

Mesh Connections Meshing Best practice



Fluid Dynamics

Structural Mechanics

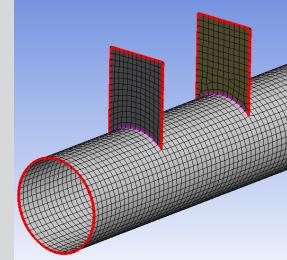
Electromagnetics

Systems and Multiphysics

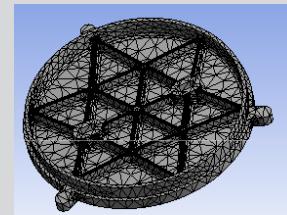
**Richard Mitchell
Joe Luxmoore
Mark Leddin**

Aim for this session

Mesh connections



Meshing tools



Best practice



ANSYS 14.0 Shell Mesh Connections



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

Mark Leddin

Agenda

Overview

Terminology

New in 14.0

Selective Meshing

Mesh Size and Topology

Visualisation

Demo

Overview



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

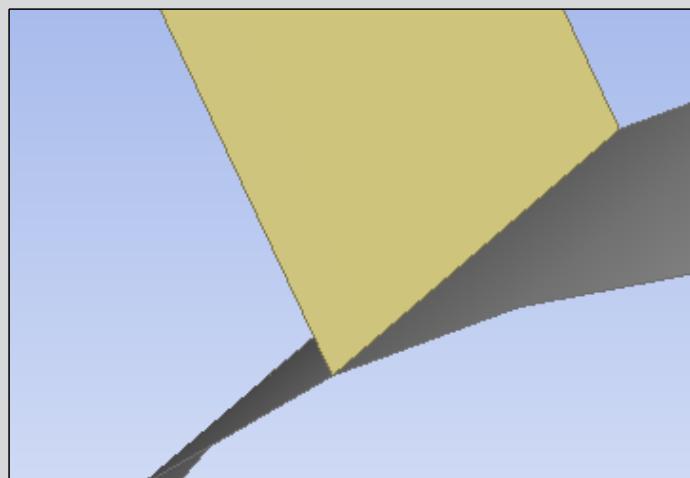
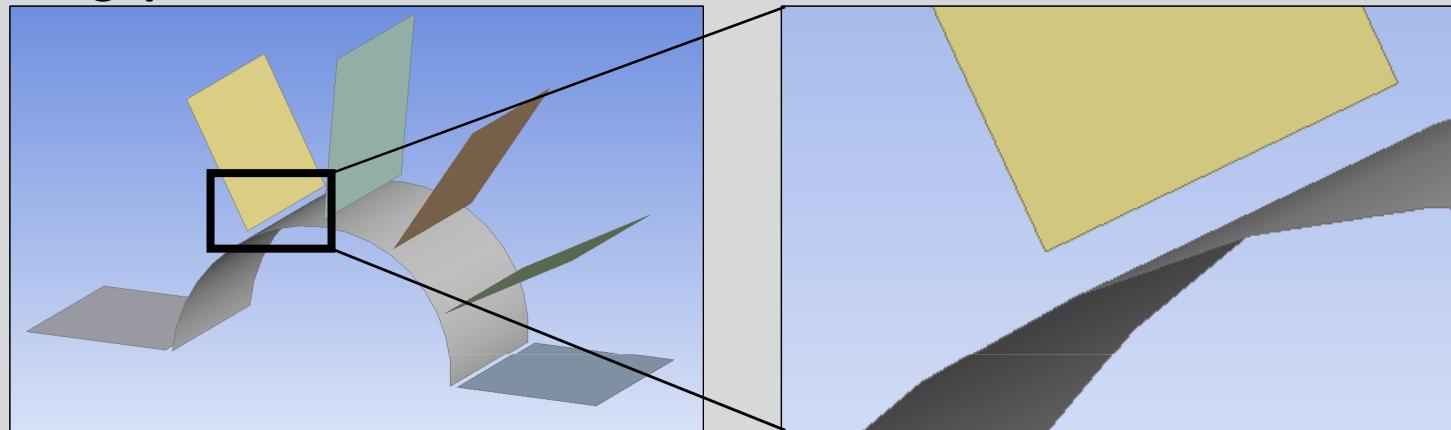
Mesh Connections Overview

Forming connections between sheet bodies with conformal mesh is a challenging task:

- In the past this has been done at the geometry level, however geometry tolerances are tighter and often lead to problems in getting conformal mesh.
- With mesh connections, the connections are made at the mesh level and tolerance is based locally on mesh size.
- There are advantages to both approaches. This material focuses on creating connections at mesh level.

Example geometry

CAD with gaps in it



Desired geometry

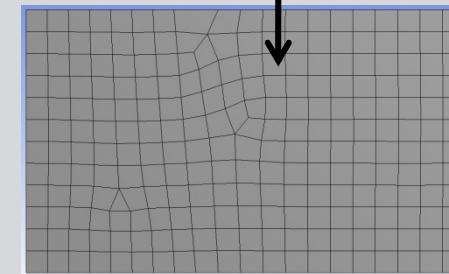
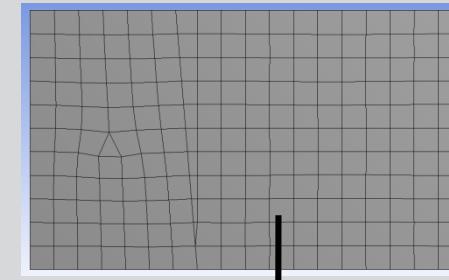
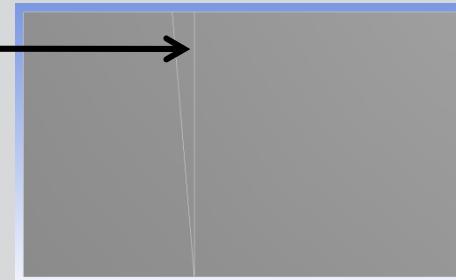
Options

- Fix CAD
- Join model at meshing stage

Mesh connections and pinch controls

Pinch controls focus on mesh based defeaturing where user may need to decide whether to keep/remove sections:

- Pinching sliver edges
- Pinching sliver sections



The software can find the problems
but user has ability to review/edit these areas:

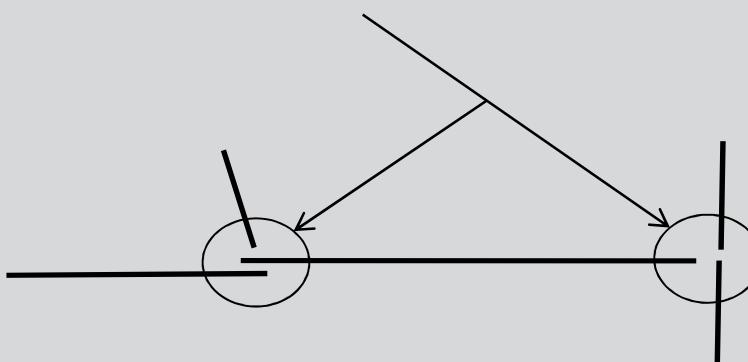
- Can adjust scoped objects
- Can adjust tolerance to limit pinching, etc.

Mesh connections and pinch controls

Mesh connections employ pinch controls behind scenes:

- Pinch is used within a body
- Mesh connections are used across bodies

In both cases, gaps, slivers, overlaps, etc. are resolved at the mesh level



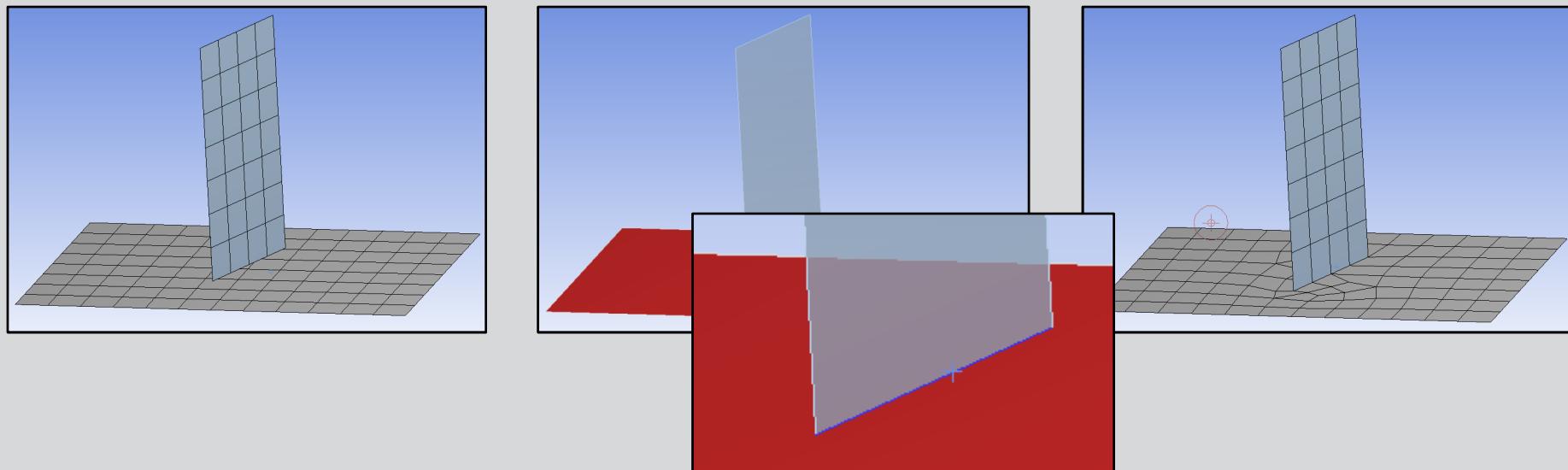
- This presentation will not discuss pinch controls for mesh based defeaturig within a body although many situations are the same as those in mesh connections.

Types of pinch controls/mesh connections

Edge to Edge: Connecting an edge of one face to an edge(s) of another face to pinch out mesh in-between based on tolerance.



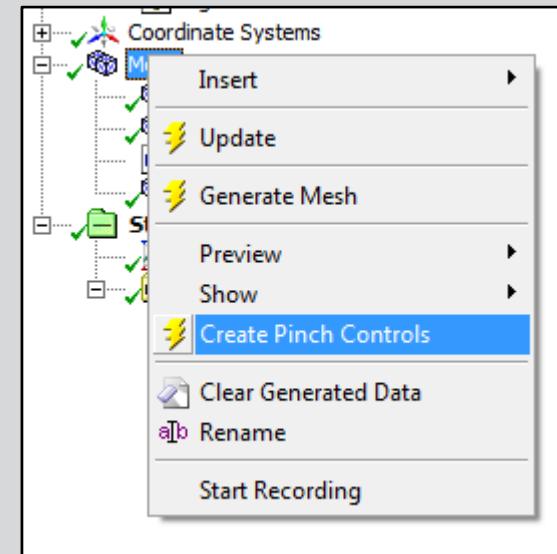
Edge to Face: Connecting an edge of one face to another face to pinch out the gap and give conformal mesh between the edge and face.



Automatic creation of mesh connections, pinch controls

There are automated detection tools to find/create mesh connections/pinch controls.

- This uses same logic as other connections:
 - Edge to edge detection
 - Edge to face detection



Terminology



Fluid Dynamics

Structural Mechanics

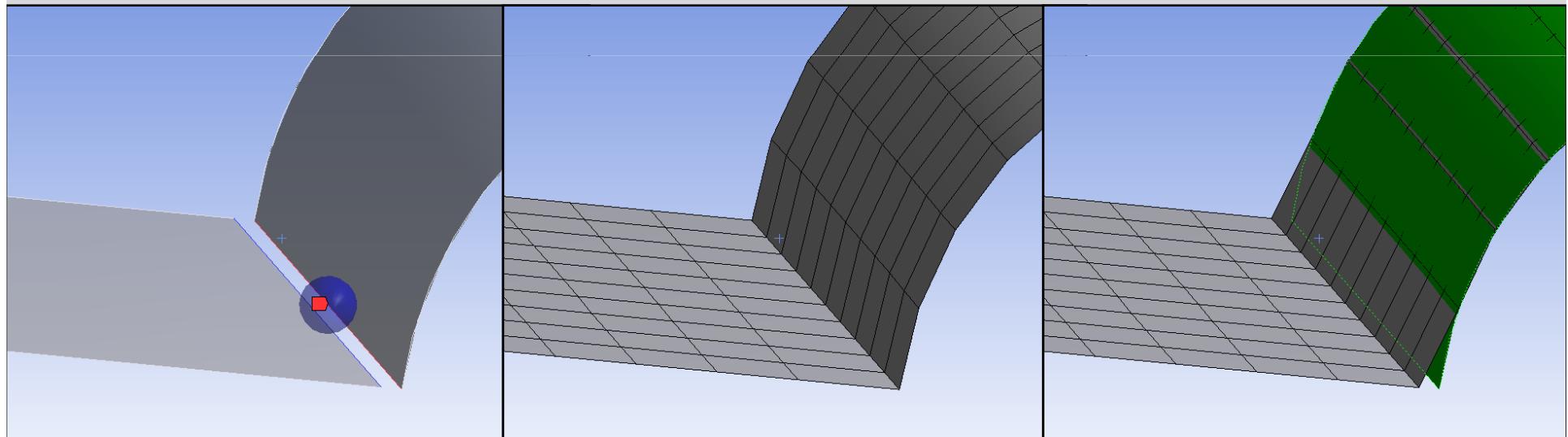
Electromagnetics

Systems and Multiphysics

Master vs. Slave

Master indicates the topology that will be captured after the operation is over.

Slave indicates the topology that will be pinched out during the operation



Pre vs. Post MCs/pinch

Pre vs. Post: Names are in relation to the Meshing operation.

- Pre indicates that the boundary mesh will be pinched prior to the face meshing.
- Post indicates that the mesh will be pinched after mesh creation (and local remeshing occurs to clean up the neighboring mesh).

In 13.0, mesh connections used the pre pinch technique, in 14.0 all mesh connections now use the post pinch technique. The pinch control allows the option for edge to edge connections, but only post pinching for edge to face.

Upon reading a 13.0 database in 14.0 MCs & pinch controls are updated accordingly.

- If the mesh is cleared and regenerated in 14.0, the new mesh will probably be different!

New in 14.0



Fluid Dynamics

Structural Mechanics

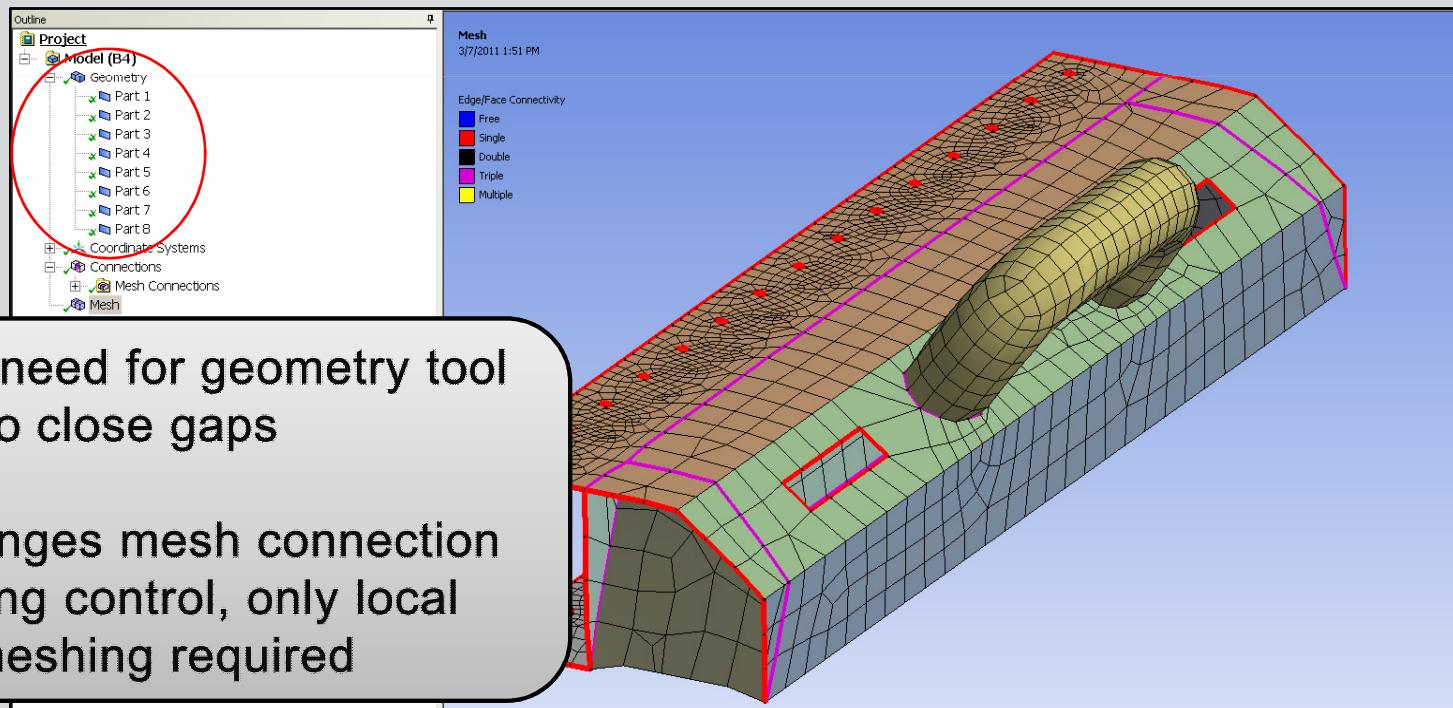
Electromagnetics

Systems and Multiphysics

Mesh Connections

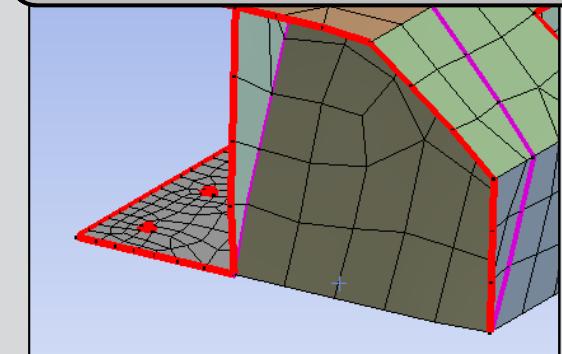
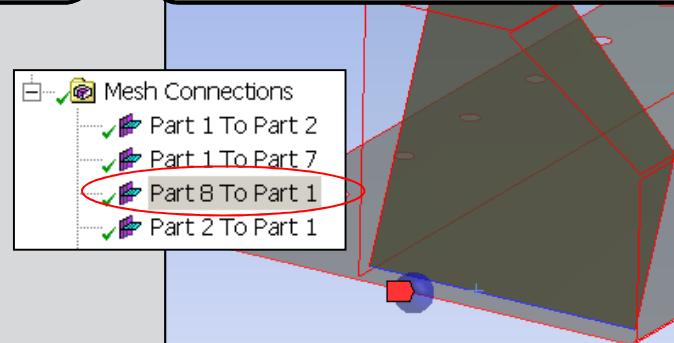
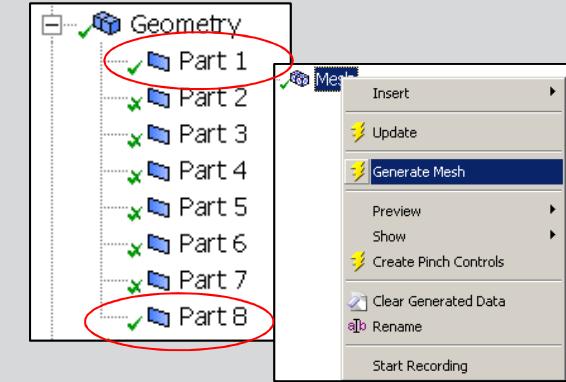
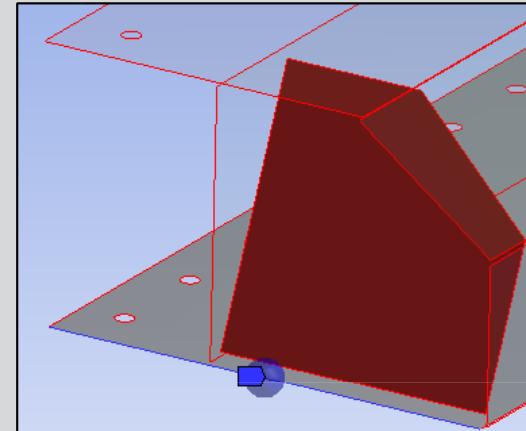
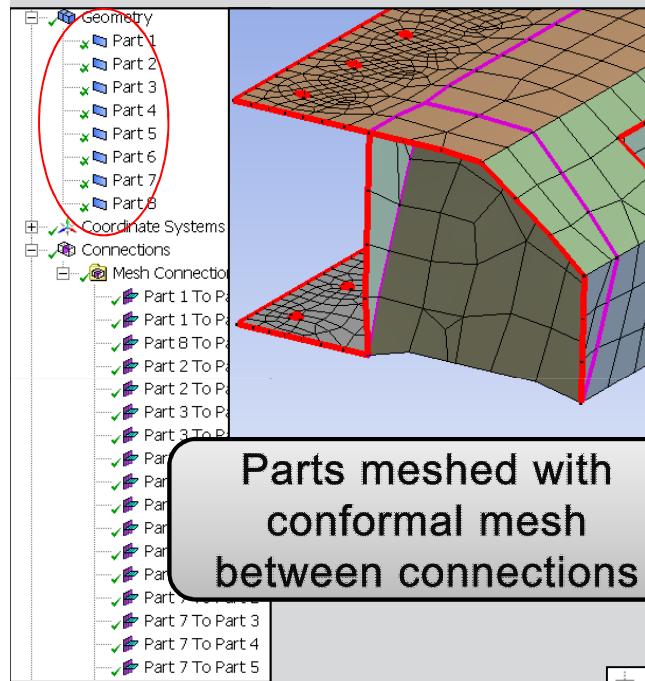
Mesh connections now work at part level:

- Post mesh connections by default (pre option still exists – used by default on multibody parts)
- Base part mesh stored to allow for quick changes



Mesh Connections

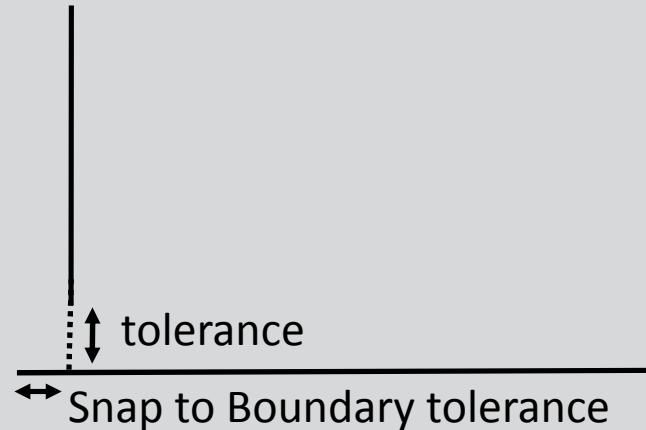
Local connection changes or mesh changes only affect local area:



Improved tolerances:

**Separated projection tolerance
and snap tolerance**

Tolerance Type	Slider
Tolerance Slider	0.
Tolerance Value	138.88 mm



Provides more robustness for bigger gaps

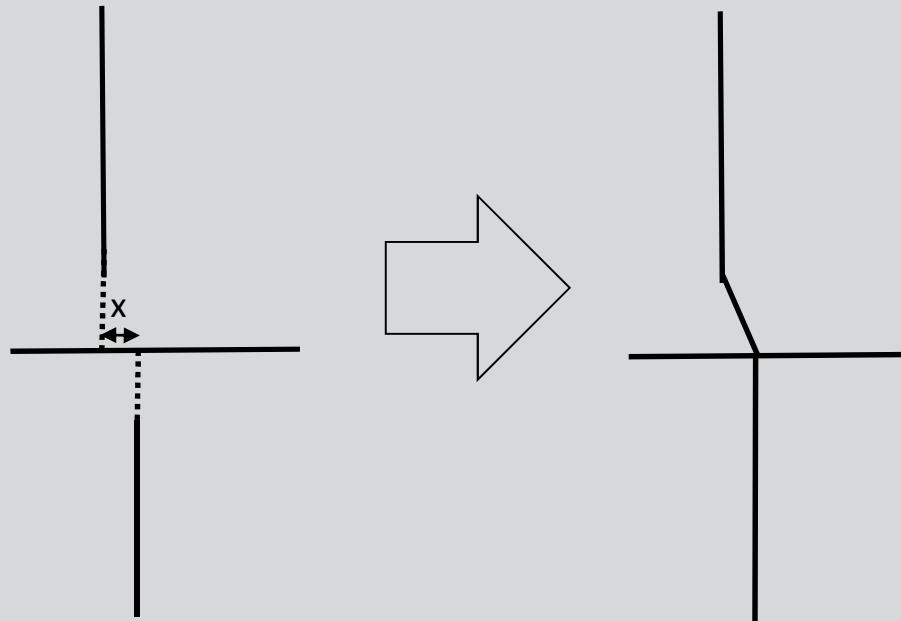
Snap to Boundary	Yes
Snap Type	Manual Tolerance
Snap Tolerance	Default

Snap to Boundary	Yes
Snap Type	Element Size Factor ▾
Master Element Size Factor	Default

Support for common imprints:

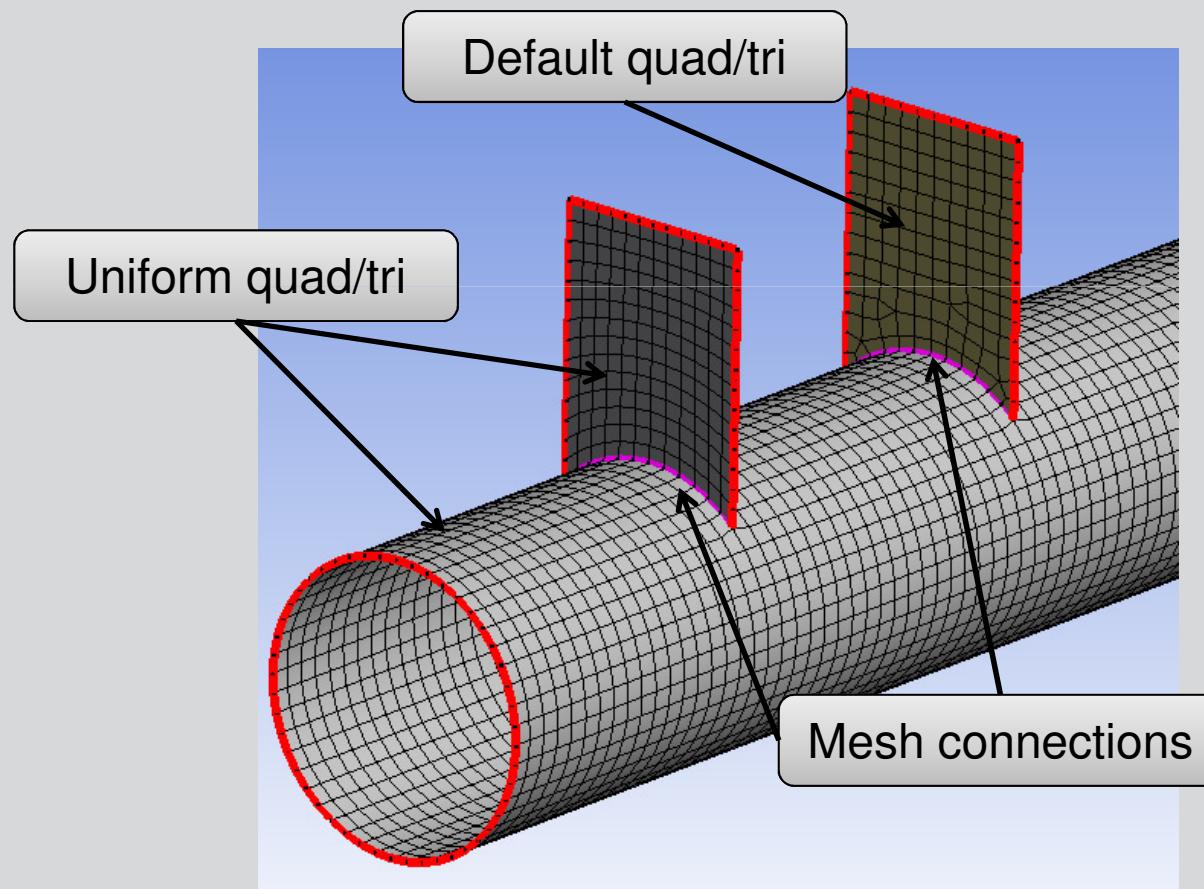
Now common imprints are shared, similar to snap to boundary

Common imprints will be made if gap between imprints (x) is smaller than or equal to minimum element size in connection region



Support for default and/or uniform quad/tri

Can mix and match mesh methods with mesh connections, and/or use selective meshing:



Selective Meshing with Mesh Connections



Fluid Dynamics

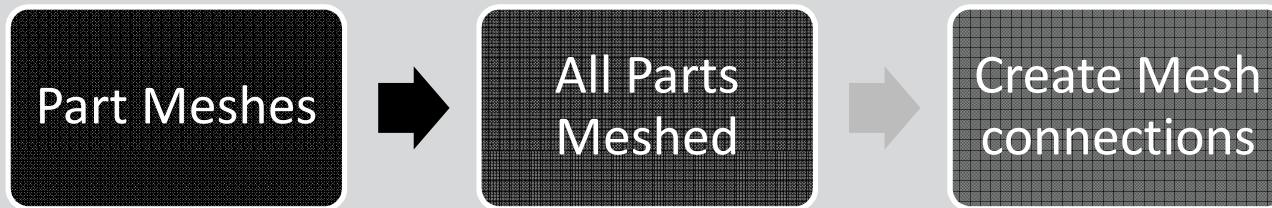
Structural Mechanics

Electromagnetics

Systems and Multiphysics

Selective Meshing with MCs

Selective body meshing works with Mesh connections as follows:



- For testing purposes if you want to just see the mesh connections for selected parts/bodies, suppress all other parts/bodies and generate mesh.

Selective Meshing: Worksheet view

Only lists body meshing steps

Mesh connection occur after all meshing steps are complete. No line item for mesh connections.

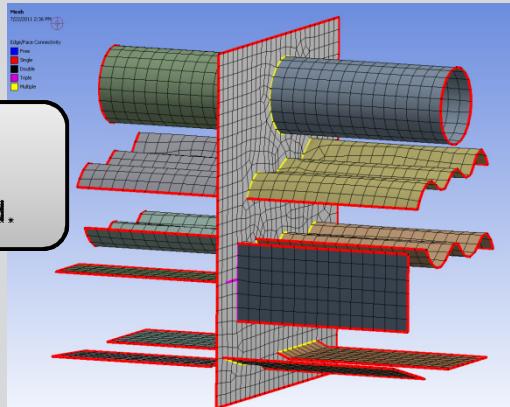
Selective Meshing: Base part mesh

The software handles this by keeping the base part mesh in memory in addition to the connected mesh.

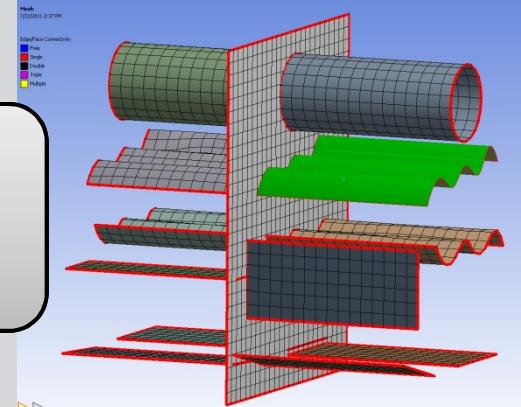
If you clear the generated data of 1 part, it will clear the part mesh for that part as well as all mesh connections.

- In other words, all the connected mesh is cleared and the base mesh of all other parts besides the cleared part is returned.

Take this meshed example, with connections formed.



Now clear the mesh on selected part (green). Notice that base mesh is returned for all other parts.



On a "Generate mesh" the software would create the base part mesh for green part and re-connect MCs.

Effect of mesh size and topology on mesh connections



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

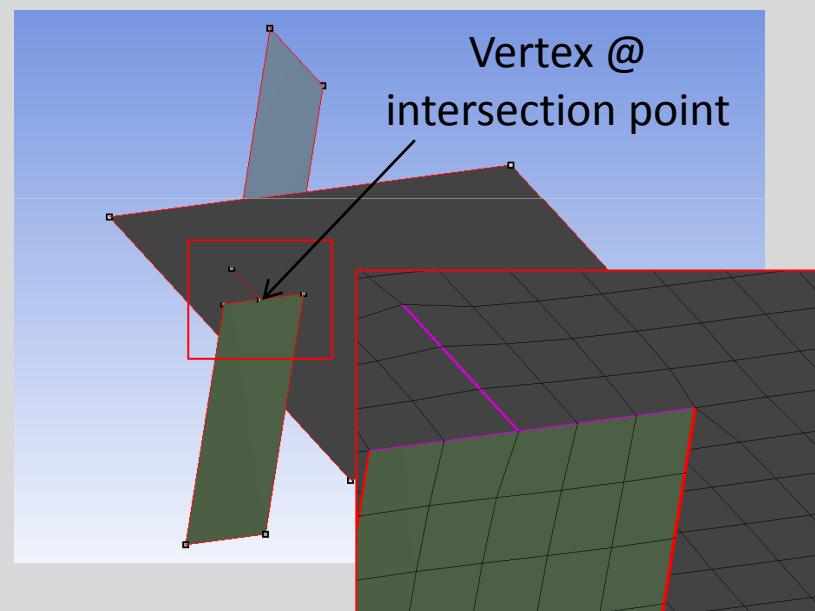
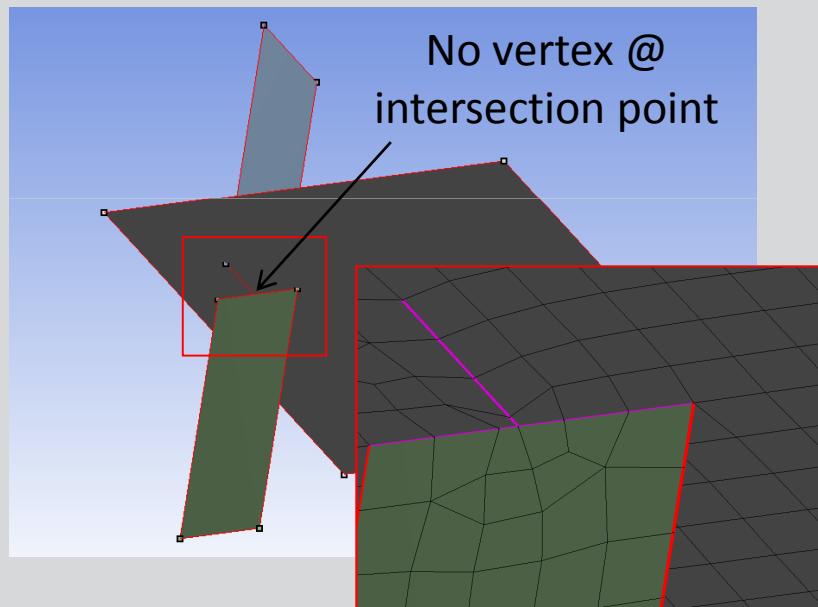
Mesh sizing w/MCs

As described under tolerances section, the mesh size has an effect on the quality and feature capturing of a MC:

- Mesh size always affects base mesh as features are only captured relative to mesh size.
- During mesh connection, the base mesh is adjusted according to the common imprint/location. In cases where there is a large projection or a large difference in mesh sizes between master and slave, the common edge between bodies can get ragged. This can often be fixed by:
 - Using more similar sizes between source and target
 - Improving the tolerance used in the mesh connection (either for projection, or for snap to boundary)
 - Adjusting the geometry's topology so that the base mesh is more accommodating for the mesh connection.

Mesh topology w/MCs

As described in selective meshing, the base part mesh is generated prior to making mesh connections. Controlling the base mesh can improve the quality of the mesh connections, for example:



Because the base mesh aligns to edge splits, having a vertex @ intersection regions, etc. can lead to better quality mesh.

Visualizing mesh connections



Fluid Dynamics

Structural Mechanics

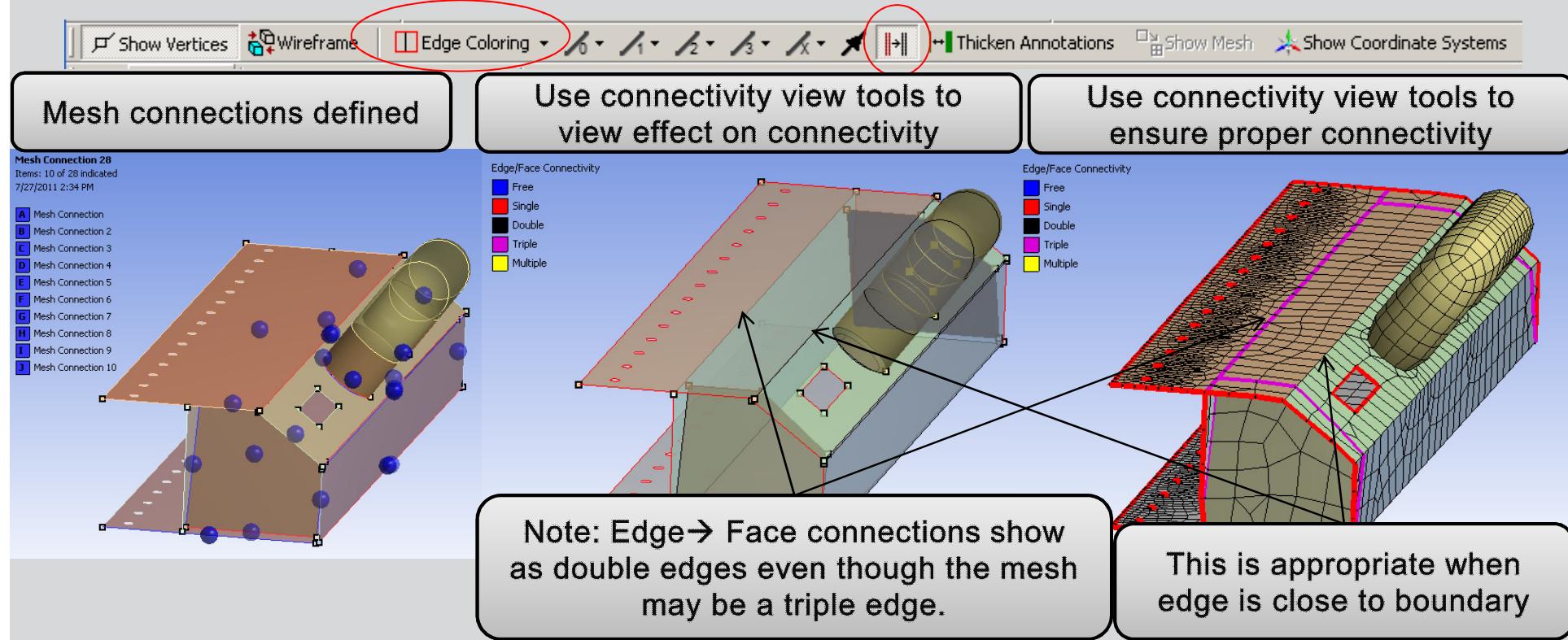
Electromagnetics

Systems and Multiphysics

Viewing Connectivity

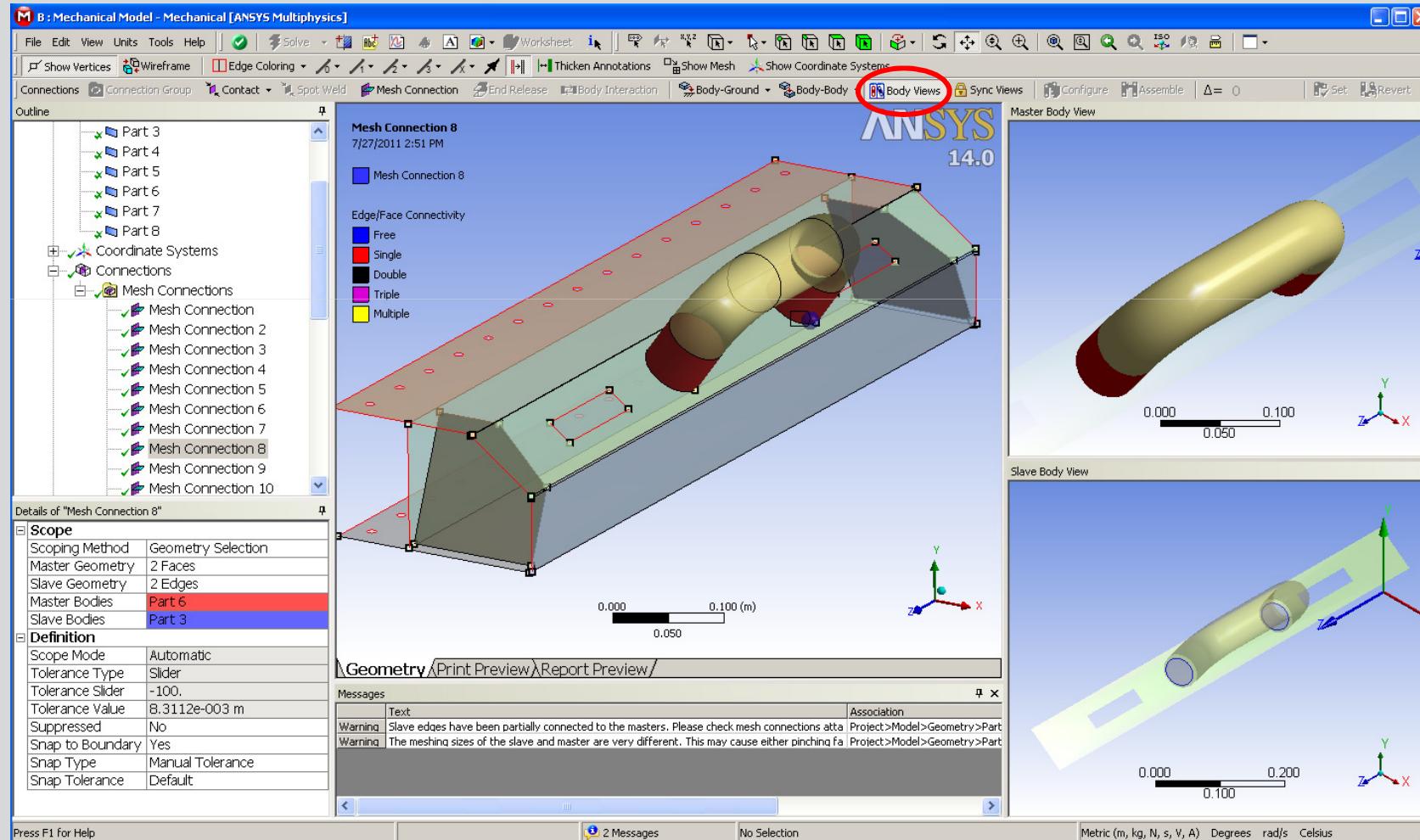
Viewing the connectivity of models is useful to diagnose the current state of connections, but geometry connections are viewed a bit differently than mesh connections.

Viewing the final mesh is best way to visualize the connectivity and to check that the mesh accurately represents the connections, welds, etc.



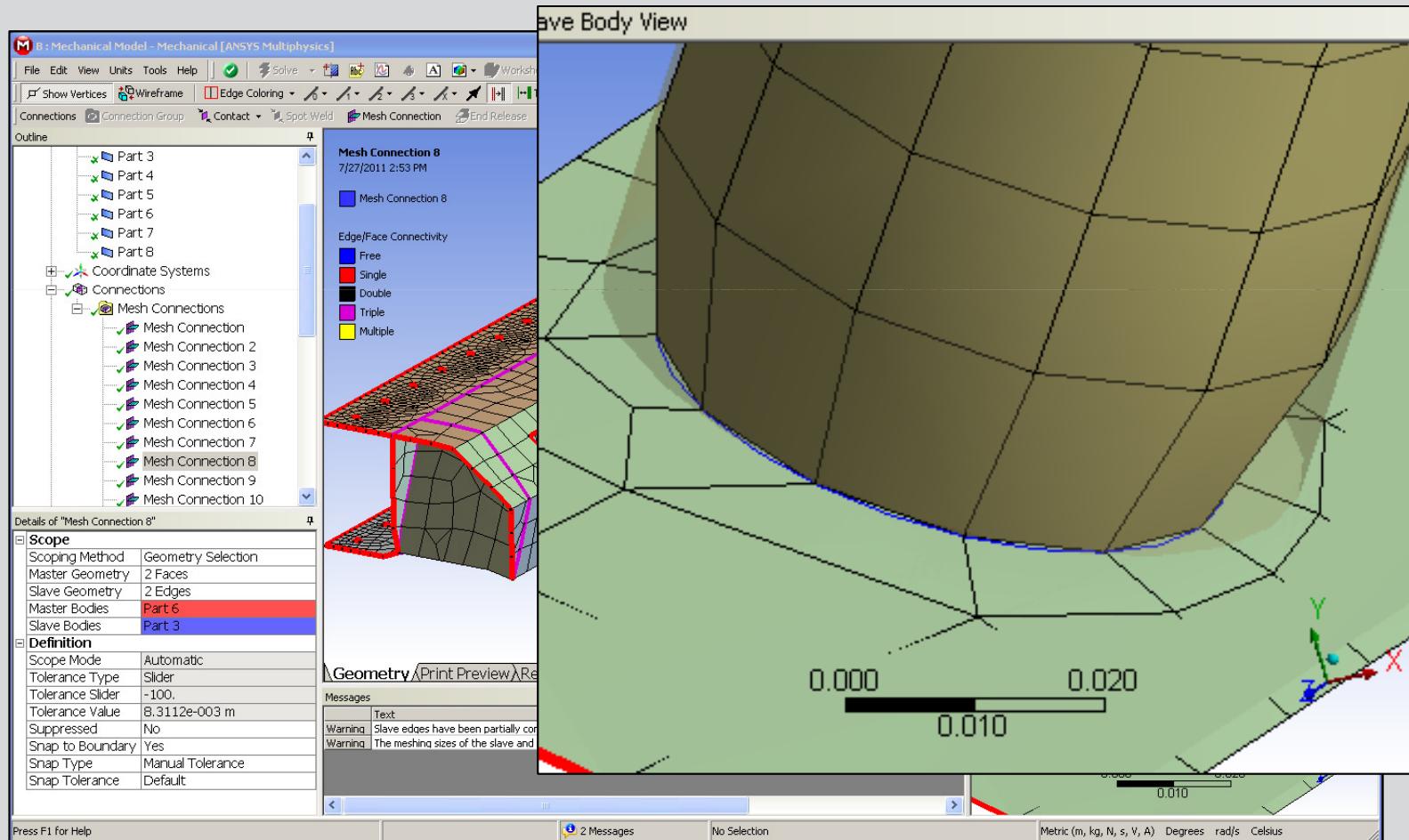
Mesh Connections: Body Views

Use Body Views option to see specific details of the mesh connection



Mesh Connections: Body Views

Use “Show Mesh” option with Body Views for closer inspection of mesh connection.



Diagnosing MCs

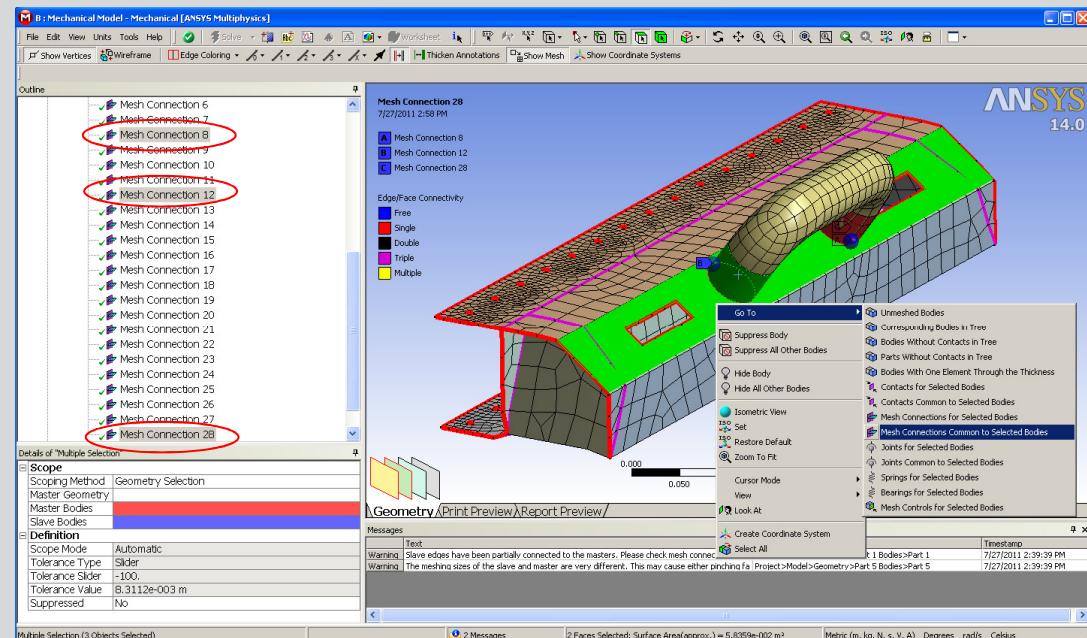
If a mesh connection fails, use the problematic geometry and RMB: Go To->Mesh connections for selected bodies.

This will highlight all mesh connections that are attached to the problematic geometry.

Review the tolerances, and mesh sizes associated with the connection.

You can also use Go To->
Mesh connections common
to selected bodies to see
all mesh connections
common to those bodies.

- **This is helpful to find spurious MCs and duplicates can be removed.**



Tolerances



Fluid Dynamics

Structural Mechanics

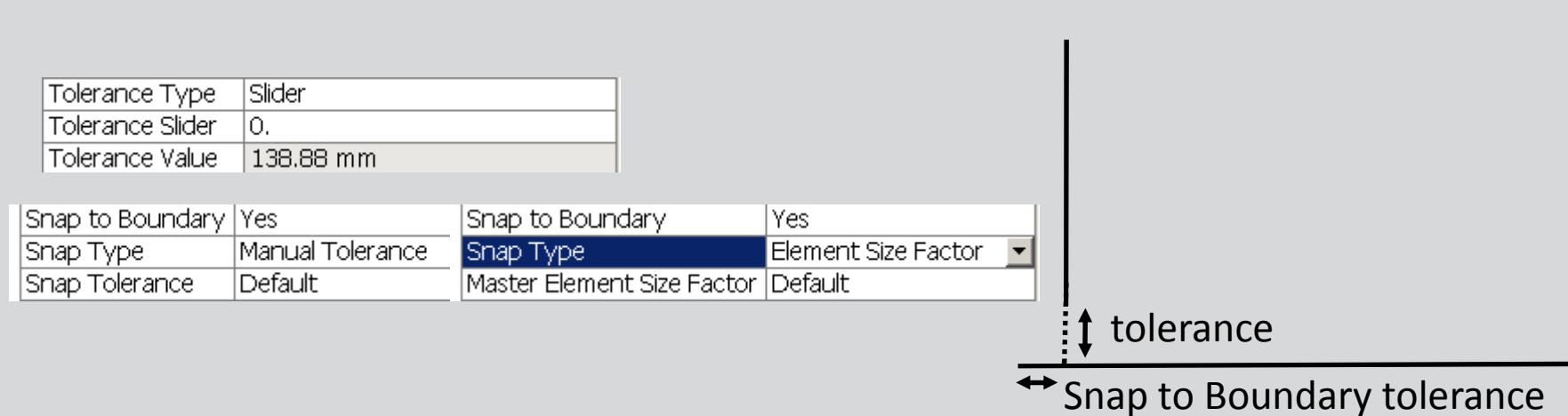
Electromagnetics

Systems and Multiphysics

There are several tolerances used in mesh connections, and setting appropriate tolerances is often critical to get high quality mesh that adequately represents the geometry you are interested in capturing.

There are 2 main types of tolerances:

- Tolerance to close gaps between bodies
- Tolerance to sew up mesh at the connection, in the interface, this tolerance is called the snap to boundary tolerance.



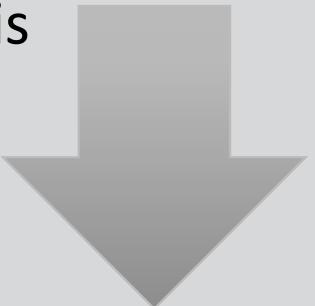
Tolerances: Balancing act

Similar to other connection options, the tolerance is used to find which bodies should be connected to which other bodies.



Setting this to a higher value will connect more bodies together, possibly incorrectly.

If it is lower it may miss connections, so there is often a motivation to set this to a larger value than needed.

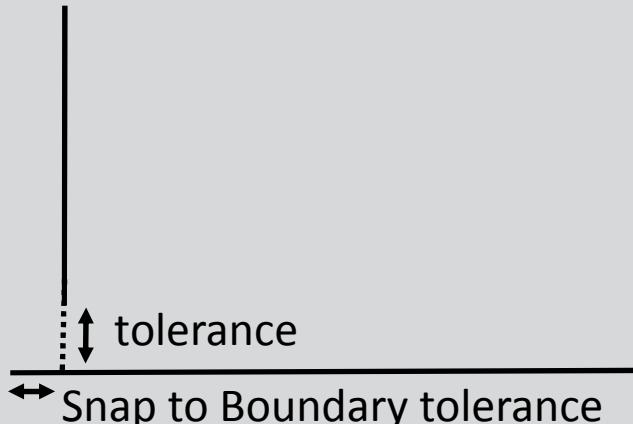


Tolerance: Large Values

For large assemblies where user doesn't want to define mesh connections manually, the automatic detection of mesh connections provides many benefits.

Setting a larger value to find these connections yields more connections and user feels more comfortable that the model is fully constrained.

- The larger values though can be problematic for the following reasons:
 - If more automatic MCs are created, more duplicates can be created and the mesher then decides which connections to create
 - Too many MCs can lead to Mesh Pockets (see later slides), Spurious mesh connections (later slides), and other such problems.
 - The snap to boundary tolerance defaults to the same value as the connection tolerance. If the tolerance is large for snap to boundary the mesher may be too aggressive in pinching out the mesh at the connection and the mesh quality and feature capturing can suffer.

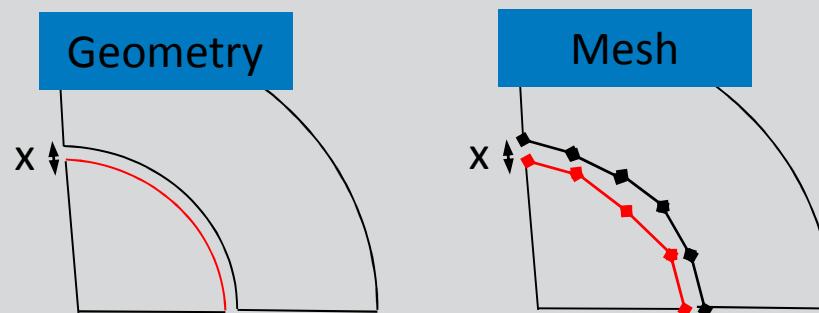


Tolerances: Small Values

So, setting large tolerances may cause problems, so what if I consider using a small tolerance?

Things to note:

- The mesh connection isn't made until meshing is carried out. When meshing, the connections are made on nodes and elements of the mesh, and the tolerances don't translate exactly.



- For example, in case above, user would want a slightly larger tolerance than the gap in the geometry. If the gap is defined as "x", and the tolerance is set as x , the Auto MC could find the connection, but the mesh could result in only partially connected mesh.

Recommendations:

As you can see setting the right tolerance can be very important.

The following are some tips and tricks to use:

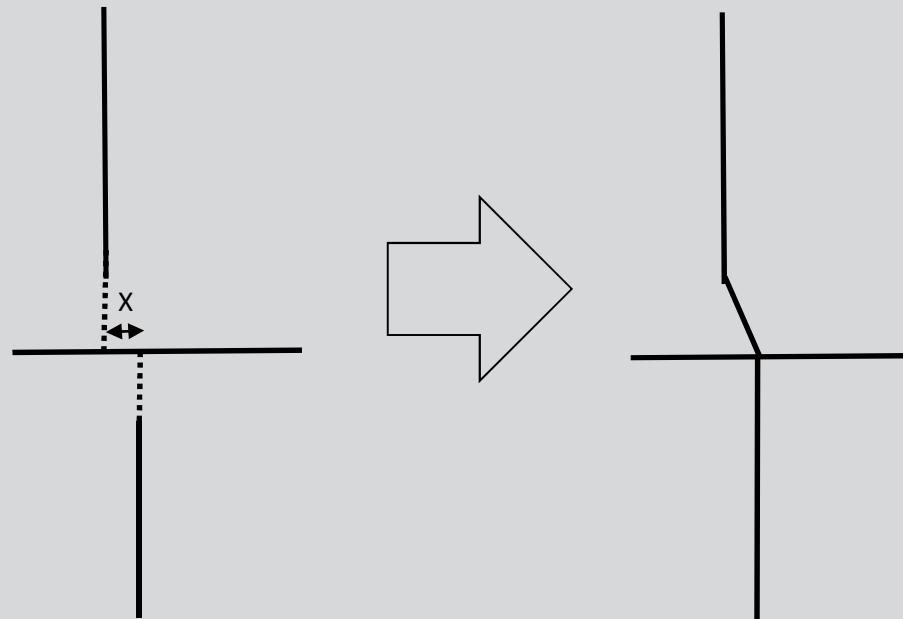
1. The tolerance used for Automatic Mesh Connections can be adjusted after the connections are found. It is sometimes a good idea to use one value to find the mesh connections, then select all mesh connections and reduce or increase the tolerance later.
2. The tolerance used for “Snap to boundary” and having this option on to begin with is not always a good thing. It depends on the model and what features a user wants to capture.
3. To set a good tolerance, you generally want to measure gaps in the geometry, and get a general impression on desired mesh size.
 1. Look at largest gaps, and smallest gaps between parts, ideally there is not a large discrepancy, if there is, you may need to use an approach to set different values for different regions.
 2. Also look at curvature and mesh size. Ideally you can use a global tolerance that is larger than the largest gap you need to resolve, yet smaller than the smallest mesh size (1/2 of smallest mesh size is often a good value)
 3. Look at overhanging sheets where “snap to boundary” tolerance will come into play. Based on mesh size, decide whether you want these overhanging regions to be removed or not. If you can do this with a global value, great, else use a value that works best for most overhanging regions and plan to adjust others manually.
4. Also keep in mind that there is some failure handling after meshing that can help. For example, in some cases you may want to use defaults, generate mesh, diagnose problems, then delete all connections and re-create with better values. Failure handling techniques are described in the section “Visualizing mesh connections”.

Common Imprints

The tolerance for common imprints comes from the minimum element size in the footprint mesh (the horizontal plate in this example). Setting the mesh size appropriately is important to control whether you want the imprints to be common or not.

For example, in this case if you want a common imprint the min element size (or element size if ASF=Off) should be $> x$.

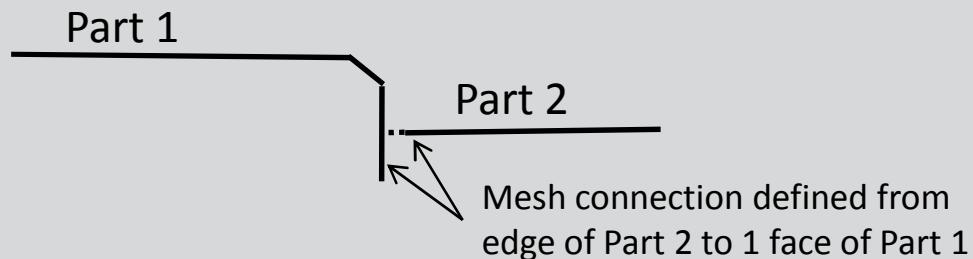
You could set a face mesh size on that plate for local control over this.



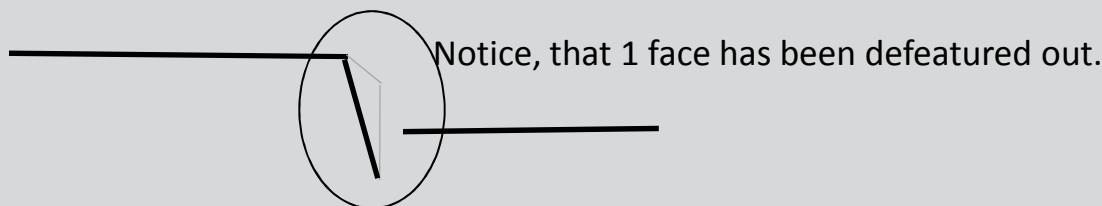
Defeathering

In cases where part mesh is significantly defeatured prior to mesh connections, mesh connections may fail.

For example, consider the 2 parts below:



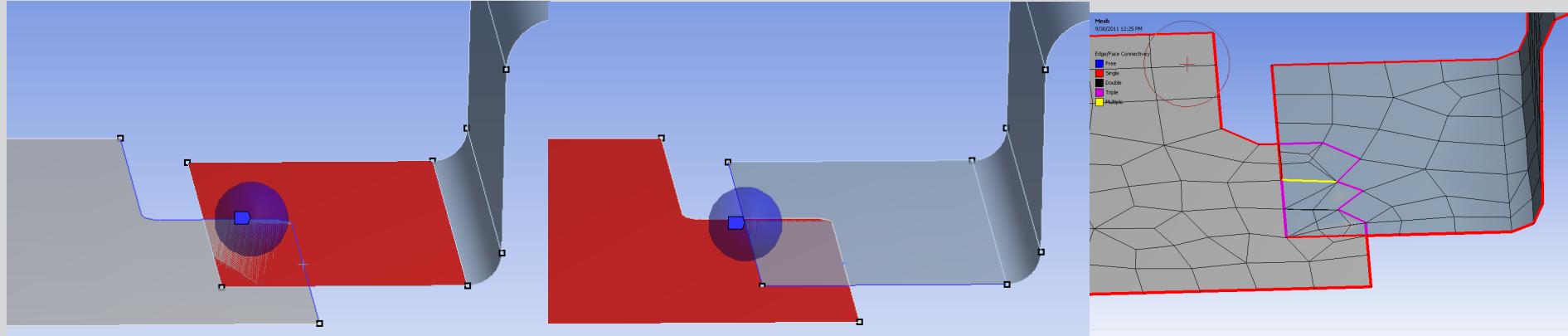
If using pinch controls, or uniform quad/tri mesh method, the part mesh could look like:



- In this case, if the defeatured face is the one defined in the mesh connection the connection will fail.
- If the other face is the one defined in the mesh connection, the connection will succeed.
- If user puts both faces in the mesh connection, the connection will succeed.
- **Since you can't always control which face is defeatured, it is recommended to put both faces in the mesh connection for most robust approach.**

Mesh Pockets in Mesh connections

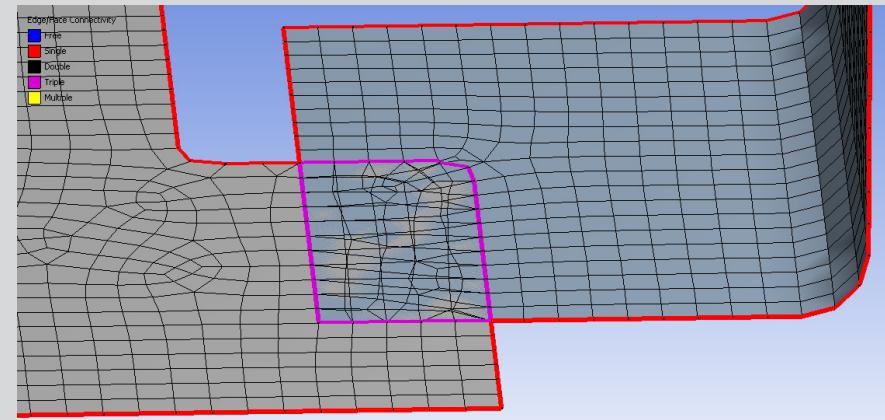
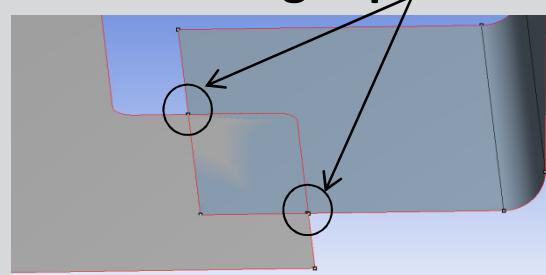
Sometimes mesh pockets can form from spurious connections:



Quality/Connectivity in mesh pockets sometimes poor

Recommendations:

- Turn off snap to boundary
- Carefully control sizing
- Create intersection VT edge splits



Demo



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

Meshing

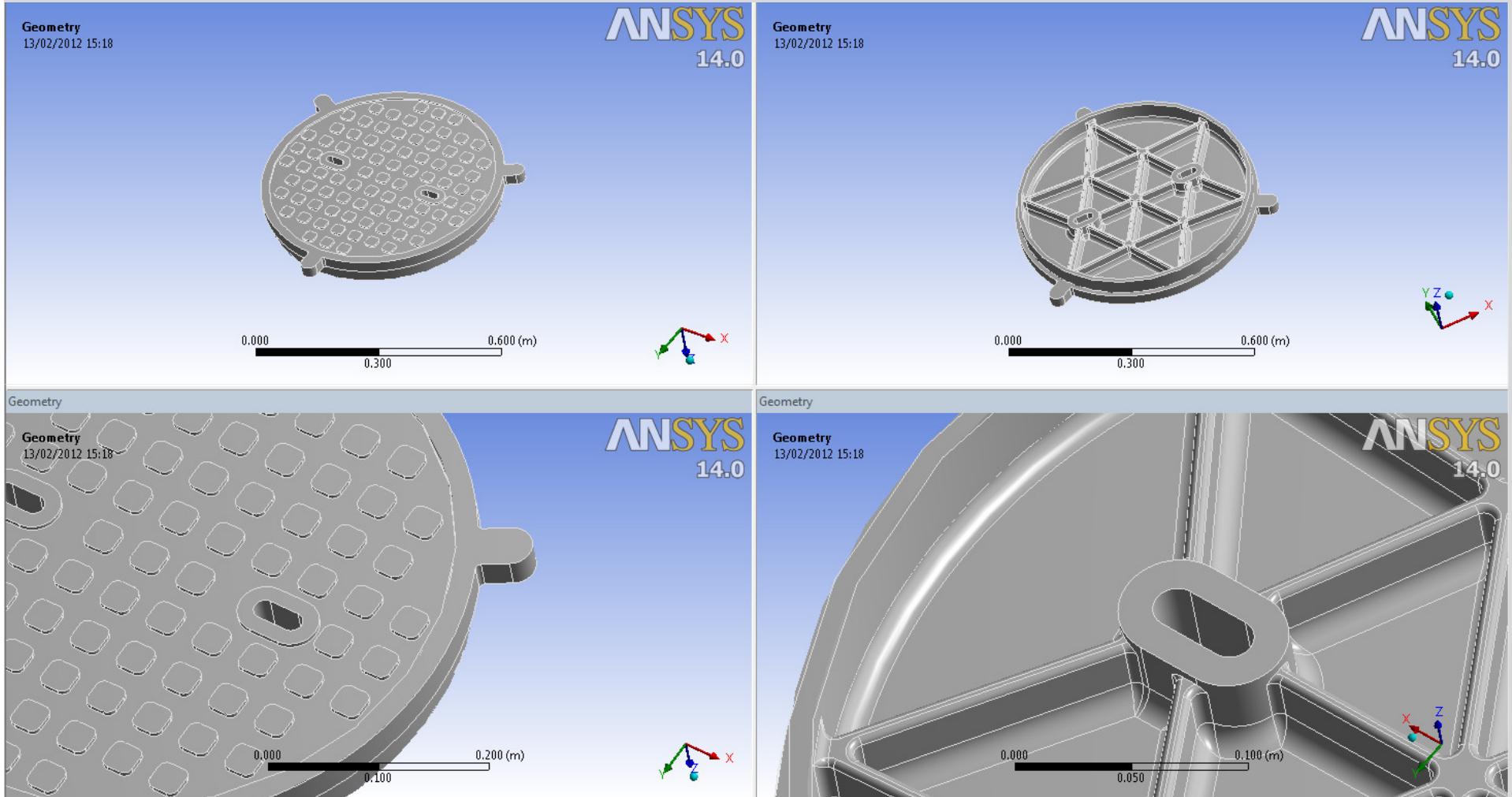


Fluid Dynamics

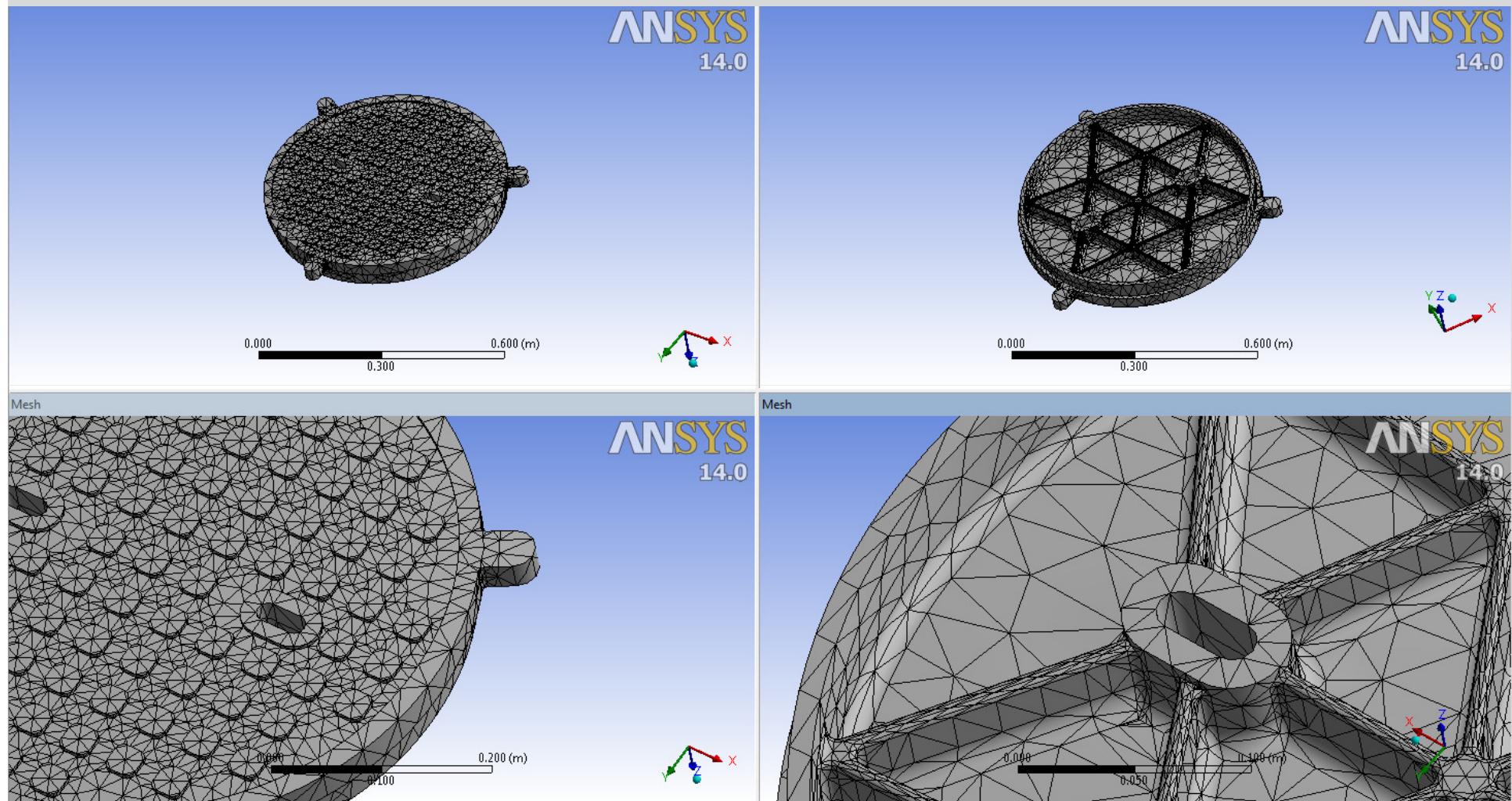
Structural Mechanics

Electromagnetics

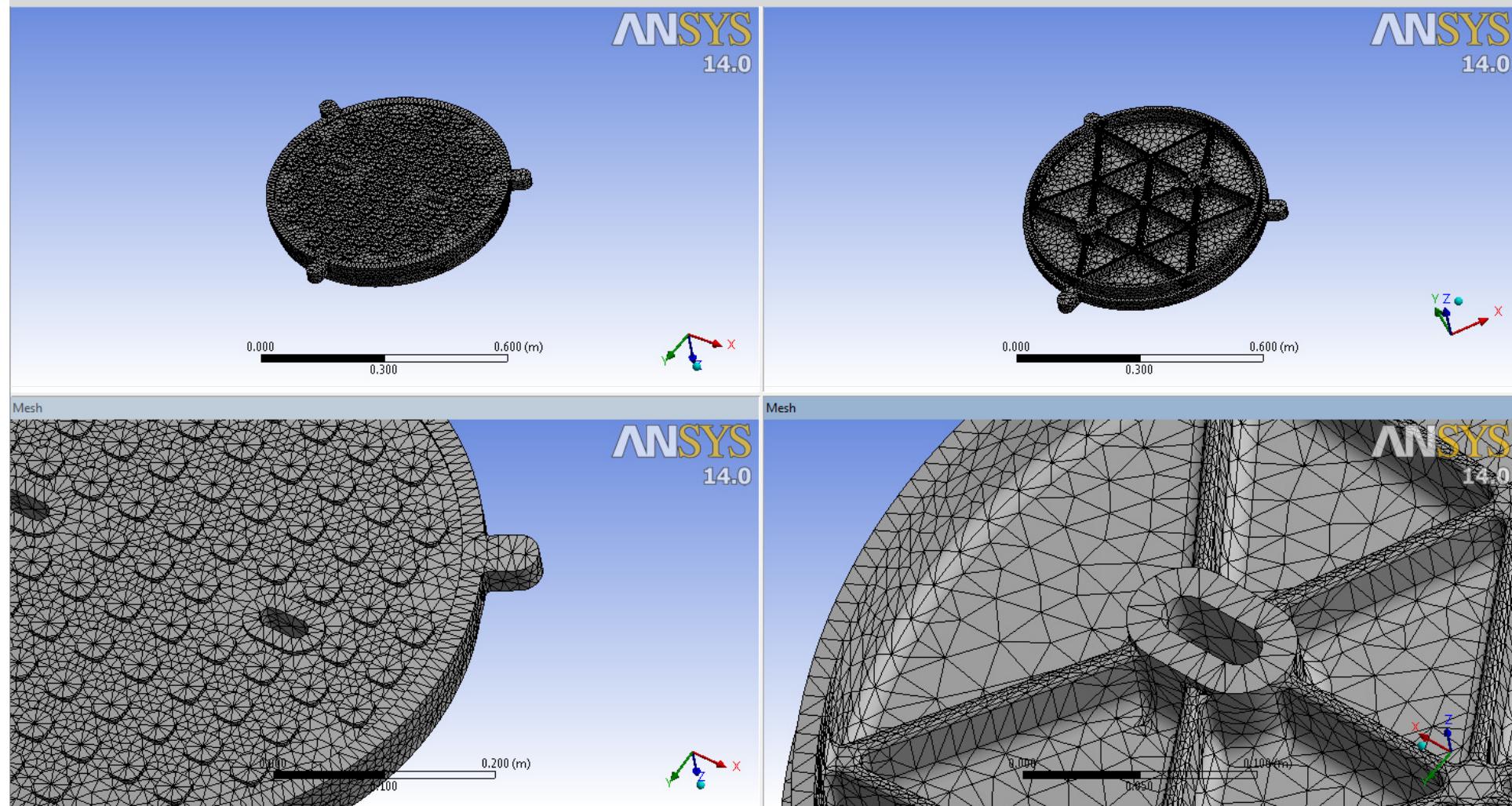
Systems and Multiphysics



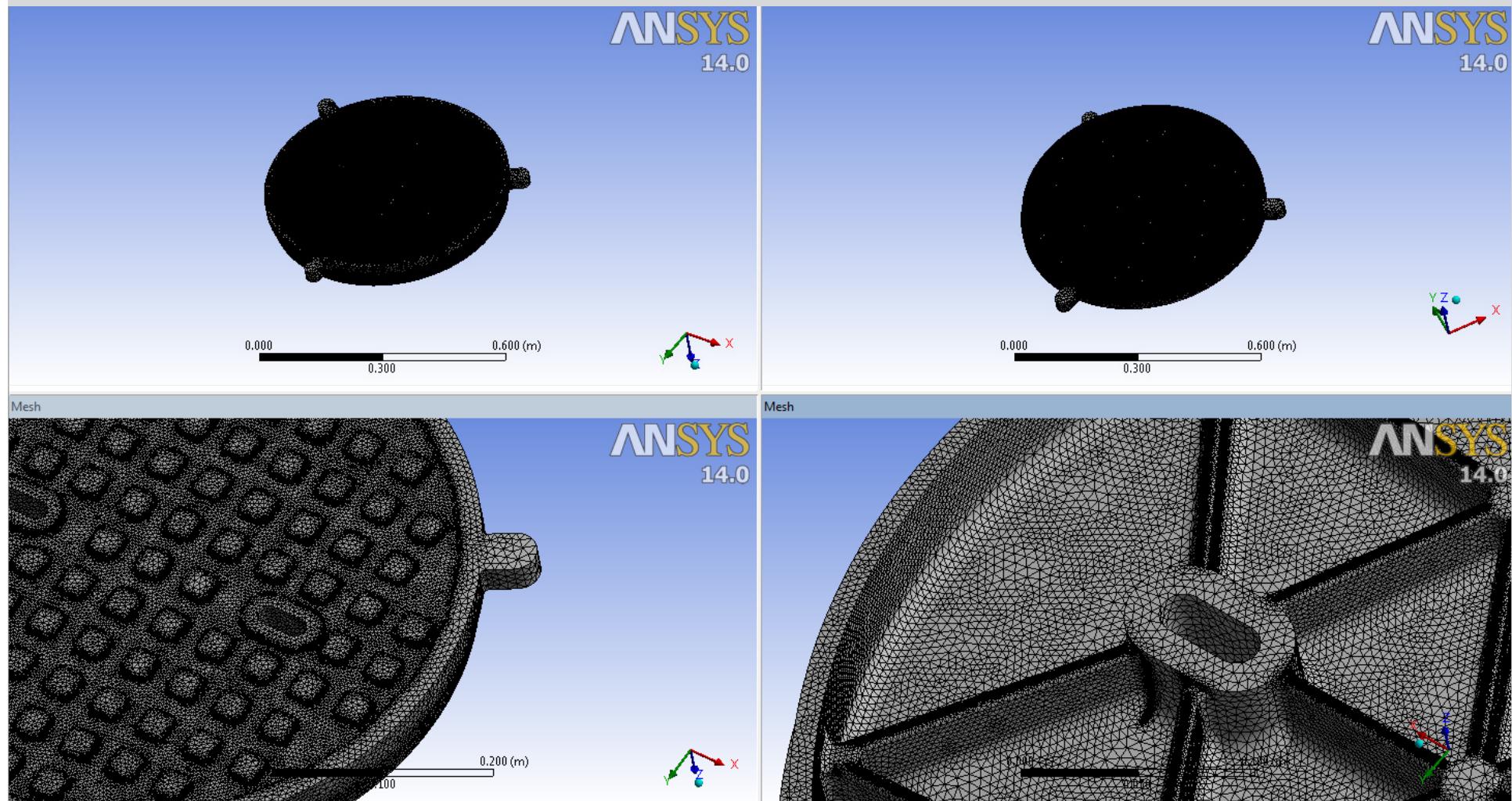
Default



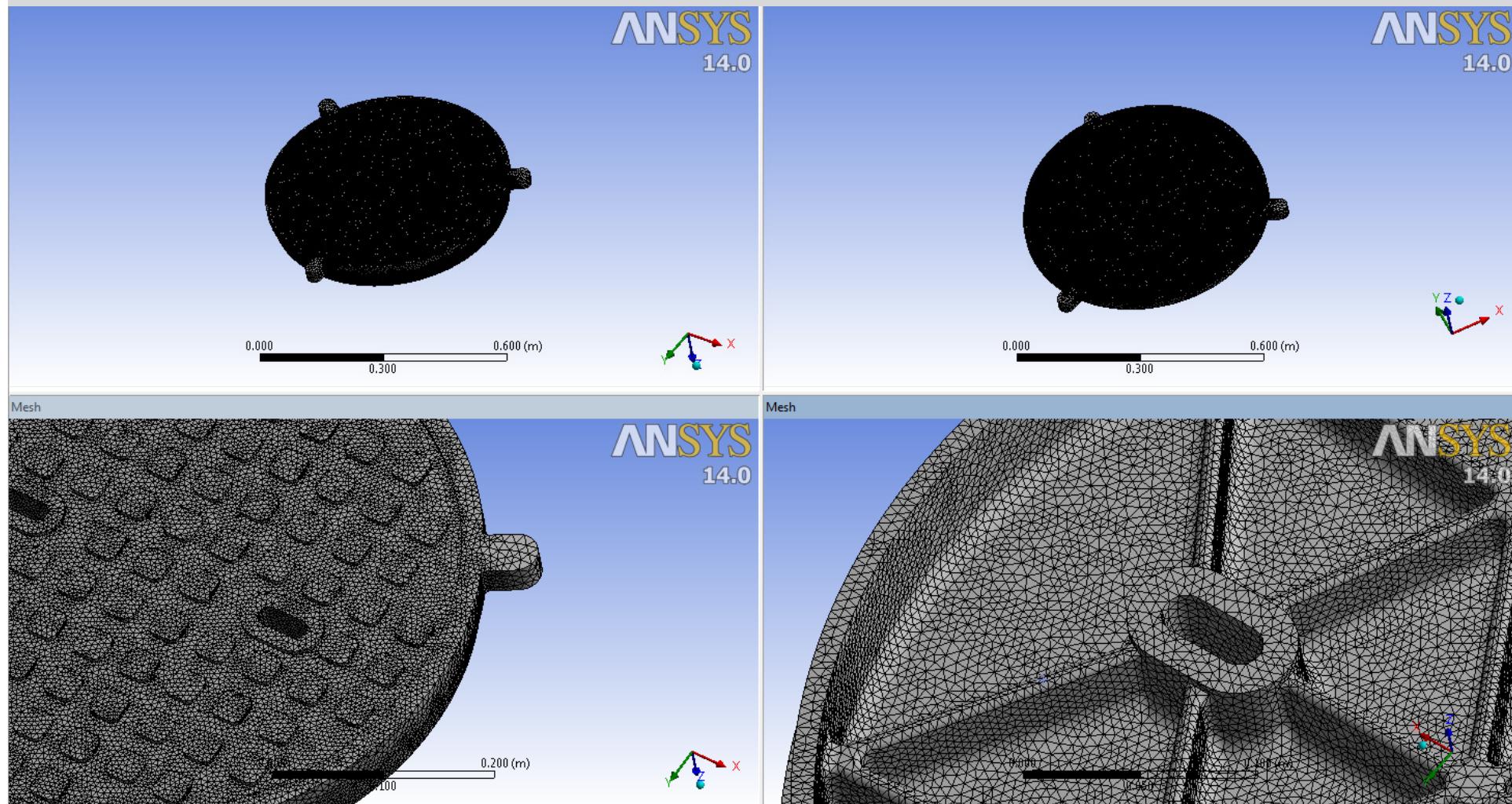
Global sizing 10mm



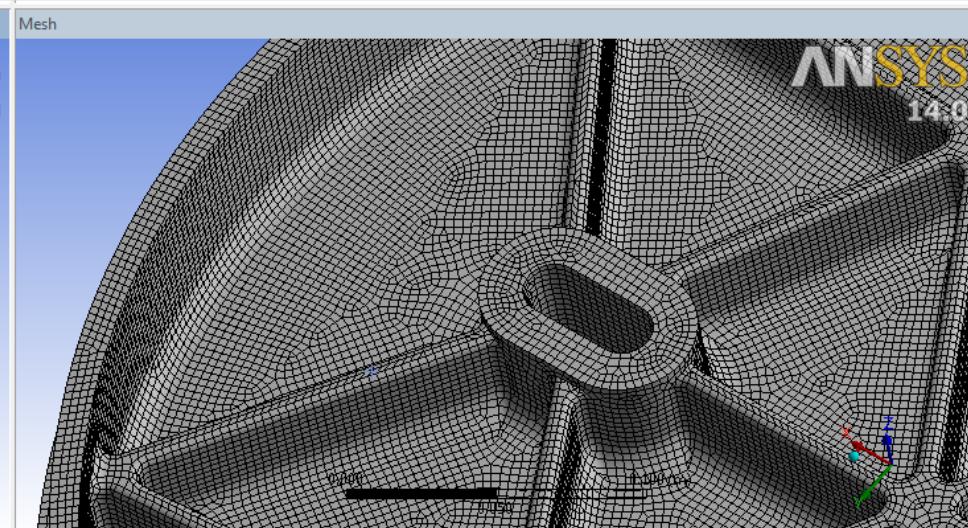
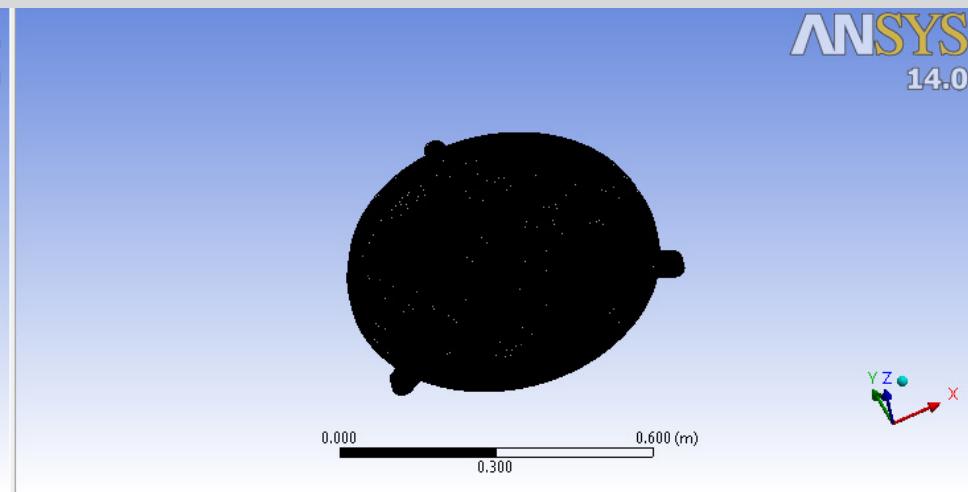
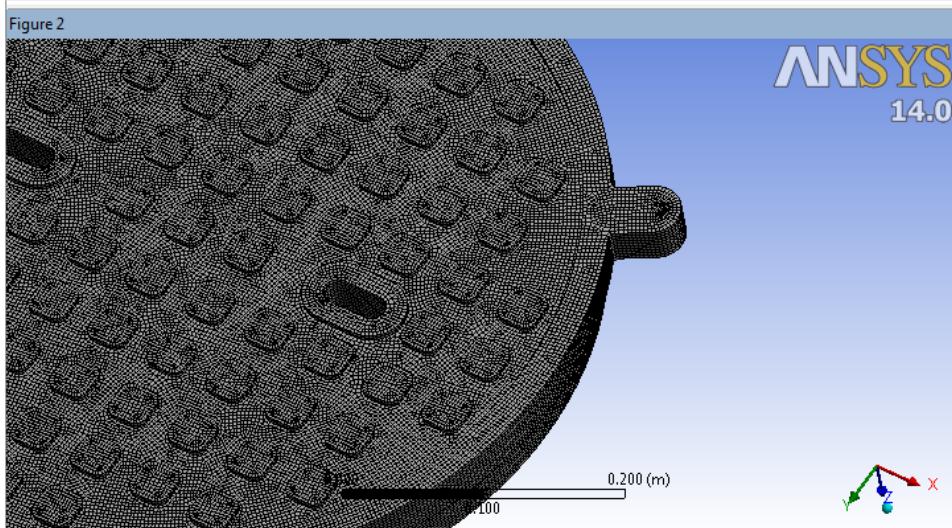
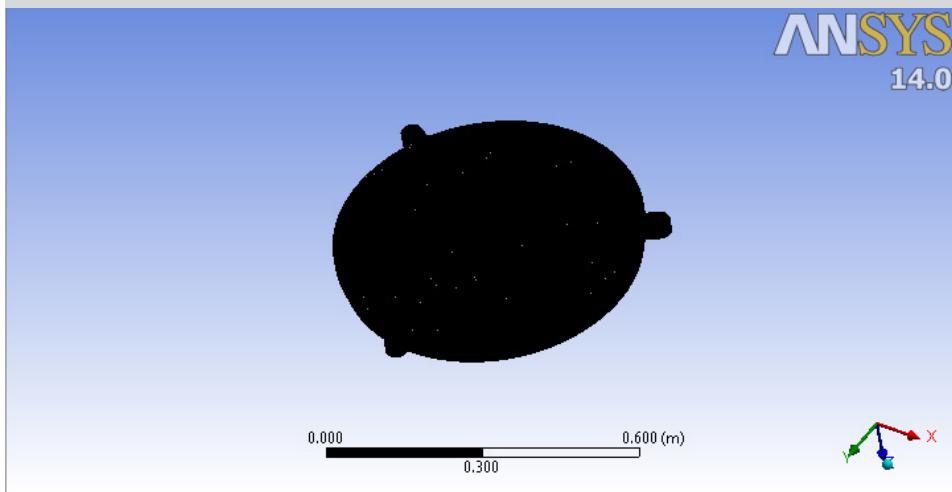
Proximity



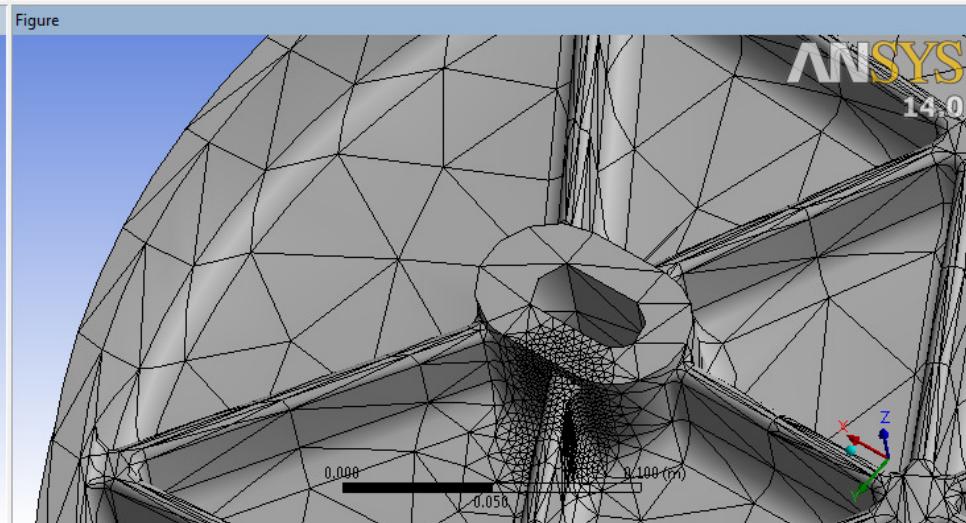
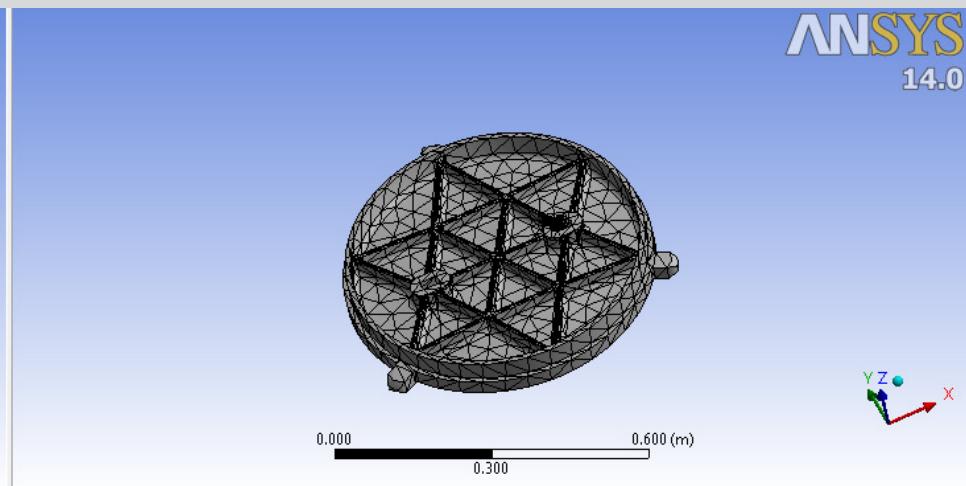
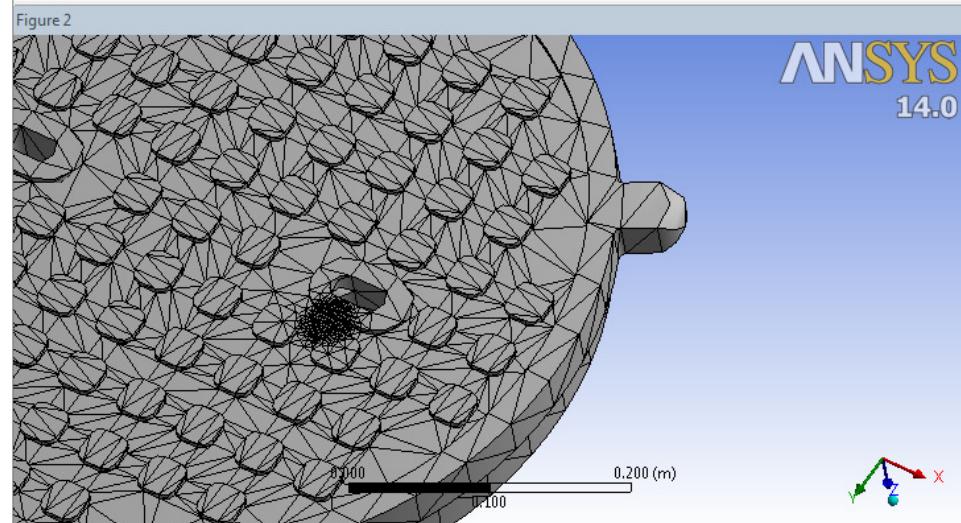
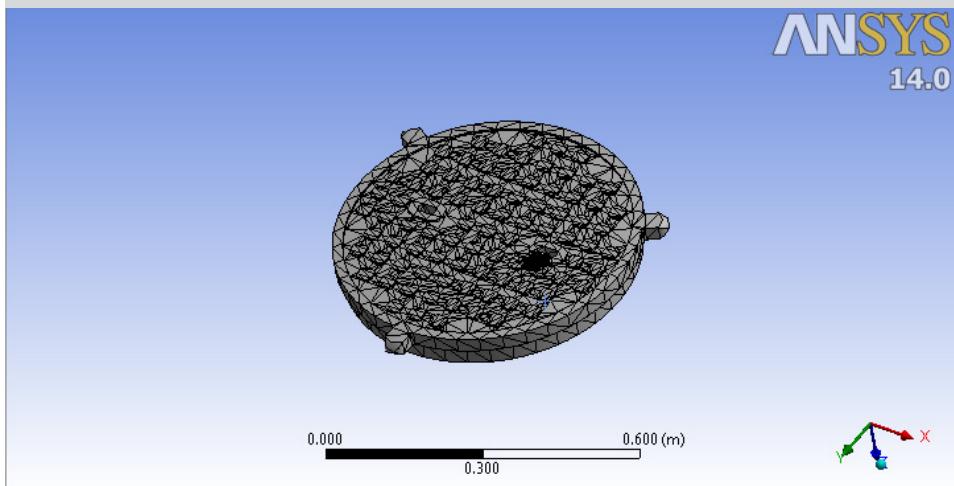
Patch Independant



Hex Dominant



Body of Influence



Body of influence with sizing

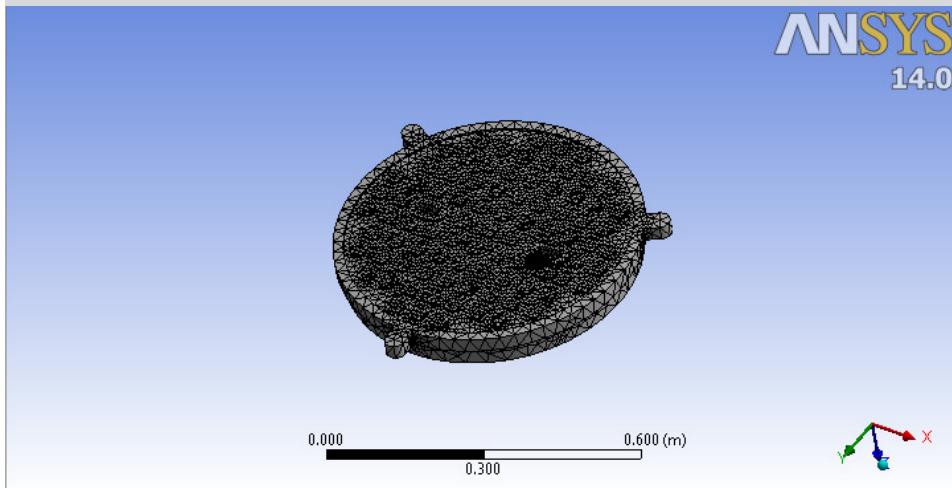
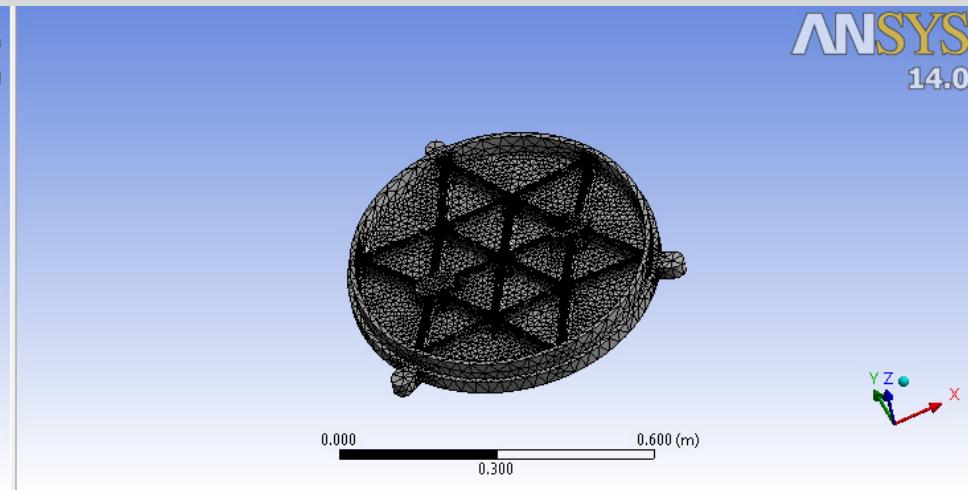
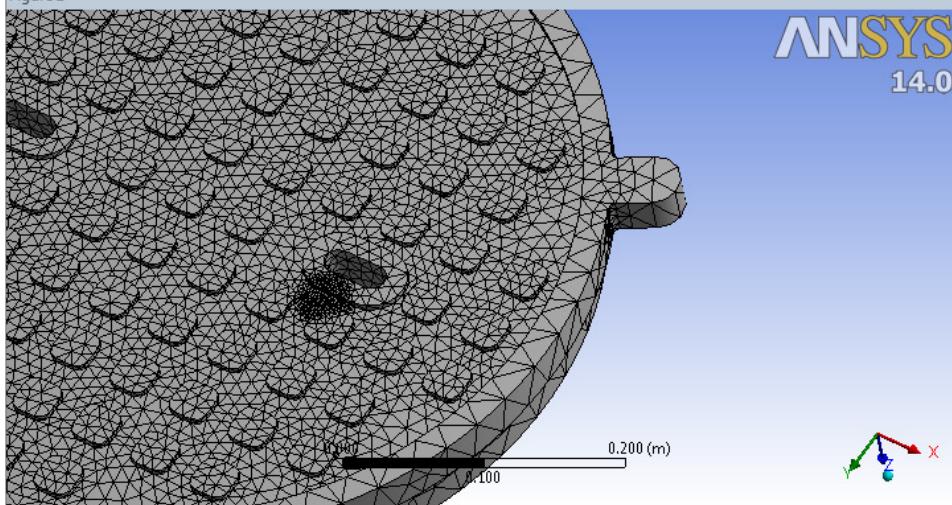
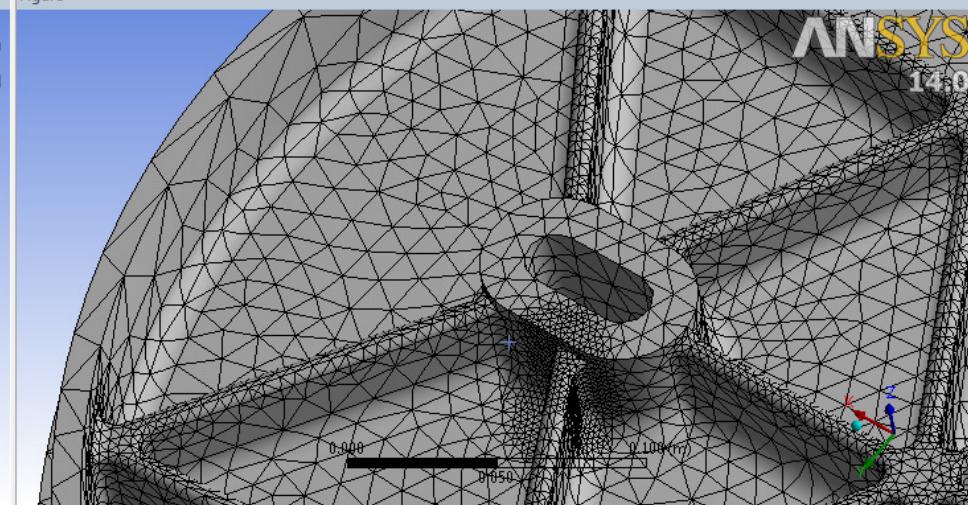


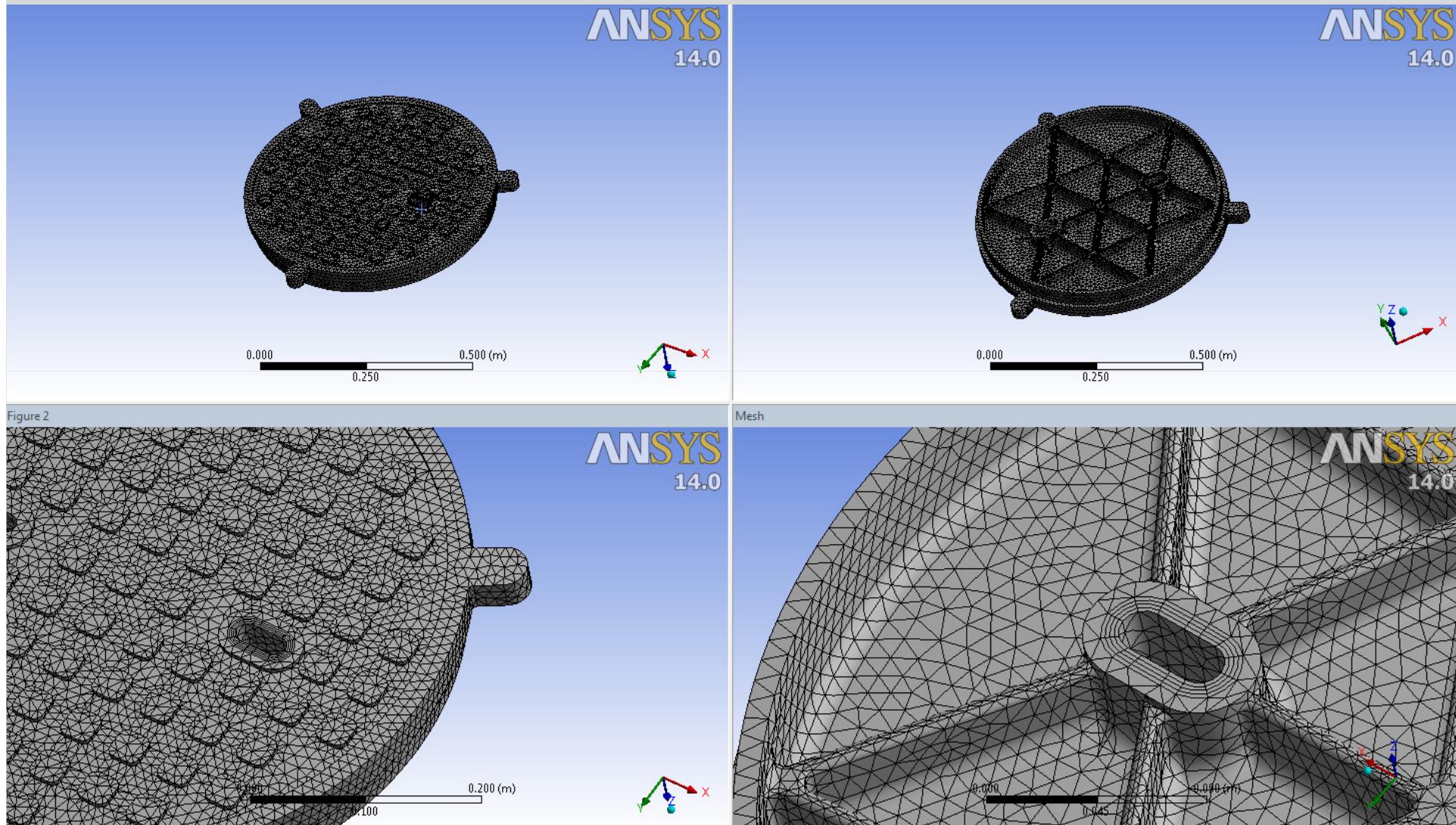
Figure 2



Figure



Inflation



Summary

Method	Nodes	Elements	+	-
Default	52k	28k	Quick	Not good for non-linear
10mm sizing	98k	53k	Quick	Can be dependant
Proximity	2500k	1600k	Minimal user input	Large model
Patch Independent	967k	642k	Removes small features	Can take time to generate
Hex dominant	1560k	424k	Good for “regular” geometry*	Beware of internal mesh
Body of Influence	146k	87k	Localised refinement	Check far field size
Inflation	130k	74k	Good quality local refinement Can be mixed with other methods	Needs good definition

* With careful sizing control

Error checking, Singularities and interpretation



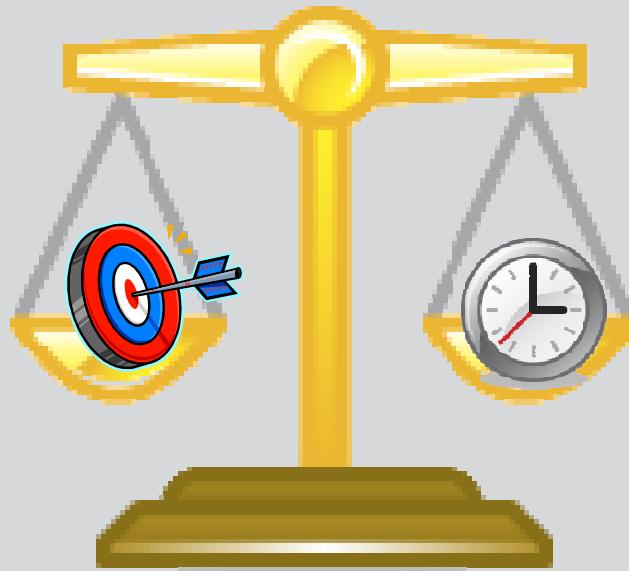
Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

Is my mesh sufficient?



Contents

Structural error

Dealing with singularities

Automatic mesh refinement

Contour averaging options

Results Comparison

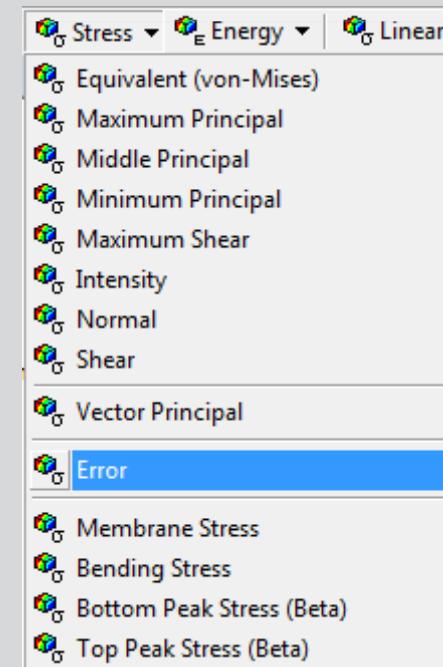
Mechanical – structural error

Structural energy error (SERR)

(Also thermal energy error (TERR))

Measure of the discontinuity of the stress field (or heat flux) from element to element

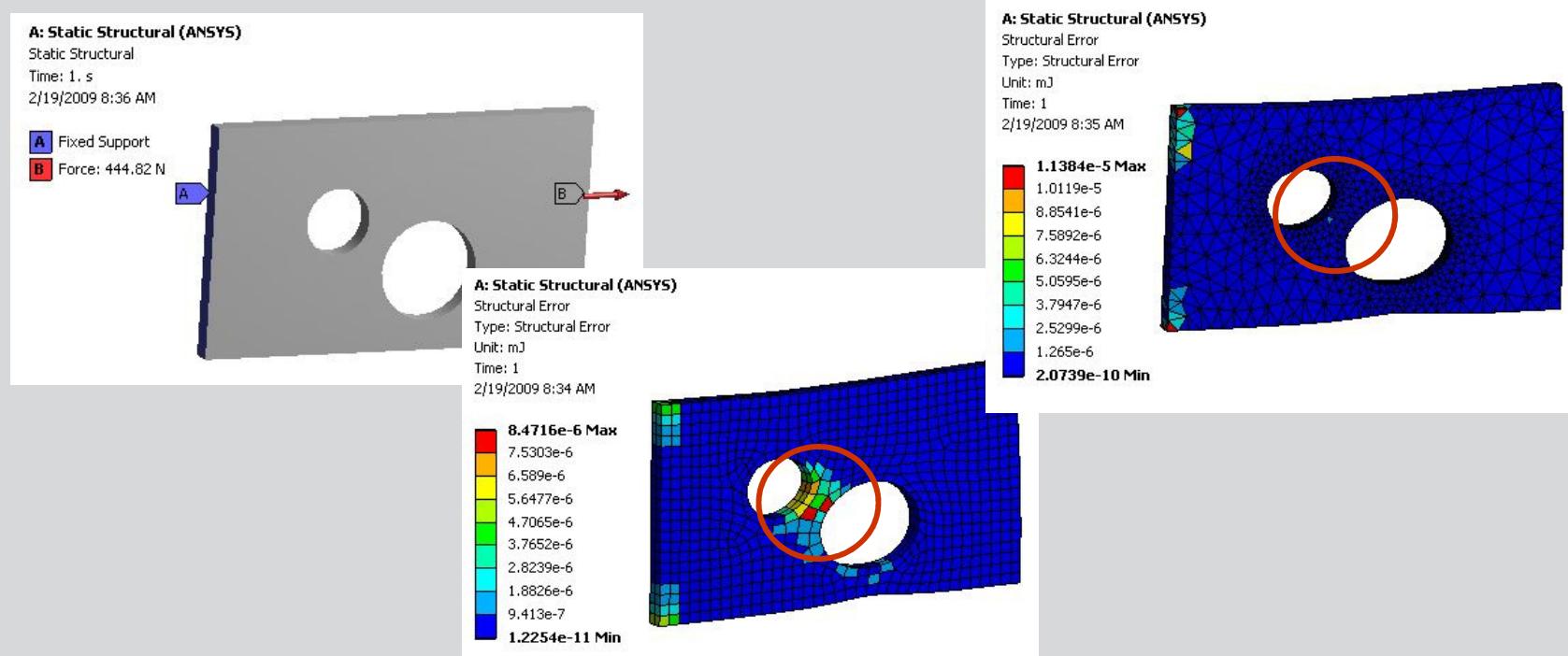
Also known as ZZ error (Zienkiewicz-Zhu)



Mechanical – structural error

Regions of high structural error indicate where the model may benefit from mesh refinement

Relative value – the absolute value is generally not significant



Singularities

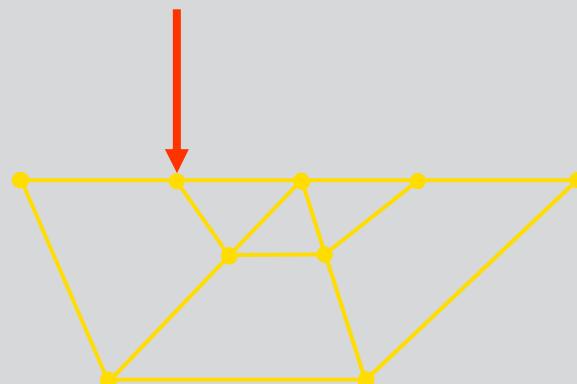
Typically, quantities that are directly solved (degrees of freedom) converge smoothly with mesh refinement

However, derived quantities (e.g. stress, heat flux, etc) may diverge due to singularities

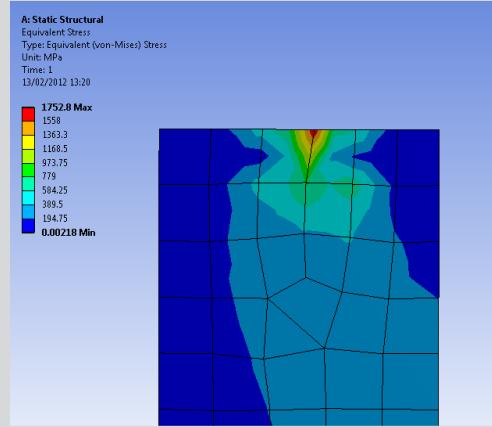
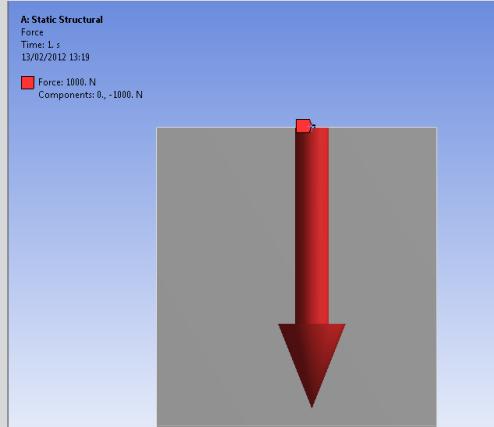
$$\sigma = \frac{\text{Force}}{\text{Area}}$$

As Area \Rightarrow Zero

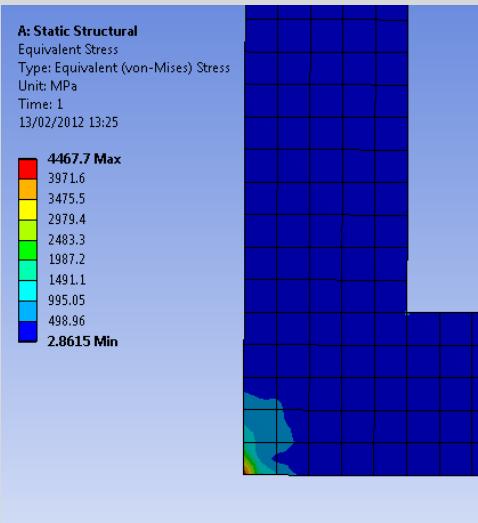
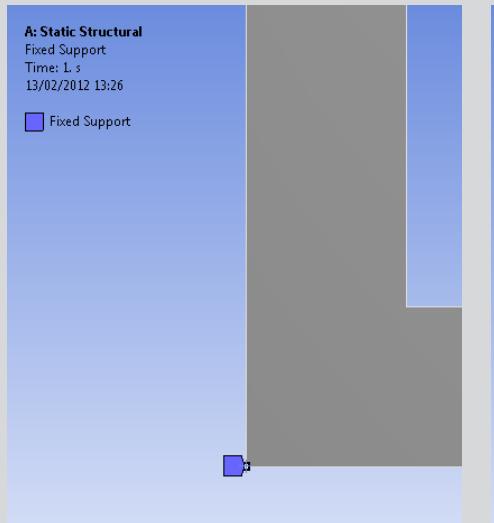
$$\sigma \Rightarrow \infty$$



Singularities

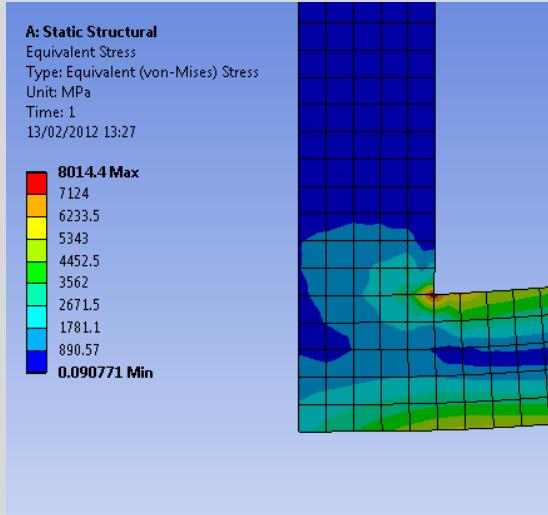


Point loads

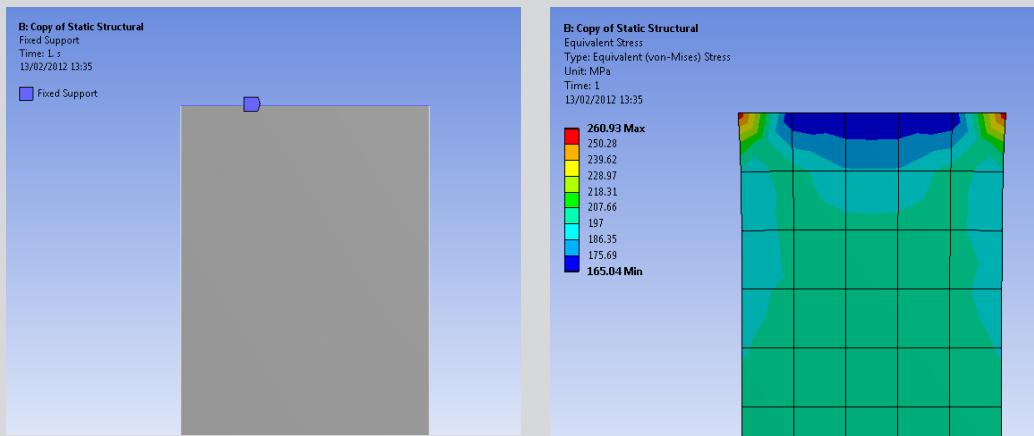


Point constraints

Singularities



Sharp corners

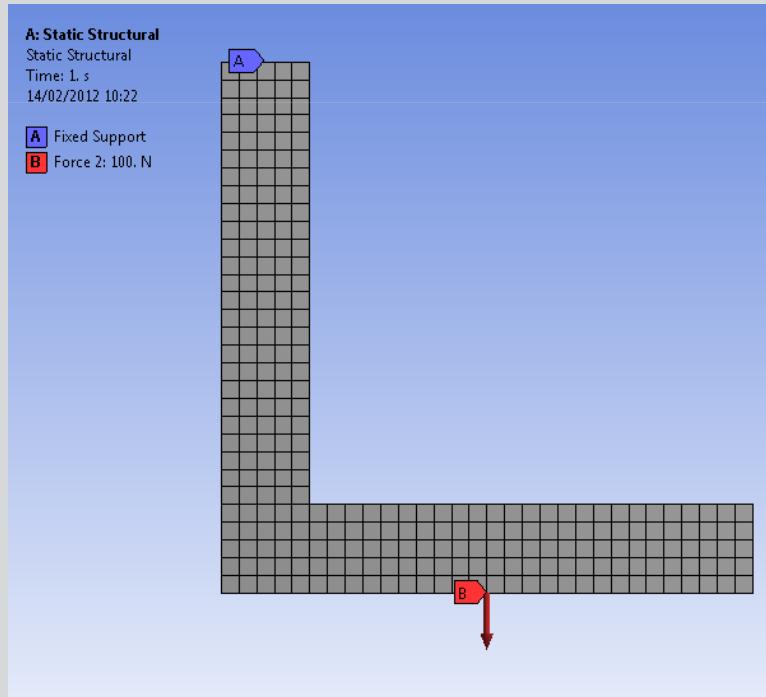


Fixed supports
(Poisson effect)

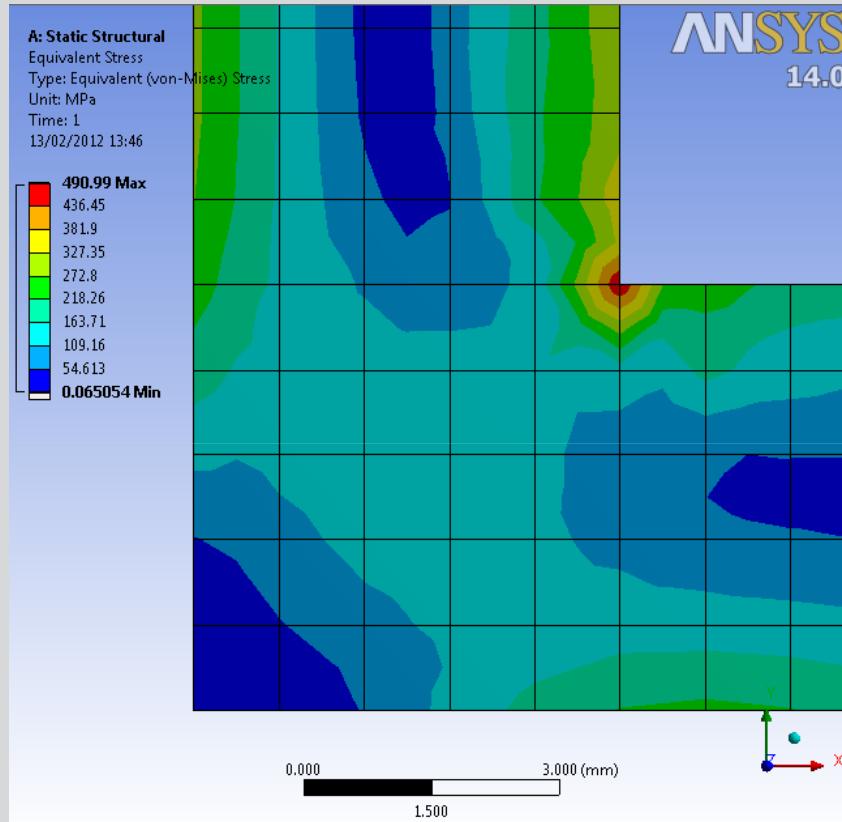
Dealing with singularities

Is it in an area of concern?

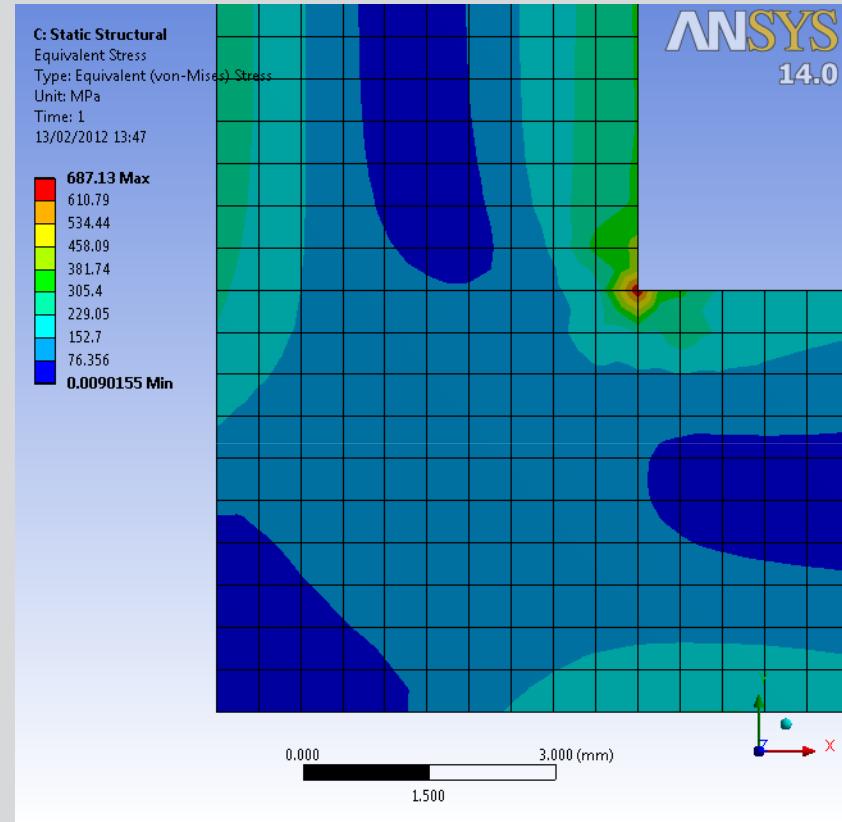
- No – use results tools to remove the effect of the singularity
- Yes – change the model setup to remove the singularity



Results tools

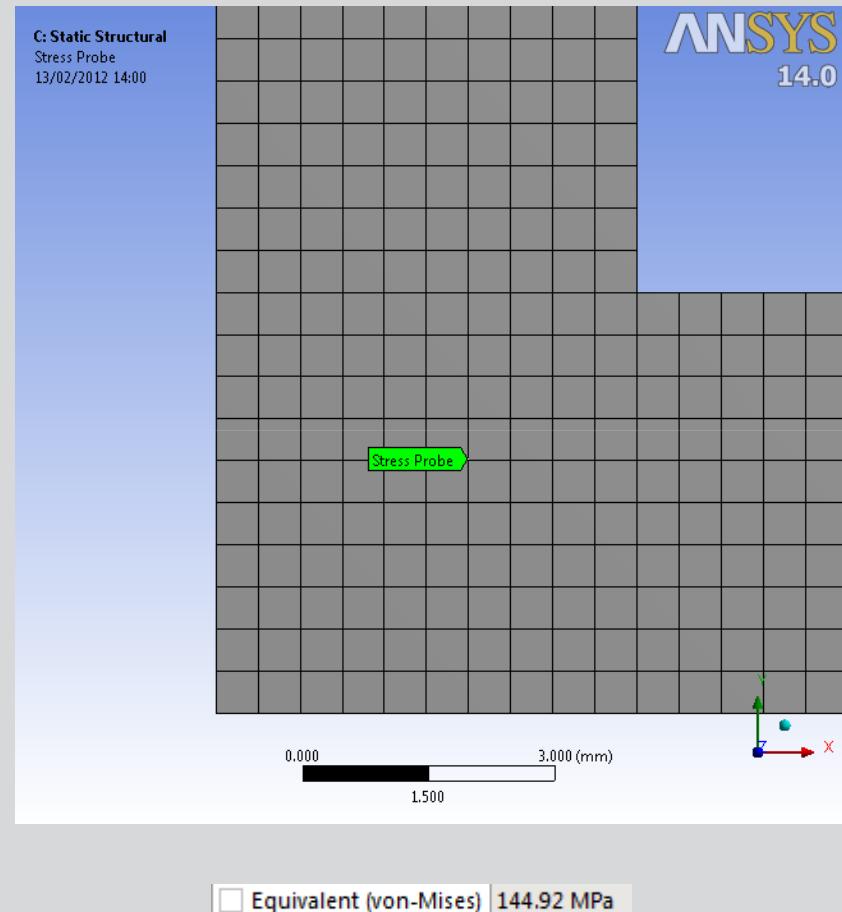
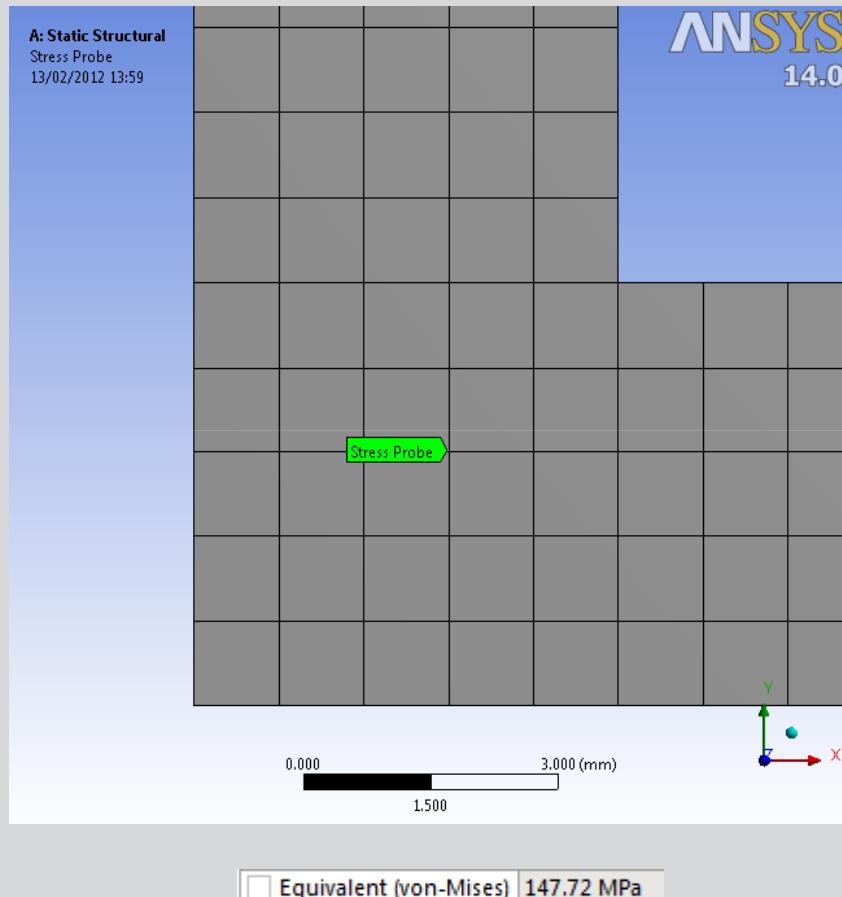


mesh size = 1mm

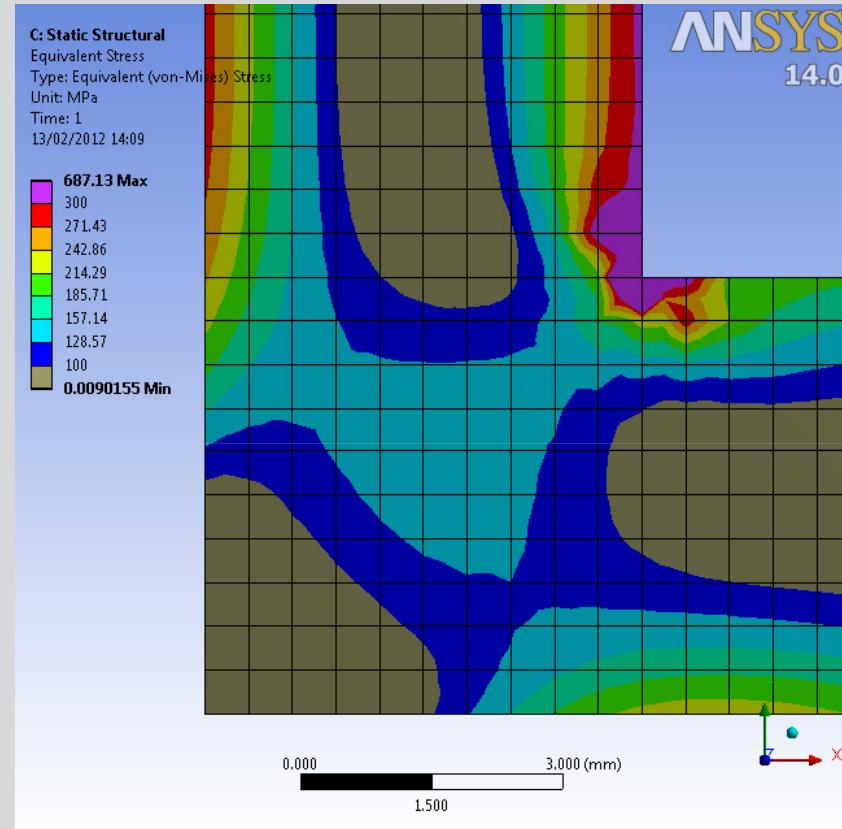
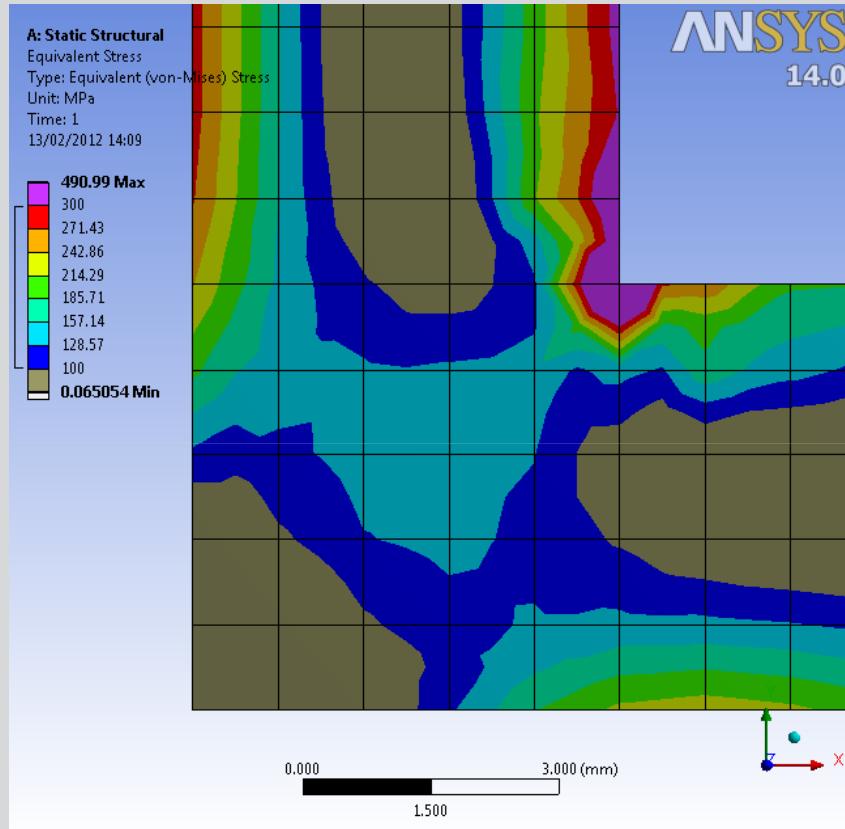


mesh size = 0.5mm

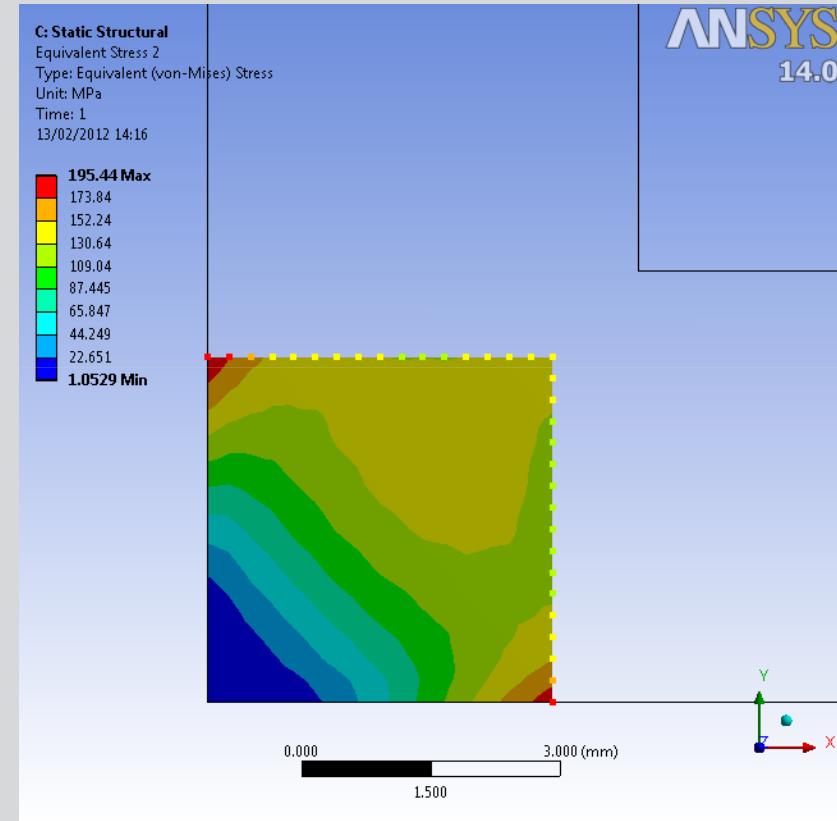
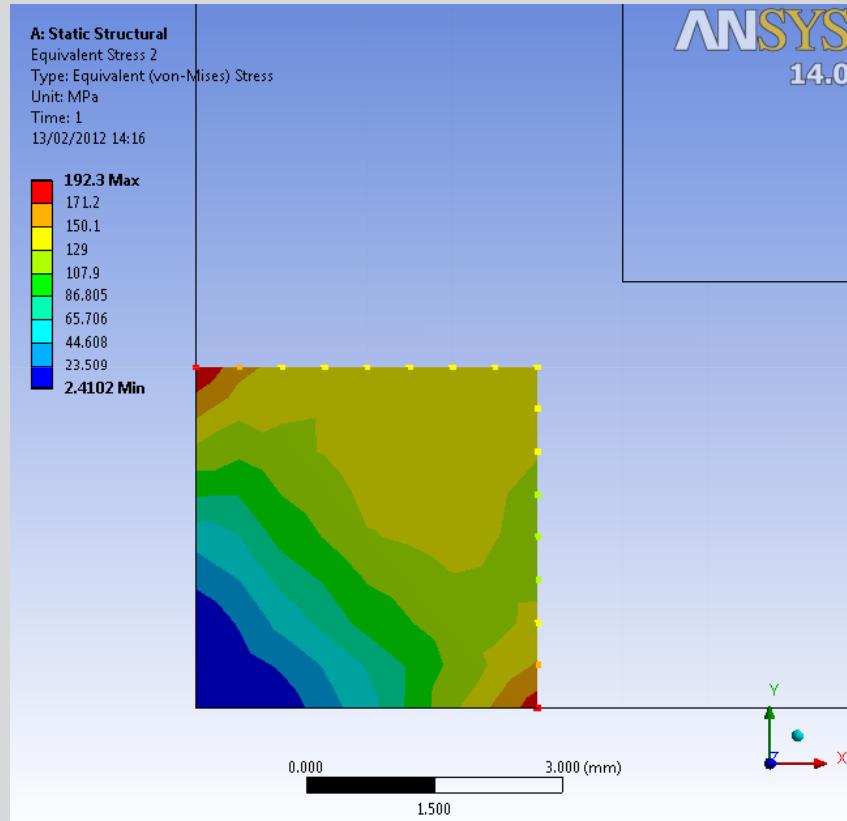
Results tools



Independence bands

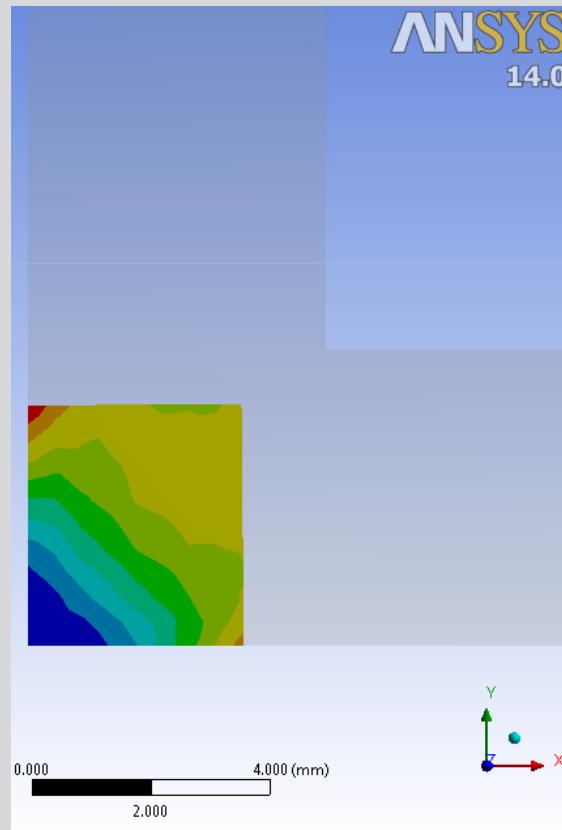
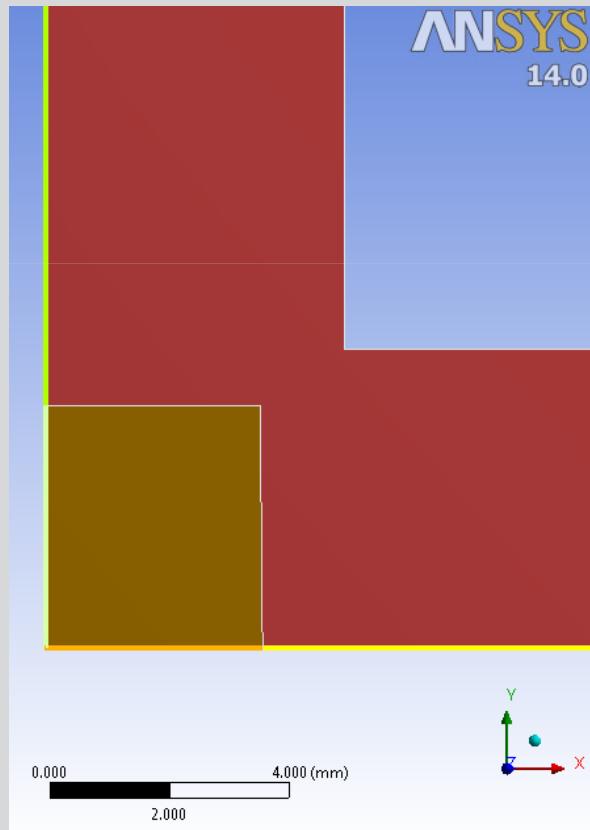


Scoped results – node/element named selections



Scoped results – virtual geometry

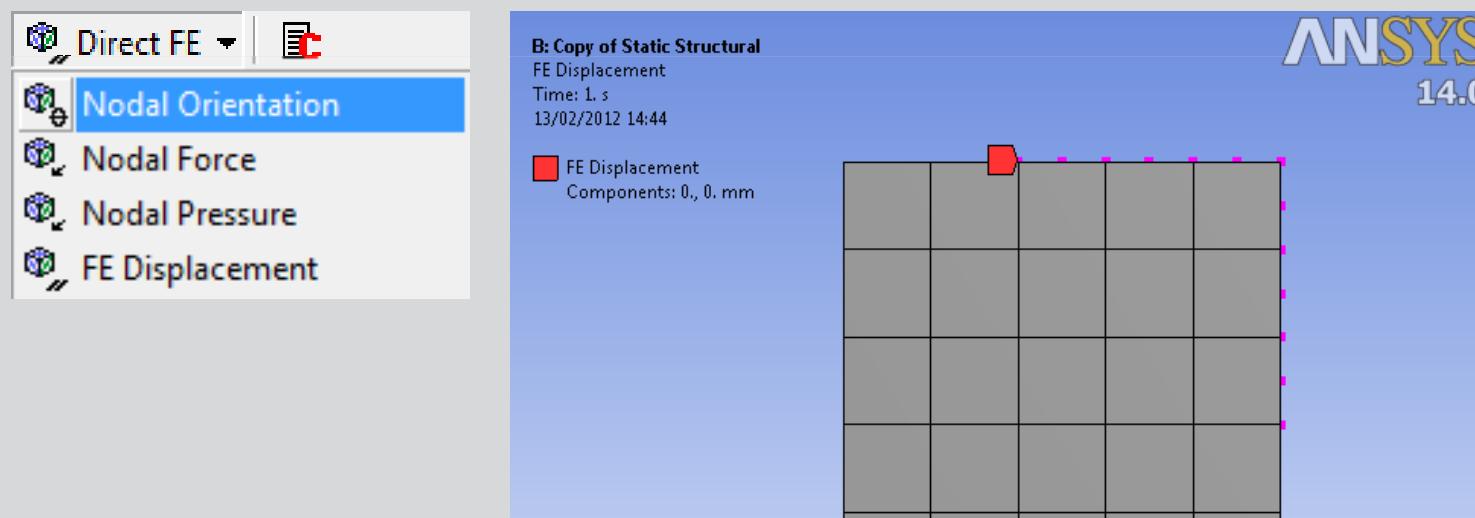
Virtual Topology Merge Cells Split Edge at + Split Edge
 Hard Vertex at + Split Face at Vertices



Changing the model setup

Distribute point loads or point constraints

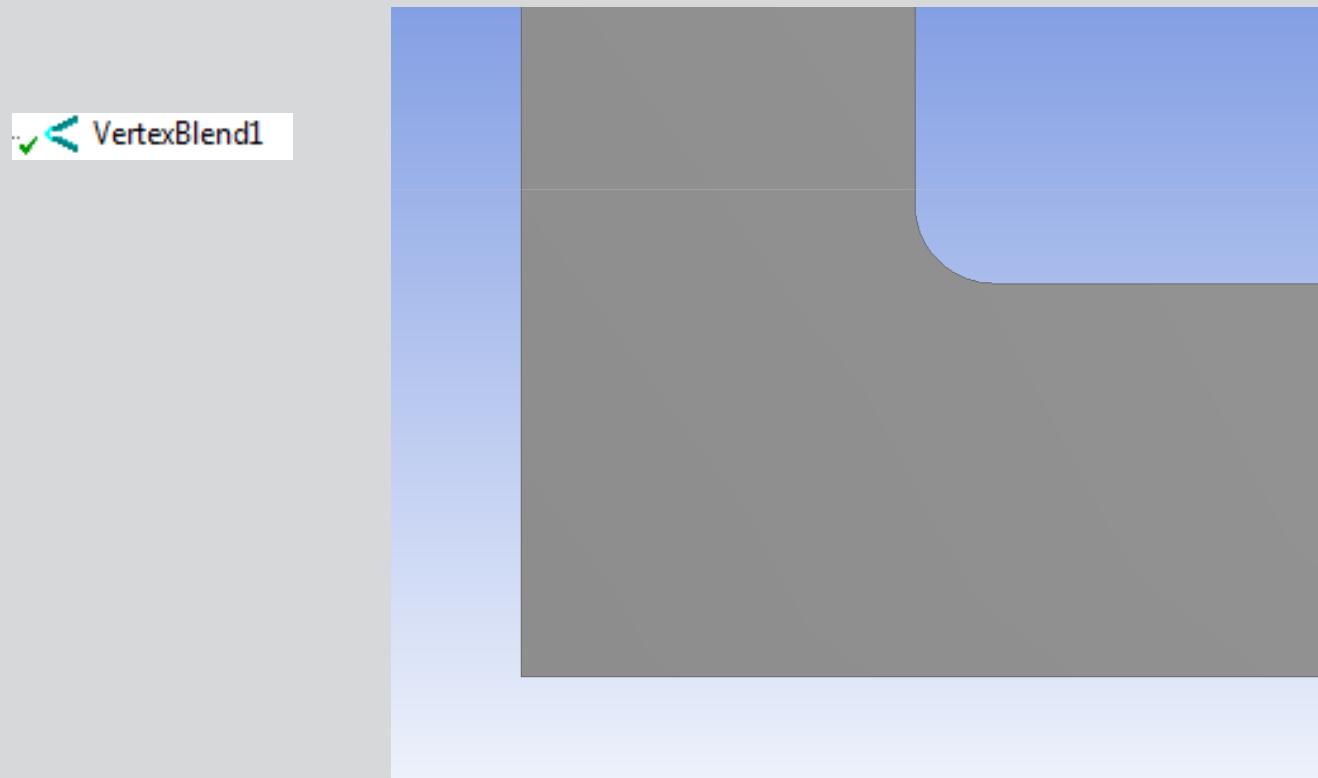
- Virtual geometry
- Nodal loads/constraints



Changing the model setup

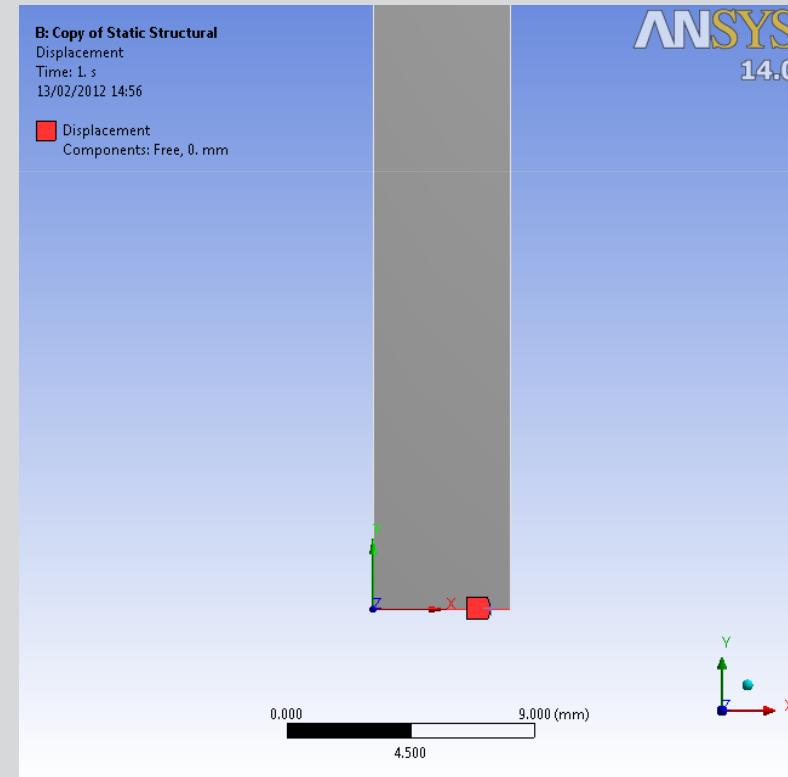
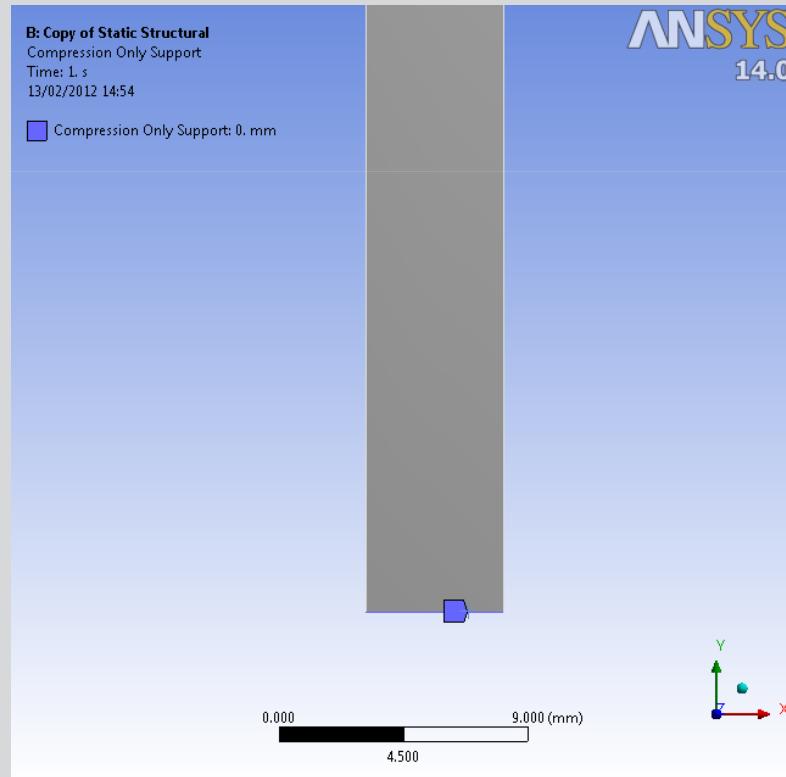
Use DesignModeler or SpaceClaim to introduce realistic fillet radii

Consider submodelling?



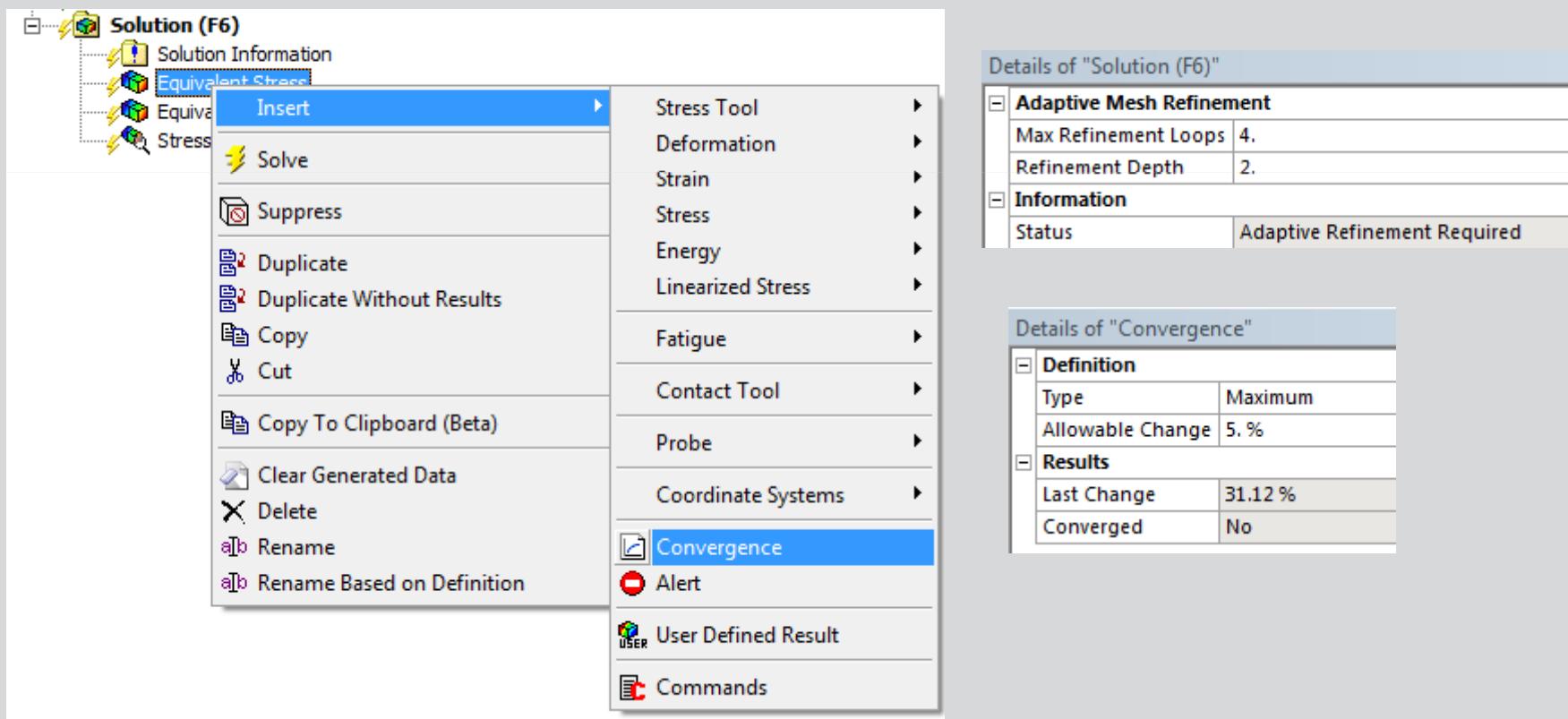
Changing the model setup

Consider using alternative constraints instead of fixed supports



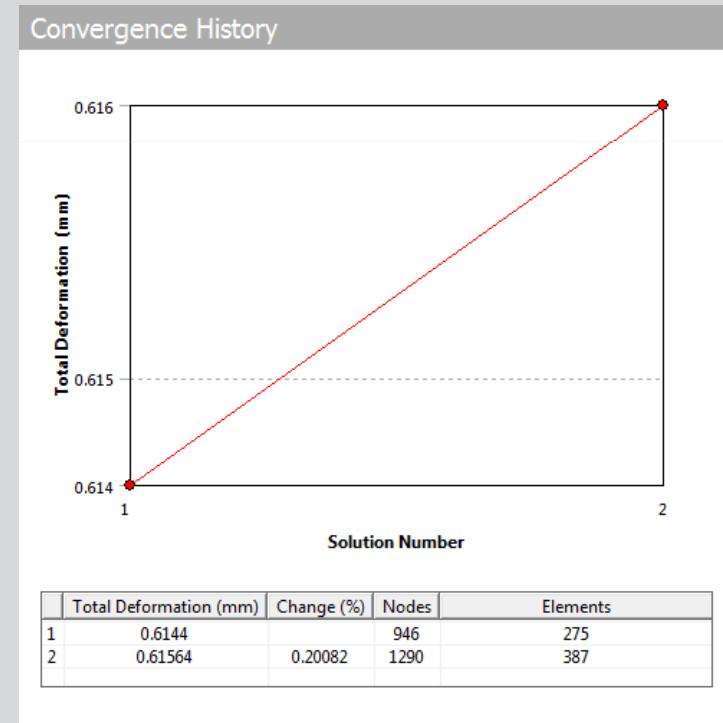
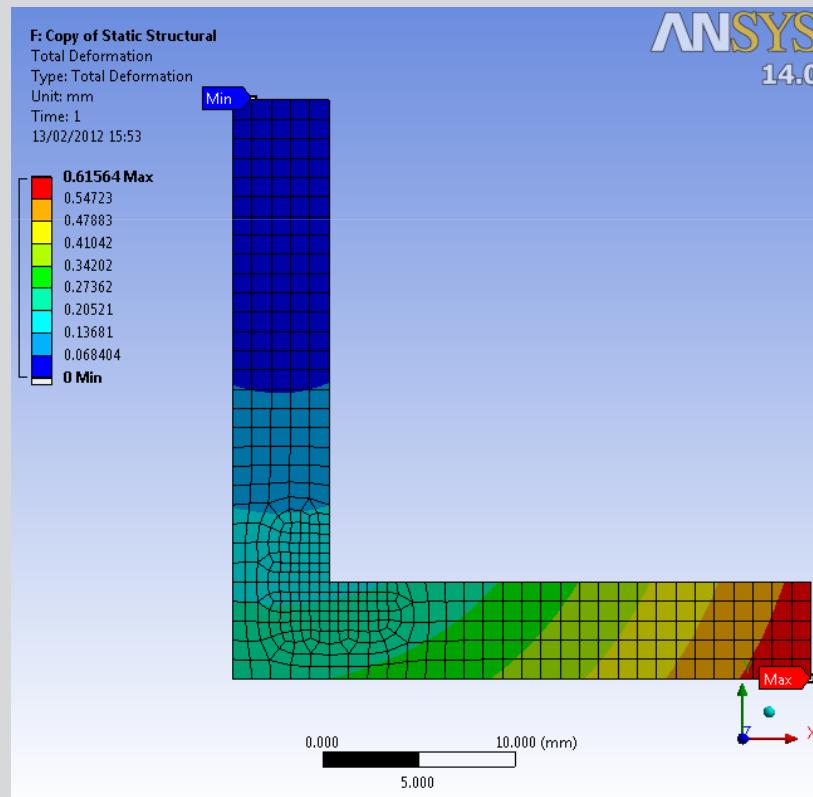
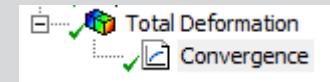
Automatic mesh refinement

Automatically refines mesh based on regions of high structural error and re-solves model on an iterative basis



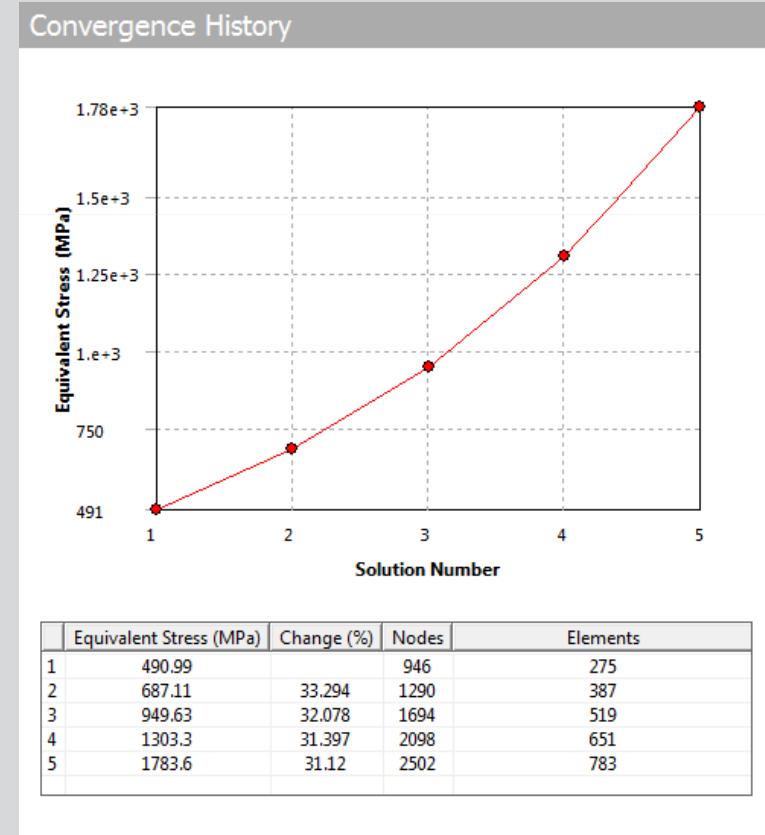
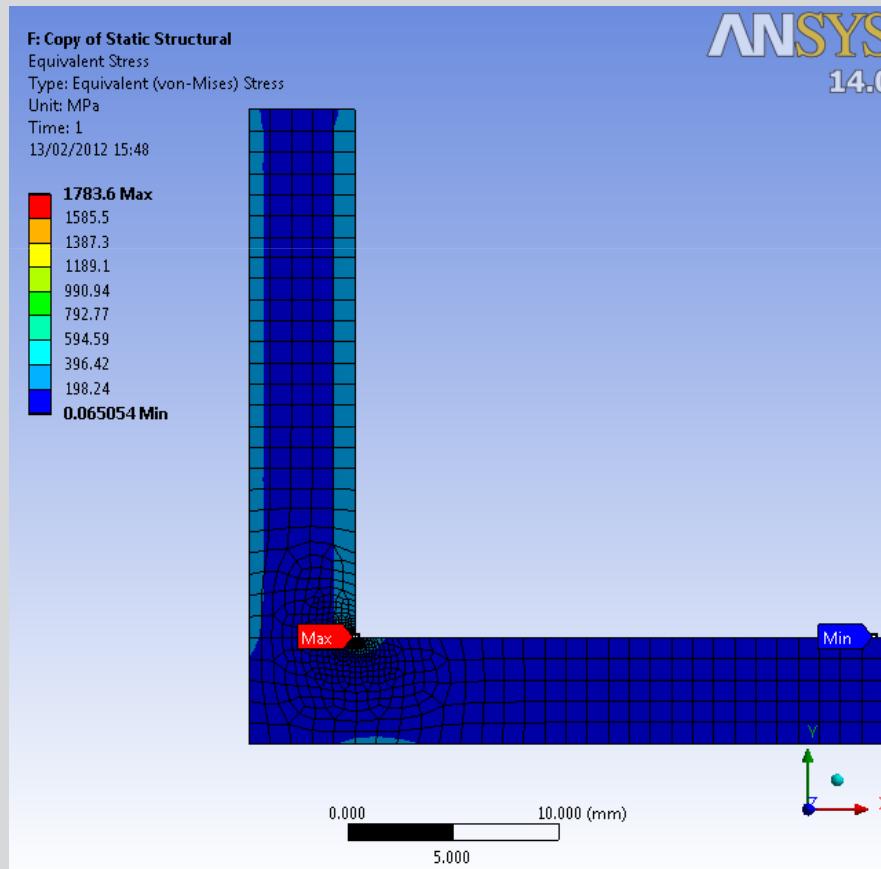
Automatic mesh refinement

Displacement results converge smoothly



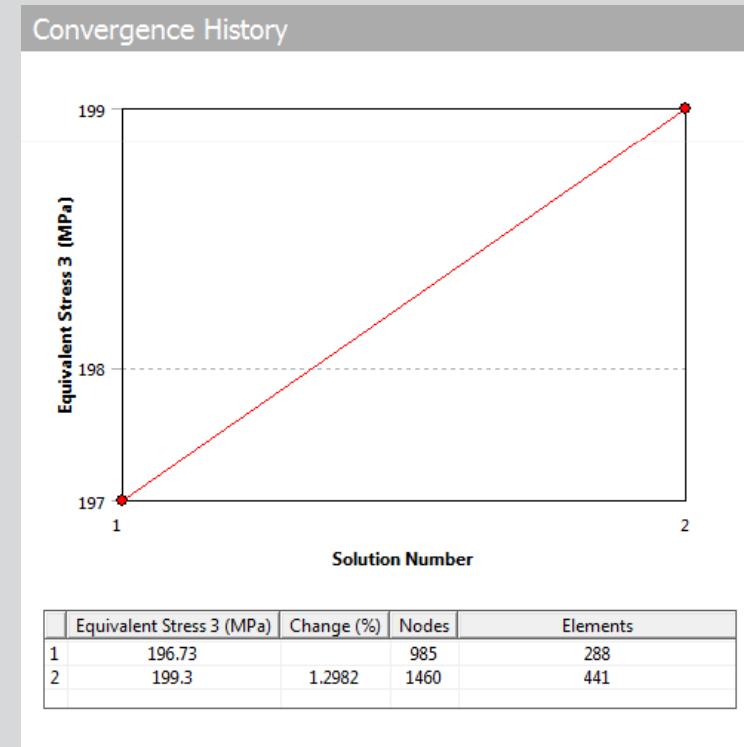
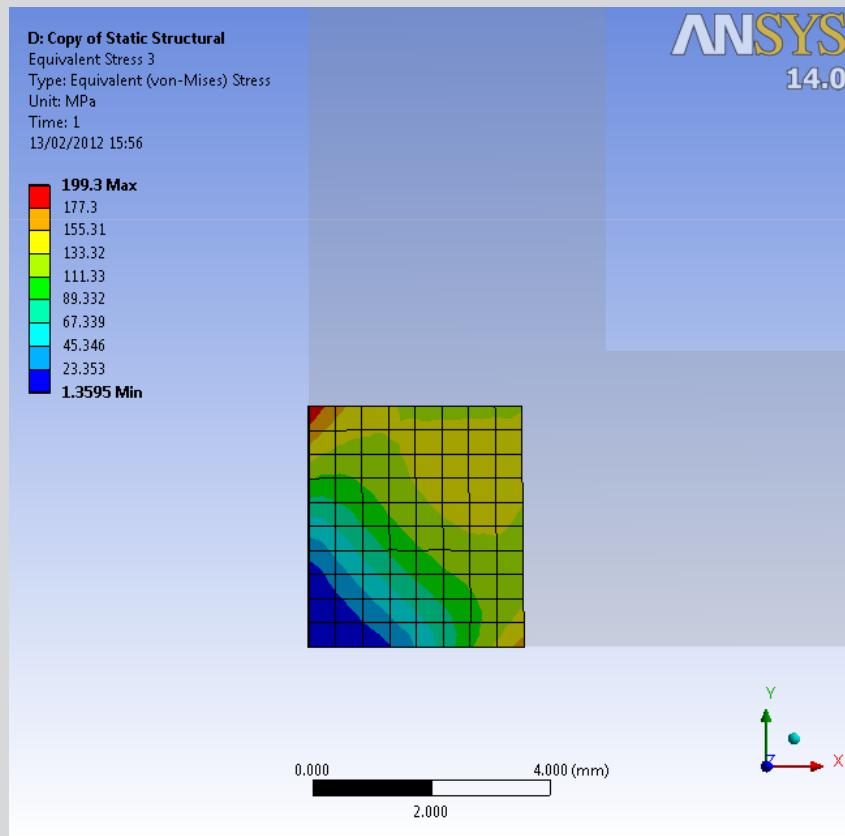
Automatic mesh refinement

Stress results fail to converge due to presence of a singularity



Automatic mesh refinement

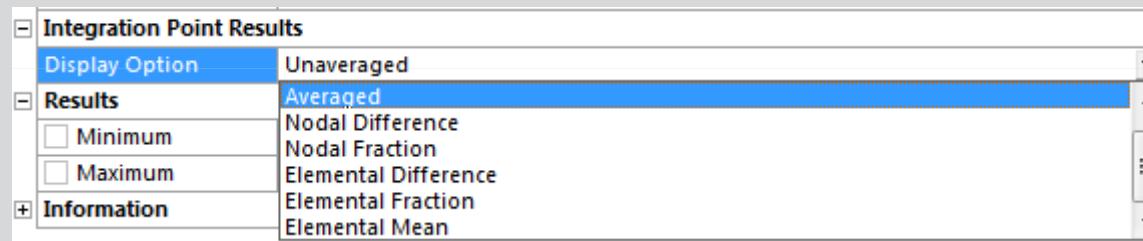
Scope result to region without singularity to obtain converged solution



Contour averaging options

Mechanical default is to display “averaged” nodal results

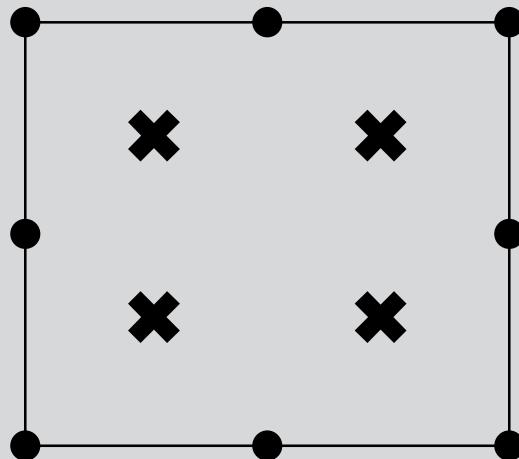
Other options can be used to understand how your mesh
is affecting your results



Contour averaging options

Typical 8-noded 2D solid element (PLANE183)

**Uses four integration points (where derived quantities
are calculated)**



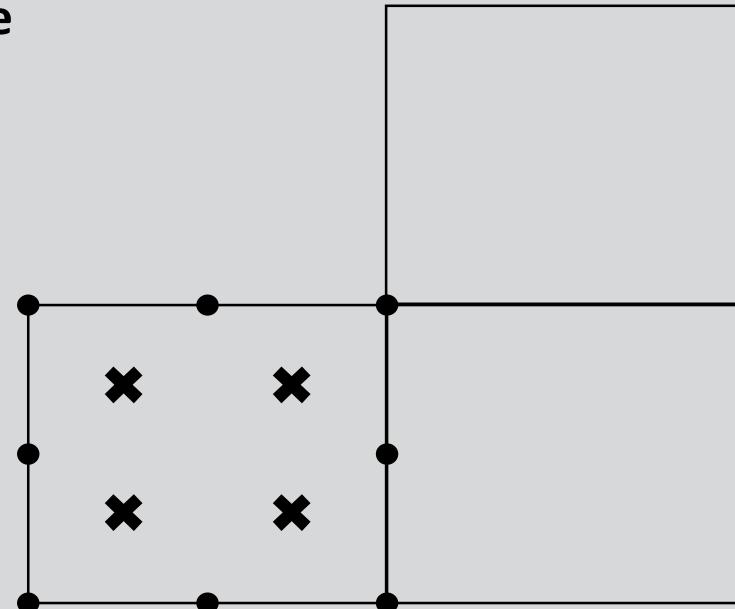
Contour averaging options

Unaveraged results – results are linearly interpolated from the integration points to the nodes

- Discontinuous contours

Averaged results – results are linearly interpolated from the integration points to the nodes. Any node that is shared between more than one element has its results averaged at the node

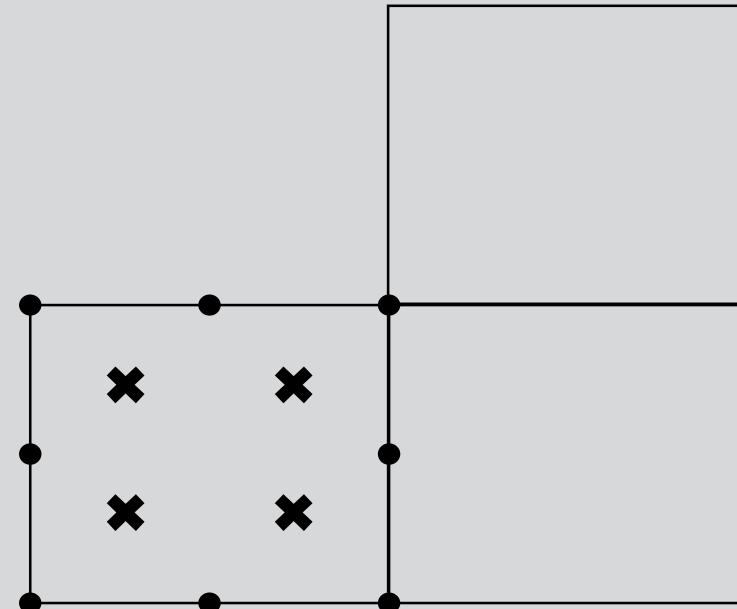
- Continuous contours



Contour averaging options

Nodal difference – maximum difference between the unaveraged results for all elements that share a particular node

Nodal fraction – ratio of the nodal difference and the nodal average

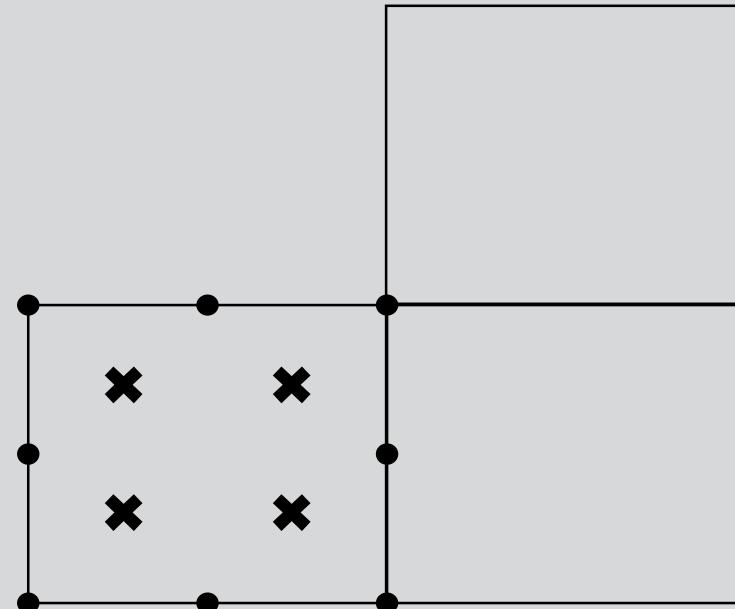


Contour averaging options

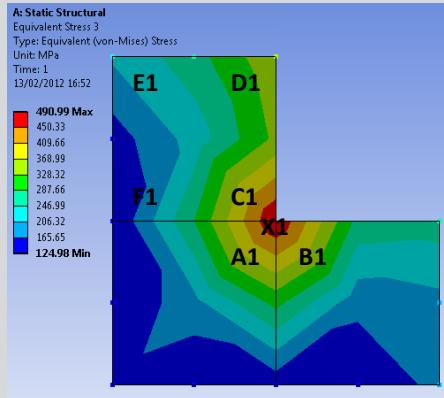
Elemental difference – maximum difference between the unaveraged results for all nodes in an element

Elemental fraction – ratio of the elemental difference and the elemental mean

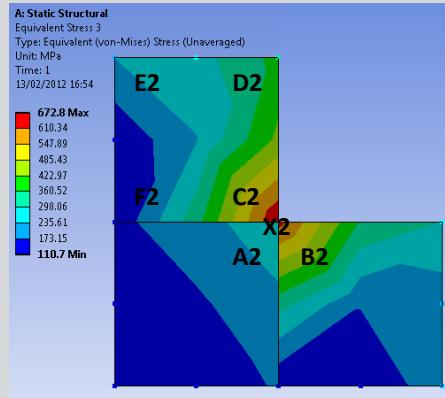
Elemental mean – elemental average from the averaged component results



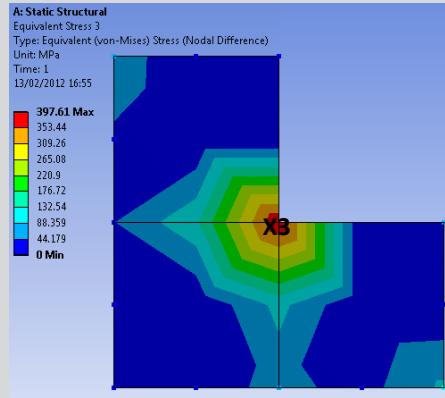
Contour averaging options



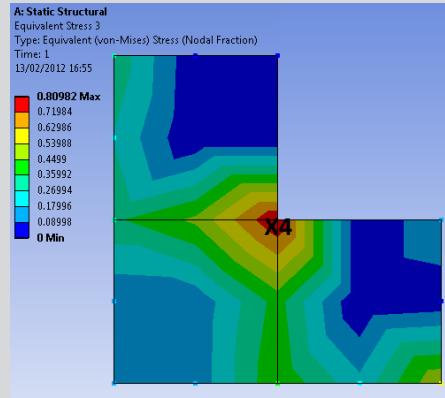
Averaged
 $X1 = (A1+B1+C1)/3$



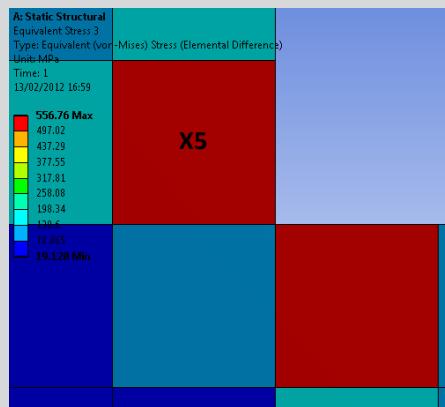
Unaveraged



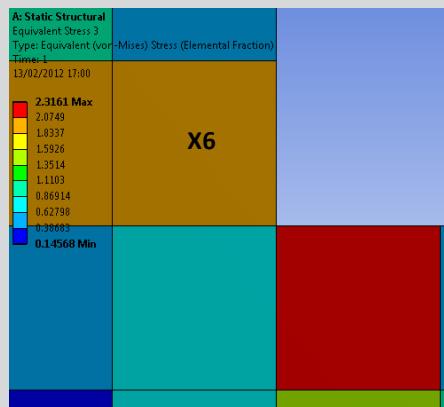
Nodal difference
 $X3 = (C2-A2)$



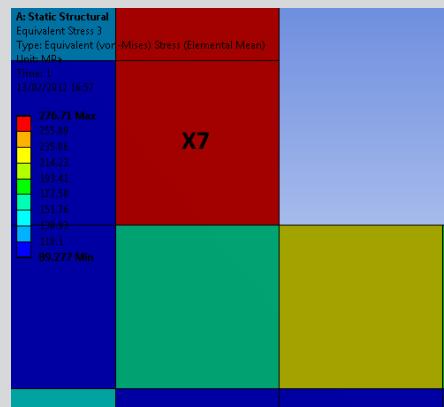
Nodal fraction
 $X4 = X3/X1$



Elemental difference
 $X5 = C2-F2$

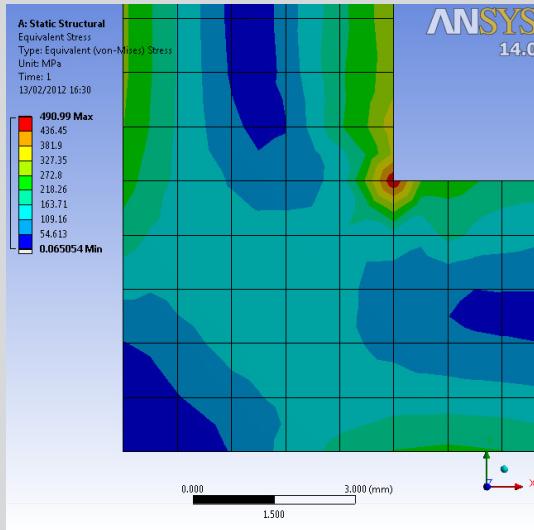


Elemental fraction
 $X6 = X5/X7$

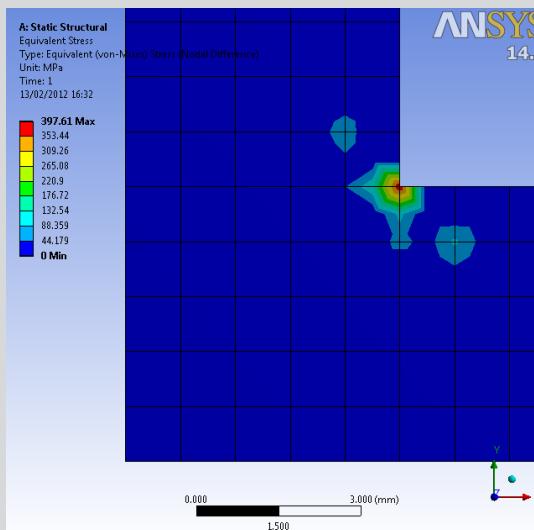
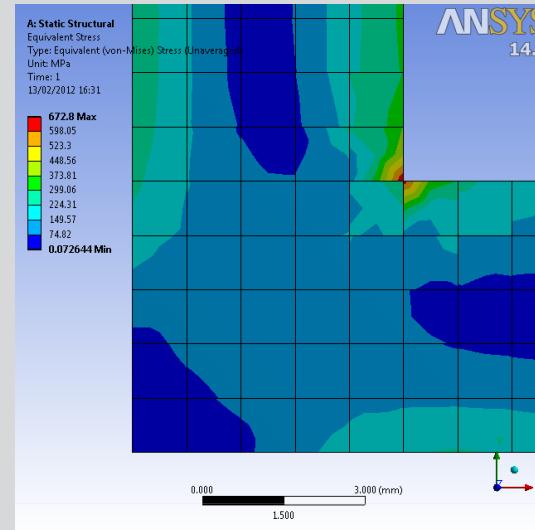


Elemental mean
 $X7 = (C1+D1+E1+F1)/4$

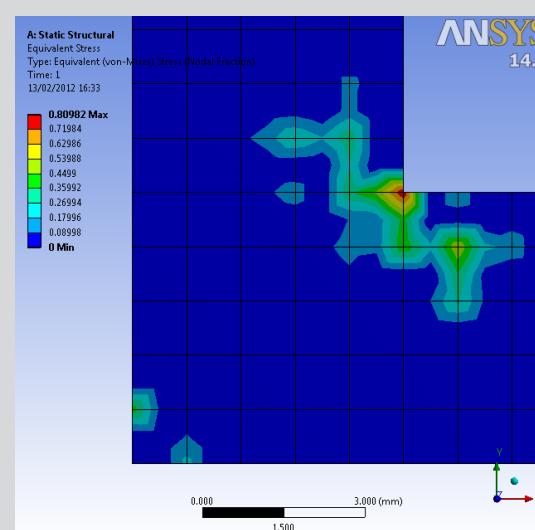
Contour averaging options



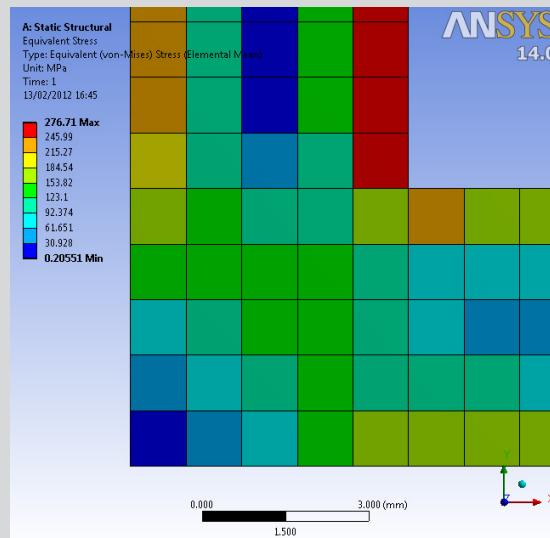
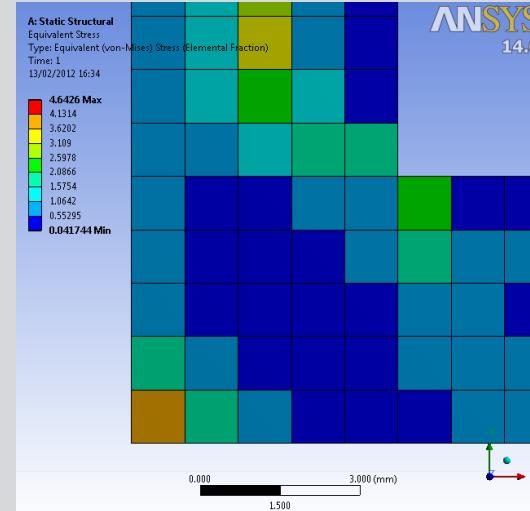
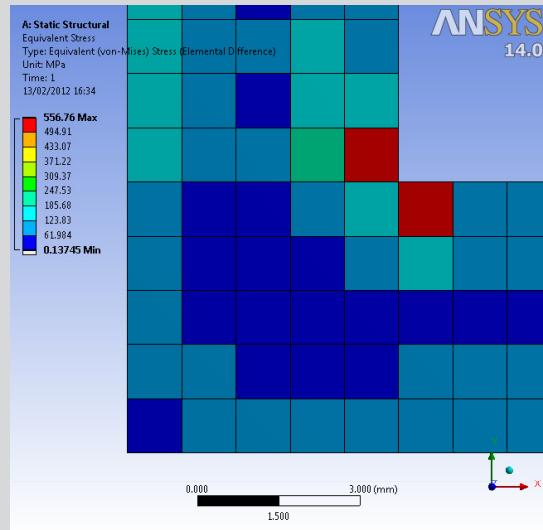
Large differences indicate areas of concern



Large values indicate areas of concern



Contour averaging options



Comparing results

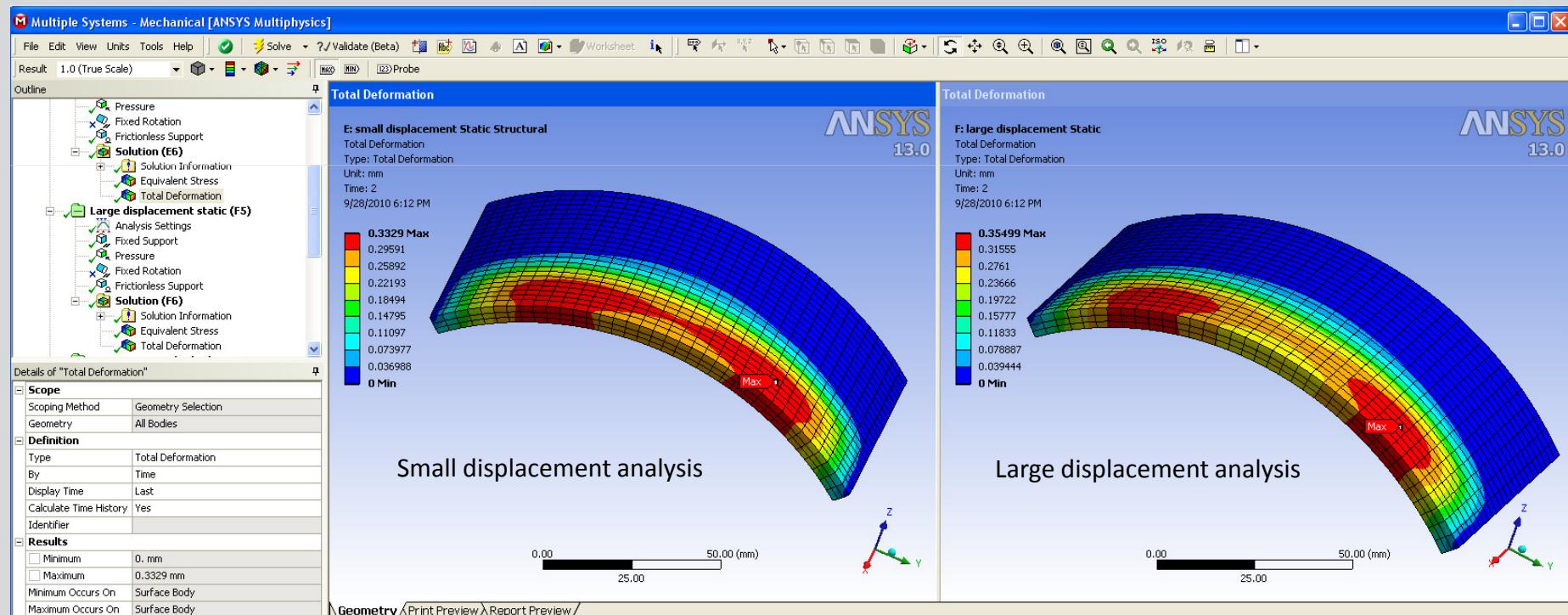
Linear solves can be significantly quicker than nonlinear solves.

Non-linearities in solves can be introduced by a number of factors.

- Materials
- Contact
- Geometry (large deformation effects)

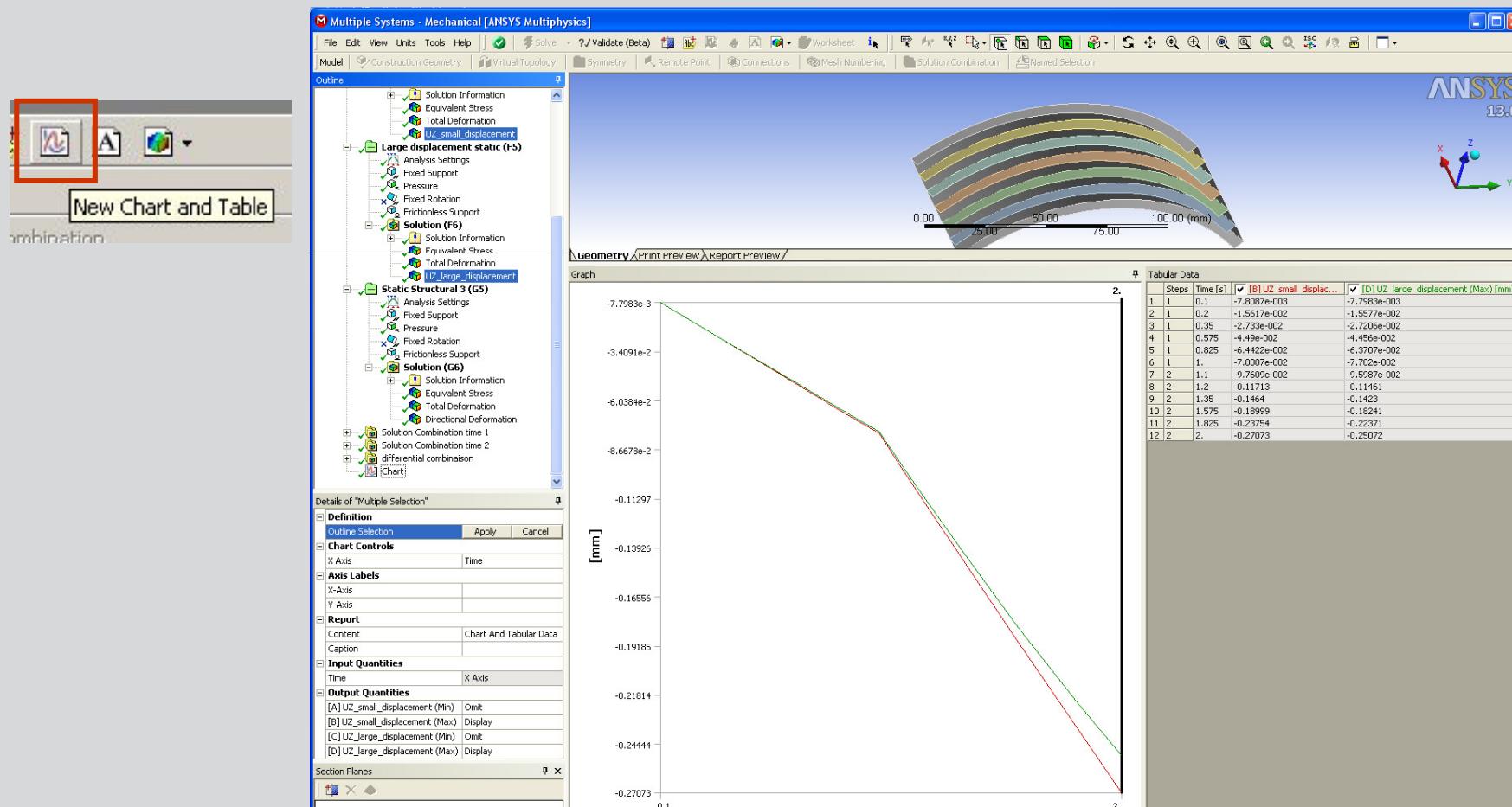
Comparing results

- The usual way to compare results from each analysis environment is to use two or more viewports and plot the variable from each environment in each view port



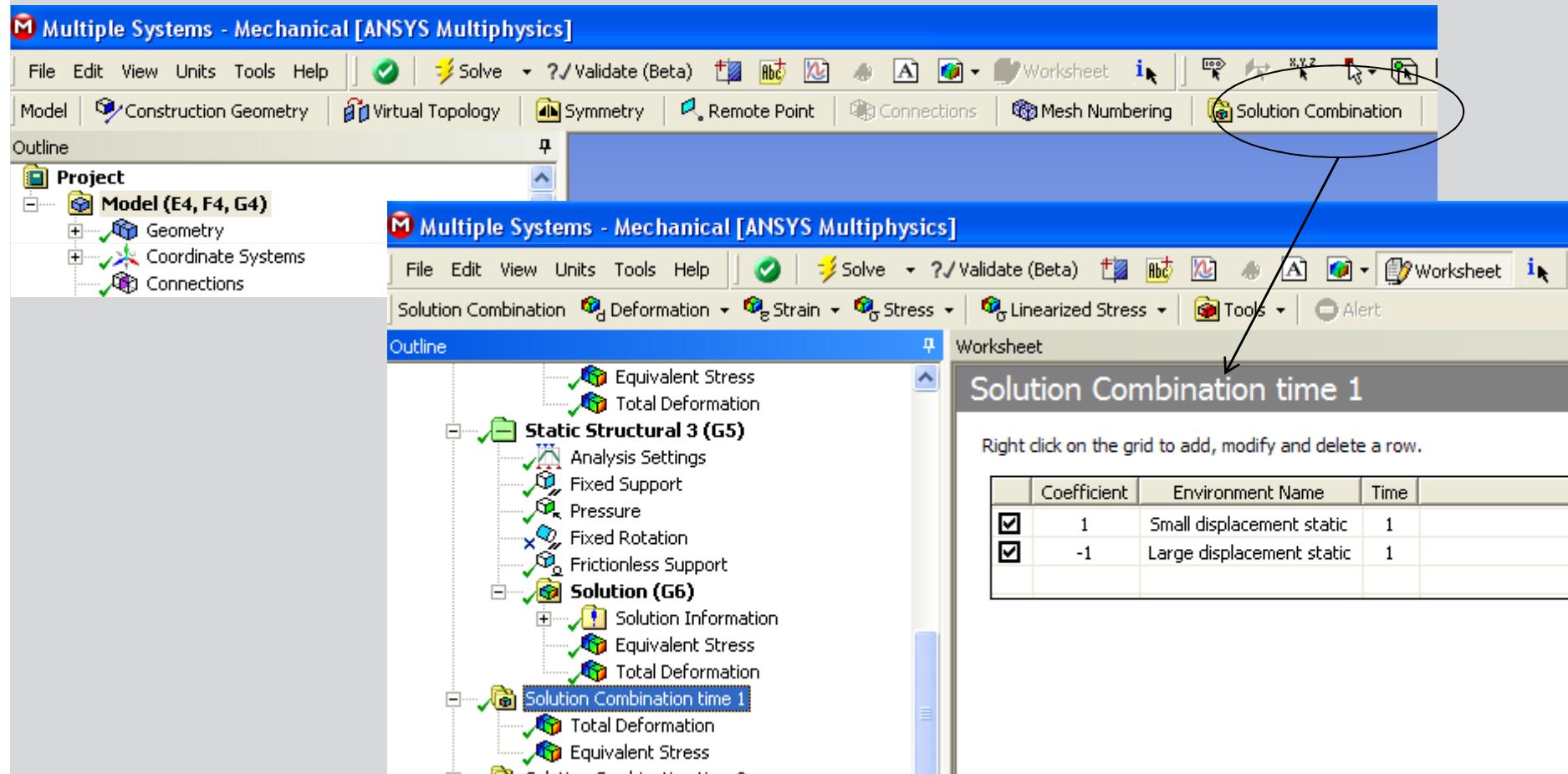
Comparing results

- Alternatively there is the Chart facility to plot solution variables at a vertex using the results from each environment



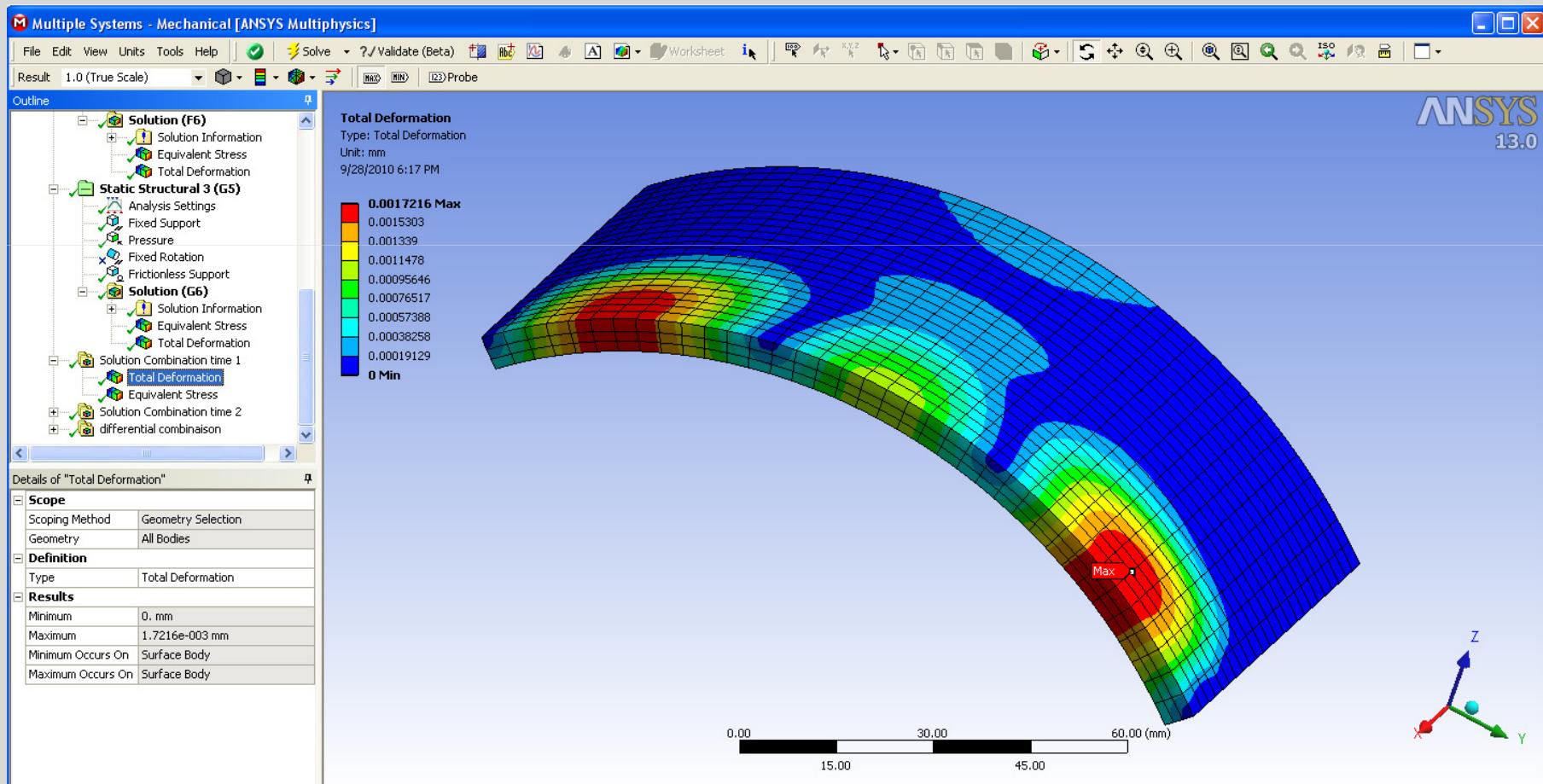
Comparing results

- However it is useful to compare results between (linear) analysis types more directly and this can be done using ‘Solution Combination’.



Comparing results

- Here the ‘Solution Combination’ has been used to combine the total deformations of the two analyses and plot the resulting difference



Summary

Mechanical contains many tools that can help you assess the quality of your mesh

- **Structural error result identifies where more elements should be invested**
- **Singularities can be dealt with during postprocessing – it is not always necessary to remove them**
- **Automatic mesh refinement can automate your mesh convergence studies**
- **Contour averaging options allow detailed assessment of the mesh influence on derived results**
- **Solution combination can show differences in results**

Thank you



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

**Please leave your feedback forms at the
registration desk**