

FEEG3001 FEA in Solid Mechanics

Lab 1: Introduction to FEA Labs & ANSYS

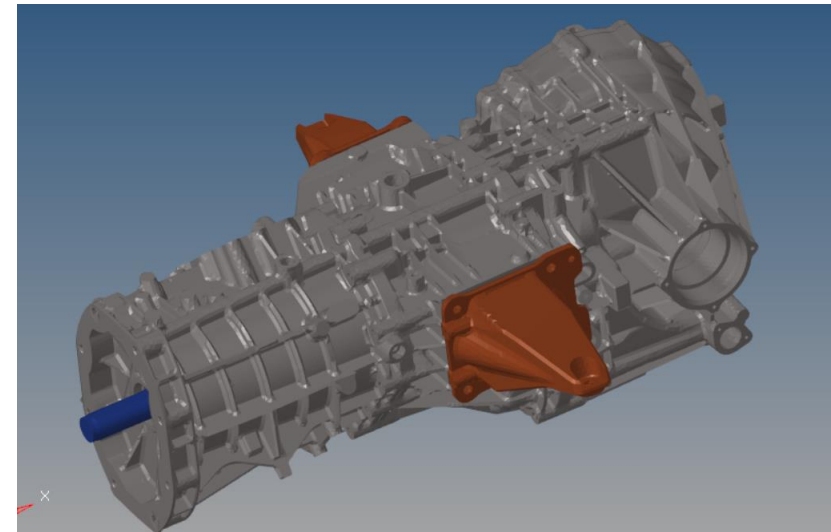
Prof A S Dickinson 2023

Part 1:

Overview

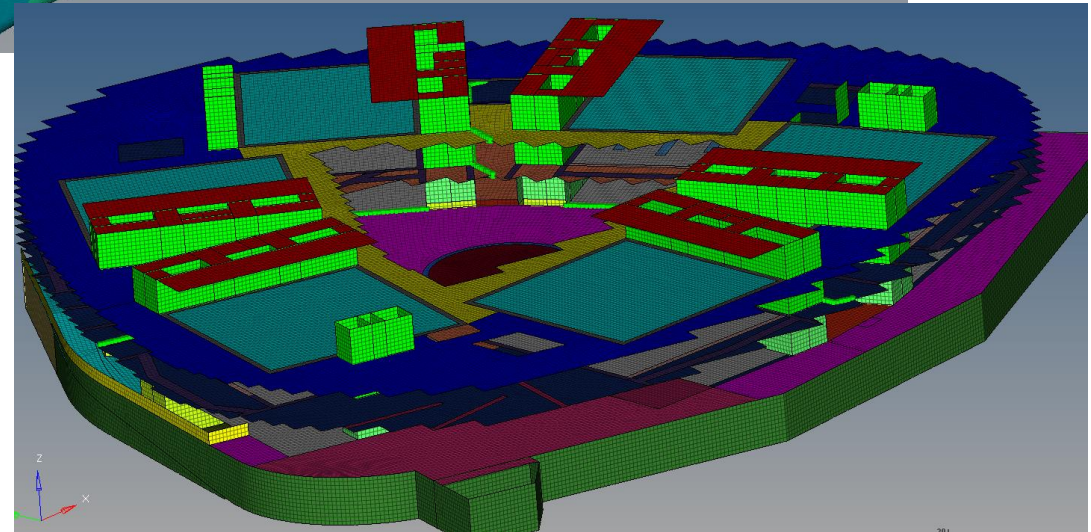
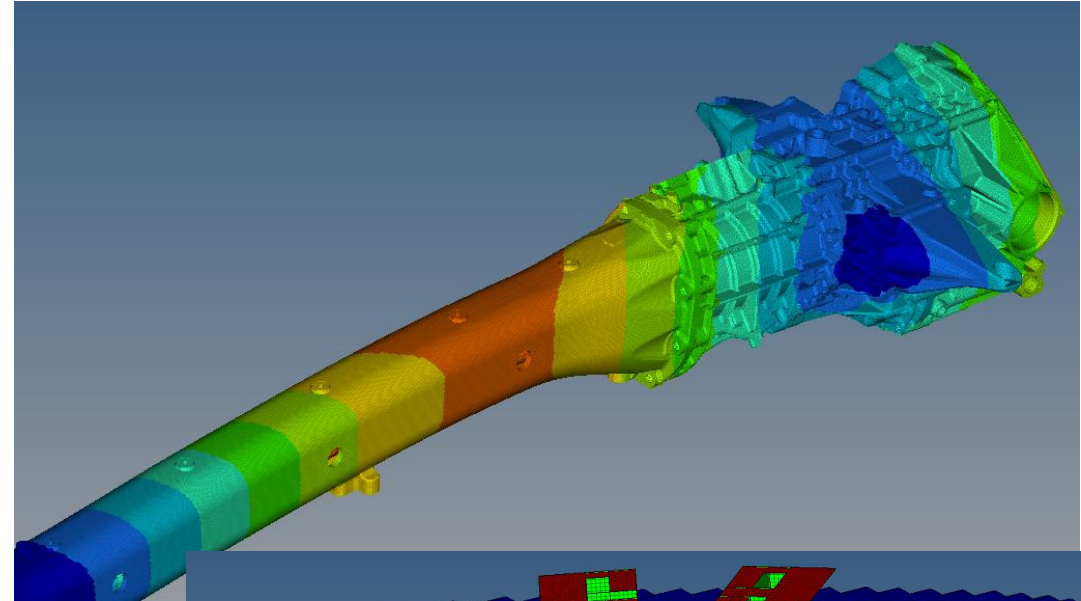
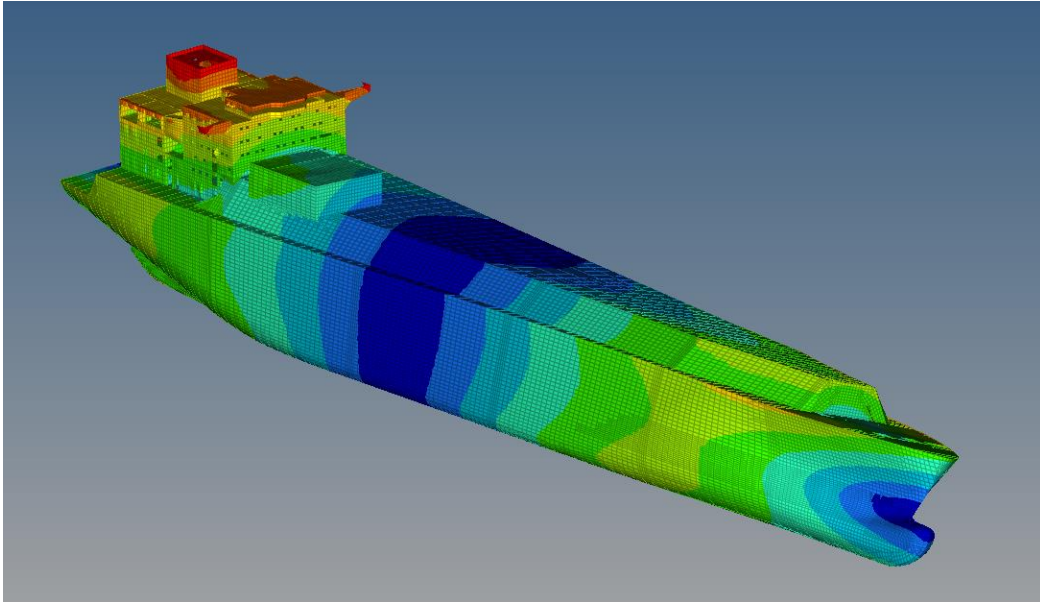
Why use Finite Element Analysis?

- Analytical Methods work well for ideal structures
- Intractable for complex structures



Typical Applications:

- 200,000 nodes, 400,000 elements



Finite Elements:

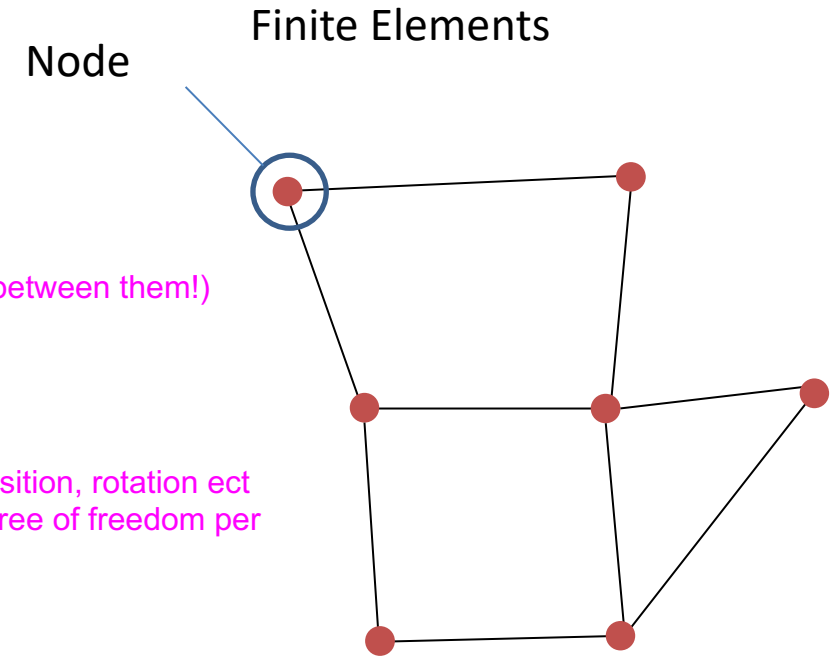
- Represent a discrete portion of a larger structure
- Response calculated at the nodes
- A 'Shape Function' defines elemental results, based on the nodal response

Data can only be extracted at nodes (of course we can interpolate between them!)

Nodes:

- Degrees of freedom (DoFs) defined at nodes
- Different Elements types have different DoFs
 - Displacement ($U_x, U_y, U_z, R_x, R_y, R_z$)
 - Temperature
 - Many more...
- Adjacent Elements Share DoFs

Nodes could have: position, rotation ect
Each adds a new degree of freedom per node





Typical Elements (ANSYS examples):

- 1D elements are long: width < 10% of length

Target Geometry

1D - Line

Typical Elements

BEAM Elements	Pictorials
BEAM188 Structural 3-D 2-Node Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
BEAM189 Structural 3-D 3-Node Beam 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	

Typical Elements (ANSYS examples):



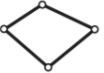

- 1D elements are long: width < 10% of length
- 2D elements are thin: thickness < 10% of edges

Target Geometry

1D - Line

2D - Surface

Typical Elements

BEAM Elements	Pictorials
BEAM188 Structural 3-D 2-Node Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
BEAM189 Structural 3-D 3-Node Beam 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
SHELL Elements	Pictorials
SHELL181 4-Node Structural Shell 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
SHELL281 8-Node Structural Shell 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	

Typical Elements (ANSYS examples):

- 1D elements are long: width < 10% of length

Target Geometry

1D - Line









- 2D elements are thin: thickness < 10% of edges

2D - Surface

- 3D elements are solid: where no dimension is small

3D - Solid

Typical Elements

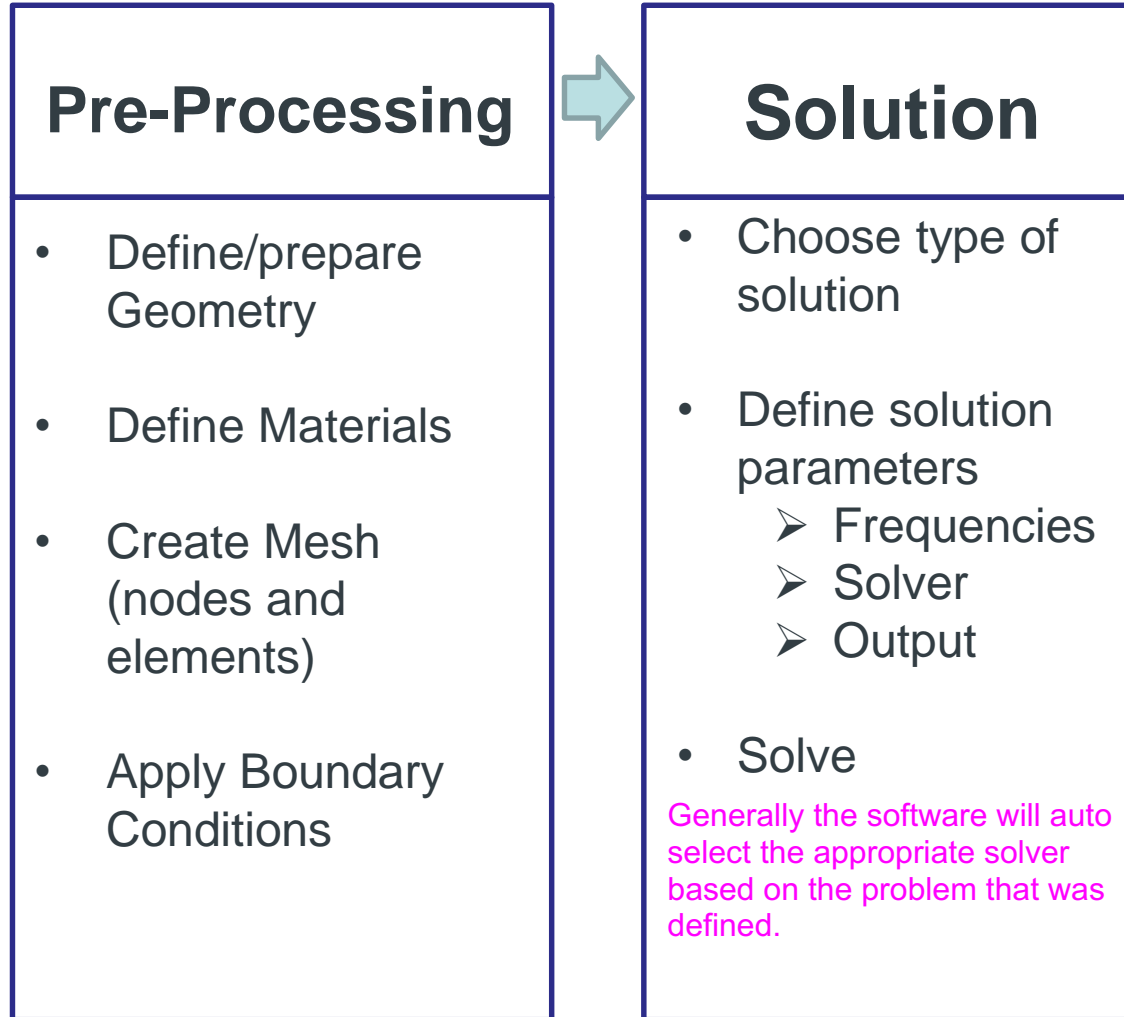
BEAM Elements	Pictorials
BEAM188 Structural 3-D 2-Node Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
BEAM189 Structural 3-D 3-Node Beam 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
SHELL Elements	Pictorials
SHELL181 4-Node Structural Shell 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
SHELL281 8-Node Structural Shell 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
SOLID Elements	Pictorials
SOLID168 Explicit 3-D 10-Node Tetrahedral Structural Solid 10 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ	
SOLID185 3-D 8-Node Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ	
SOLID186 3-D 20-Node Structural Solid 20 nodes 3-D space DOF: UX, UY, UZ	
SOLID187 3-D 10-Node Tetrahedral Structural Solid 10 nodes 3-D space DOF: UX, UY, UZ	

Tasks involved in running an FEA:

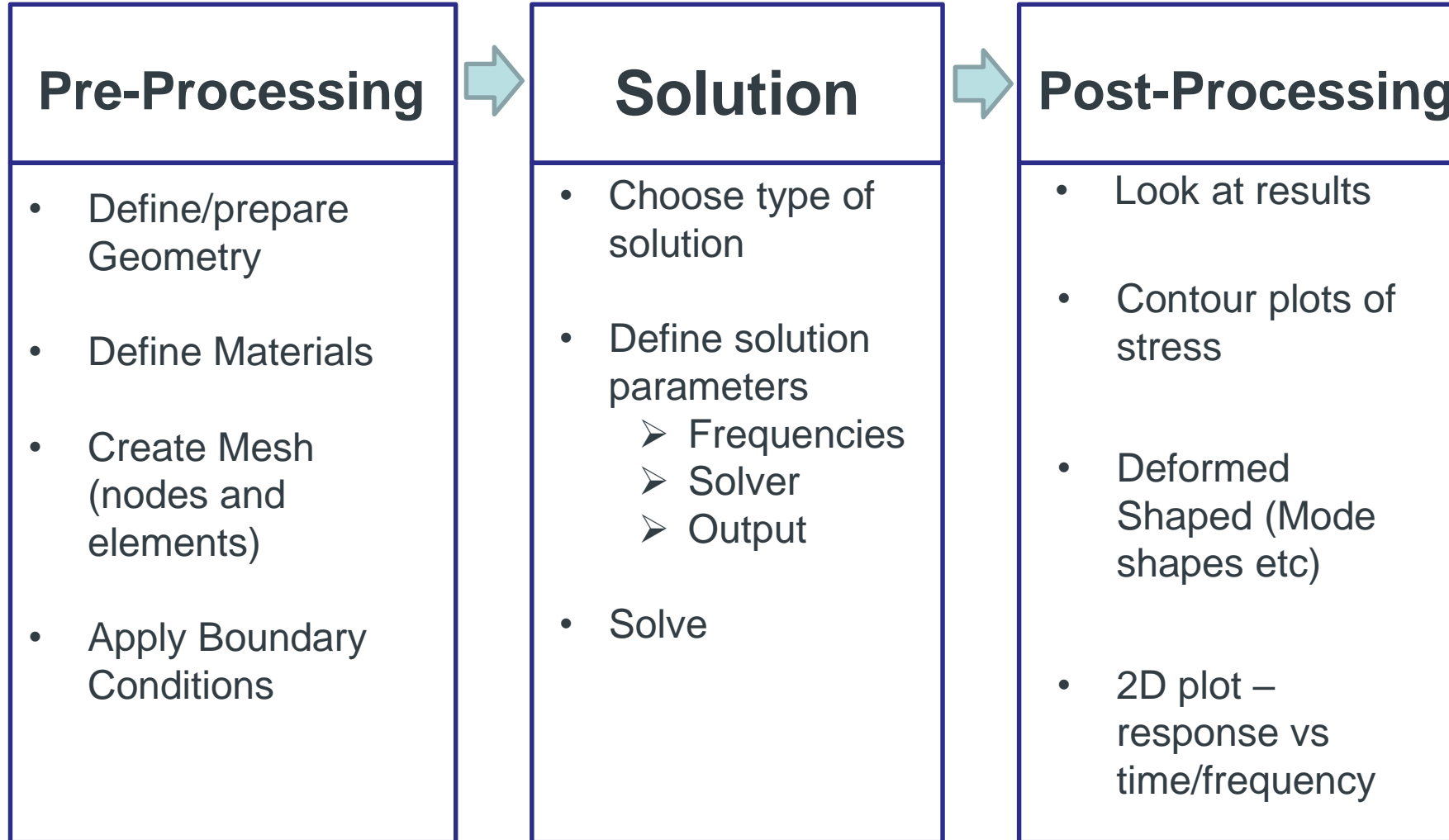
Pre-Processing

- Define/prepare Geometry
- Define Materials
- Create Mesh (nodes and elements)
- Apply Boundary Conditions

Tasks involved in running an FEA:



Tasks involved in running an FEA:



Many commercial FE packages including:

www.abaqus.com

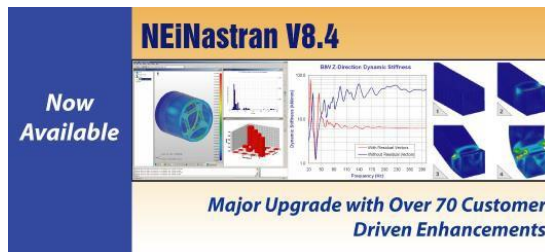


www.mscsoftware.com



MSC Nastran MSC Patran

www.neinastran.com

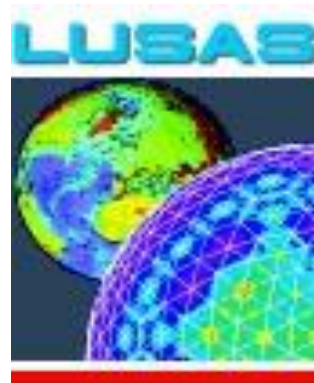


Ansys

www.ansys.com



www.adina.com



PAM-CRASH

get it right®

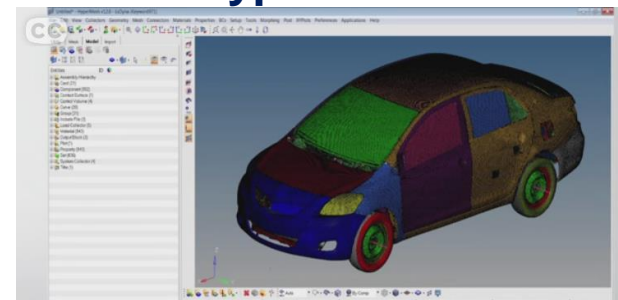
<http://www.esi-group.com/>

LS-DYNA



<http://www.lstc.com/>

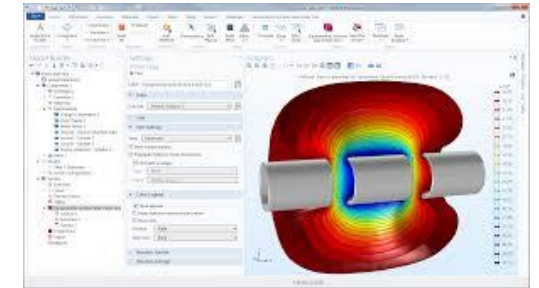
HyperWorks



<http://www.altair.com/>

UNIVERSITY OF
Southampton

COMSOL



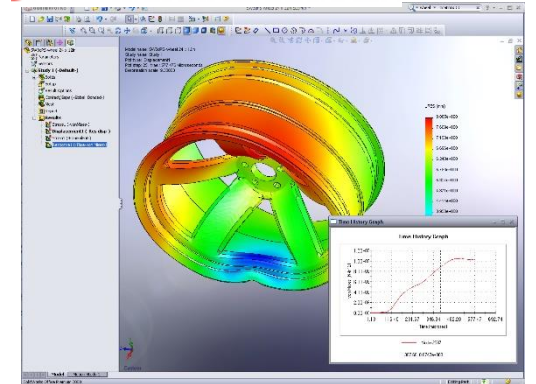
<http://www.comsol.com/>



FEBio Software Suite

<https://febio.org/>

SOLIDWORKS

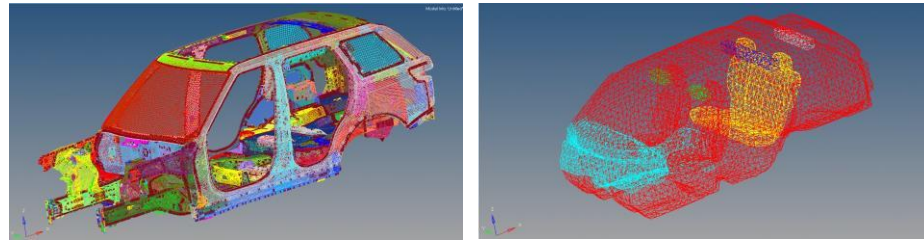


www.solidworks.com

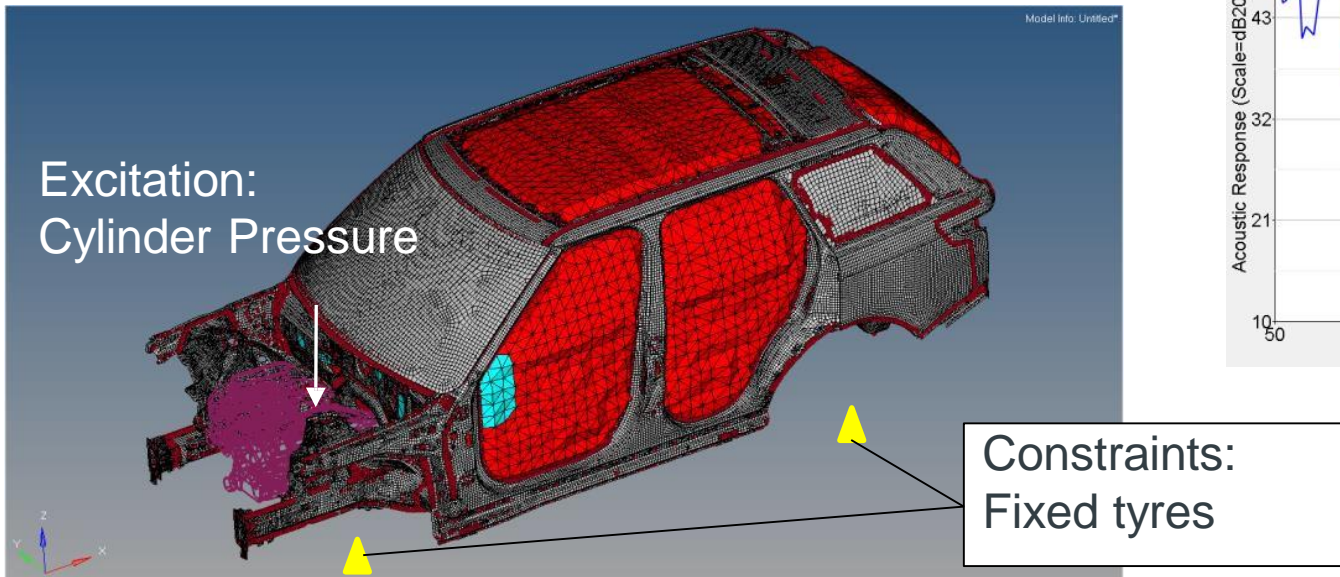
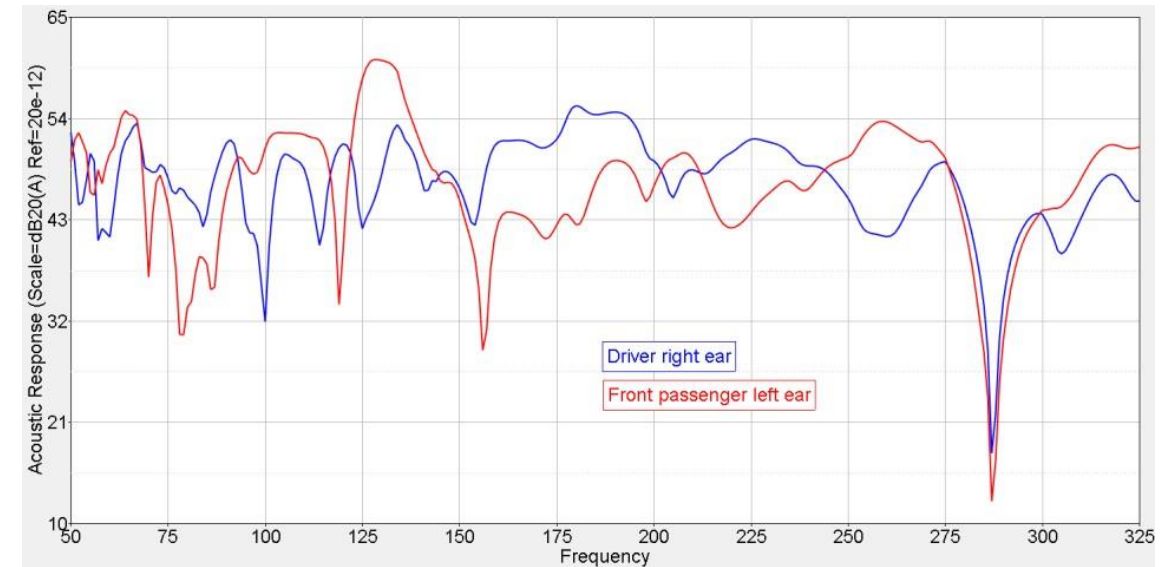
An Example you might not have thought of:

Here a acoustic model of the air inside in response to the engine is being modelled.

- Frequency Response Analysis for Acoustic Cavity:

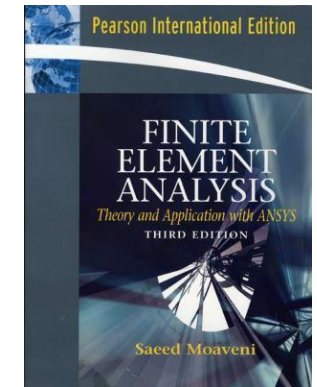
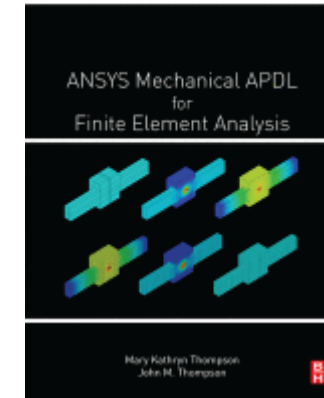


Results:
Acoustic Response - Sound pressure level



Resources:

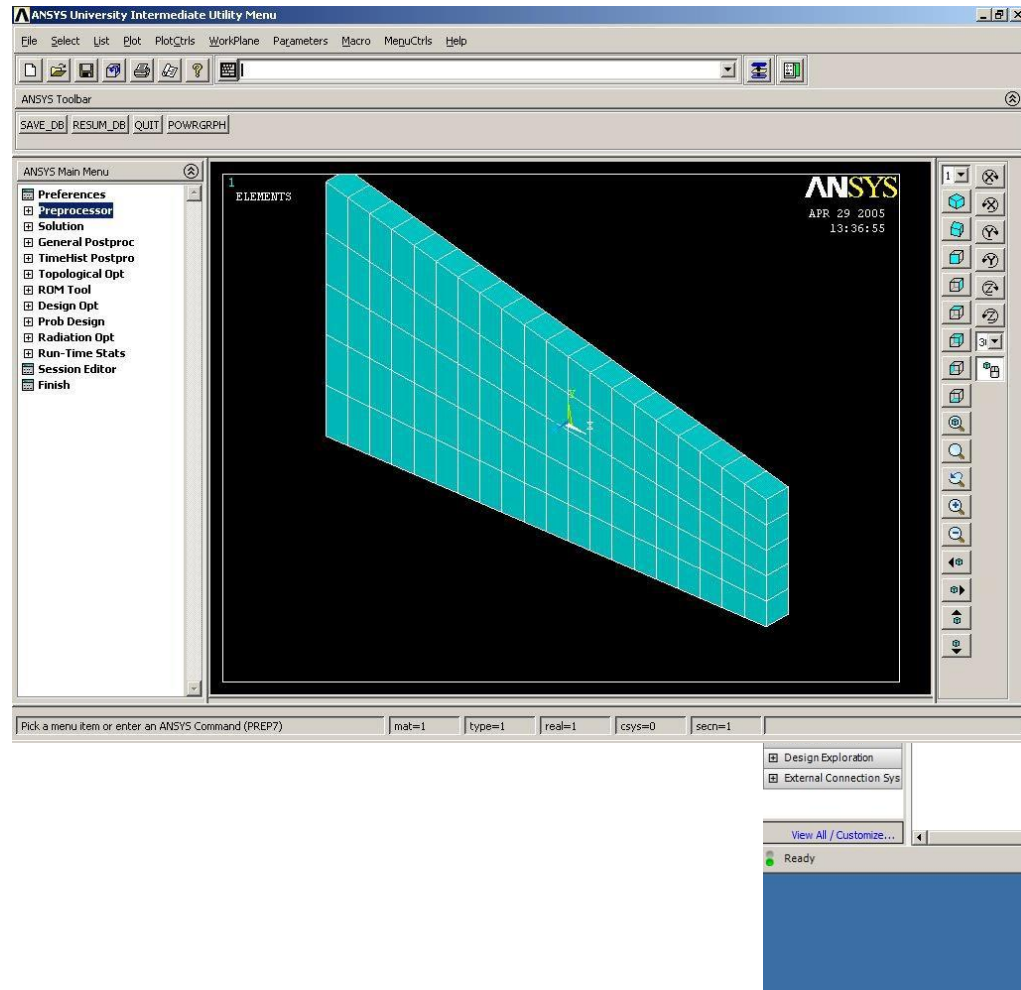
- Lots more background in FEEG2001 Systems Design and Computing notes
- SolidWorks Simulation tutorials
- M Thompson, *ANSYS Mechanical APDL for Finite Element Analysis* (e-book)
- S. Moaveni, *Finite element analysis : theory and application with ANSYS*, Pearson



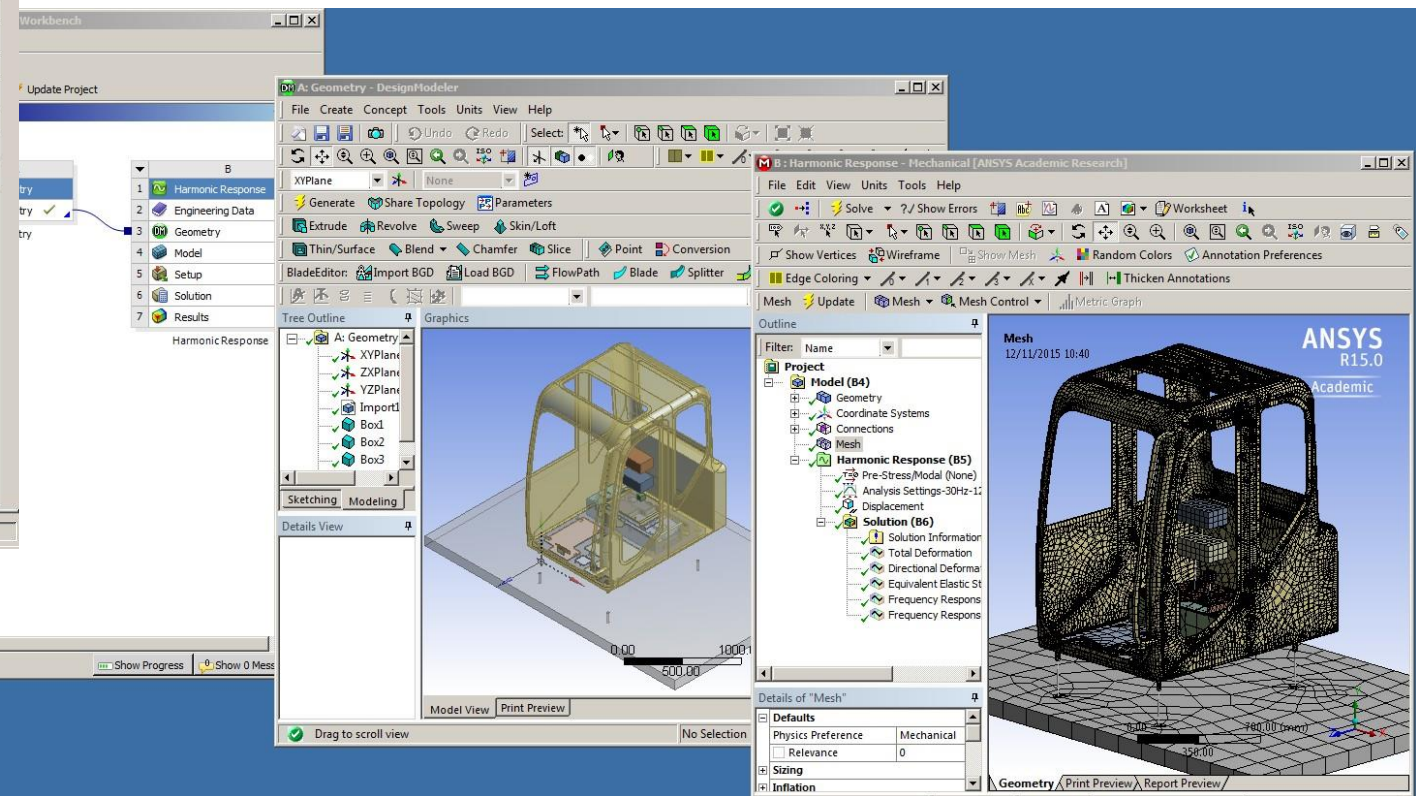
Part 2:

Introduction to ANSYS

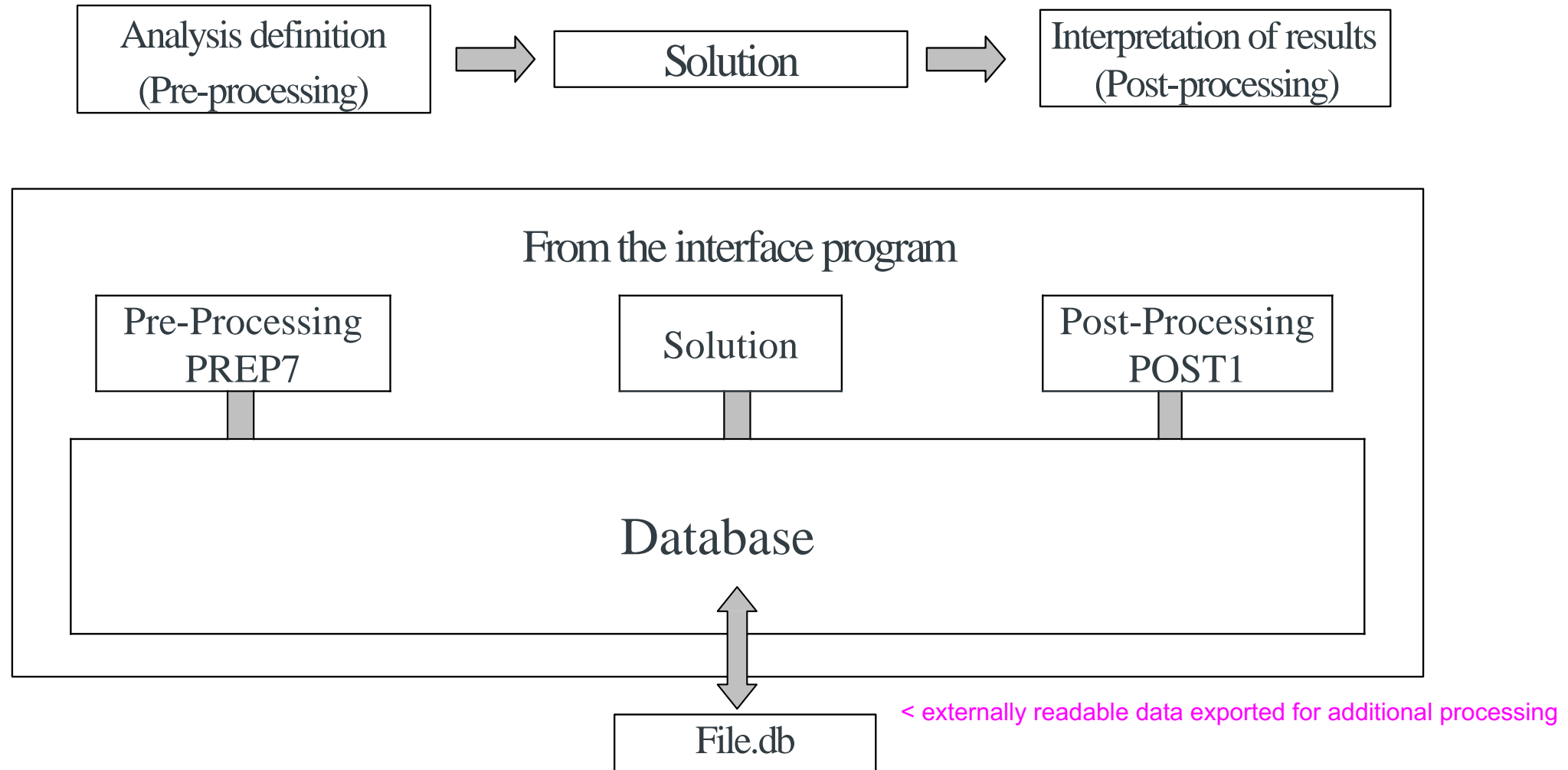
ANSYS 'Classic' (APDL) and Workbench



Workbench is a convenient wrapper for ansys, uses same solver



The ANSYS Data Structure



This week's activity:



<https://commons.wikimedia.org>



Report of the Investigation Into the Sinking of the “MOL Comfort” in the Indian Ocean, Bahamas Maritime Authority, Sept 2015

This Week's Activity:

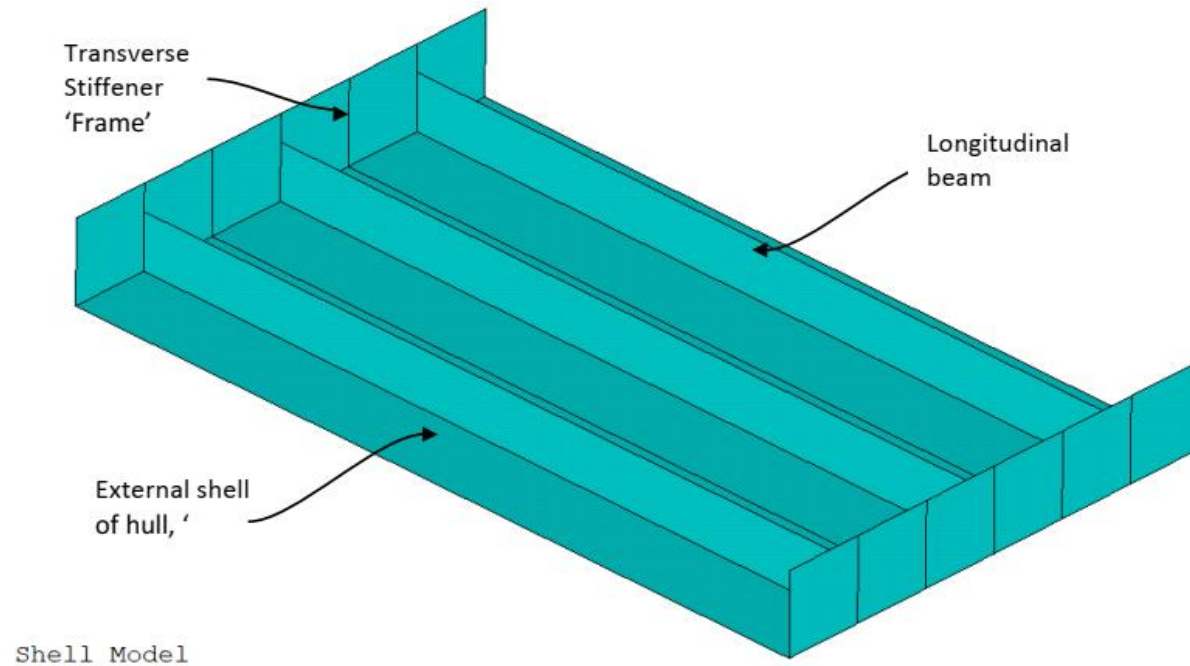
- Solid modelling
- Applying constraints and loads

Diagram shows the external skin with the internal ribs, similar to what was seen in aircraft structures

For our model we're flatteneing a part of the hull and then simulating it



dhwelding.com



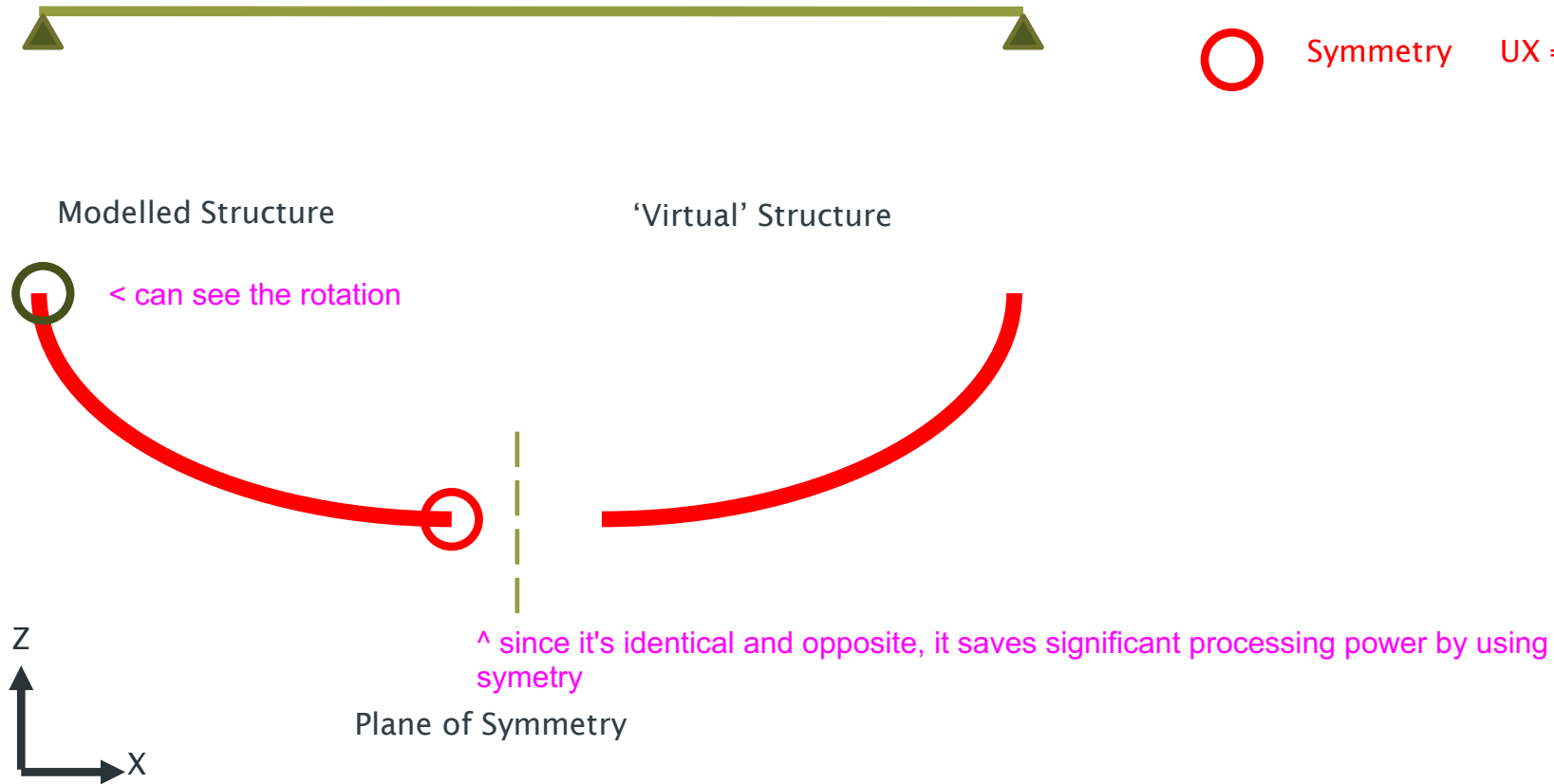
- Displacements
 - Applied to individual DoF at a specified node
- Forces / Moments
 - Applied to individual DoF at a specified node
- Pressure
 - Applied to Elements
- Temperature
- Gravity
- Inertia

- Constraints represent the boundary conditions of the structure
 - Simply supported, Encastre, Pin jointed etc
 - Sometimes represent the structure around the portion you are modelling
- There should be enough constraints to prevent any rigid body motion
 - Requires constraints for translation and rotation of the structure as a whole
- 3D model: nodes have up to 6 DoF
 - Translations UX, UY, UZ
 - Rotations ROTX, ROTY, ROTZ (for some elements)
- One or more of these can be constrained at each node

Symmetry

 Pin Joint $U_Z = 0$

 Symmetry $U_X = 0, R_{OTY} = 0$

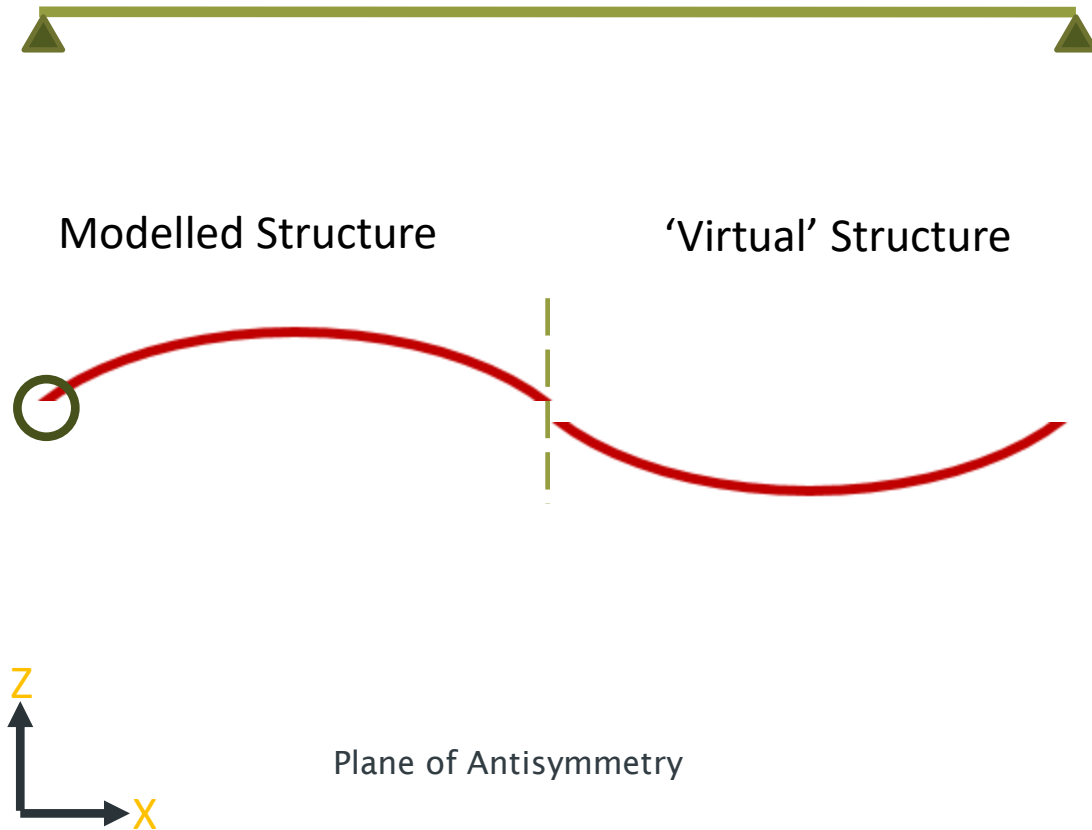


Antisymmetry

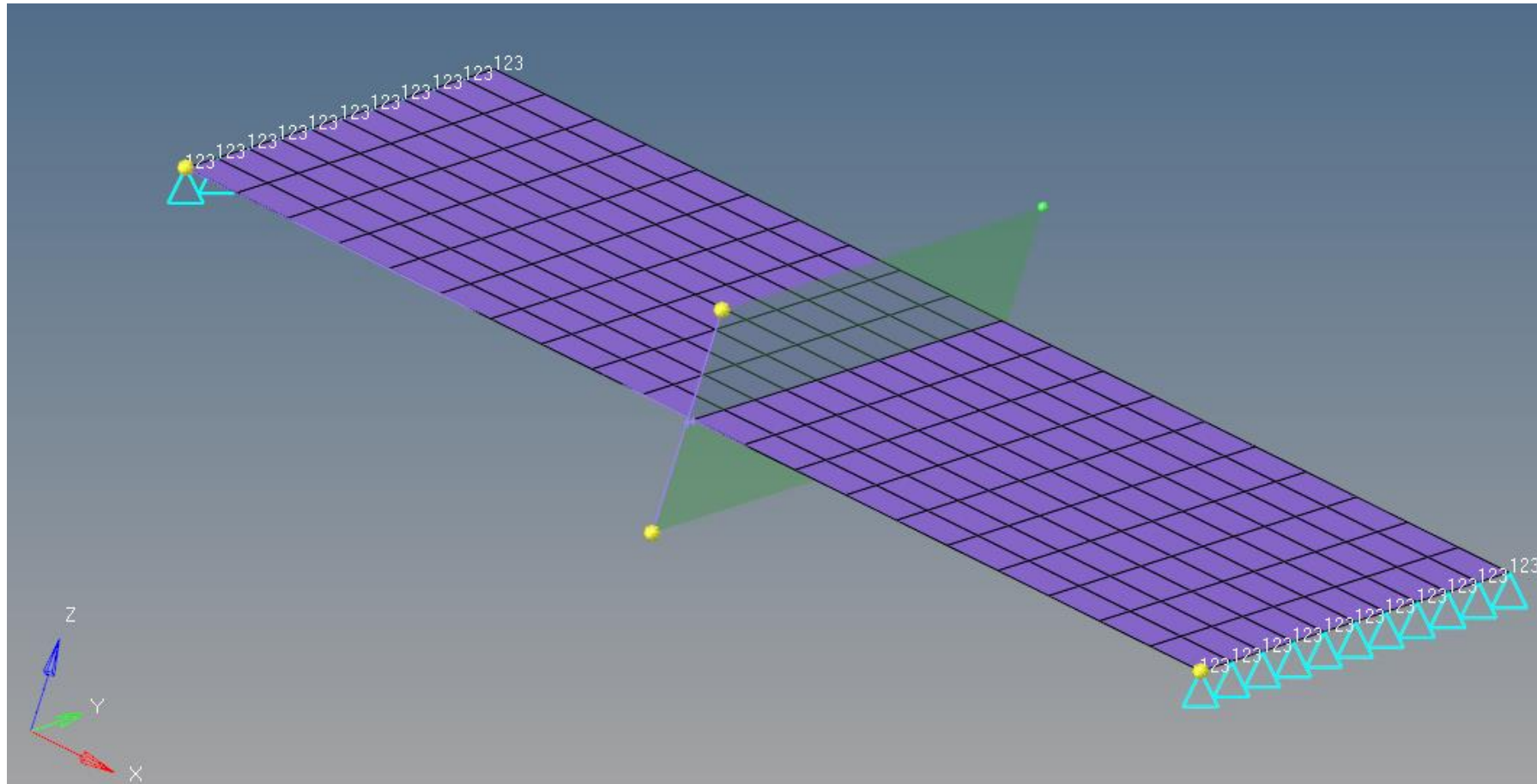
Here we're doing dynamics, clearly the previous symmetry thing would cause issues!

○ Pin Joint $U_Z = 0$

○ Antisymmetry $U_Z = 0$



3D Symmetry / Antisymmetry



Symmetry
 $UX = 0$
 $ROTY = 0$
 $ROTZ = 0$

Antisymmetry
 $ROTX = 0$
 $UY = 0$
 $UZ = 0$

Part 3:

Model Verification

Model Verification Analysis

- Validation: is my modelling physics correct?
- Verification: is my modelling maths correct?
- Many aspects of model verification:

Validation: use of comparing models to ensure correctness

- Many aspects of model verification, most of which don't need reporting:

- Is the geometry the right size?
- Are the correct materials applied, to the right regions?
- Is the model continuous, or feature connections?
- Is the model adequately constrained?
- Do the reaction forces equal the applied loads?
- ...

Increased mesh resolution, increases the numbers of DOF and hence becomes closer to reality.

- **But we always assess and report the FE mesh.** Why?
- Why does mesh size influence the model predictions?

Element Size:

Large elements:

- Small models – solve quickly
- May not track geometry well
- Can give distorted elements
- Are less accurate:
 - Artificially stiff
 - Dispersed stress



Element with large aspect ratio

Small elements:

- Large models – solve slowly
- Can track geometry well
- Better mesh quality
- Are more accurate:
 - More true stiffness
 - Localised stress



Element with aspect ratio close to 1.0

Element Size – Dynamics

Stuff about harmonics

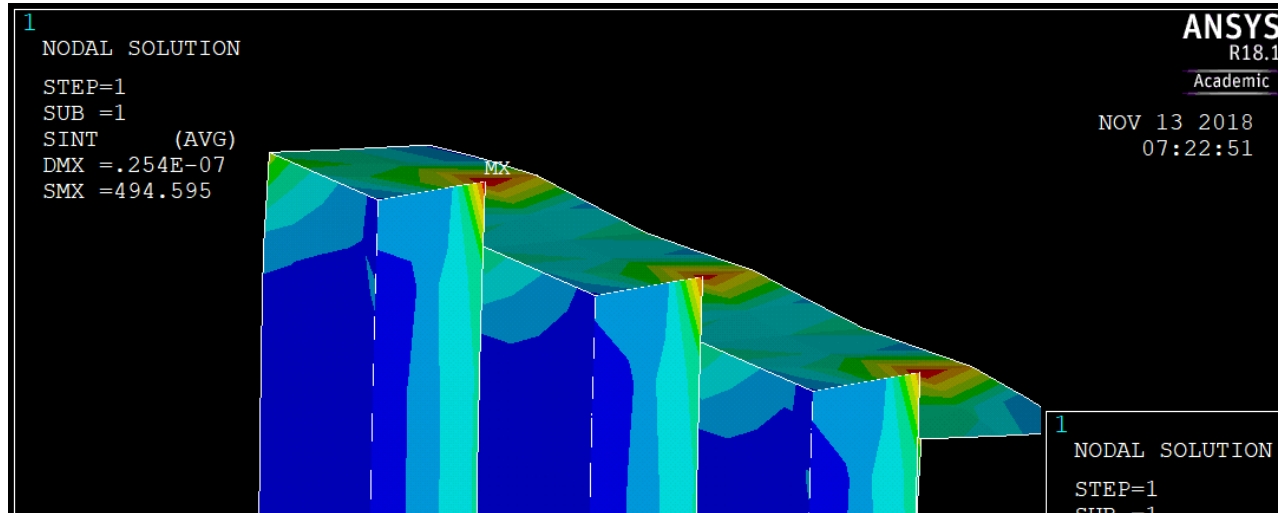
Too few nodes -> (incorrectly) high stiffness -> oscillates at too high a frequency (because it's too stiff)

- As the number of elements increases the natural frequencies tend to fall
- Finer mesh gives more accurate results for higher order modes
- Rule of thumb – 8 to 10 nodes per wavelength
- For dynamics solutions a uniform mesh is usually used
- Model size increases rapidly with mesh density

Simply supported beam

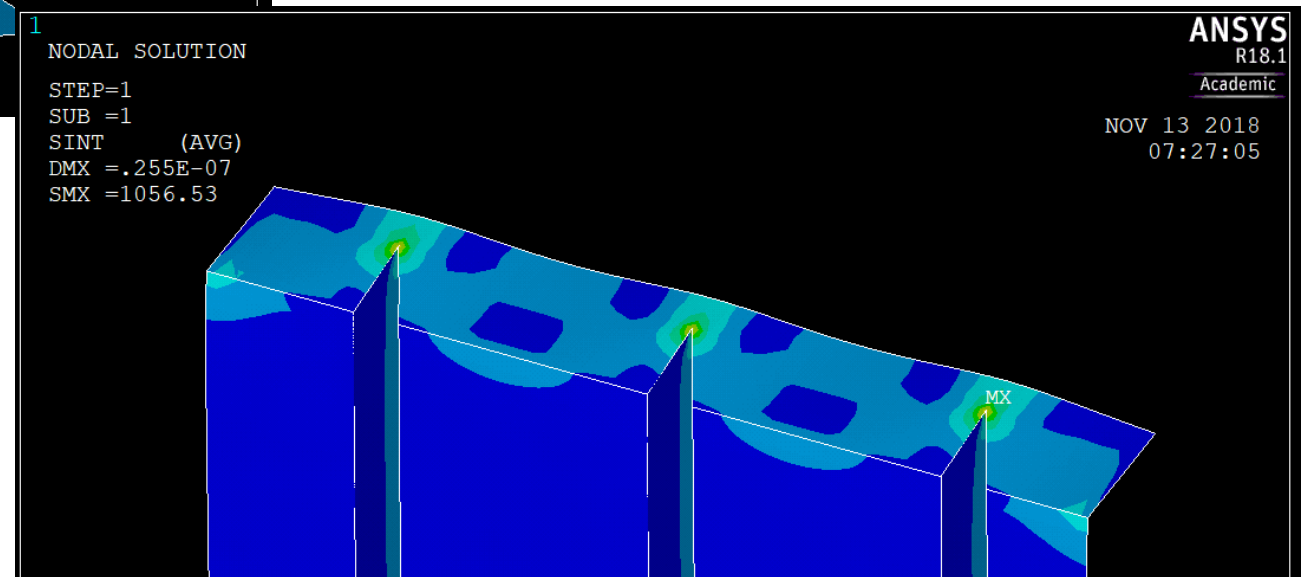
Mode	No. of Elements				
	2	3	4	6	10
1	10.74	10.70	10.70	10.70	10.70
2	47.36	43.20	42.80	42.70	42.68
3	118.50	106.10	97.30	96.00	95.65
4	131.60	129.80	129.10	128.70	128.40
5	214.60	196.00	187.50	170.90	169.20
6	459.80	347.50	295.60	268.90	263.00
7		424.40	407.33	394.80	376.80
8		474.50	462.70	414.80	388.50
9		769.90	683.00	554.90	510.60
10			739.90	687.60	658.10

Element Size – Stress Analysis



Finer meshes improve *stress gradient* prediction, offering greater *spatial resolution*

So, as element size decreases, stress becomes more localised and maximum levels increase



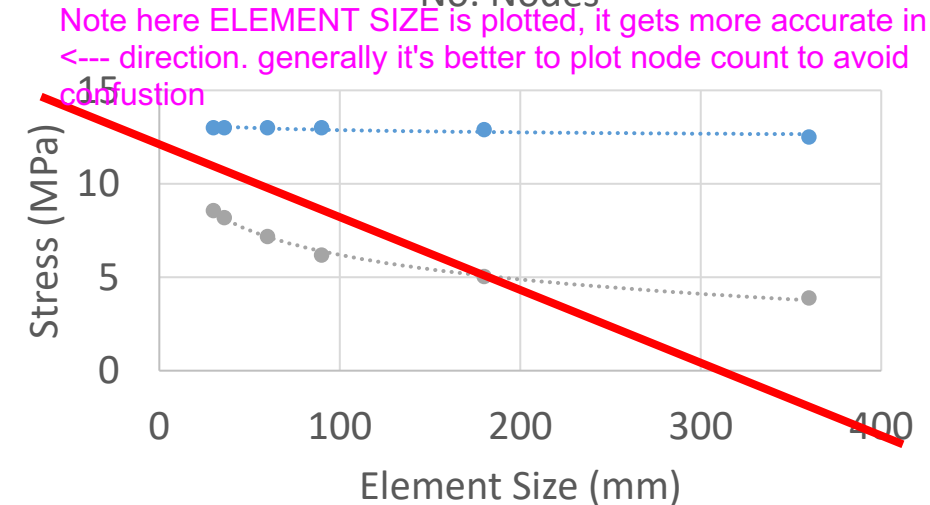
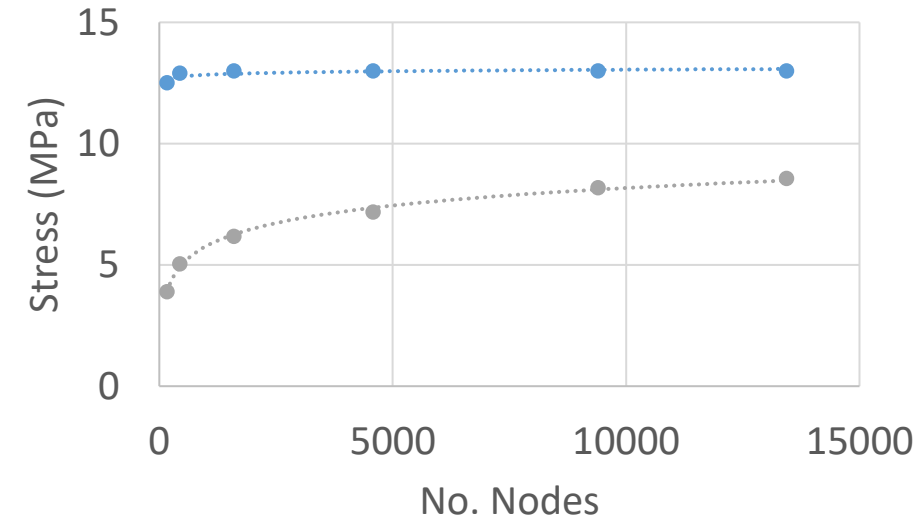
Element Size – Stress Analysis

Mesh convergence analysis, looking for at what mesh size do we no longer get significant changes in results!

We can see in the graph, that depending on where in the model we look, the required mesh to capture accurate behaviour changes!

- As element size decreases, normally:
 - peak stress increases
 - peak deflection increases
- Stress should converge on the ‘correct’ solution
- Simple: do the nodal and elemental results agree (~5%)?
- Thorough: **convergence analysis** – run a number of models with various element sizes – to find the correct size to use
- **Verify whichever result you plan to use**

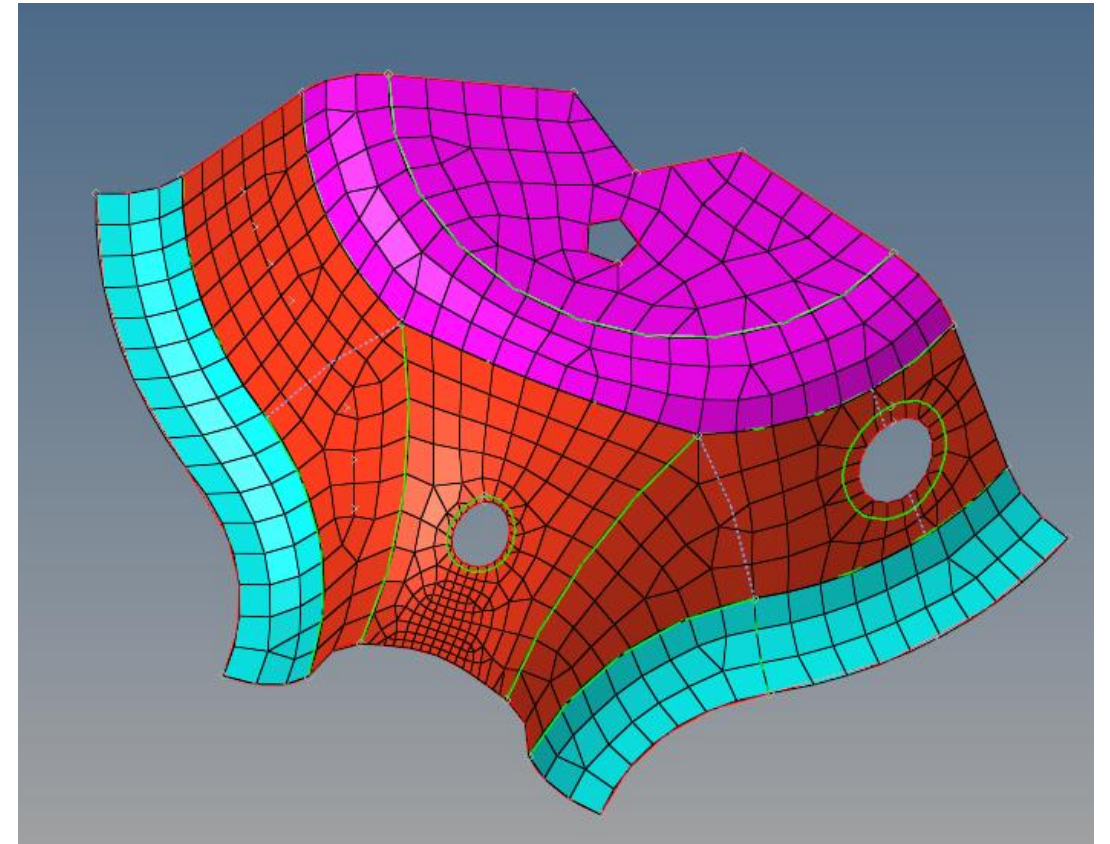
Your not intrested on convergence for every node, just convergence relevant to the result your intrested in!



• SMax.Span • SMax.Root

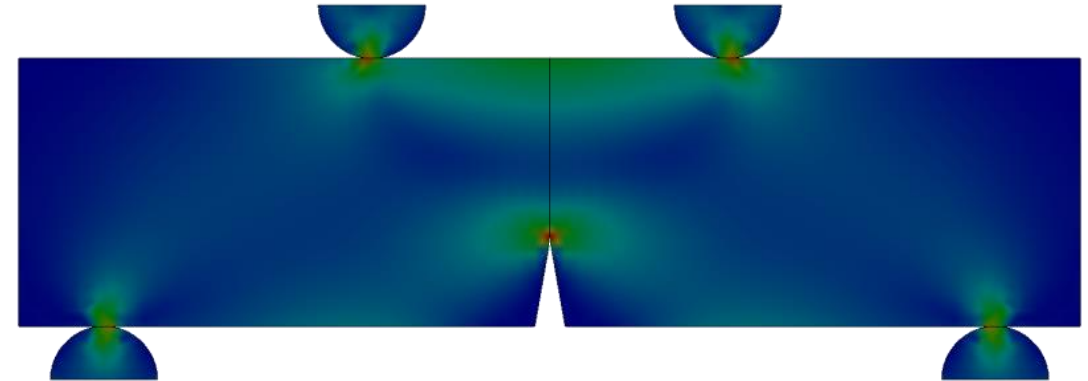
Element Size – Stress Analysis

- Different results will converge at different mesh sizes
- For stress analysis element size can vary significantly across model
- Localised refinement where stress levels are highest
- Very fine meshes can be run without excessive model size



When does mesh convergence fail?

- Be aware of stress singularities
- e.g. Discontinuities and sharp corners
- Theoretically infinite stress
- Mesh convergence will not occur
- Several options:
 - Assess: is it a boundary condition artefact, that I can ignore? (easy but risky)
 - Model failure processes... (can be difficult)
 - Accept ‘I need to change my design’ (also difficult, but better use of your time!)



What is coming next?

- This week we will build a Solid model of the ship hull section, and perform a mesh convergence analysis.
- Save your files periodically, with version history for re-starts.
- Next time we will look at two more solution types, and postprocessing

Thank You

UNIVERSITY OF
Southampton

