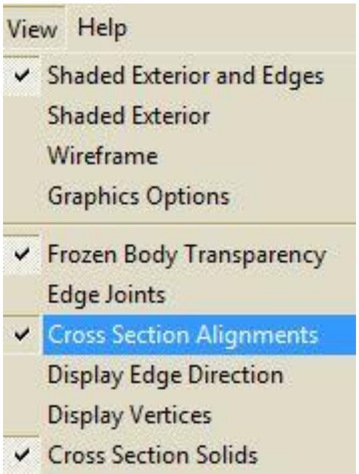
A few more options we can add



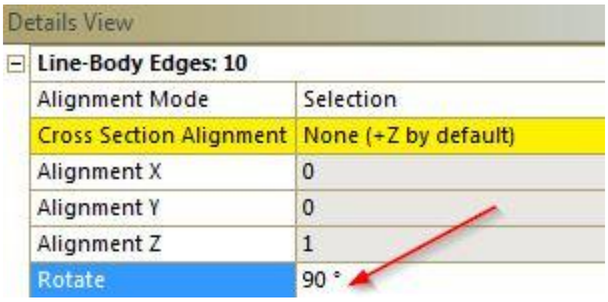
Position Cross Sections Properly

In the example below the right I is not in the correct position, it should be vertical.



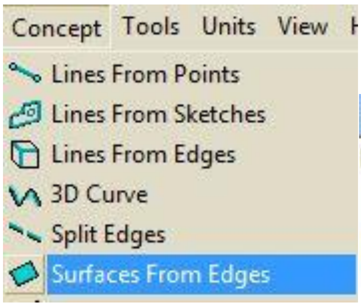


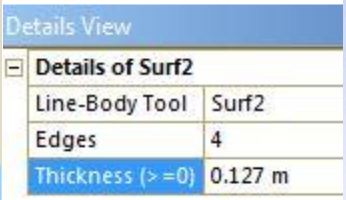
Select the line body

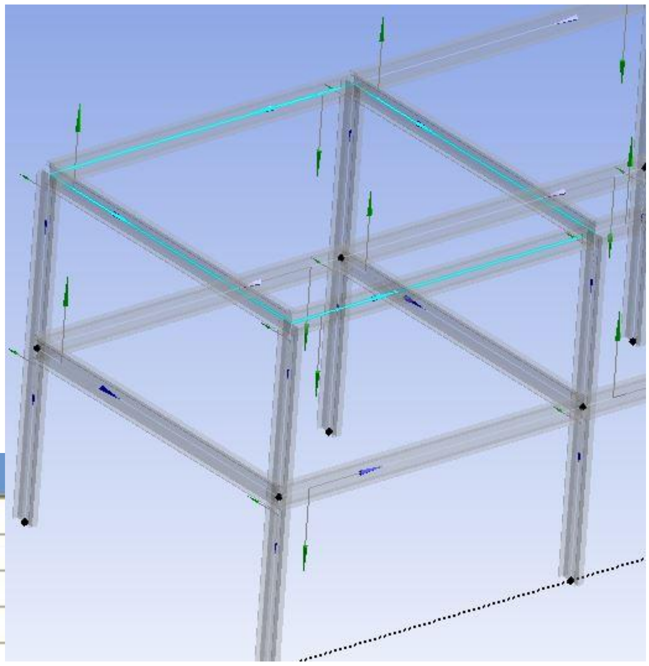


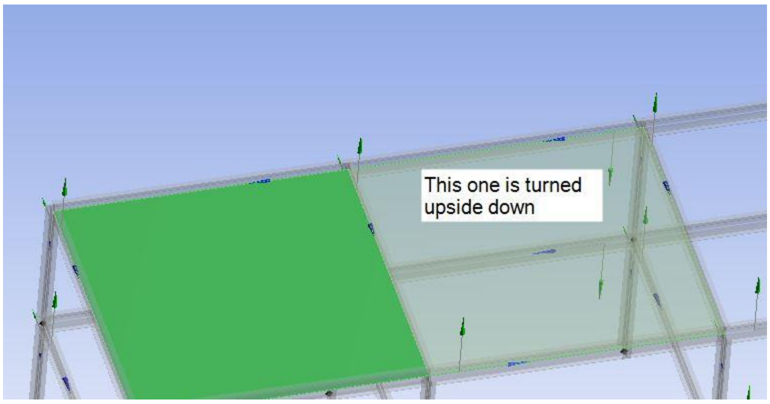
Details View	
Line-Body Edges: 10	
Alignment Mode	Selection
Cross Section Alignment	None (+Z by default)
Alignment X	0
Alignment Y	0
Alignment Z	1
Rotate	90 °

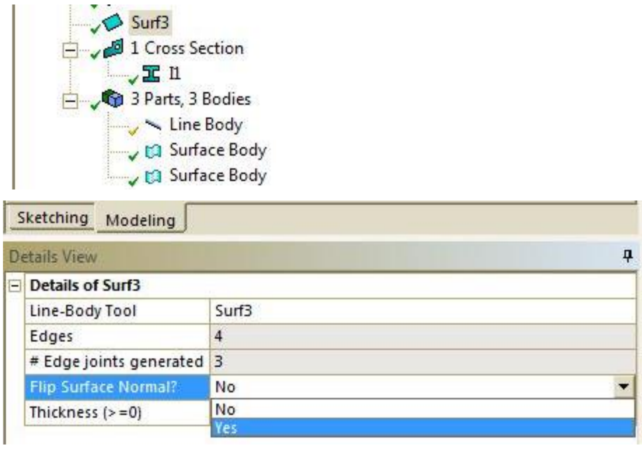
Adding Surface Body to a line body





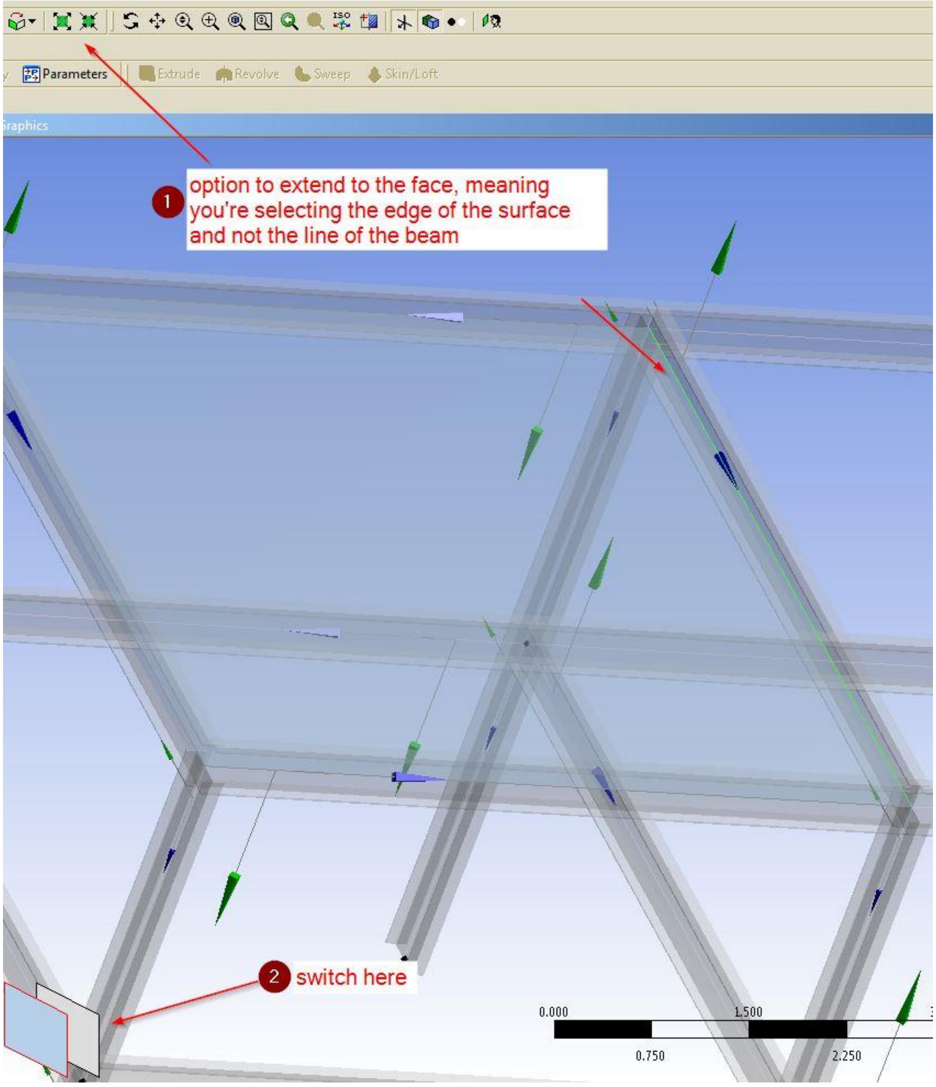




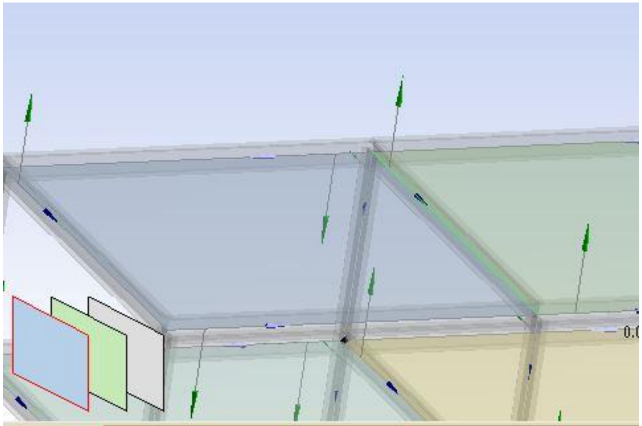


Direction of Surface Bodies

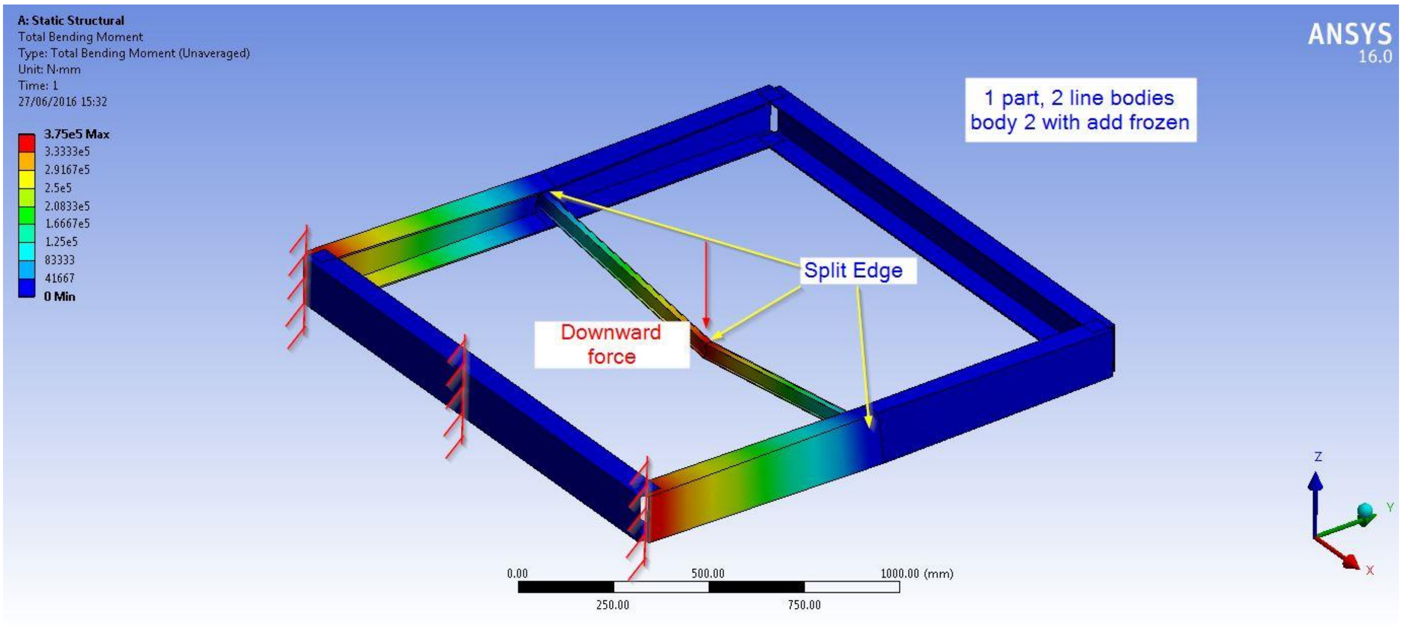
All Surface Body have a positive and a negative side. In Mechanical, the pressure is always applied on the positive side. When selecting faces, the positive side has a stronger green color than the negative (see above).



Or you can notice that the plane is blue just like the surface, which means you want the other.



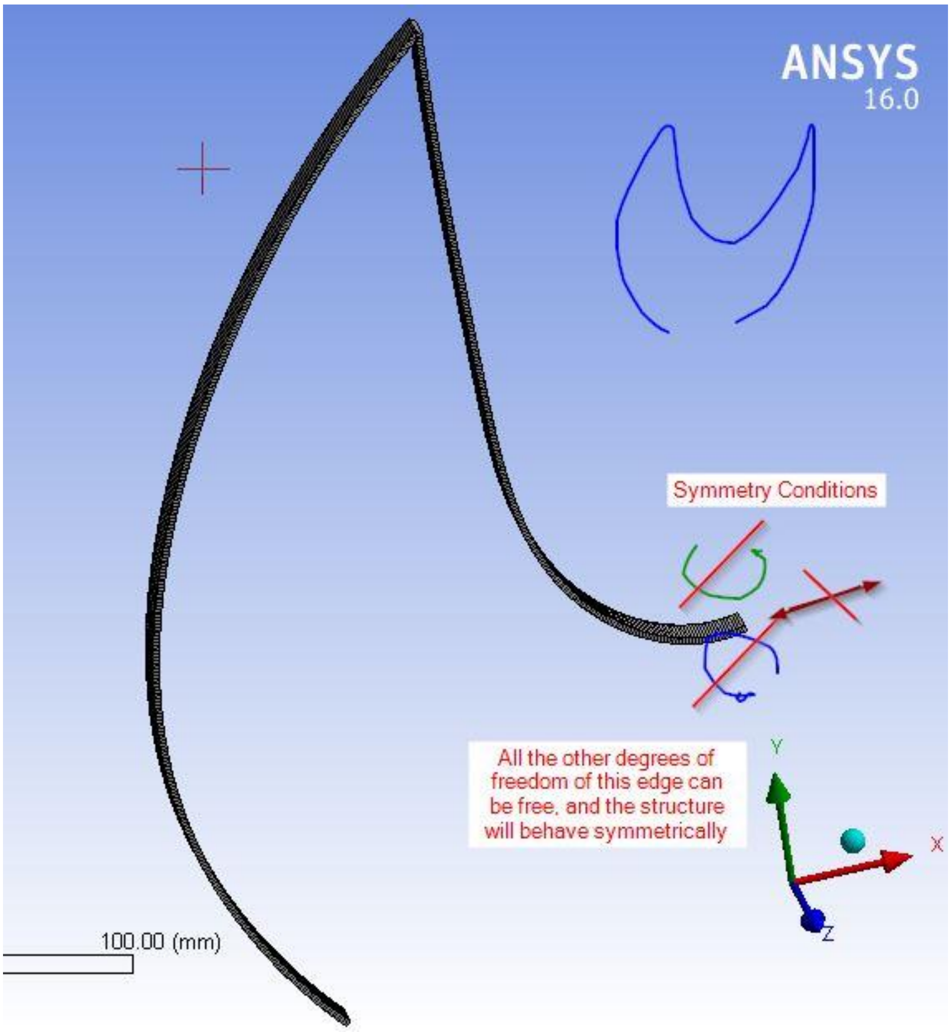
When you convert multiple parts into one, the previous parts are now called *bodies*, and they become bonded together and the mesh becomes compatible between the body boundaries.



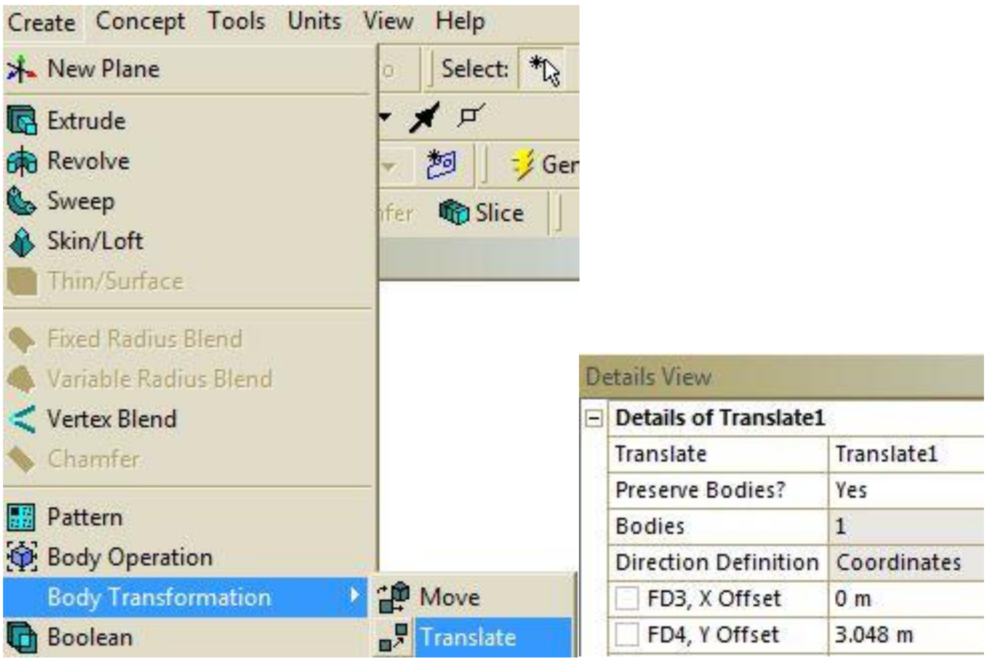
Using Symmetry by Setting Appropriate Boundary Conditions

A general rule of symmetry for shell and beam elements is to fix (that is, set as zero) the out-of-plane translations and in-plane rotations.

In the case below the symmetry plane is the YZ. So the out-of-plane translations are those along the X axis, and the in plane rotations are around the Y axis and the Z axis.



Copying a body



Buckling

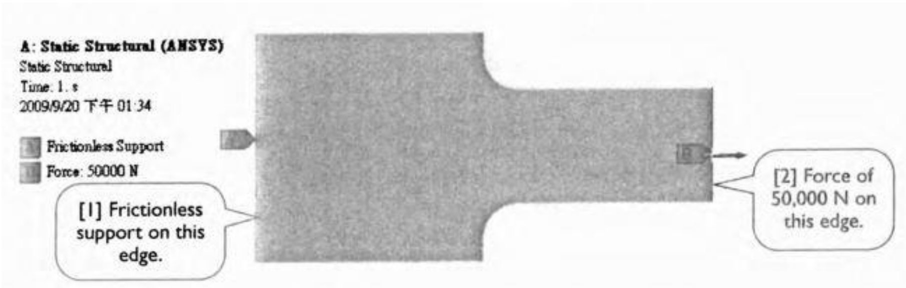
As mentioned in the beginning of Section 1.4, functionality, safety, and reliability are the main purposes of structural simulations. Stresses usually relate to the safety and reliability. In the 3D truss example (Section 7.2), calculated stresses are well below the material's yield strength. Can we conclude that the design is safe? Not yet. For any structural members (particularly those slender or thin members) subject to compressive stresses, we need to check their stability before concluding their safety. This chapter mainly deals with *stability analysis*, or *buckling analysis*.

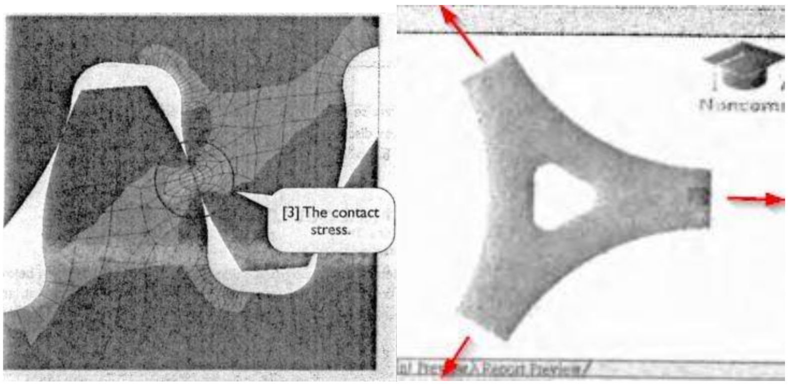
Buckling can be viewed as an ultimate case of a more general effect, called *stress stiffening*: a slender or thin structure member's bending stiffness increases with increasing axial tensile stress, and, on the other hand, the member's bending stiffness decreases with the increasing compressive stress. Buckling occurs when the compressive stress reaches a level such that the bending stiffness vanishes; the applying load is called a *buckling load* and the deformation is called a *buckling mode*. The purpose of buckling analyses is to find the buckling loads and the buckling modes.

Summary of Analysis Type

Can you cut a section of your part and the analysis you perform on it will be almost the same as the whole part ? --> 2D analysis (The mesh elements will be solid bodies)

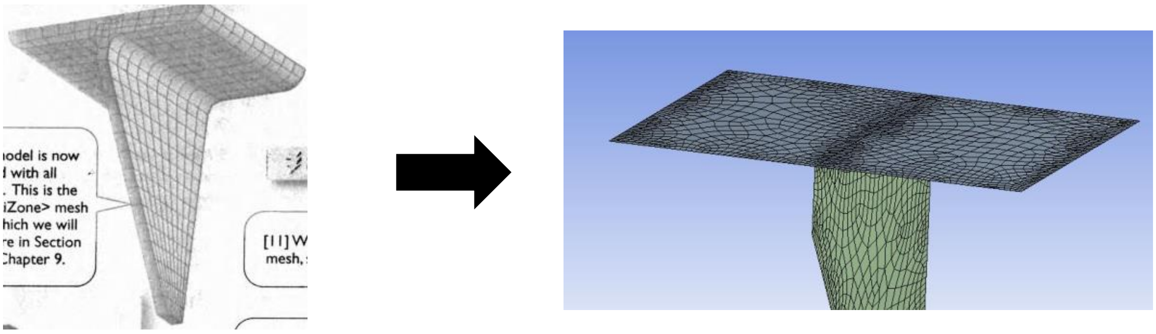
Examples:

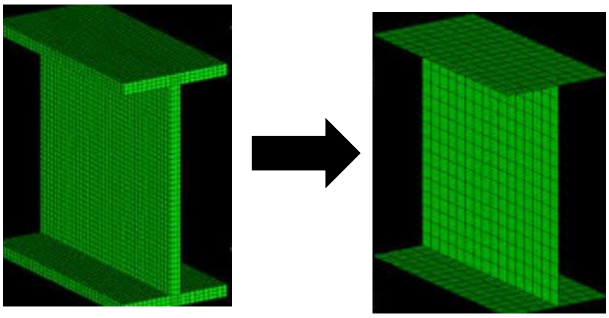




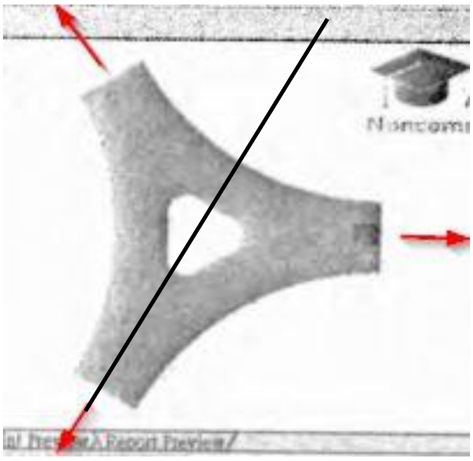
Is the structure composed of thin features, and which could be substituted by planes? --> 3D surface (The mesh elements will be shells)

Example:

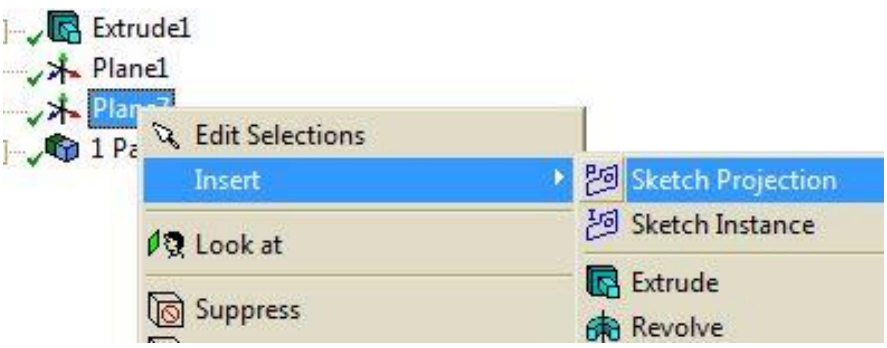




Are there planes of symmetry that you can use to cut the piece in 1/2 or 1/4 ?

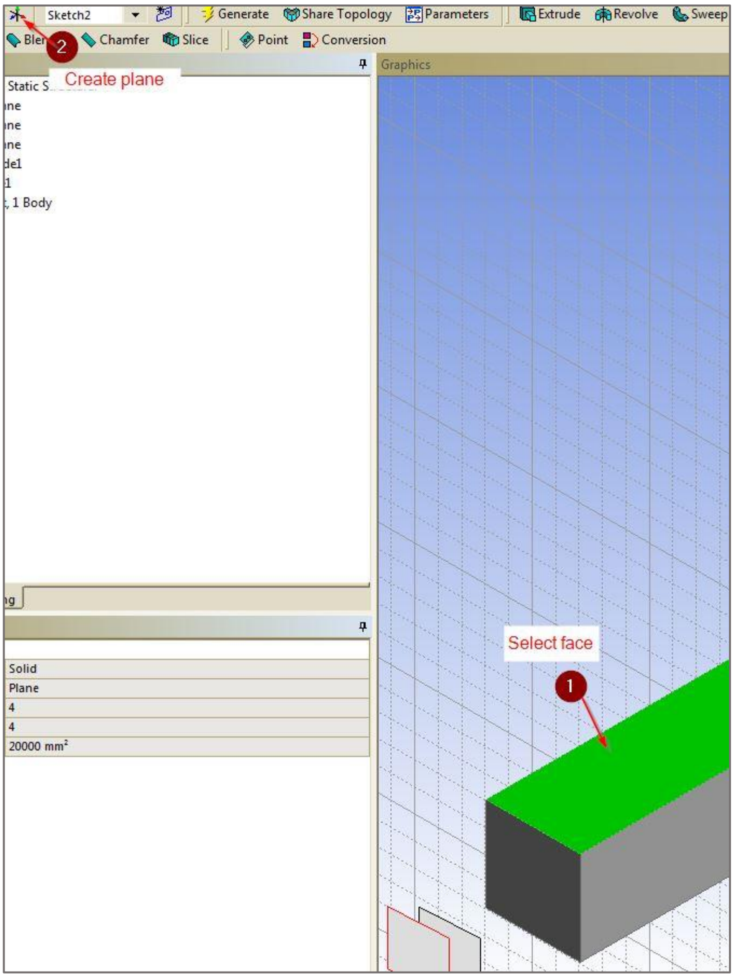


Does it have small lateral dimensions compared to longitudinal and uniform cross-section? --> Line body (The mesh elements will be beams)



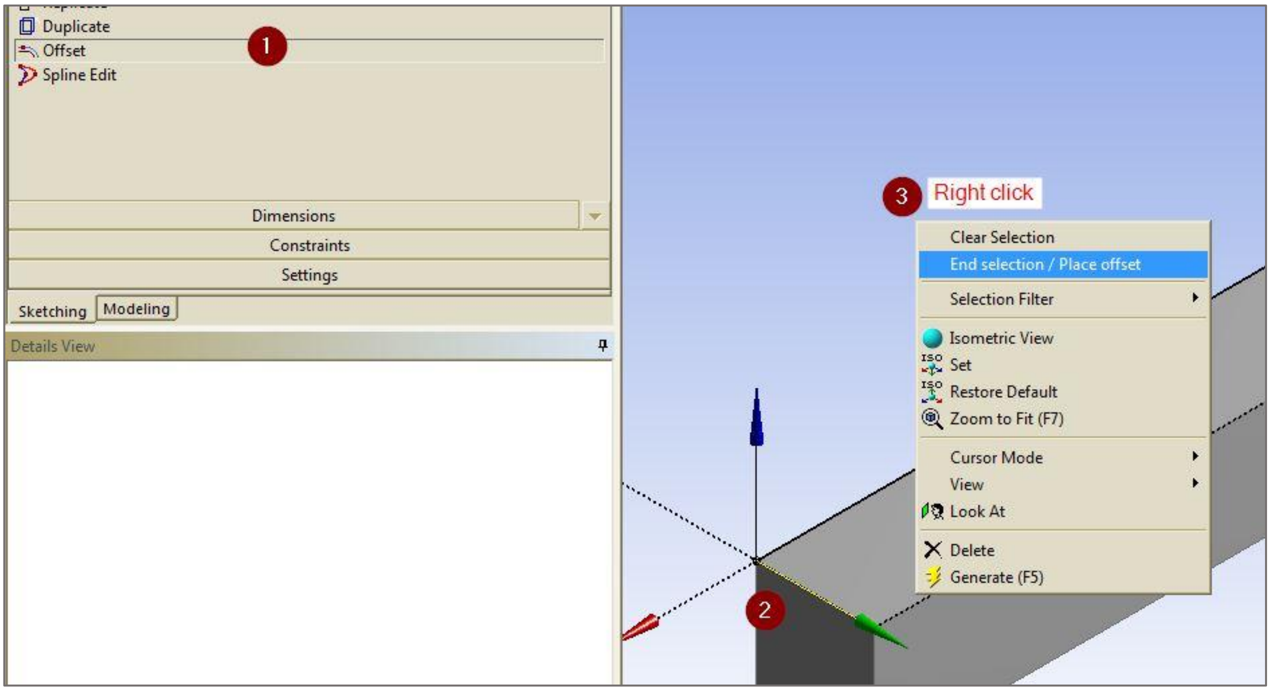
Applying a Force in a 3D Face

For that we need to create a line.

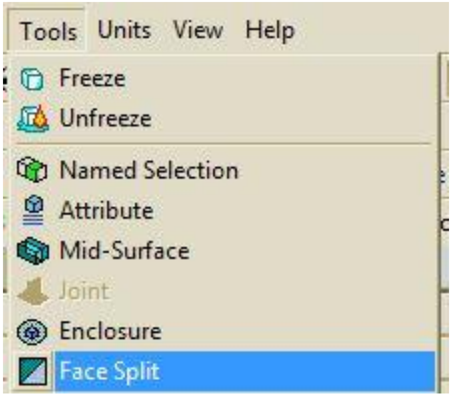


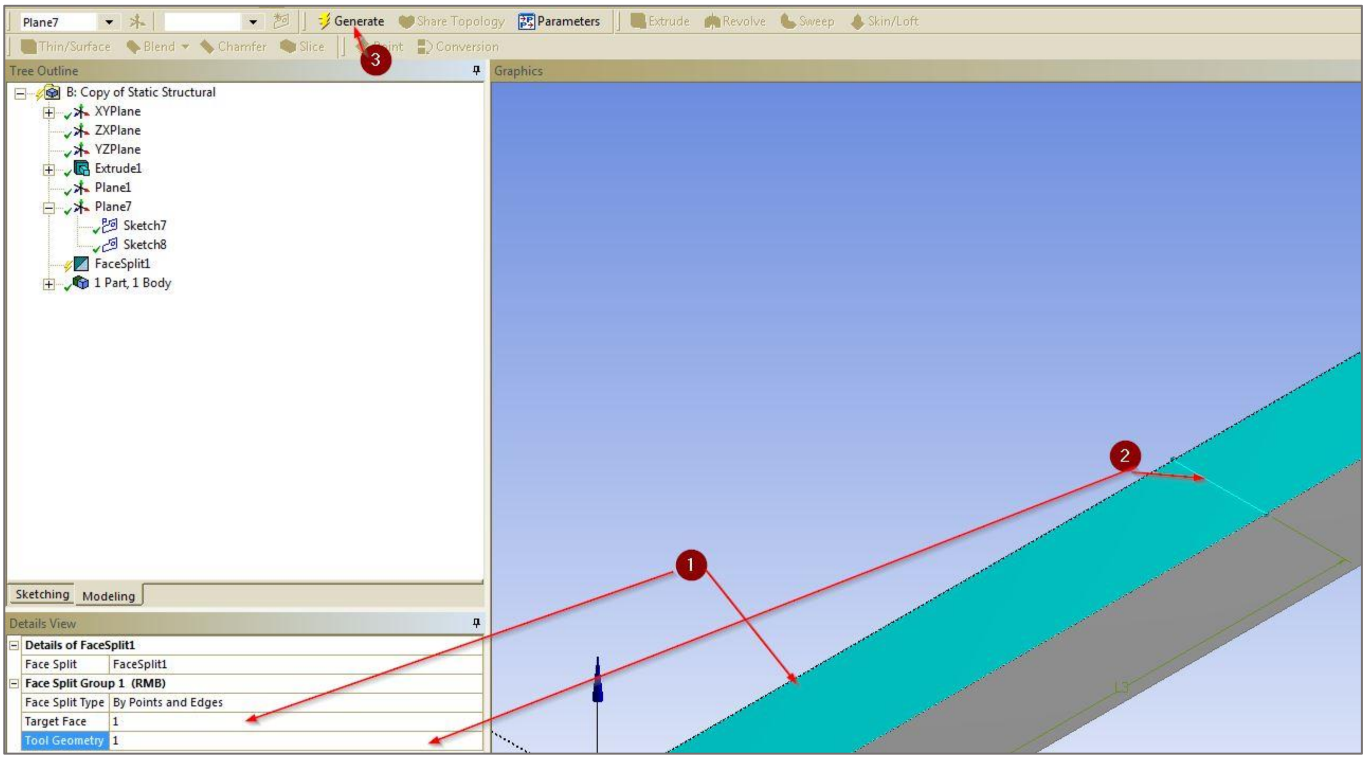
Then click generate.

This is the equivalent of SolidWorks Convert Entities



Then with the general dimensioning tool specify the precise distance that you want. Then split the faces in two, so that the line appears in the model.

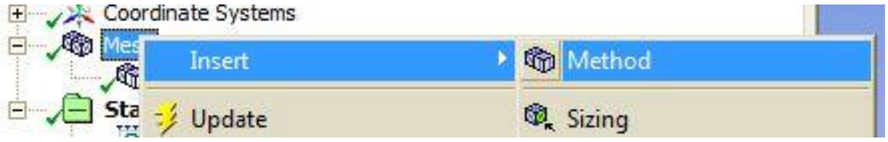




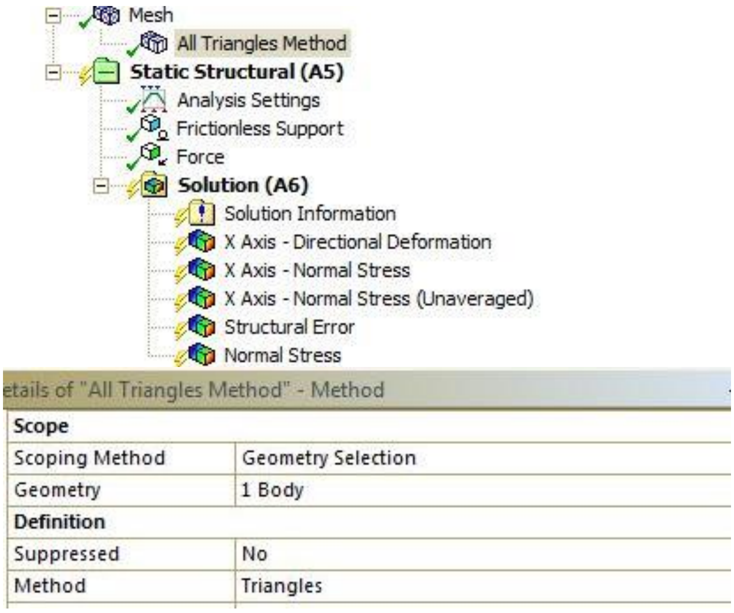
Mesh

It’s good practice to first see what the default mesh looks like and then decide what mesh controls are needed.

Changing Element Shape



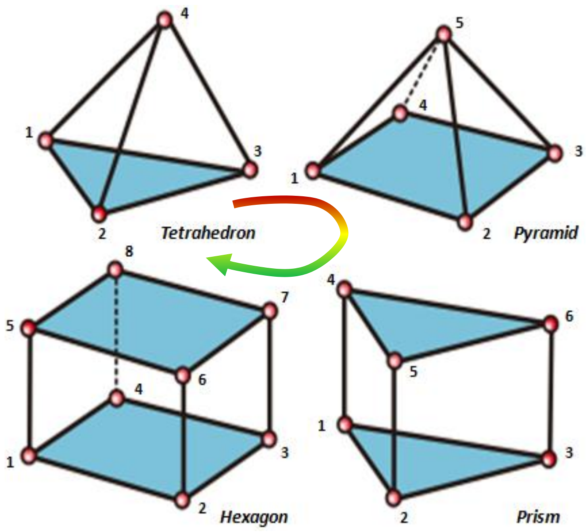
For triangles (2D) this would be



Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Triangles

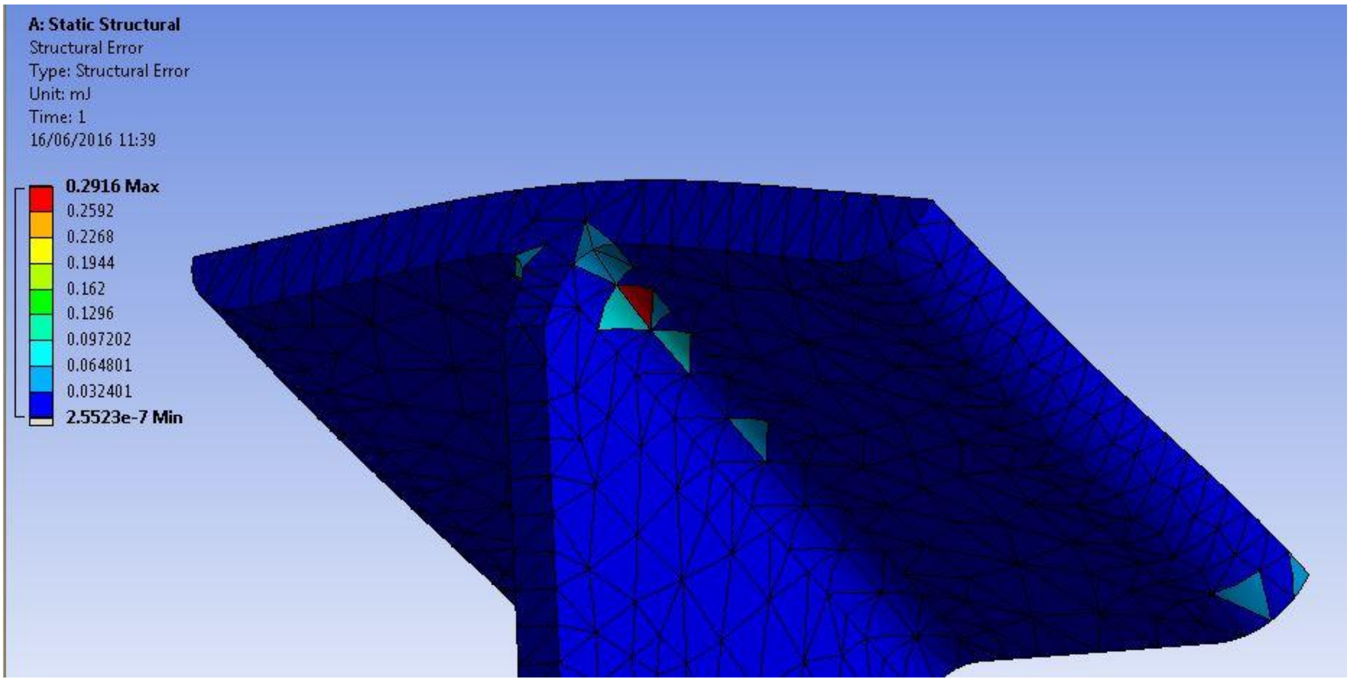
Speed of Results’ Convergence According to Element Shape

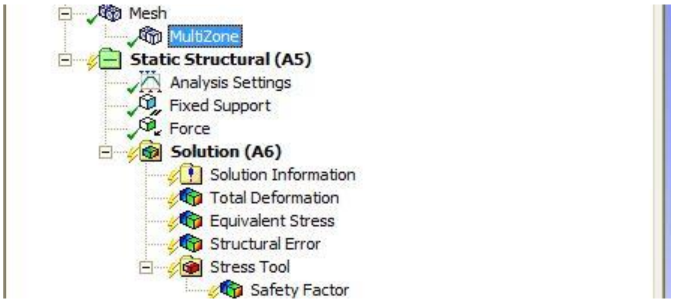
In 2D quadrilateral elements converge faster than triangular. In 3D, from faster to slower convergence are the hexahedral, prism, pyramid, and tetrahedral.



Besides converging faster, by changing the element shape you can also reduce structural error.

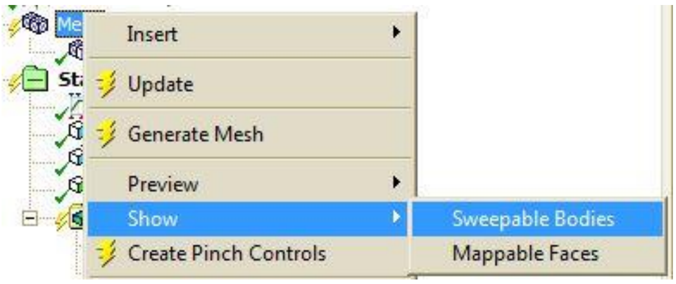
Let’s change the element shape from tetrahedral to hexahedral.



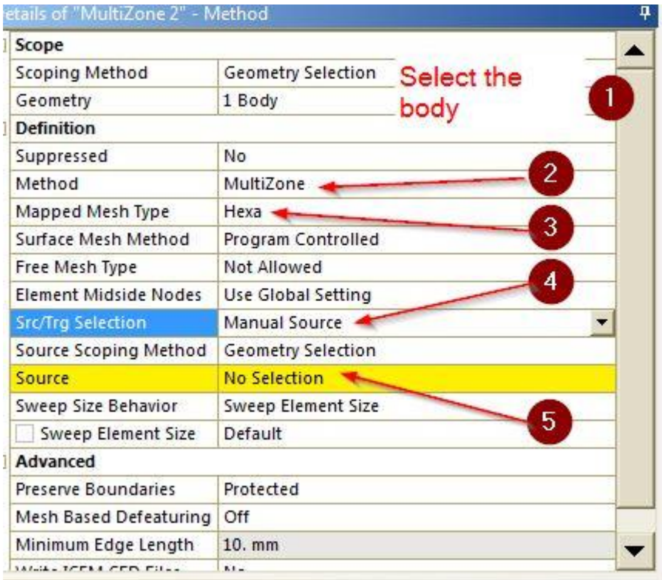


Sweeping

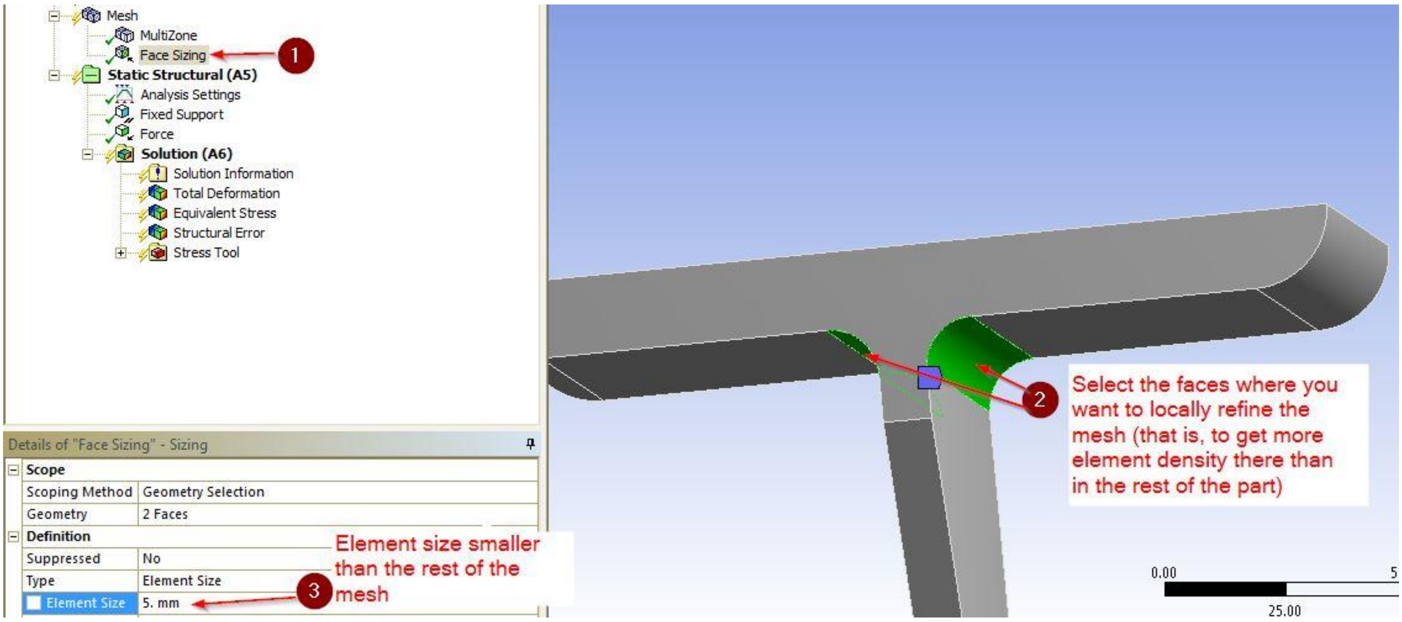
If the part is simple we can simply sweep it. But for that we need to see if there are sweepable bodies



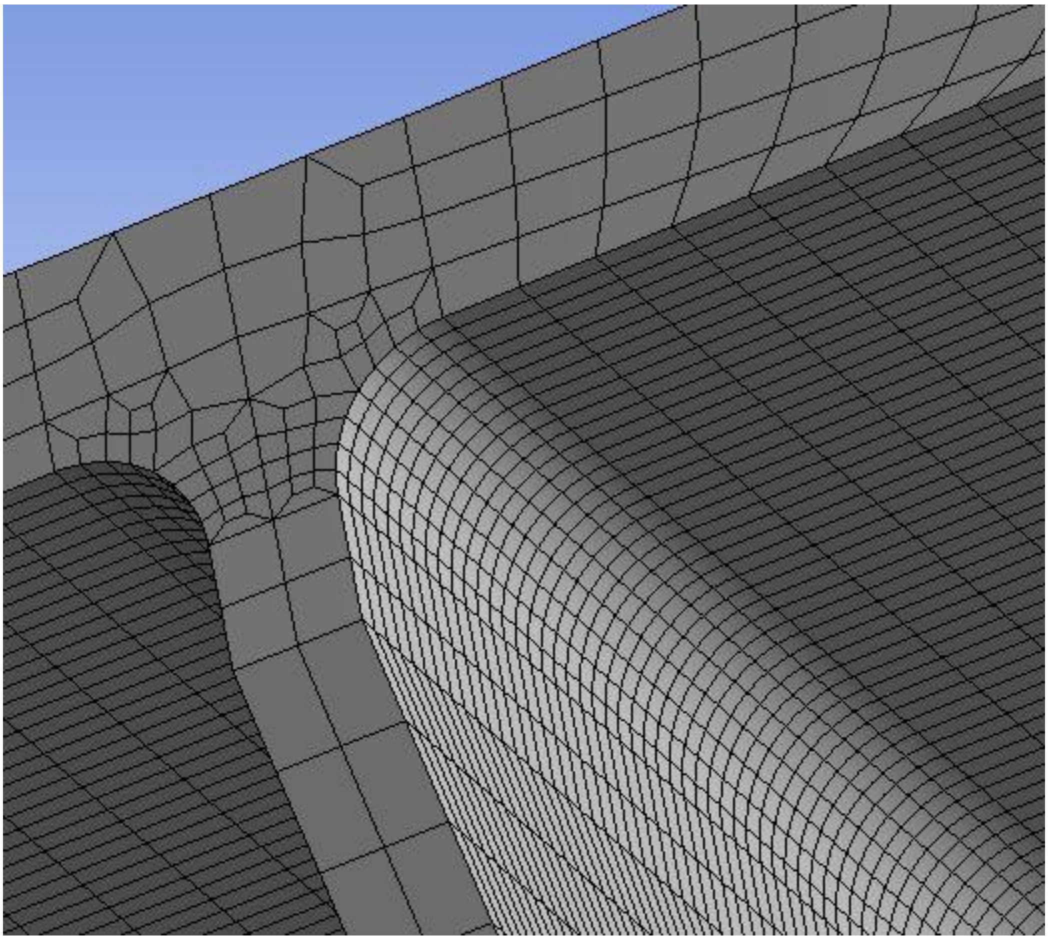
If the part is more complex, then it’s better to divide it into several zones to be sweeped — the MultiZone method.



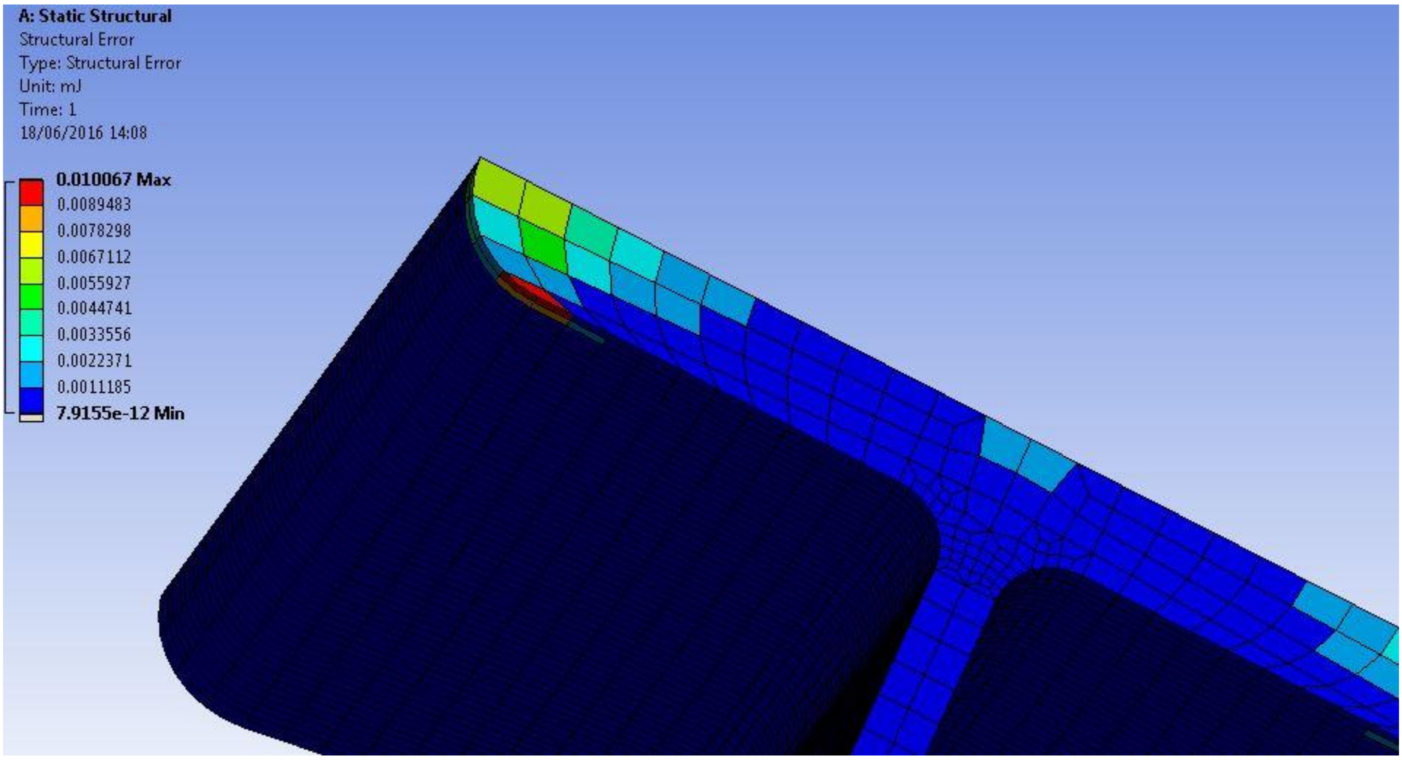
If on no. 4 we let "Program Controlled" it's fine. By selecting "manual source" you will then (5) chose faces/edges/... to give ANSYS priority in making well shaped elements. Because the structural error is bigger in the 2 fillets we can also increase the number of elements there.



Result



Now see the structural error significantly decreasing:

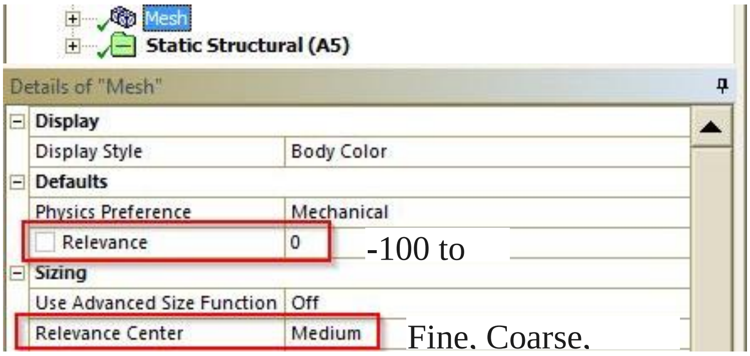


Besides, the structural error now occurs on a singularity point, so it's not as important. The singularity point is simulating a right angle that in real life doesn't exist (it's rounded, even if only very slightly, so the stress is not infinity).

See more about the MultiZone Method below. Use it where the structural error is bigger. More on Structural Error below.

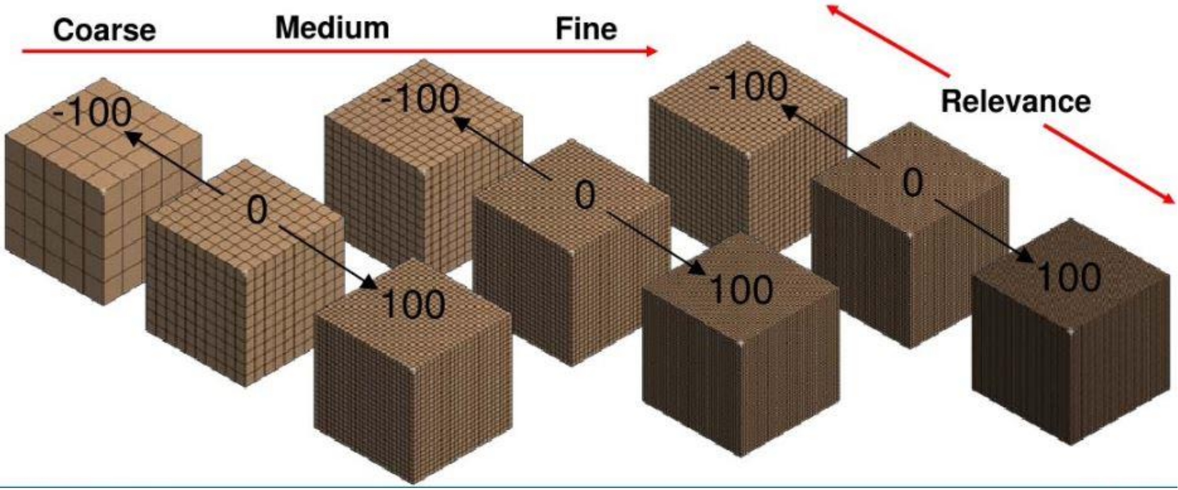
Mesh Properties

Relevance and Relevance Center



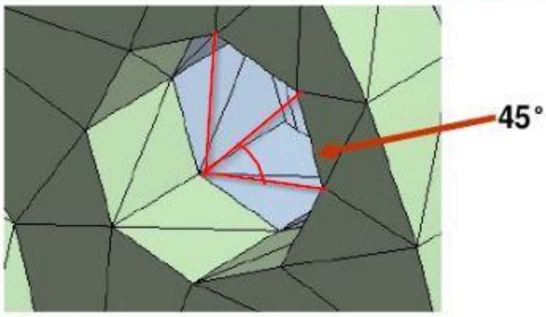
Relevance and Relevance Center, together, provide a way of global size mesh control. The larger the value in relevance, the smaller (/finer) the elements.

finer. These two values are related roughly as follows: (Coarse, 0) = (Medium, -100), (Coarse, 100) = (Medium, 0) = (Fine, -100), and (Medium, 100) = (Fine, 0).

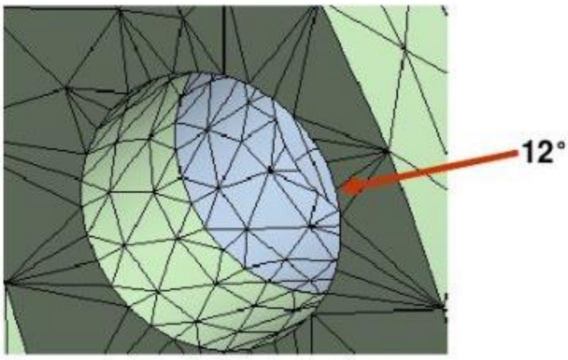


Span Angle Center

Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	5.9847e-002 m



Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Fine
Minimum Edge Length	5.9847e-002 m



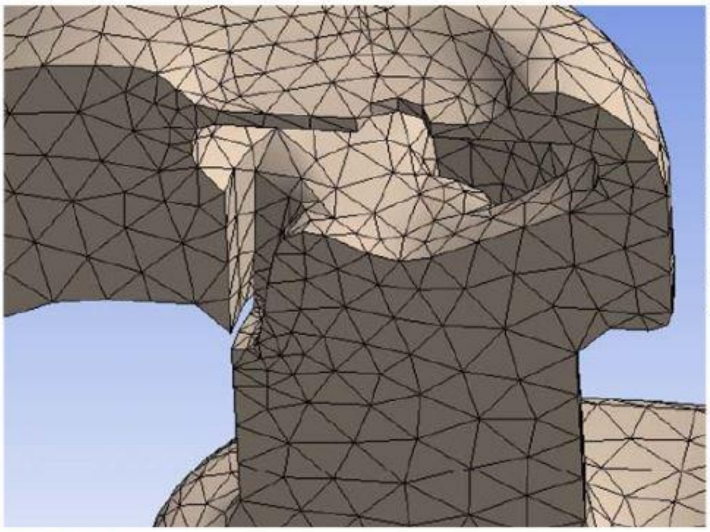
Advanced Size Function

- Without the advanced size function, edges are meshed according to the specified element size, refined for curvature and proximity, adjusted for defeaturing or pinch controls, and then passed to the face and volume meshers

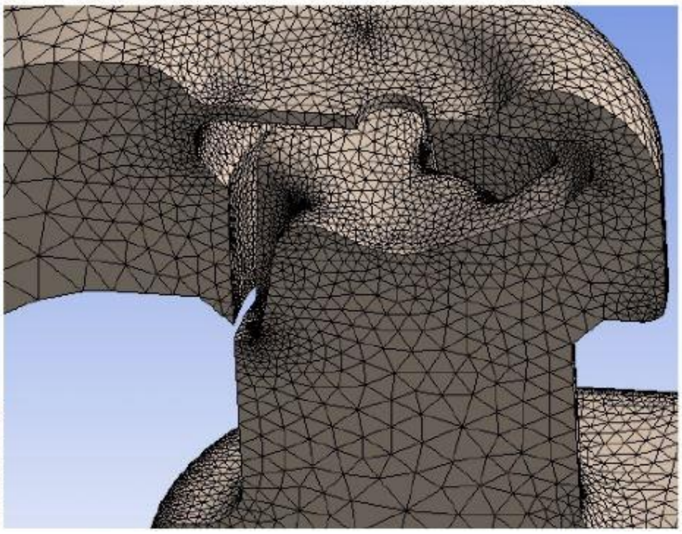
Details of "Mesh"

Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Off
Initial Size Seed	On: Proximity and Curvature
Smoothing	On: Curvature
Transition	On: Proximity
	On: Fixed

- Standard Size Function



- Advanced Size Function



Hex Dominant Method

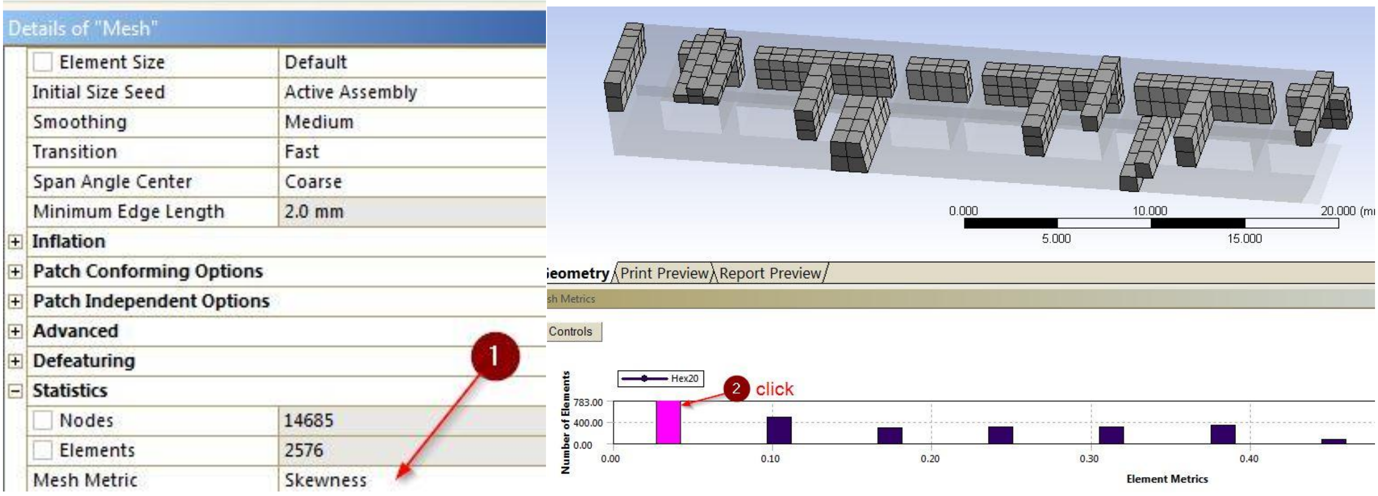
Details of "Hex Dominant Method" - Method

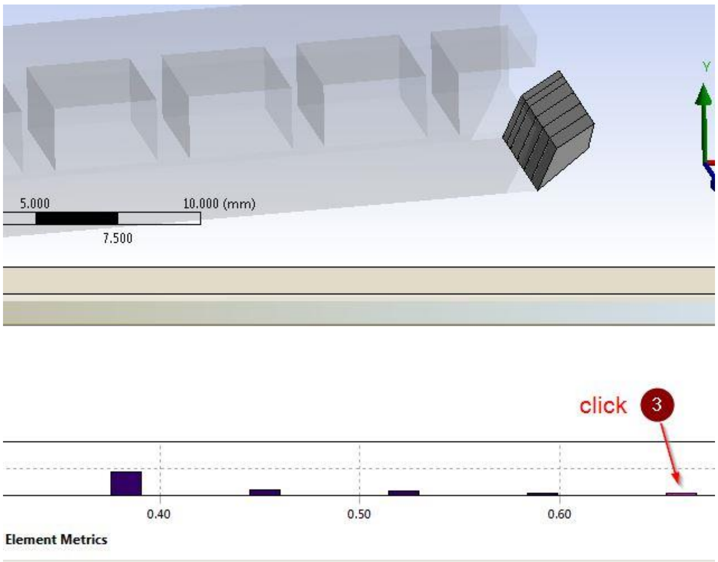
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Automatic
Free Face Mesh Type	Tetrahedrons
Control Messages	Hex Dominant
	Sweep
	MultiZone

An idea of <Hex Dominant> is to mesh the body with <Patch Conforming> first and then combine tetrahedra to form hexahedra. It usually leaves some tetrahedra that cannot be combined to form hexahedra; that is how the name <Hex Dominant> comes from. After forming hexahedra, the algorithm tries to adjust the nodes to improve the mesh quality further.

Note that, <Hex Dominant> method, by its nature, is a method of patch conforming, that is, the faces are not distorted. In fact, all methods except <Patch Independent> are patch conforming.

Analyzing Mesh Quality





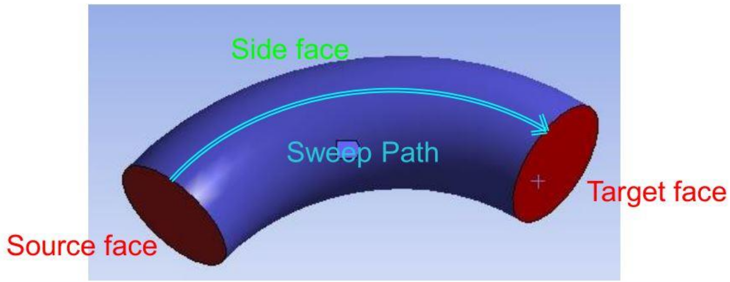
What is the next move?

Workbench can't properly divide the body into appropriate sweepable zones, meaning we need to do it manually. We'll divide the part into two bodies. As long as they're a single part, the mesh between the bodies, in the boundaries between both, will always be compatible, as they will share the same nodes.

How does sweeping work?

In evaluation of which method to use there are some important terms to consider/understand.

- When creating a hex mesh, a source face is meshed and then extruded to the target face
- Other faces are called side face(s)
- The sweeping direction or path is defined by the side face(s)
- The layers of elements between the source and target faces are created through interpolation schemes and projected to the side face(s).



The Thin Sweep and MultiZone methods were introduced to help resolve some of the difficulties with the general sweep approach.

General Sweep Method:

- Sweeps a single source/face to a single target/face.
- Does a good job of handling multiple side faces along sweep.
- Geometry needs to be decomposed so that each sweep path is represented by 1 body.

Thin Sweep Method:

- Good at handling multiple sources and targets for thin parts.

MultiZone Method

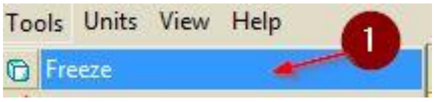
- Provides free decomposition approach: attempts to slice up the model without having to do this manually to the geometry.
- Supports multi-source and multi-target approach.

The basics of the Multizone Method is decompose a non-sweepable body into several sweepable bodies, and then apply the Sweep method on each body.

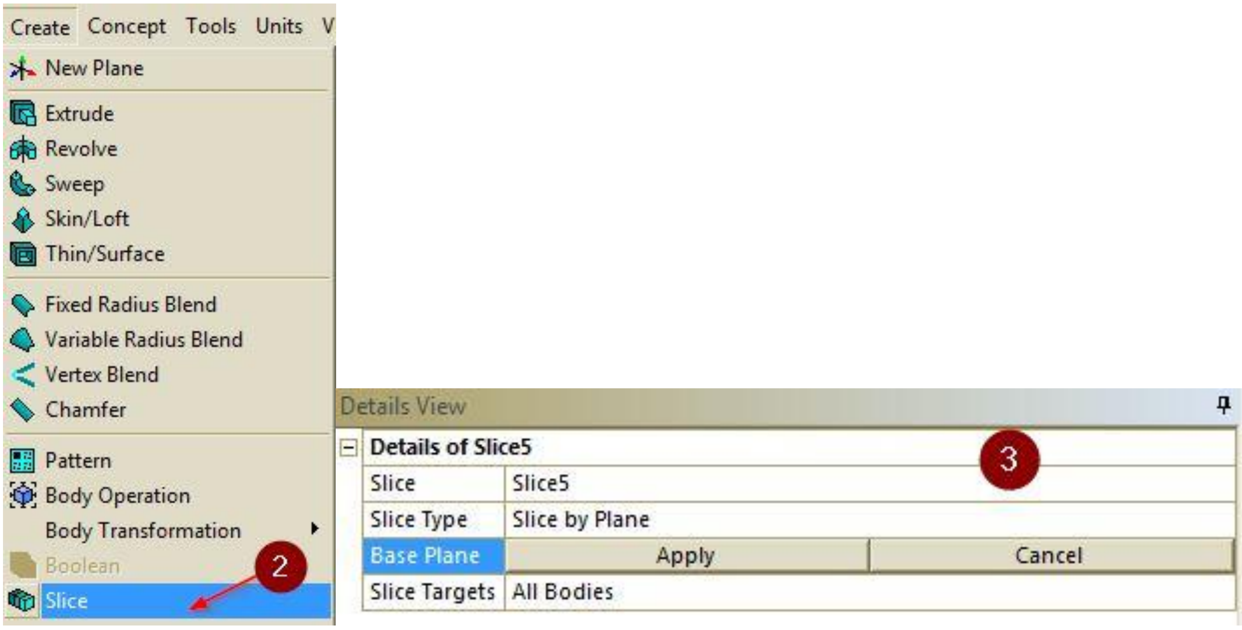
Workbench is usually smart enough to decompose the body into sweepable bodies. But, since these bodies are integral part, the boundaries between the bodies must be conformal (i.e, the boundaries must have the same surface mesh), these constraints may complicate the meshing task.

Splitting a Body to Improve the Sweep

You'll slice using a plane. So first you need to create the plane where you want to make the slice. Having done that:



All bodies need to be frozen for the slice.



You'll get 2 bodies. Merge them into a single part (for the reasons already explained).

Meshing With Patch Conforming Method

<Patch Conforming> and <Patch Independent> Methods

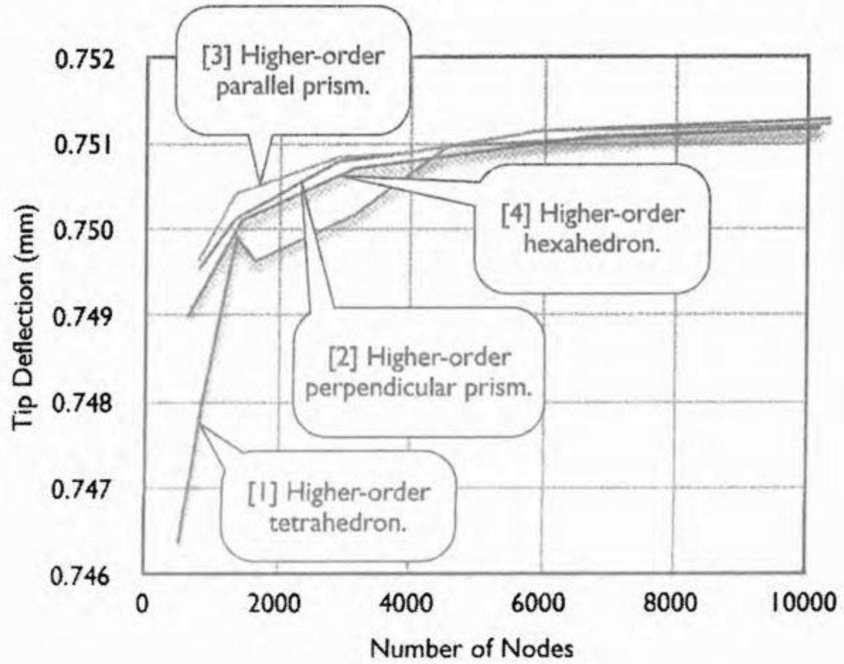
In CAD jargon, faces of solid bodies are also called *patches*. The basic idea of <Patch Conforming> is to mesh all the faces of the body with triangles and then "grow" inward to create tetrahedra. In this way, the shapes of the faces are respected (preserved); that is how the name <Patch Conforming> comes from. For complicated geometry, this is the default method.

On the other hand, <Patch Independent> creates tetrahedra from inside out. The outermost nodes are then projected onto the boundary faces and the element edges are created. In this way, the mesh's outline may be different from the original geometry; that is how the name <Patch Independent> comes from.

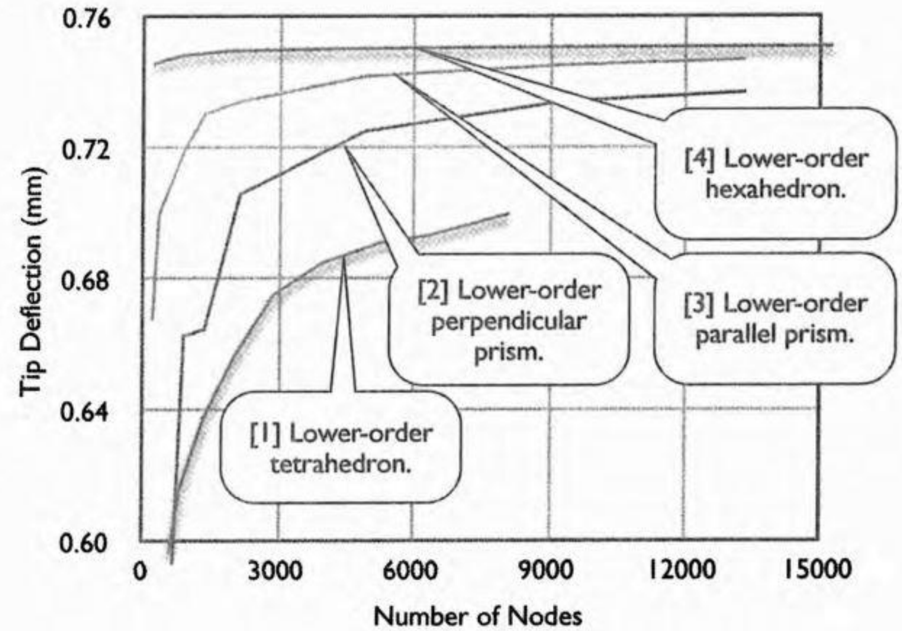
In some cases when too many details exist that cause meshing difficulty, we may resort to <Patch Independent> algorithm and ignore these details. However, it is your responsibility to make sure that ignoring those details wouldn't distort the geometry too much and raise an accuracy issue.

Higher vs Lower Order Elements

The chart below is made by a collection of the convergence curves in Sections 9.3-9, 9.3-10, 9.3-11, and 9.3-12 to compare the convergence behaviors of the higher-order elements. Except the tetrahedron, the order of the convergence speed among other elements is not obvious. The tetrahedron performs poorly only when the mesh is coarse (below 5000 nodes, for this case), otherwise it may be as good as other elements. Contrasting to the lower-order tetrahedron, the higher-order tetrahedron is still practically useful as long as the mesh is fine enough.



The chart below is made by a collection of the convergence curves in Sections 9.3-5, 9.3-6, 9.3-7, and 9.3-8 to compare the convergence behaviors of the lower-order elements. The order of the convergence speed is, from fast to slow, hexahedron, parallel prism, perpendicular prism, tetrahedron. The differences between them are obvious and quite evenly spaced. The lower-order tetrahedron converges so poorly that it is not practically useful. As a guideline, NEVER use lower-order tetrahedral elements.



Combining the observations in Section 3.5 and this section, we may summarize the conclusions as follows: (a) Never use lower-order tetrahedra or triangles. (b) Higher-order tetrahedra or triangles can be as good as other elements as long as the mesh is fine enough. In cases of coarse mesh, however, they perform poorly and are not recommended. (c) Lower-order prisms are not recommended. (d) Lower-order hexahedra and quadrilaterals can be used, but they are not as efficient as their higher-order counterparts. (e) Higher-order hexahedra, prisms, and quadrilaterals are among the most efficient elements so far we have discussed. Mesh your models with these elements whenever possible. If that is not possible, then at least try to achieve a higher-order hexahedra-dominant or quadrilateral-dominant mesh.

Besides the above guidelines, mesh quality requirements, in terms of mesh metrics such as skewness, should also be met.

Solution

In ANSYS ***von Mises stress***, or effective stress, is referred to as ***equivalent stress***.

Some Theory to Remember

von Mises, Equivalent, Maximum Principal Stress

The beauty of Von Mises stress is that in the real world "everything" fails by shear. That's why it has emerged as the favorite failure theory. Having said that, the world of material failure is highly stochastic — subject to statistical variation. So as good as the theory is, you still need significant factors of safety if you don't want your project to come crashing down.

You find Von Mises stress from the principle stresses by using a big ol gnarly equation or three. It is always a smaller value than maximum principle stress (by definition) BUT it is aligned in the direction that has to support the maximum shear load. This can be very helpful in design.

Maximum Principal Stress Theory - According to this theory failure will occur when the maximum principal stress in a system reaches the value of the maximum stress at elastic limit in simple tension. This theory is approximately correct for cast iron and brittle materials generally.

Von Mises Stress (Distortion Energy Theory) - This theory proposes that the total strain energy can be separated into two components: the volumetric (hydrostatic) strain energy and the shape (distortion or shear) strain energy. It is proposed that yield occurs when the distortion component exceeds that at the yield point for a simple tensile test.

Stress Fields

After the displacement fields are calculated, the strain fields are calculated using Eq. 1.2-7(1), and stress field are in turn calculated using Eq. 1.2-8(1).

The calculations are element-by-element. The figure in the right is a typical results of stress calculation [2, 3]. Note that a node may have multiple stress values, since the node may connect to multiple elements, and each element calculation results a value.

This behavior is not difficult to understand. Since differentiations are involved in the Eq. 1.2-7(1), this makes the strain field and the stress fields become piecewise continuous only. They are not continuous across the element boundaries.

By default, stresses are averaged on the nodes, and the stress field is recalculated. After that, the stress field is continuous over the body.

Strain-Displacement Relations

ANSYS calculates the displacement for each node. Knowing that then it can calculate the deformations by using the relations:

$$\varepsilon_X = \frac{\partial u_X}{\partial X}, \quad \varepsilon_Y = \frac{\partial u_Y}{\partial Y}, \quad \varepsilon_Z = \frac{\partial u_Z}{\partial Z}$$

$$\gamma_{XY} = \frac{\partial u_X}{\partial Y} + \frac{\partial u_Y}{\partial X}, \quad \gamma_{YZ} = \frac{\partial u_Y}{\partial Z} + \frac{\partial u_Z}{\partial Y}, \quad \gamma_{ZX} = \frac{\partial u_Z}{\partial X} + \frac{\partial u_X}{\partial Z}$$

Stress-Strain Relations

To get the stress values we need to call Hooke's law:

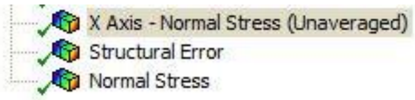
$$\varepsilon_X = \frac{\sigma_X}{E} - \nu\frac{\sigma_Y}{E} - \nu\frac{\sigma_Z}{E}$$

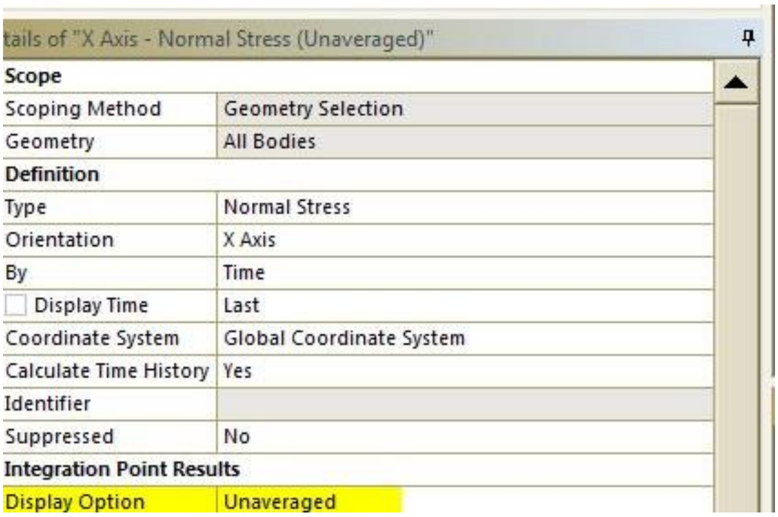
$$\varepsilon_Y = \frac{\sigma_Y}{E} - \nu\frac{\sigma_Z}{E} - \nu\frac{\sigma_X}{E}$$

$$\varepsilon_Z = \frac{\sigma_Z}{E} - \nu\frac{\sigma_X}{E} - \nu\frac{\sigma_Y}{E}$$

$$\gamma_{XY} = \frac{\tau_{XY}}{G}, \quad \gamma_{YZ} = \frac{\tau_{YZ}}{G}, \quad \gamma_{ZX} = \frac{\tau_{ZX}}{G}$$

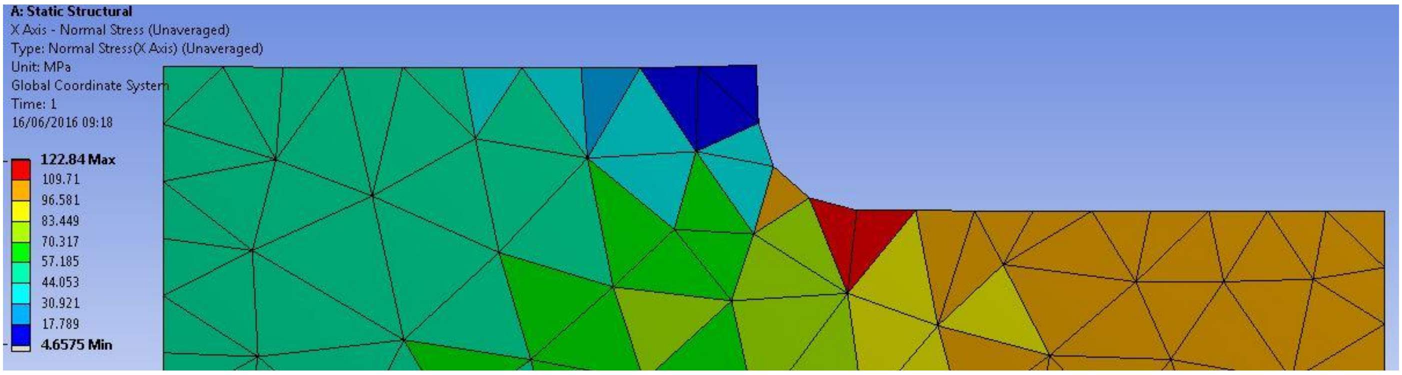
For what I'm understanding, the displacement values are calculated for each node. But then the strain is calculated for the element. Then introducing E and G we get the stress, for each element as well. After this, each node may have different values. Because of that, after these calculations, these nodal values are averaged.





tails of "X Axis - Normal Stress (Unaveraged)"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Normal Stress
Orientation	X Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Unaveraged



In general, as the mesh is getting finer, the solution is more accurate, and the stress discontinuity is less obvious. Thus, stress discontinuity can be used as an index for solution accuracy: the less discontinuous of the stress field, the more accurate of the solution.

Types of Nonlinearities

When the responses (deflection, stress, strain, ...) of a structure are linearly proportional to the loads, the structure is called a ***linear structure*** and the simulation a ***linear simulation***. If not, then they're ***nonlinear structure*** and ***nonlinear simulation***.

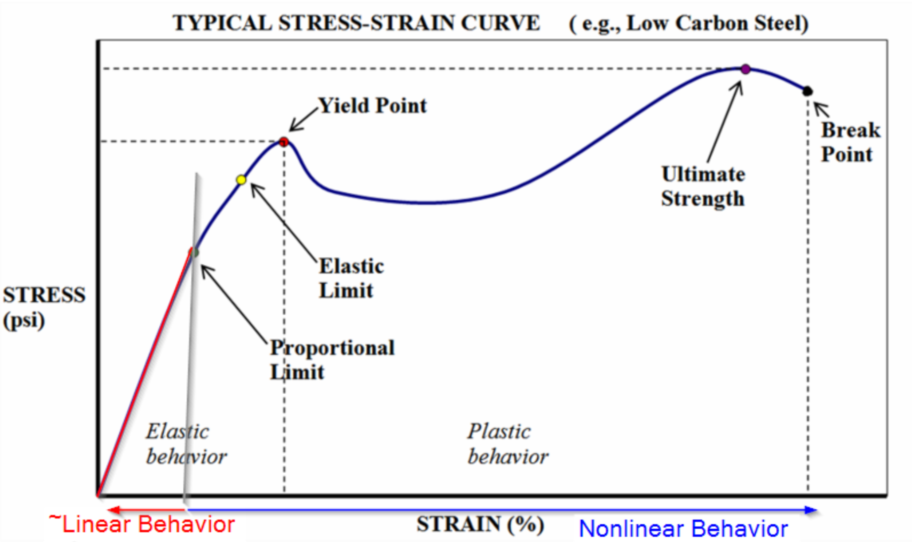
There are 3 main sources of structural nonlinearities:

1) Due to large deformation – ***geometric nonlinearity***. In a geometrically linear setting, the equations of equilibrium are formulated in the undeformed state and are not updated with the deformation. In most engineering problems, the deformations are so small that the deviation from the original geometry is not perceptible. The small error introduced by ignoring the deformations does not warrant the added mathematical complexity generated by a more

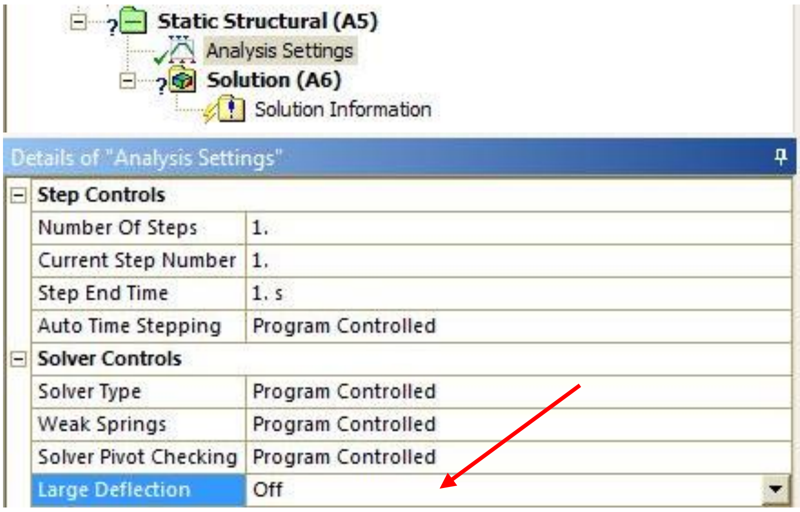
sophisticated theory. This is why a vast majority of analyses are made with an assumption of geometric linearity.

2) Due to topological change of the structure, e.g., contact nonlinearity which happens because of change of status of contact – ***topologic nonlinearity***

3) Due to the nonlinear stress-strain relationship of the material – ***material nonlinearity***



In a simulation where we have large deformations/deflection we should turn that on in the analysis settings, so that ANSYS updates the model during the simulation. This will give more accurate solutions (and in fact even when the deformations are small), but takes more computing time. To evaluate the need to include geometric nonlinearity try to run the simulation with it on and off. A substantial difference indicates the need for the account of geometric nonlinearity.

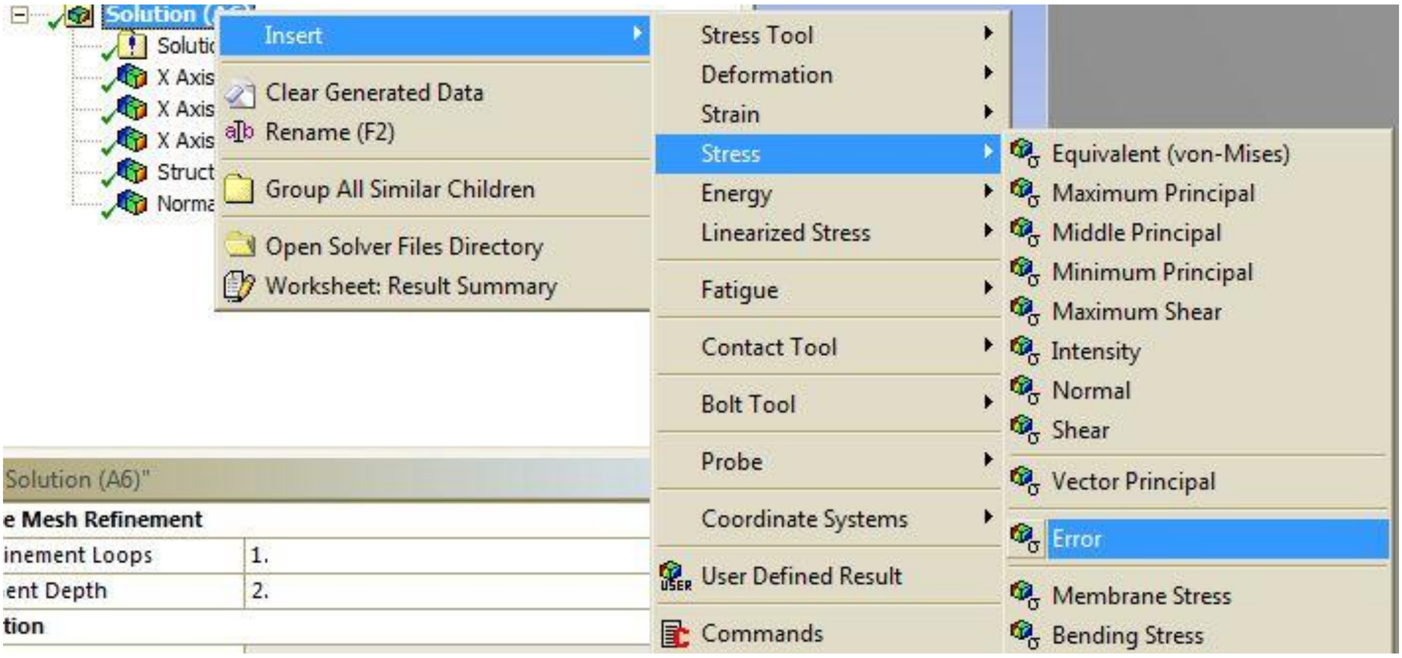


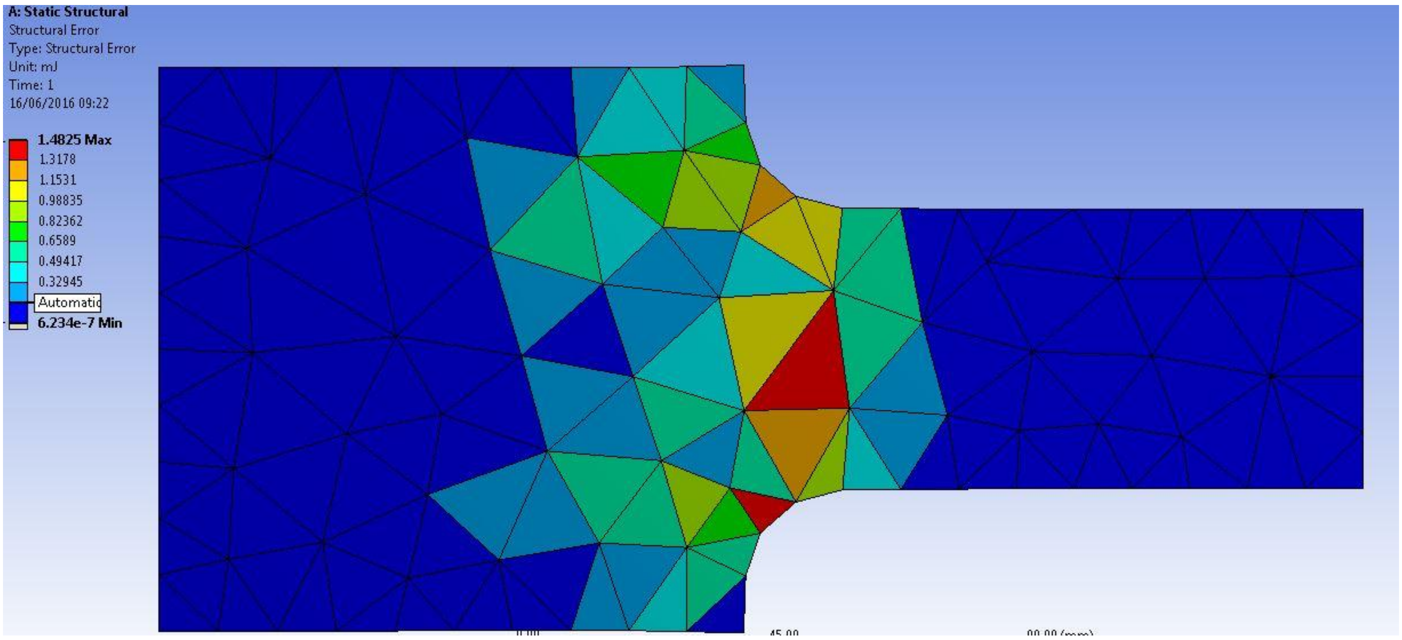
Structural Error

For an element, strain energies calculated using averaged stresses and unaveraged stresses respectively are different. The difference between these two energy values is called <Structural Error> of the element. The finer the mesh, the smaller the structural error.

The structural error can be used for two purposes: (a) As an indicator of global mesh adequacy. In general, we want the values as small as possible. Refining the mesh globally is a way of reducing structural error. (b) As an indicator of the local mesh adequacy. In general, we want the structural error distribution as uniform as possible to maximize the efficiency of computing resource usage. This implies that in the region of large values of structural error we need to reduce the element size while in the region of small values we may enlarge the element size.

How to show Structural Error:



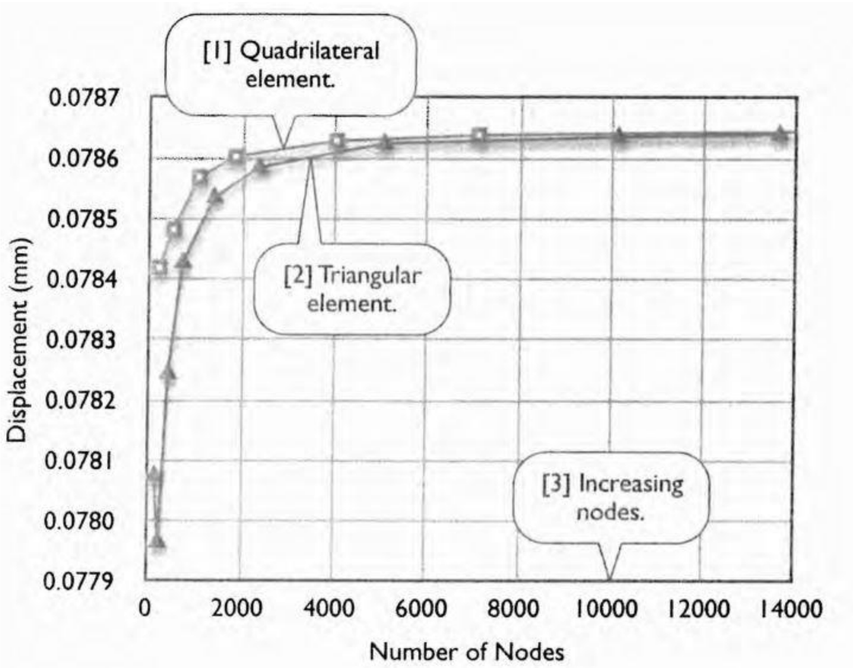


The more you reduce structural error, therefore, the more accurate the results will be. However remember that there’s no point in refining the mesh too much at stress singularity points, as the stress will keep increasing until the theoretical infinity.

For our case, we will not go further to improve the mesh. The reason is that the maximum structural error now occurs on a region of stress singularity. Improving mesh on a region of stress singularity makes little sense.

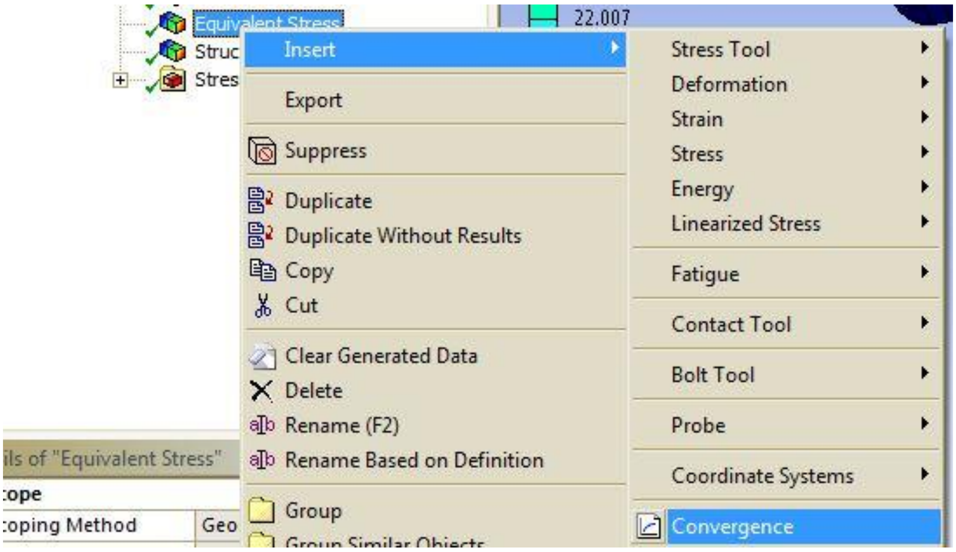
Convergence

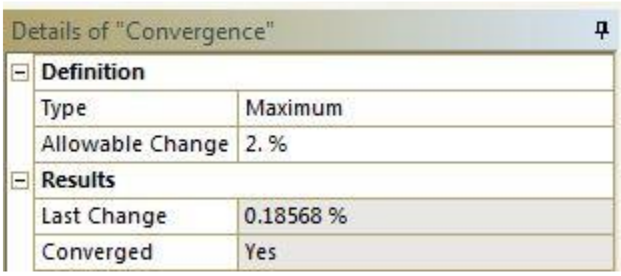
For 2D analysis squares converge faster than triangles. Also, the deformation values are always underestimated because the stiffness matrix is overestimated.

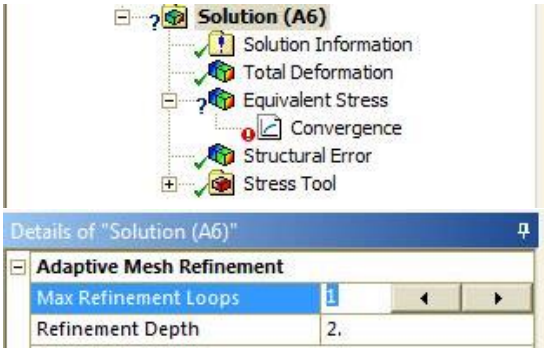


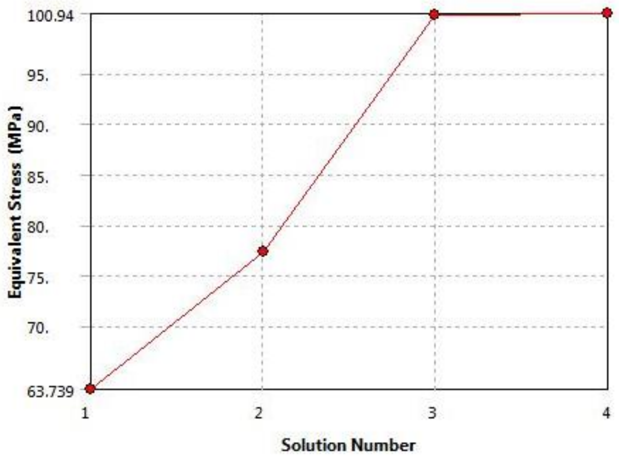
How to verify if results is OK or not?

- Method 1
  - Check Error Plot
    - Stress should be continuous. There should not be any sudden change of stress between adjacent mesh
    - Error plot will check the continuity. High value means the stress are not continuous, thus require mesh refinement.
    - Manually refine the mesh and check the error plot
- Method 2
  - Check Results Convergence
    - As mesh density increases, the results should be more accurate
    - Therefore, as we increase the mesh, the results should converge to a single value
    - Set the convergence criteria and check the results as well as error plot

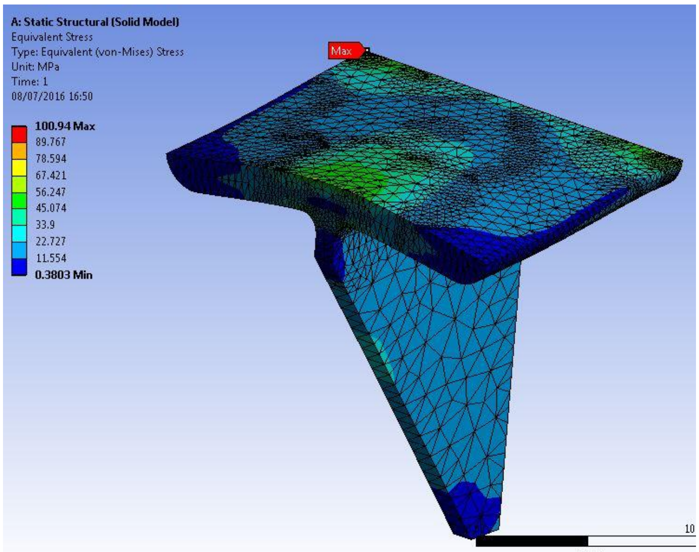








	Equivalent Stress (MPa)	Change (%)	Nodes	Elements
1	63.739		5496	2851
2	77.384	19.337	14844	8846
3	100.75	26.237	43651	28583
4	100.94	0.18568	107681	73277



Basically refines the mesh in the places where the change in that result (in this case equivalent stress) is above the defined maximum (in this case 2%). In the last loop the equivalent stress only changed 0.2% (< 2%) so the analysis stopped.

Stress Singularity

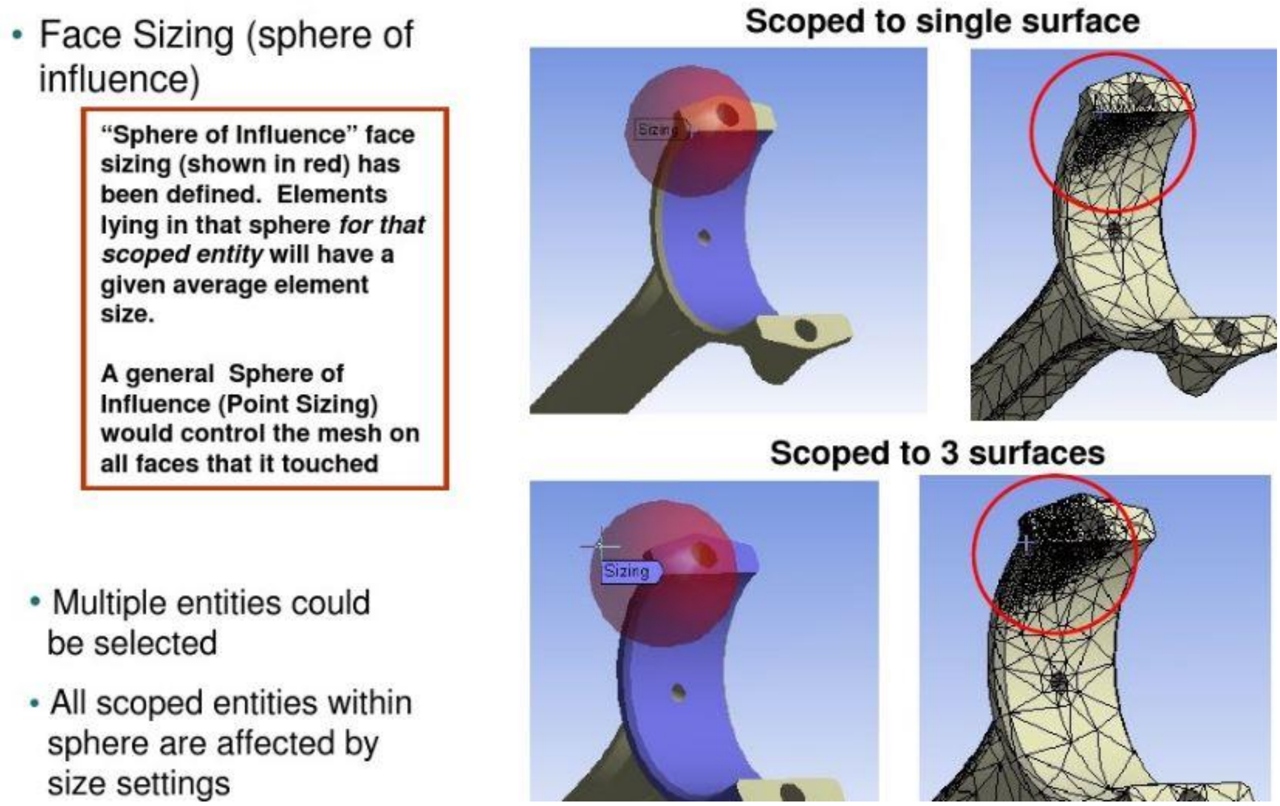
The concept of stress concentration. Stress is larger near corners than away from them. So in order to accurately evaluate the concentrated stress, there needs to be a finer mesh used around it. The less round the corner is, the higher the stress concentration. When the corner has no radius, then in that sharp corner the stress is infinite, and the corner is called a point of stress singularity.

In the real world, though, corners like that don't happen as it's near impossible to manufacture such a sharp corner as the precisions of current machining don't allow for that (maybe with 3D printing ;)).

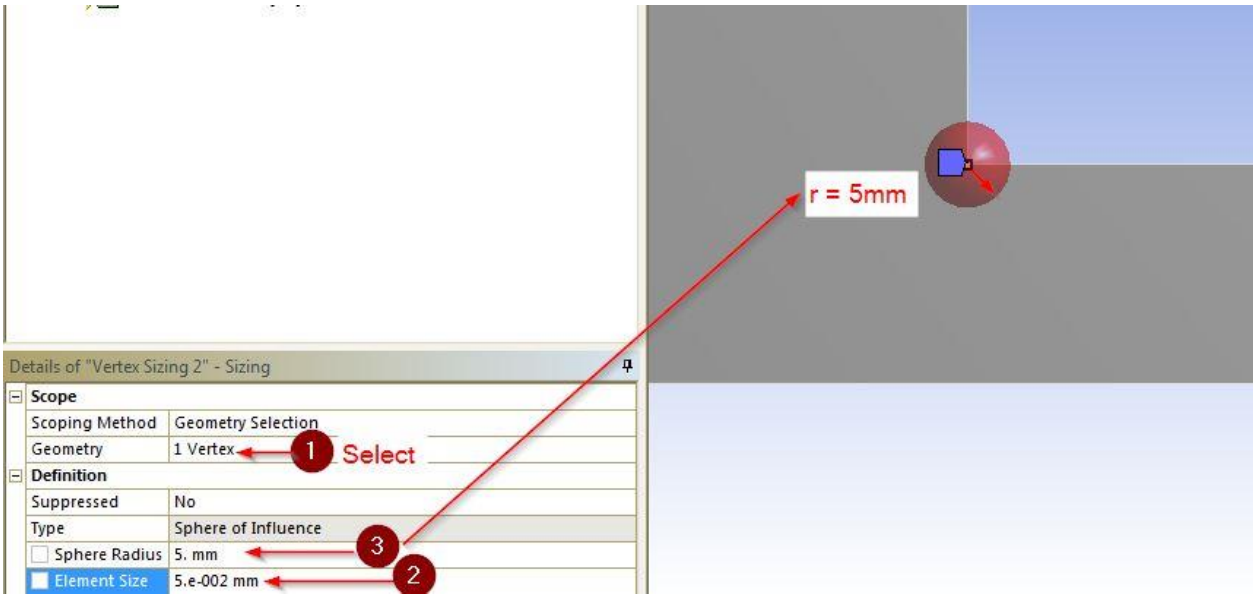
In the computer though, since we're dealing with the theoretical, zero-radius fillets are frequent. More, because certain small features such as small fillets don't contribute much to the global behavior of the structure, they are often not included in the model for simulation.

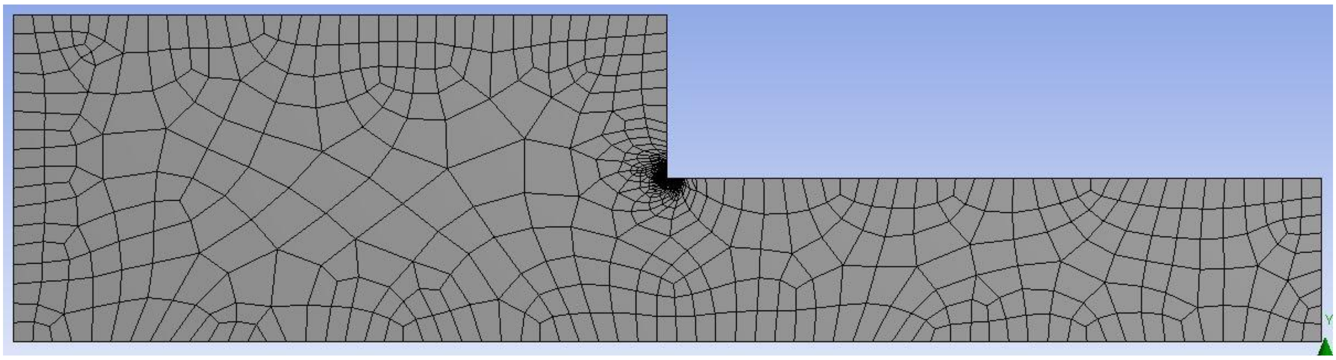
It becomes, then, important that engineers are aware of the existence of these points. They may mistakenly use the maximum stress as the design stress, but glazing over the fact that it came from a singularity – it wouldn't exist in the real world. So always check if the maximum stress comes from a singular point.

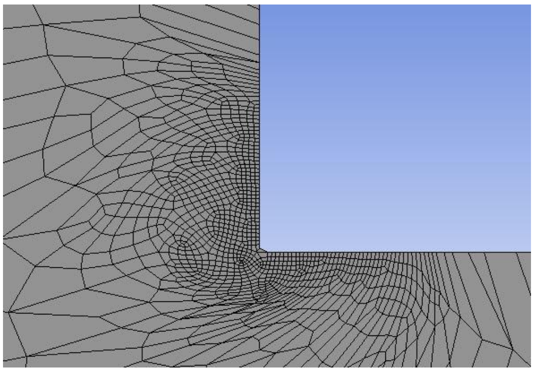
So what can we do about this in ANSYS ? We can increase the mesh around what we believe can be a singularity to assess whether it is one or not:



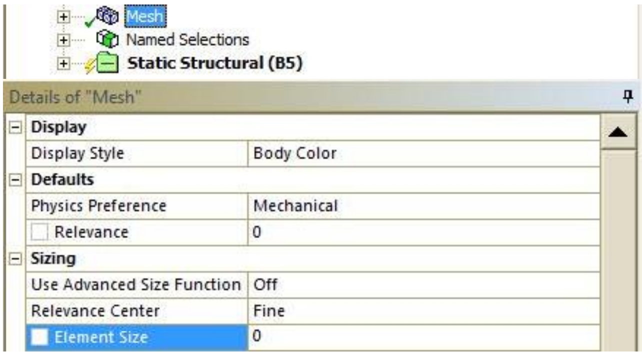
We want to have at least 10 elements of the small size. In this case if we chose an element size of 0,05mm then we should chose a sphere radius of at least (0,05*10=0,5mm). Because we suspect this is a singularity then we don't need that many elements around it because the high stress value should be in a very small point, and there's no need to increase computing time with needless elements. In this case 5mm is more than enough.



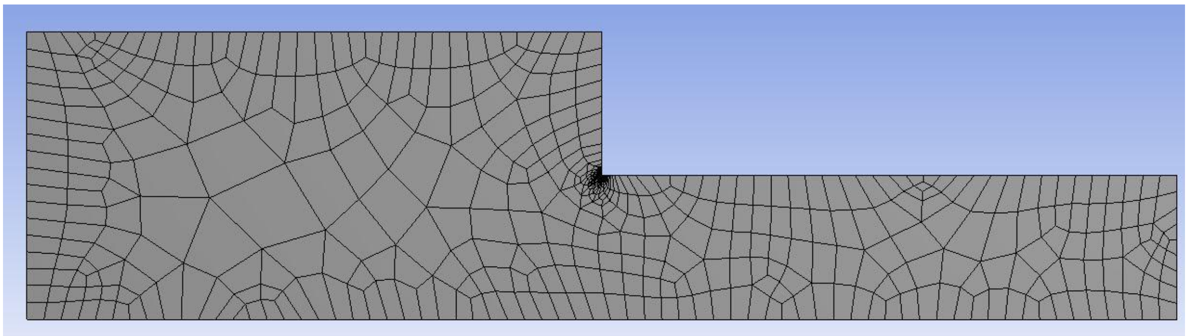




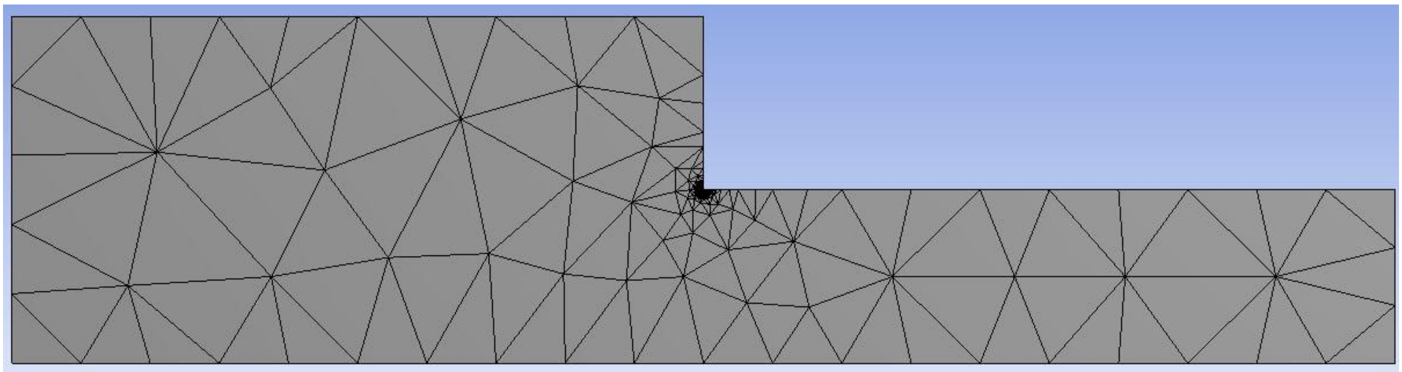
As for the mesh on the rest of the part, put 0 on element size (which for ANSYS = default) meaning that you'll let ANSYS decide the size of the mesh on the rest of the part. If you specify a value, ANSYS will try to make all elements of that size (except the ones which have sizing).



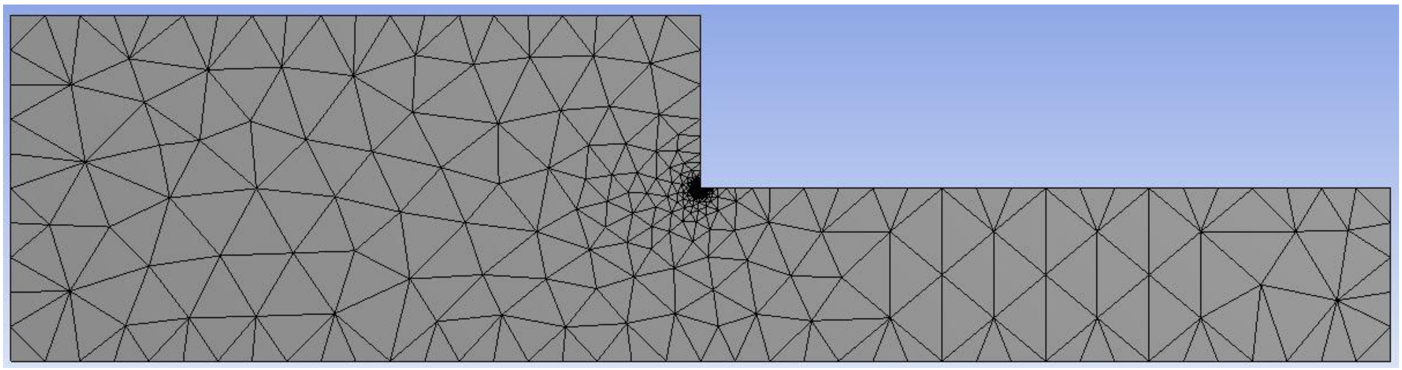
Default element size



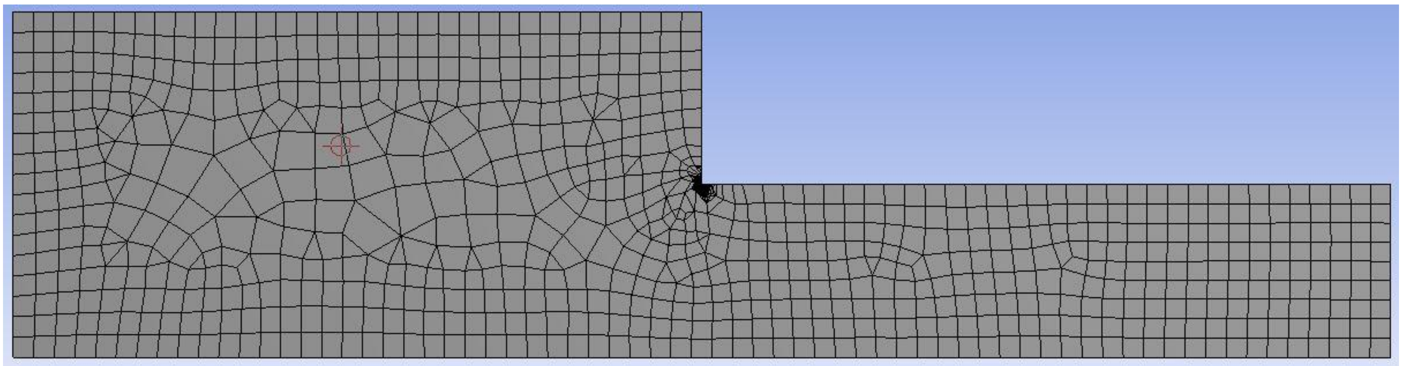
Element size = 10mm



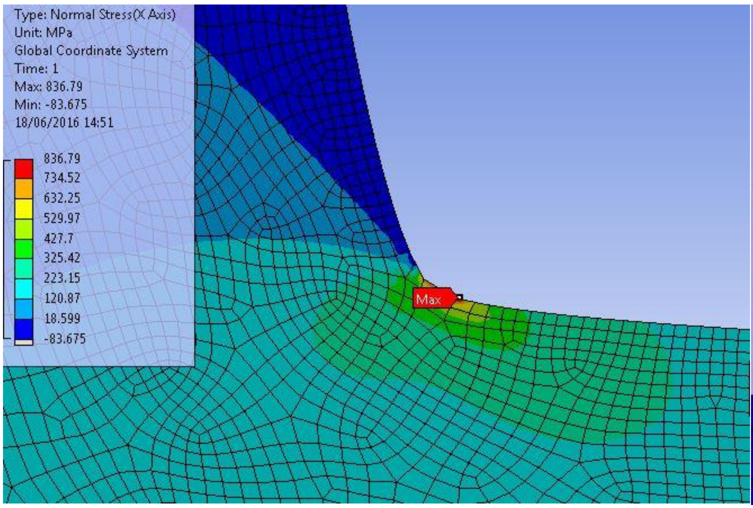
Element size = 5 mm

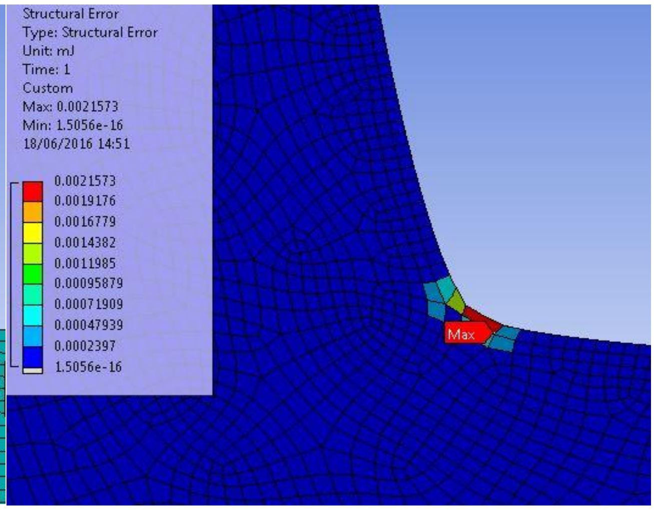


Element size = 3 mm (ANSYS converted to quadrilaterals by itself)

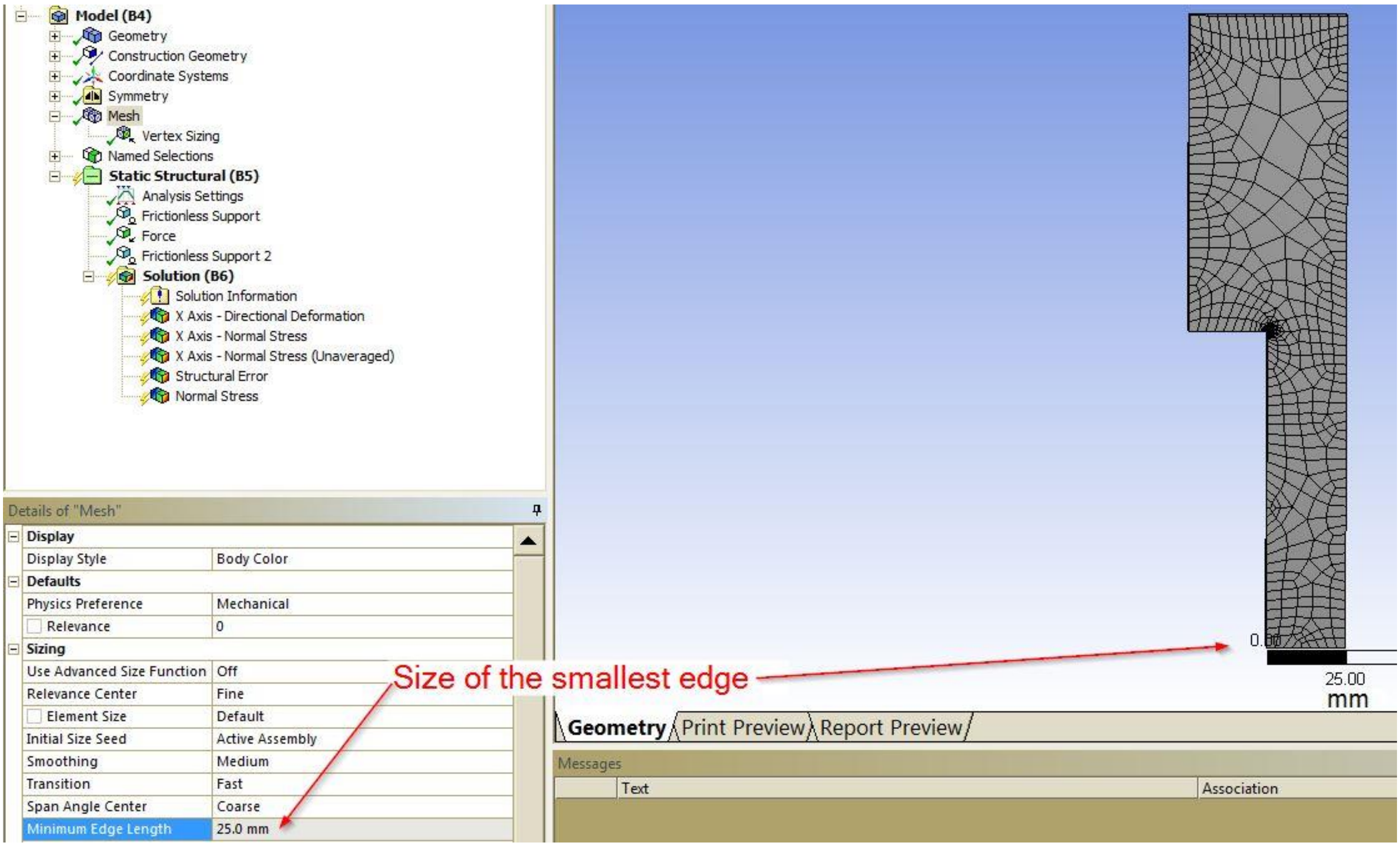


Anyway, singularity confirmed:

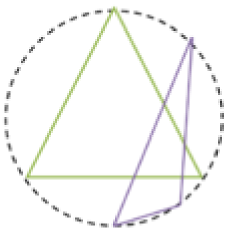




By the way,



Smoothness: From what I understood, ANSYS will try to reduce skewness by moving the nodes High means less skewed.



Skewness

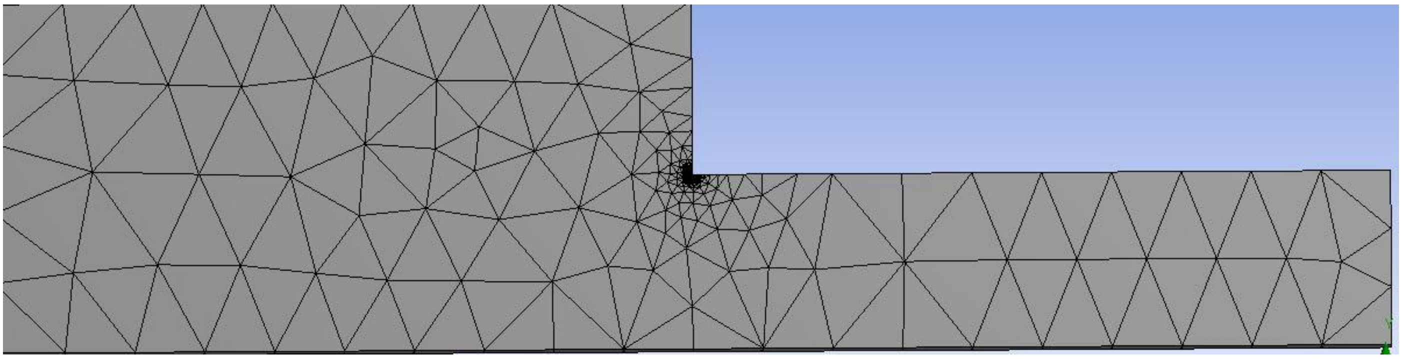
Skewness, a measure of mesh quality, can be calculated for each element according to its geometry. A detailed definition can be found in the on-line documentation[1]. For now, all you need to know is that it is a value ranging from 0 to 1, the smaller the better, and, as a guideline, elements of skewness of more than 0.95 are considered unacceptable.

Details of "Mesh"

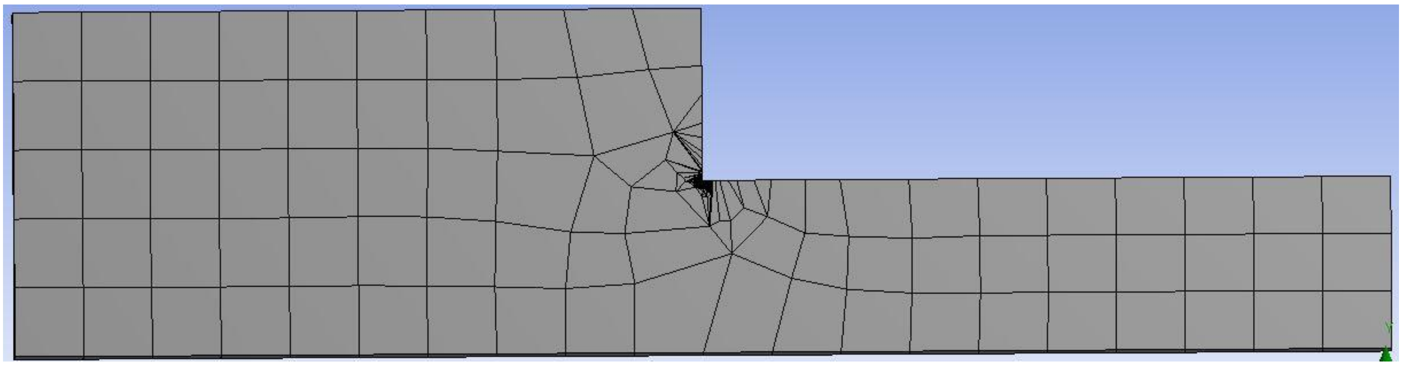
Display	
Display Style	Body Color
Defaults	
Physics Preference	Mechanical
Relevance	35
Sizing	
Inflation	
Patch Conforming Options	
Patch Independent Options	
Advanced	
Defeaturing	
Statistics	
Nodes	9786
Elements	5276
Mesh Metric	Skewness
Min	4.3459e-003
Max	0.94002
Average	0.40537
Standard Devi...	0.18784

Transition: Fast means that there may be large jumps in element size. Slow means opposite, that is, the transition from big to small elements is smooth (with medium sized elements in between).

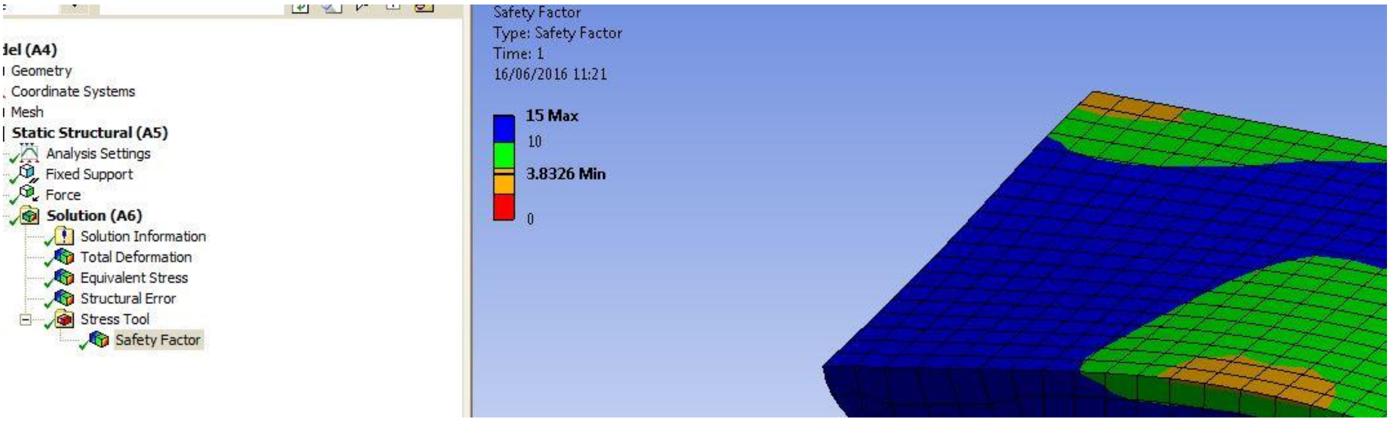
Slow transition

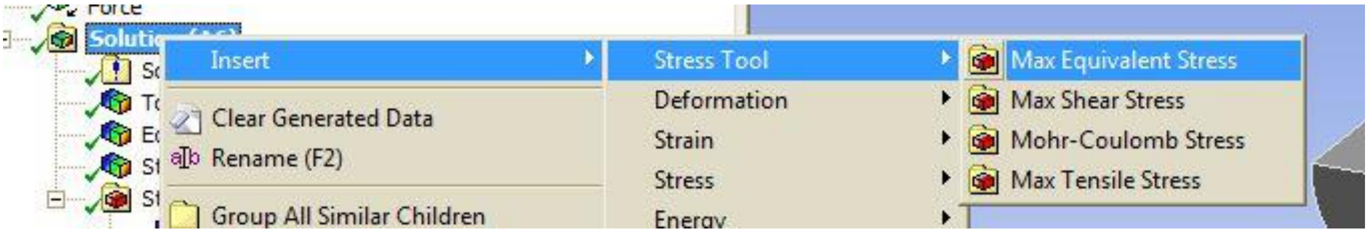


Fast transition



Adding Safety Factor Analysis





$$N = \frac{\sigma_y}{\sigma_{eq}}$$

More About Mesh Controls

We mentioned in 5.1-12[2] that hexahedral elements are generally more desirable than other shapes, such as tetrahedral. Similarly, in 2D cases, quadrilateral elements are more desirable than triangular. The main reason is that hexahedral (or quadrilateral in 2D) has better convergence behavior. That implies: (a) with the same problem size, hexahedral or quadrilateral gives more accurate results, and (b) it needs fewer iterations in a nonlinear simulation. Besides, mesh quality (5.3-1[2]) is also important though. A mesh of hexahedral elements with poor mesh quality is usually less desirable than tetrahedral with good mesh quality.

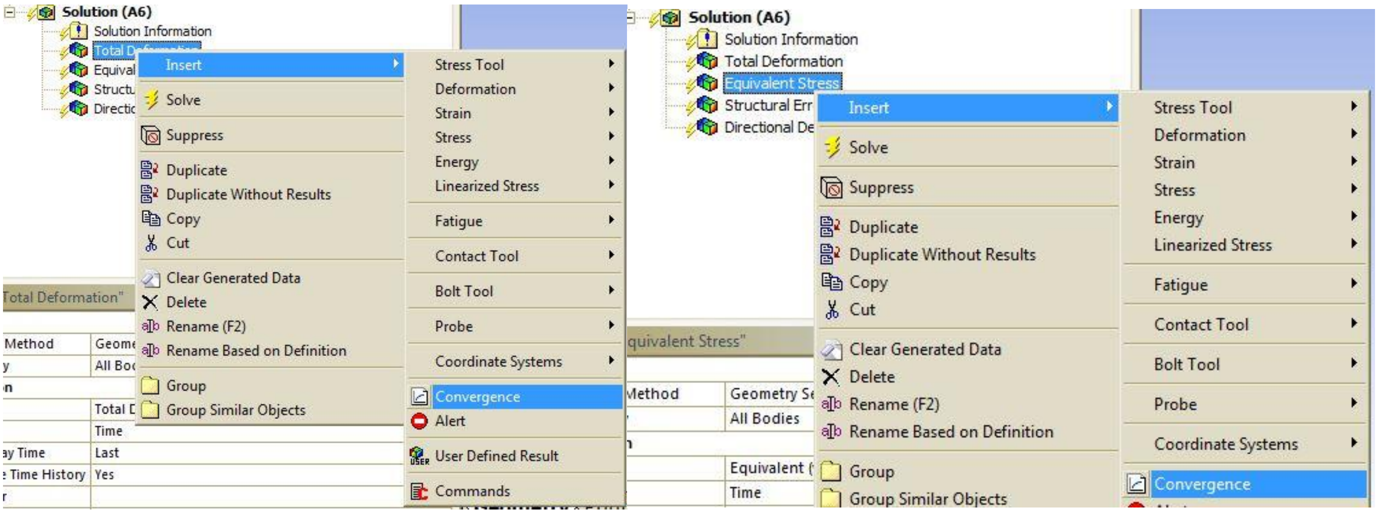
For 3D models, meshing is much more challenging. In Section 5.2, the model (the cylinder cover) is so complicated that the Workbench choose to mesh with all-tetrahedral.

A simple idea of creating hexahedral elements is to mesh a face (or faces) of a body with quadrilaterals and then "sweep" along a path up to the other end faces of the body. The starting faces are called a *source* faces and the ending faces are *target* faces. The source or target faces can be either manually or automatically selected. Not all bodies are sweepable. In Section 5.1, the body (the bracket) is not sweepable without further decomposition.

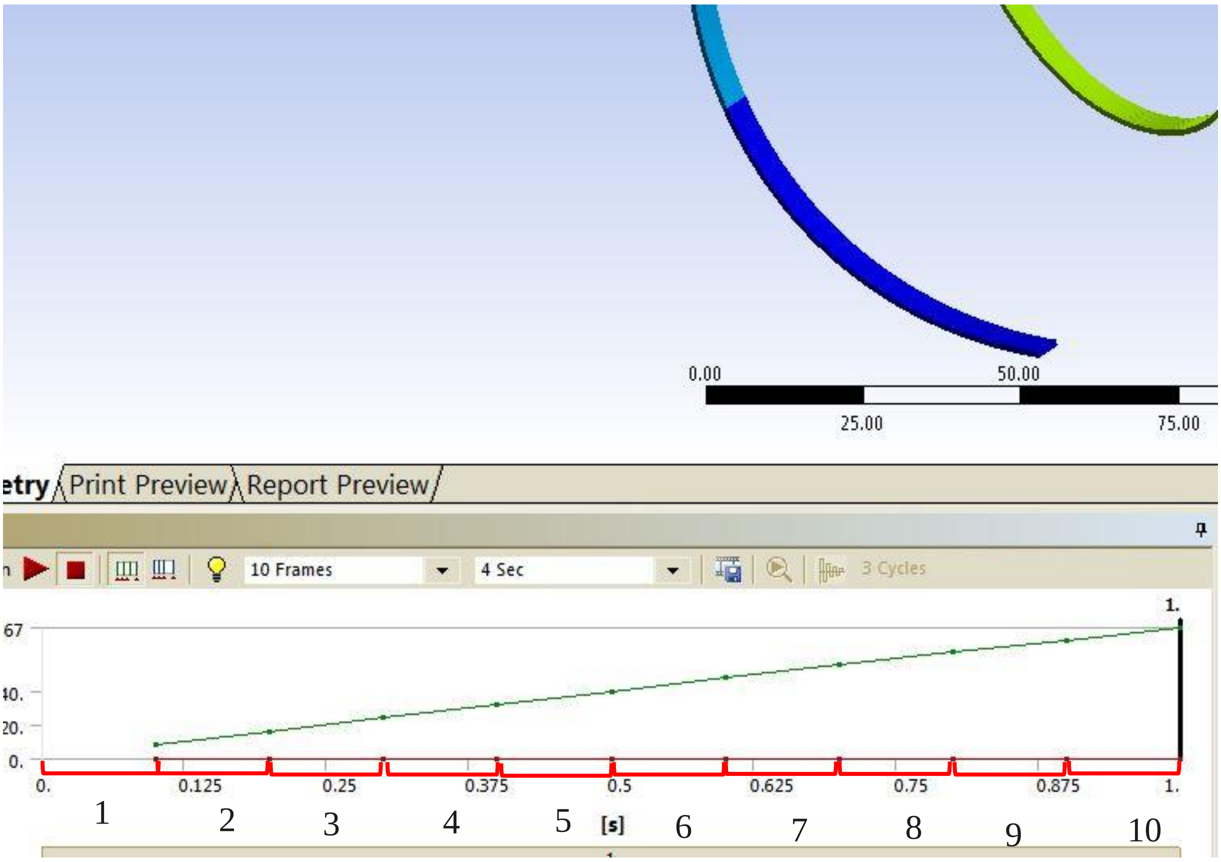
The idea of <MultiZone> method is to decompose a non-sweepable body into several sweepable bodies, and then apply <Sweep> method on each of bodies. This is what we had done in Section 5.1-12, where we inserted a <MultiZone> method and manually selected the source faces. According to these information, Workbench decomposed the body into several sweepable bodies and easily "sweep" each body with hexahedral elements. The result is an all-hexa mesh.

We had demonstrated the procedure of finite element convergence study in Sections 3.5-8 through 3.5-10. Theoretically, given an error, say 5%, we may refine mesh until the accuracy reaches this level. Performing these tasks manually is cumbersome and inexact. Workbench provides a tool to automate the mesh refinement until a user-specified level of accuracy is reached. This idea is termed *adaptive meshing*. Internally, Workbench uses the structural errors (Section 3.5-7) to help adjust the mesh, that is, it refines the mesh size in the area of large structural errors.

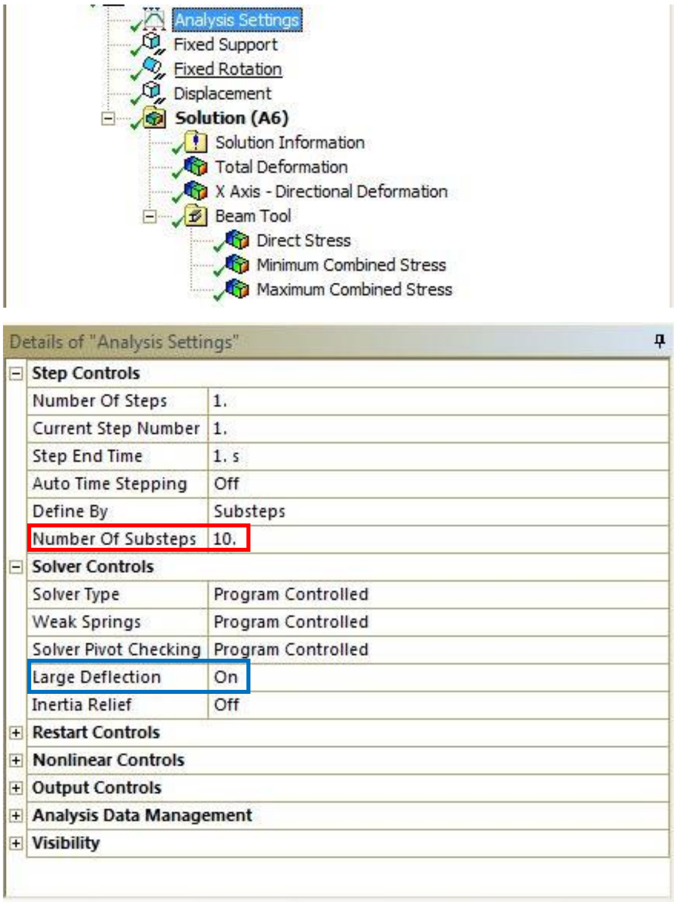
To use this tool, right-click a results object and select <Insert/Convergence> [1]. In the details view, specify the accuracy, or <Allowable Change> [2]. Also, highlight <Solution> in the project tree and, in the details view, specify the maximum number of mesh refinement loops. When solving the model, Workbench will iterate to refine the mesh until the difference between two iterations is less than the <Allowable Change> or the <Max Refinement Loops> reaches.



The Meaning of Substeps

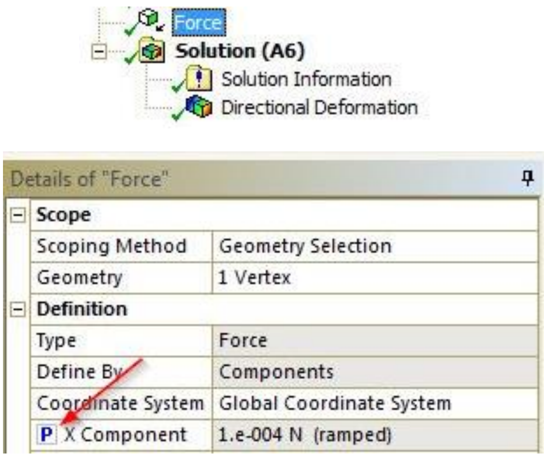


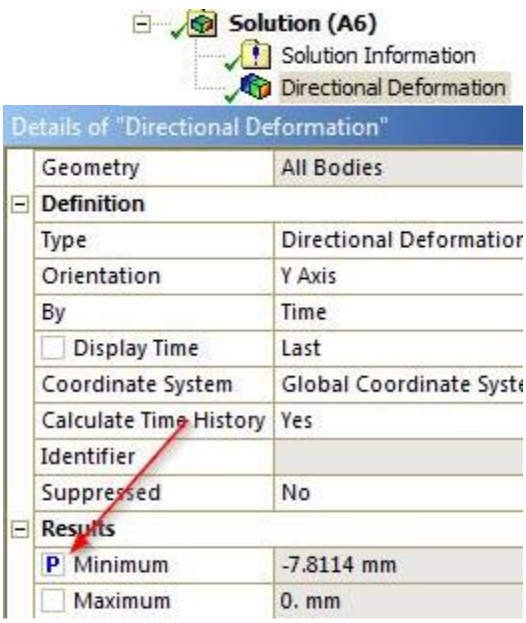
We had made the right hand move 50mm upwards (y-direction). So by using 10 substeps, that means each substep is 5mm of input vertical (y-direction) displacement.



Automatic Change of Variables

Let’s say we want to study how a certain variable, e.g. *displacement*, varies with another one, e.g. applied force.

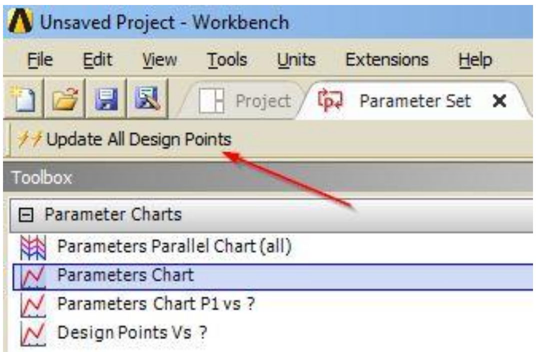




Put the values of the force to apply

Table of Design Points

	A	B
1	Name	P1 - Force X Component
2	Units	N
3	DP 0 (Current)	0.0001
4	DP 1	100
5	DP 2	200
6	DP 3	300
7	DP 4	400
8	DP 5	500
9	DP 6	1000
10	DP 7	-100
11	DP 8	-200
12	DP 9	-300
13	DP 10	-400



Displacement values for those x values will be calculated

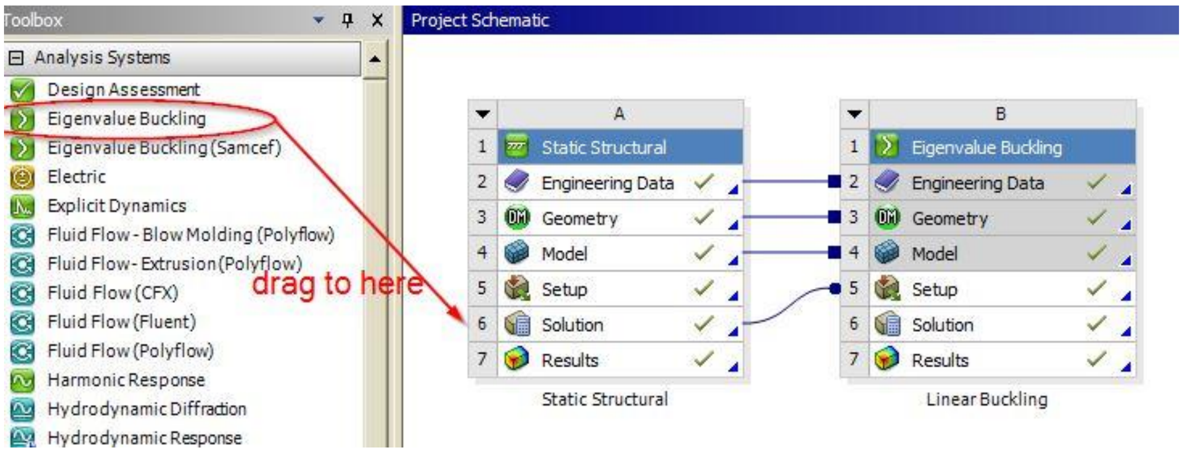
Table of Design Points

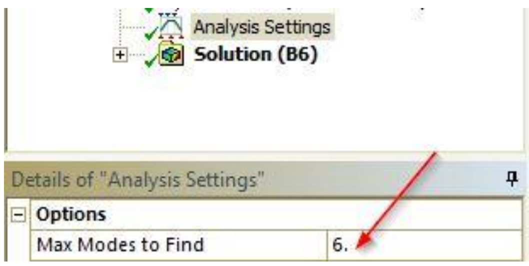
	A	B	C
1	Name	P1 - Force X Component	P2 - Directional Deformation Minimum
2	Units	N	mm
3	DP 0 (Current)	0.0001	-7.8114
4	DP 1	100	-7.3626
5	DP 2	200	-6.9624
6	DP 3	300	-6.6034
7	DP 4	400	-6.2796
8	DP 5	500	-5.9863
9	DP 6	1000	-4.8551
10	DP 7	-100	-8.3184
11	DP 8	-200	-8.8953
12	DP 9	-300	-9.5571
13	DP 10	-400	-10.321

These values can then be exported to excel to be plotted.

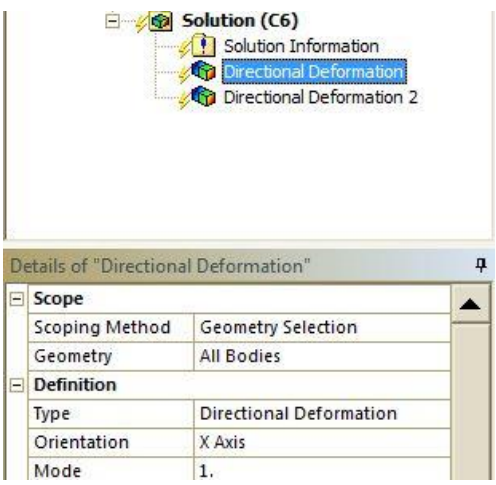
Buckling

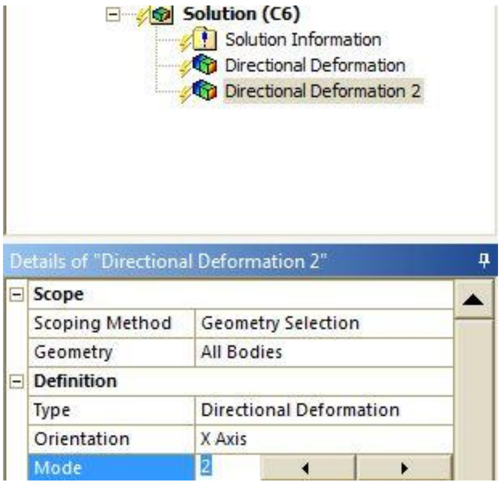
The stress of a structure may be below what the material can withstand but there could still be problems. One of those is buckling which tends to happen in slender structures under compressive stress. So after we study the stress of a slender structure it's good practice to analyze whether buckling can occur whenever there's compressive stress.



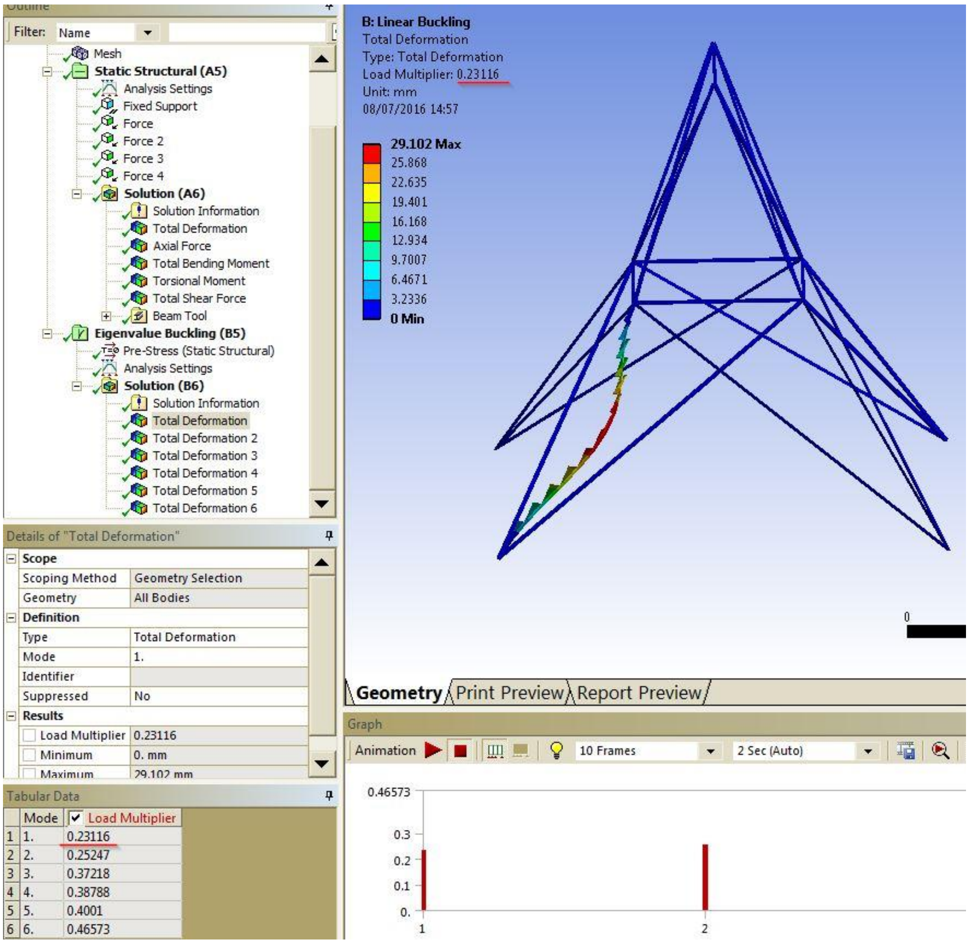


6 or less, depending on the case.





X axis or another, or even total deformation.

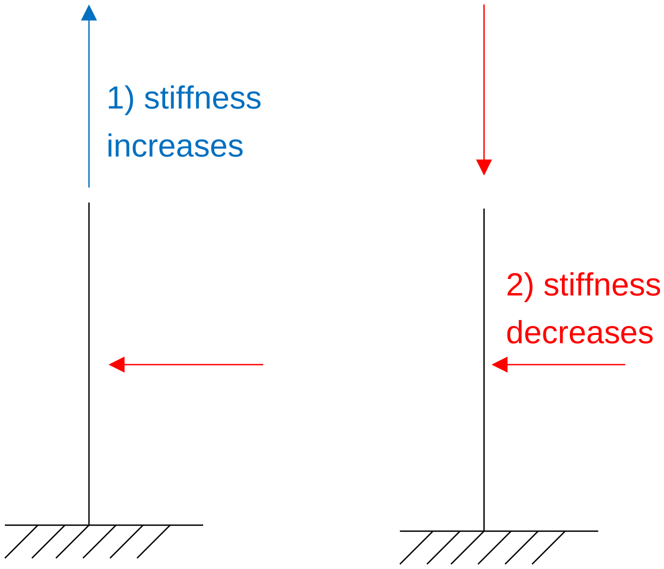


The first buckling mode occurs for 23% of the design load meaning the structure is not safe, and for that load, buckling will occur.

Also remember two important effects:

1) The stress stiffening effect —axial tension increases bending stiffness;

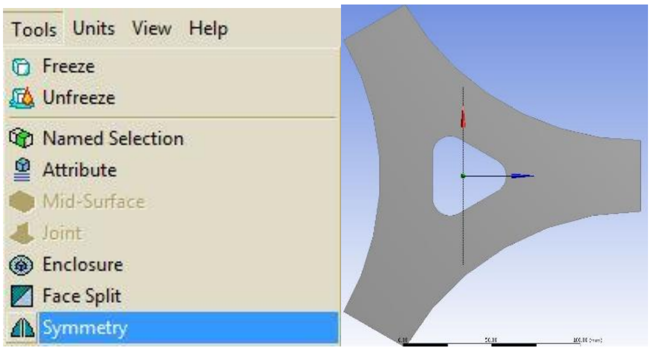
2) Lateral forces may change buckling load.

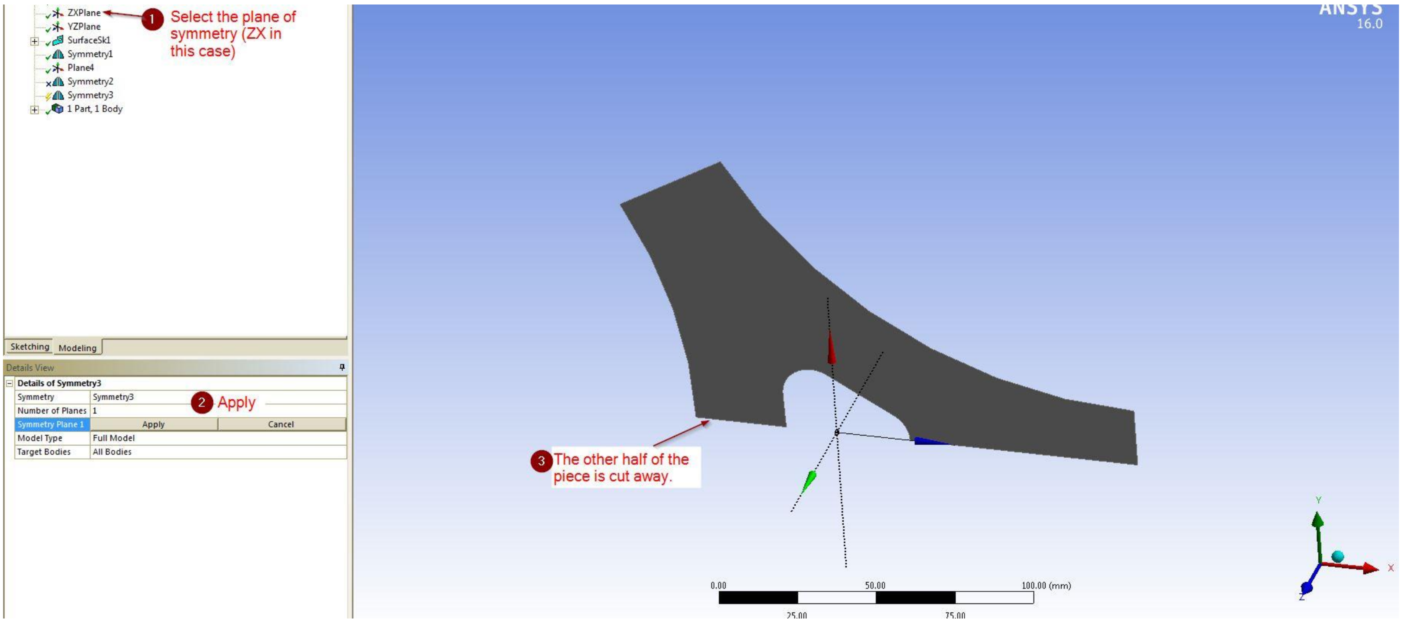


Other Stuff

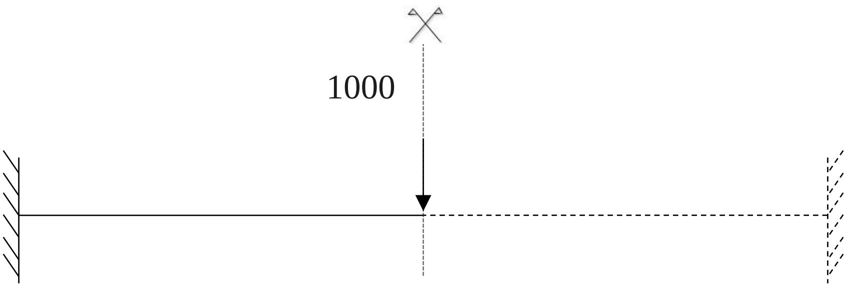
How to Take Advantage of the Symmetry of your Model

Method #1 — Symmetry tool in Design Modeler





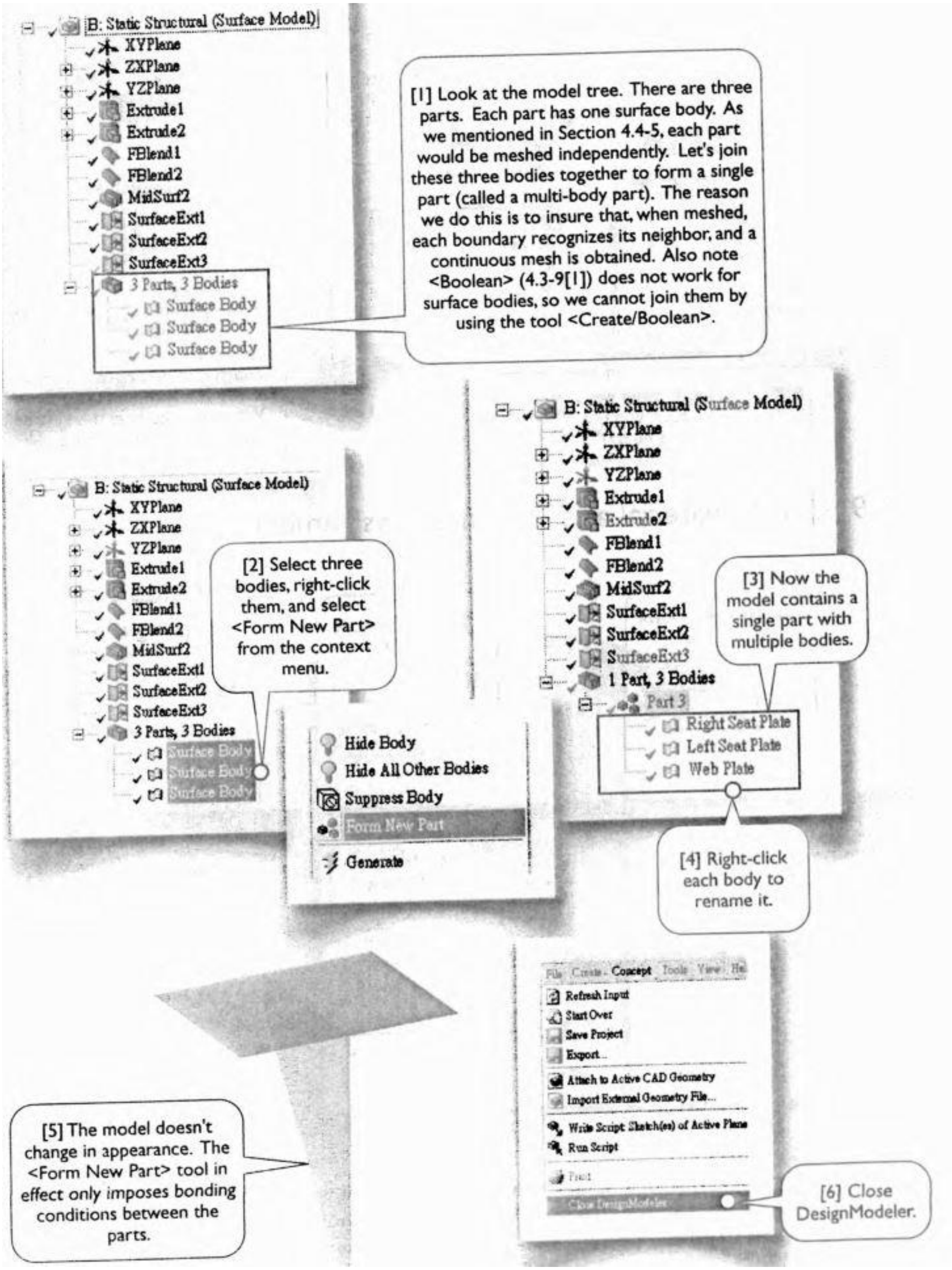
When you use the symmetry in this way (Tools > Symmetry) then everything else becomes symmetric, including forces and boundary conditions that you will apply. But if you apply a force in the plane of symmetry, then ANSYS won't double it.



Note: I think we can't use symmetry this way in line models.

Method #2 — Symmetry through appropriate boundary conditions

Merging Bodies to form a Single Part



Edge Joints[2]

In Section 6.2-7, we formed a single part by grouping three surface bodies to ensure the meshing continuity. Another way is using the <Edge Joints> feature. Edge joints are the glue that holds together bodies where a continuous mesh is desired. Edge joints can be created manually by pull-down-selecting <Tools/Joint>, and can be viewed by turning on <View/Edge Joints>. Explore this feature yourself.

Working With Things from SolidWorks

Note: to save some headaches, make sure the axis and views in SolidWorks are positioned correctly before saving, as ANSYS will read them like you saved on SolidWorks.

Importing a sketch from SolidWorks to ANSYS

1. Save the Solidworks sketch as .dwg or .dxf
2. Import it to the Design ANSYS
3. Put “Yes” on “Line Bodies”
4. Generate

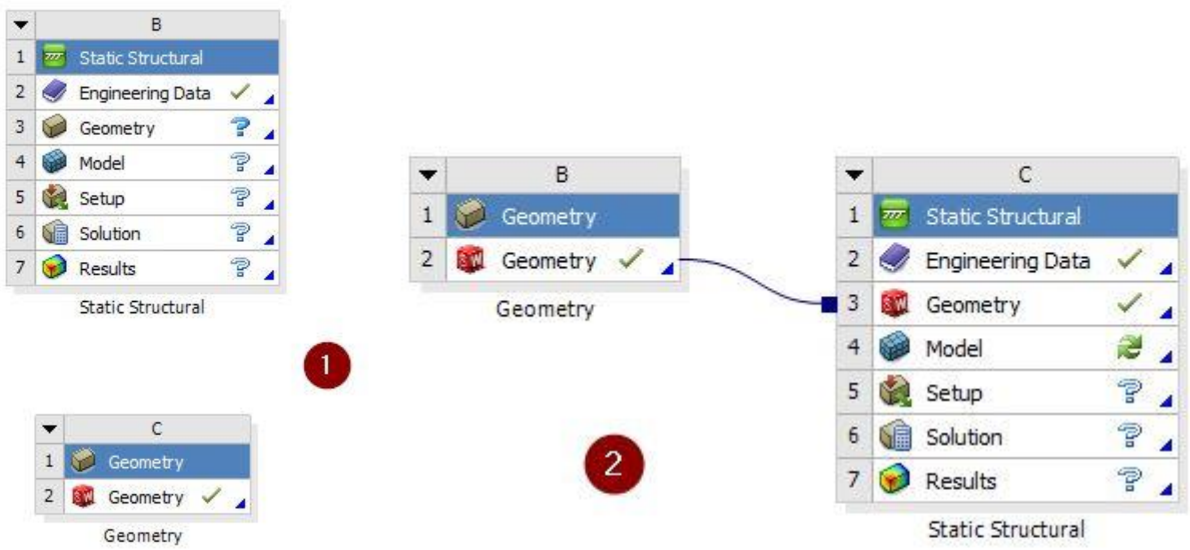
Then if you want to edit it you need to convert the body into an ANSYS-usable sketch, and for that you project the lines into a plane resulting in an editable sketch.

Importing a 3D Part Directly from SolidWorks

Method #1 (With this method you can only edit the part in Solidworks)

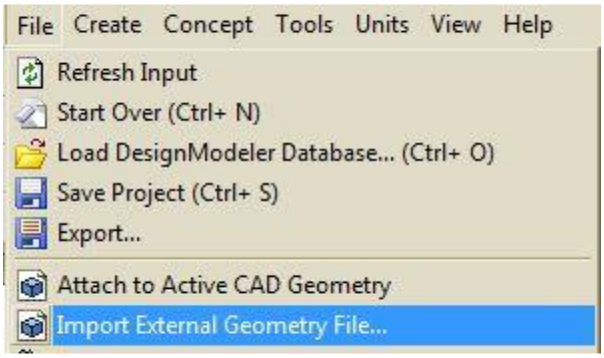
Create project (Static Structural in this case) then drag SolidWorks part file into it (1);

Drag the Geometry with Solidworks icon to the Geometry of the Static Structural (2).



Method #2 (Importing part into DM)

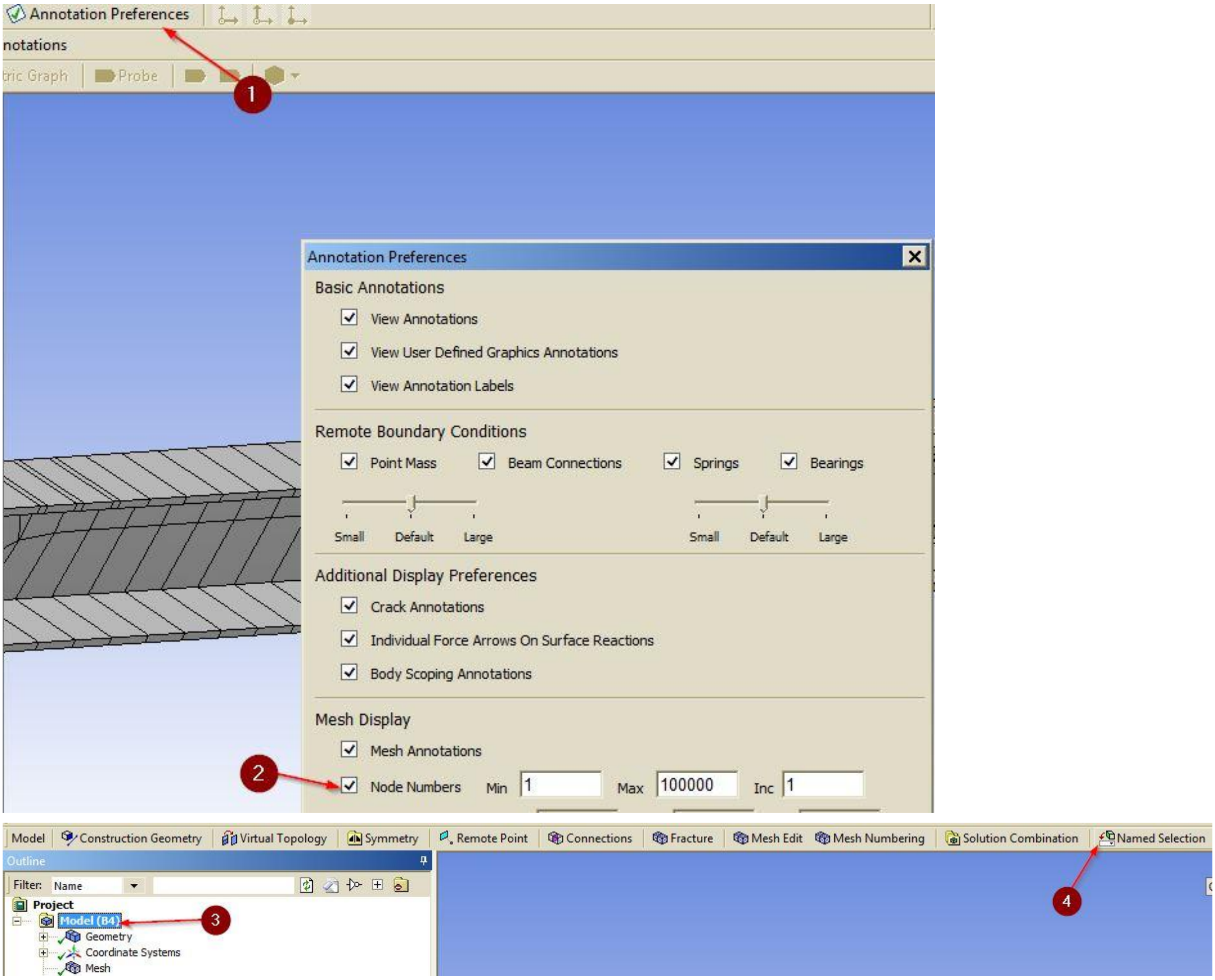
Go to DM

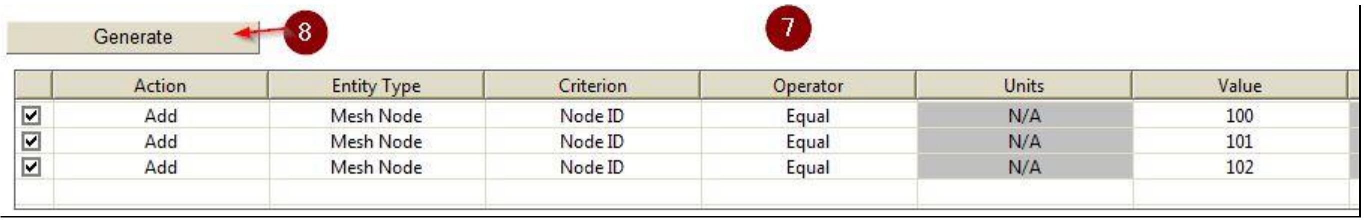


Creating a Surface from a 3D Part from Solidworks

1) Import the Part using Method #2
2) Proceed as normal

Selecting Individual Nodes by Number





	Action	Entity Type	Criterion	Operator	Units	Value
☑	Add	Mesh Node	Node ID	Equal	N/A	100
☑	Add	Mesh Node	Node ID	Equal	N/A	101
☑	Add	Mesh Node	Node ID	Equal	N/A	102