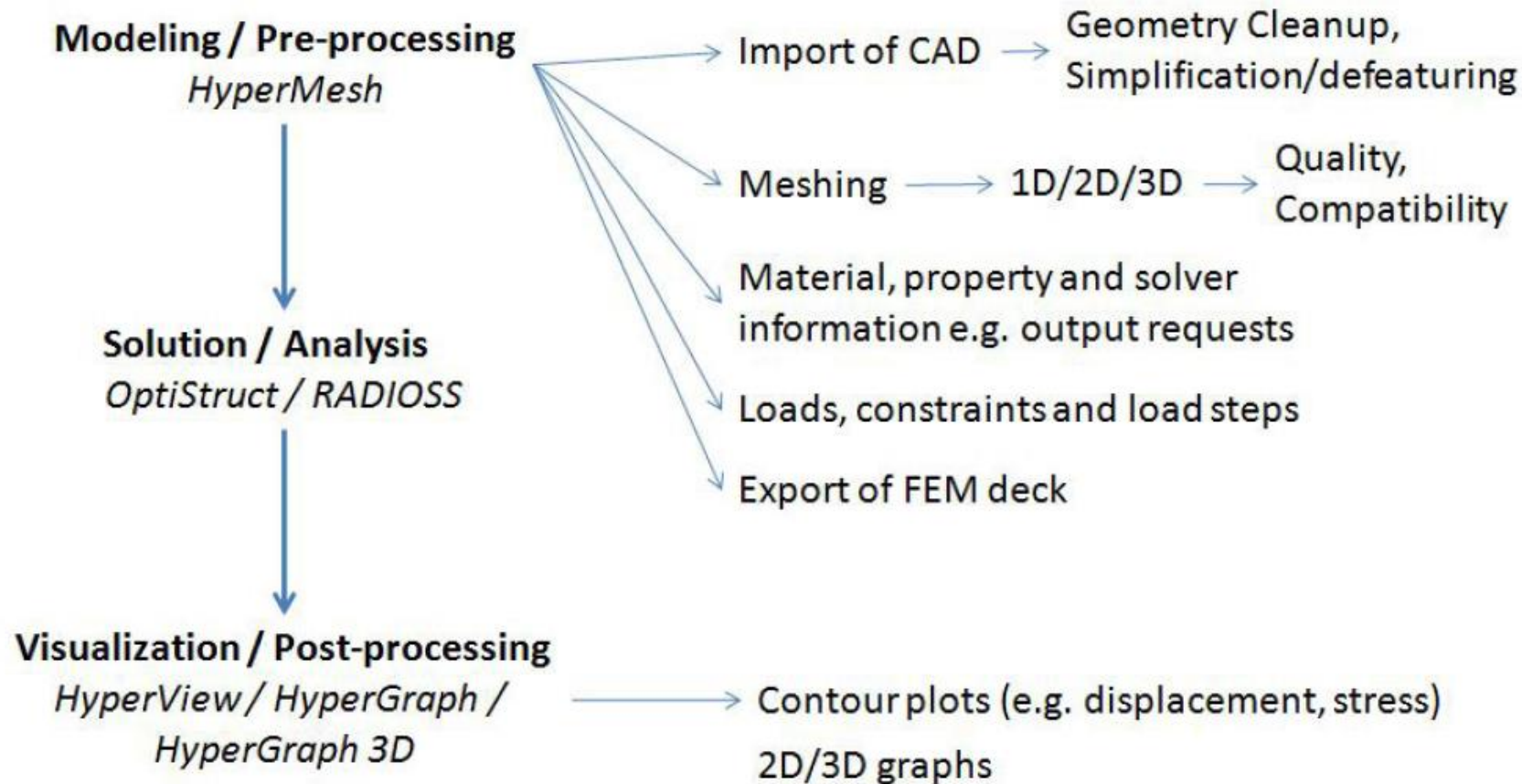


Creating a CAE FE model

Standard FEM Process



What are you trying to do?

- Is FEA appropriate/necessary? Eg. CFD?/Hand calcs?/ Physical testing?
- How much time do you have?
 - Accuracy/complexity v time
 - Is initial “coarse” model required?
 - Symmetry of model/constraints? Front v Rear crash?

“It is often less time consuming to build a large model by using an automatic mesher, and increased computer capacity has lessened the need to use special techniques to minimize the number of elements.”

- One model with many subcases v many models (Solve the stiffness matrix once!)

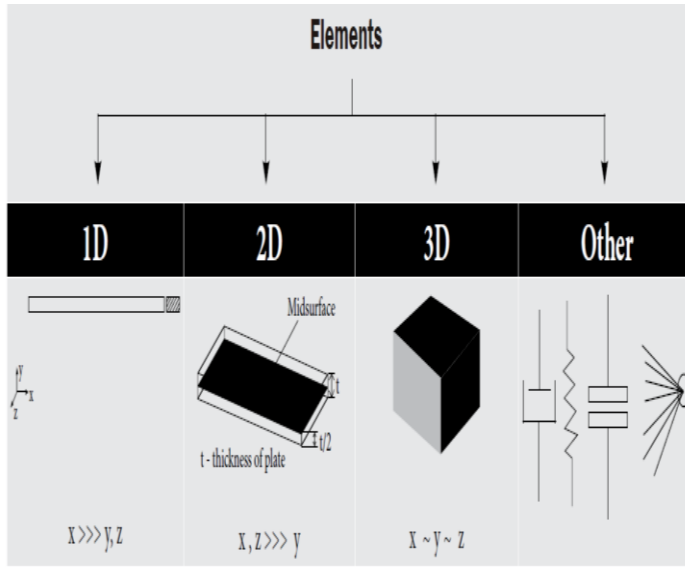
“the size of the problem grows with the square of the number of nodes in the model, and the solution time grows with the cube of the number of nodes”

- Nodes double: Memory requirement is 4 times as much, and run-time is 8 times as much!
- Component/sub-system/system analysis

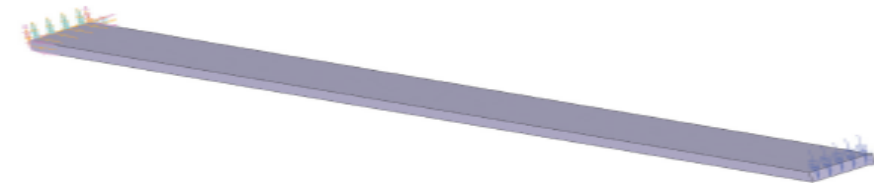
What are you trying to do?

- Maximum deflections
- Peak stresses
- Stiffness under load
- Ultimate strength
- Fatigue life
- Vibration characteristics
- Thermal stresses
- Combinations of several requirements
- Optimization

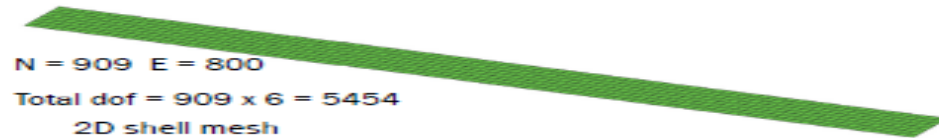
Pre-Processing: Choosing an element type



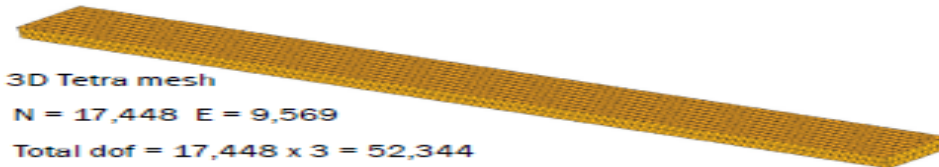
- Choice of element type: Does this influence your geometry generation/clean-up?



1D beam model
 $N=2$ $E=1$
 Total DOF = $6 \times 2 = 12$



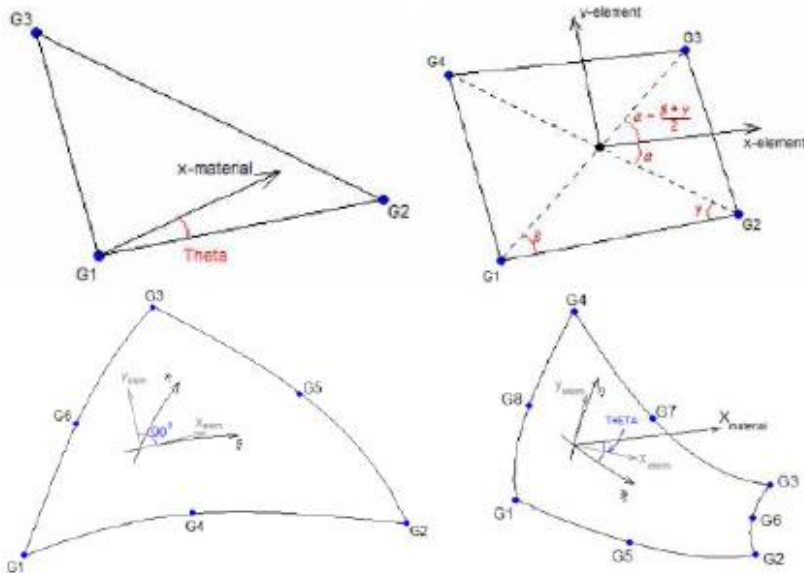
$N = 909$ $E = 800$
 Total dof = $909 \times 6 = 5454$
 2D shell mesh



3D Tetra mesh
 $N = 17,448$ $E = 9,569$
 Total dof = $17,448 \times 3 = 52,344$

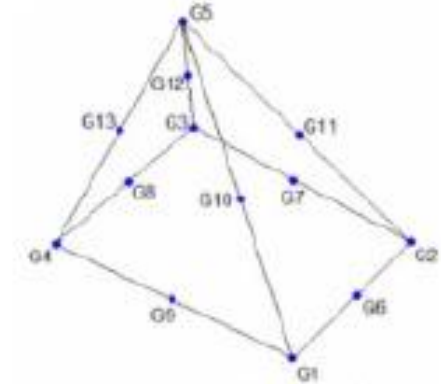
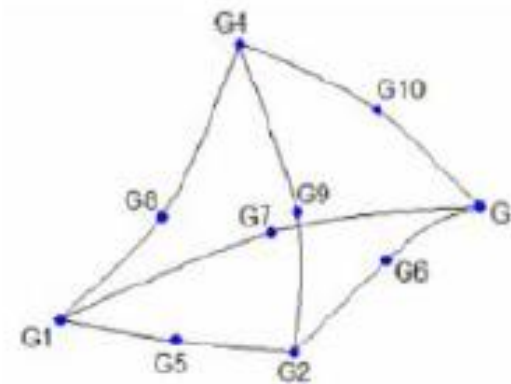
	Nodes	Elements	Stress N/mm ²	Displacement mm
Analytical	–	–	105	4.23
1D	2	1	105	4.23
2D	909	800	103	4.21
3D	17,448	9,569	104	4.21

Pre-Processing: Choosing an element type



Example of **shell** elements (CTRIA3, CQUAD4, CTRIA6, CQUAD8)

First Order	4 or 3	nodes
Second Order	6 or 8	nodes
DOFs	6	degrees of freedom per node



Example of **solid** element (CTETRA3, CTETRA10)

First order	4	nodes
Second order	10	nodes
DOFs	3	degrees of freedom per node

CONSIDER THIS FOR YOUR BCs: NODES ATTACHED TO SOLID ELEMENTS HAVE NO ROTATIONAL STIFFNESS!

Pre-Processing: Meshing – What size to use!

- **Based on previous experience**
- **Type of analysis:** Linear v non linear, number of nodes?
- **Hardware configuration**

Still do not know?

- accept the default element length.
- run the analysis and observe the high stress regions.
- Remesh the localized areas of high stress (with smaller element length) and solve again.

- **Subjective check?**

“If neighboring elements display large differences in stress, the gradient was probably not captured in these areas, therefore some refinement is recommended.”

- **Quantifiable Check?**

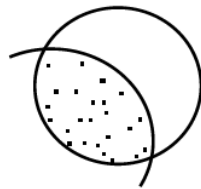
- Compare the difference in the original and the new results.
- Continue the process until convergence is achieved
 - (5 to 10% difference in strain energy / maximum stress value).

Pre-Processing: Meshing – What size to use!

- **Automate?**
- H-Mesh Refinement?
 - Automatically reduce mesh size based on the error norm
- P-Mesh Refinement?
 - Increase the element order
- H-Mesh refinement v P-Mesh refinement

Pre-Processing: Meshing – Things to consider

Why Do We Carry Out Meshing?



No. of Points = ∞
DOF per Point = 6
Total equations = ∞

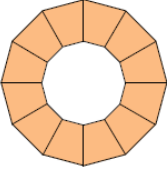
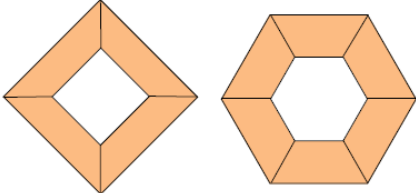
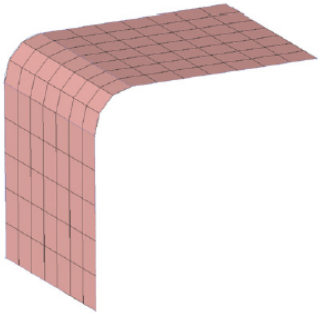
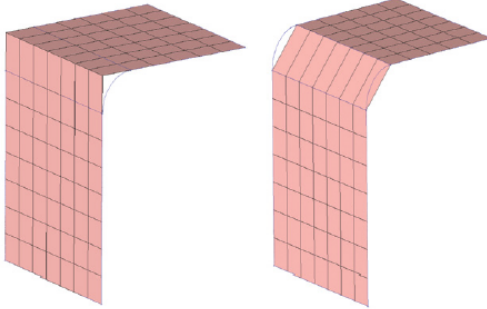


No. of Nodes = 8
DOF per Node = 6
Total equations = 48

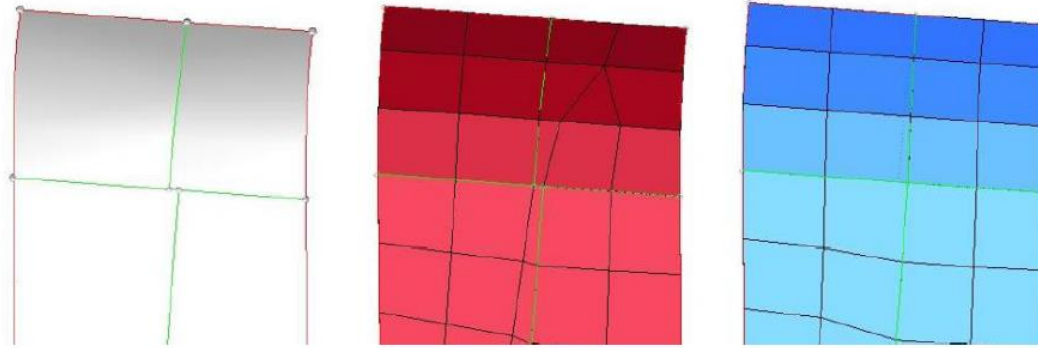
- Mesh Quality (see mesh quality lecture)
- Mesh connectivity
- Element Normals (Contacts, top/middle/bottom stress)
- Element Orientation (non-isotropic materials)
- Automated v user driven?
- Linked to CAD? (Data management/version control)

Pre-Processing: Meshing

- How important is the area being considered?

Critical area	General area
 Minimum 12 elements around the hole	 4 to 6 elements
 Minimum 3 elements on fillet.	 Suppress small fillets, 1 element for large fillets.

Pre-Processing: Geometry Clean-up



- **“Is the Geometry needed?”**
- Free edges
- Scar lines
- Duplicate surfaces
- Small fillets
- Small holes
- Beads
- Intersection of parts (assembly of components)
- **CAD for Manufacturing engineer (part) v CAD for CAE engineer (idealised part)**
 - Version control?

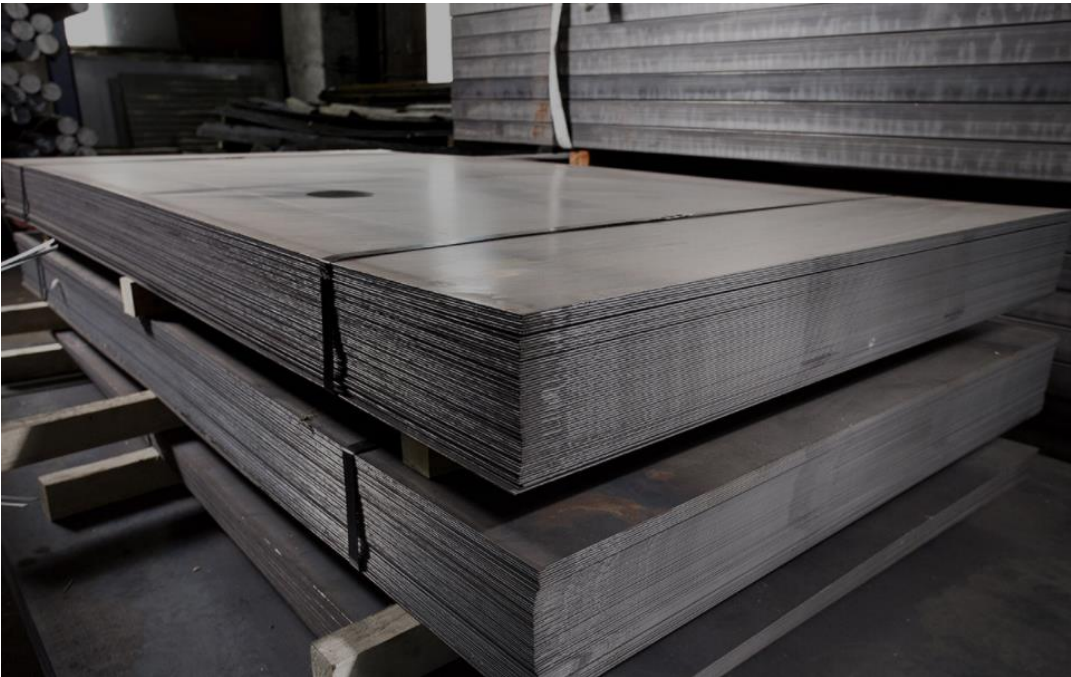
Pre-Processing: Material Info

- Manufacturing variability
- e.g. tolerances for material grades? Coils of steel downgraded



Pre-Processing: Physical Information

- Shell thickness (Tolerances/Corrosion?)
- Beam section (Orientation/Tolerances)



Pre-Processing: Loading & BCs

- Loads
 - Magnitude?
 - Direction?
 - Linear? (Linear or non-linear solver?)
 - Static or dynamic?
 - Applied at a point? (Stress=force/area!)
- Boundary Conditions
 - Held in 6 dofs?
 - Stiffness of the attachment points?
 - Displacement v load driven?
 - Constrain the part fully! (“negative pivot ratios”)
- [EXPORT YOUR DECK AND SOLVE > “Typical” Nastran deck](#)

Know the following...

1. What is the problem you are trying to solve?

- Problem statement
- Objective
- Constraints
- Design Variables (i.e. what is to be changed)

2. What geometry are you using?

- Do you trust the fidelity?
- Is the geometry changing?
- Can you change the geometry?

3. What solution type are you using?

- Is it consistent with your problem?

4. What is your material model?

5. What are your element selections?

6. What are your constraints?

- How are you holding the model?
- Over constrained or under constrained?

7. What are your loads?

8. How are you combining the loads and the constraints?

9. What are the results?

- What results are important for this problem?
- What is success?
- Are the results correct?
- How do you know?

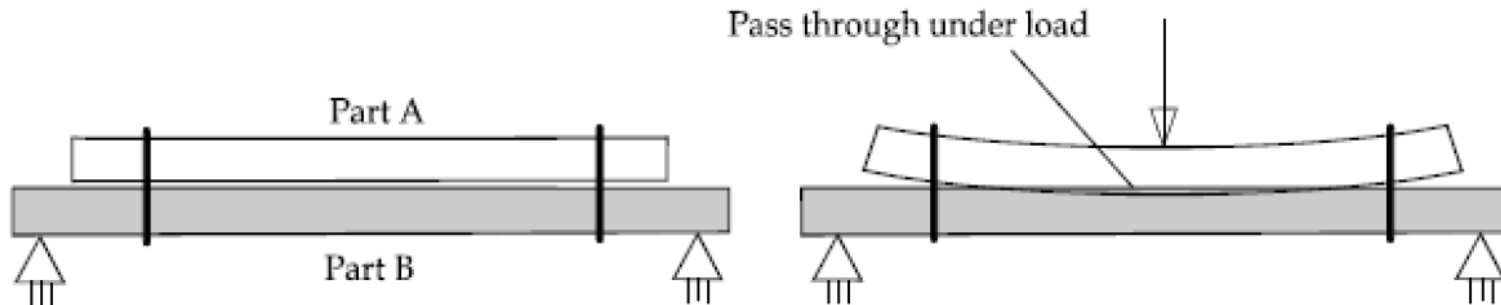
10. What are your recommendations?

What if your job falls over?

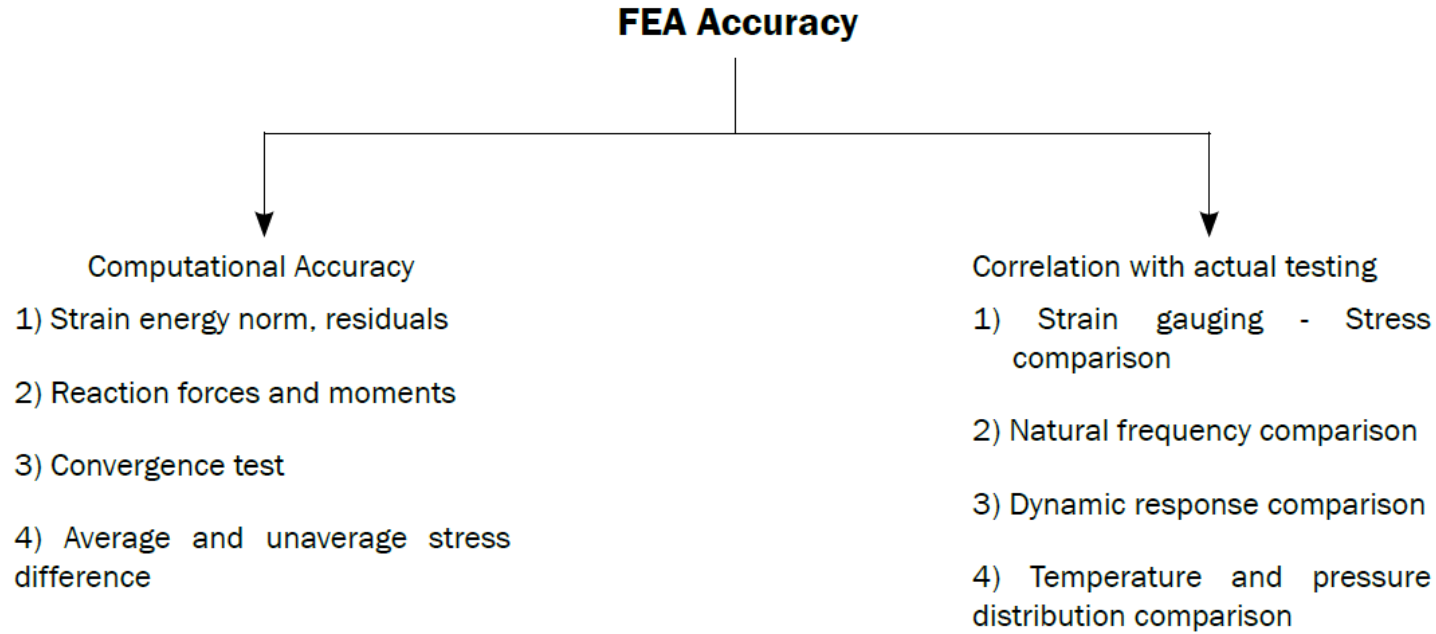
- Check the above! Error messages? *.f06/*.error files
- See “common issues” lecture

Evaluate

- Correlate to test/Hand calcs?
 - Boundary conditions!
- Use your judgement
- Is the problem really linear?
 1. Deflections should be small relative to the structure.
 2. Rotations should be less than 10 -15 degrees.
 3. Material should be linear elastic.
 4. Boundary conditions should be constant.

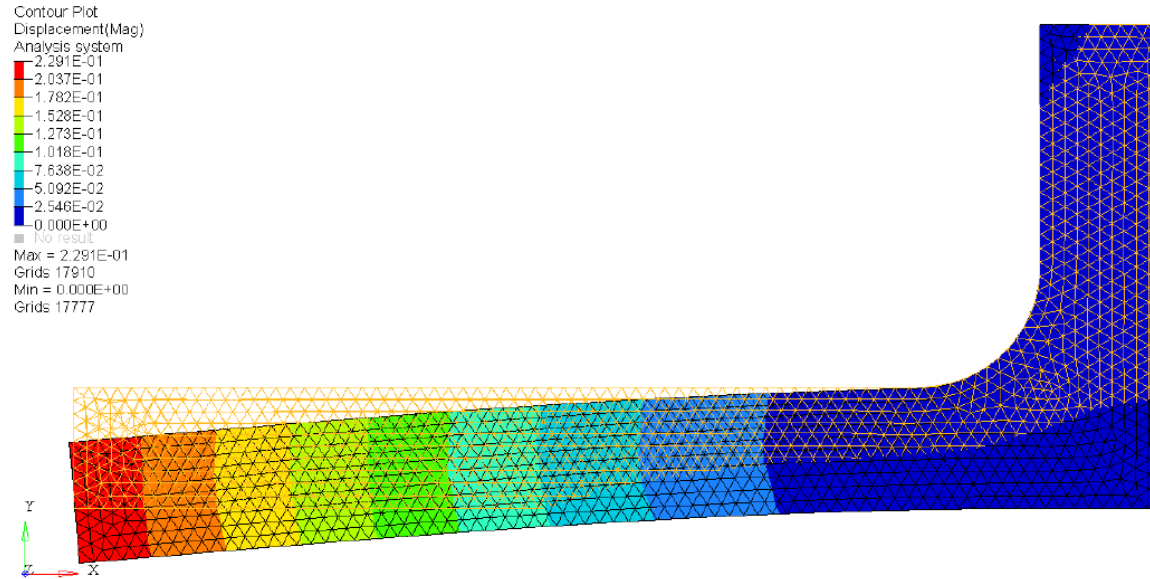


Correlating your model



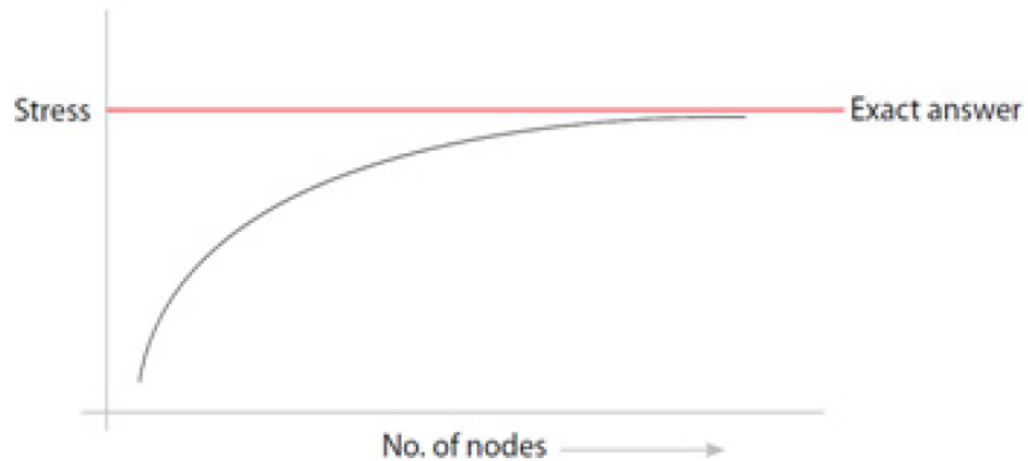
- Visual check – Discontinuous or an abrupt change in the stress pattern across the elements in critical areas indicate a need for local mesh refinement in the region.
- 10 to 15% difference in FEA and experimental results is considered a good correlation
- Probable reasons for more than 15% deviation – wrong boundary conditions, material properties, presence of residual stresses, localized effects like welding, bolt torque, experimental errors etc.

Viewing Displacements

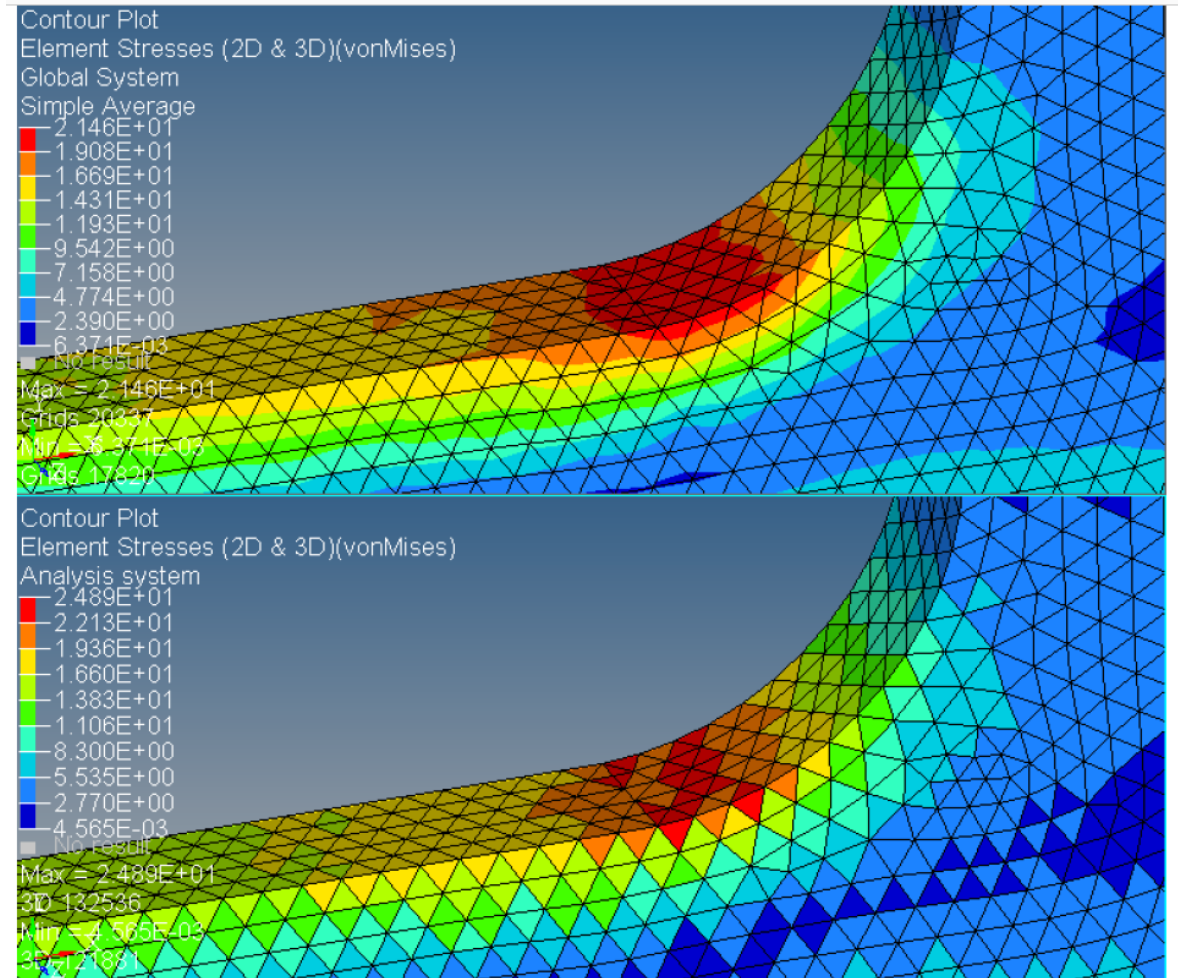


- Generally magnified to see deformed result.
- Animations are linear Interpolation of known points.

Viewing Stresses



- Averaged v un-averaged results
- VonMises stresses: ductile materials
- maximum principal stresses for brittle materials



FEA is SIMPLE

S I M P L E