FEA Best Practices

Introduction

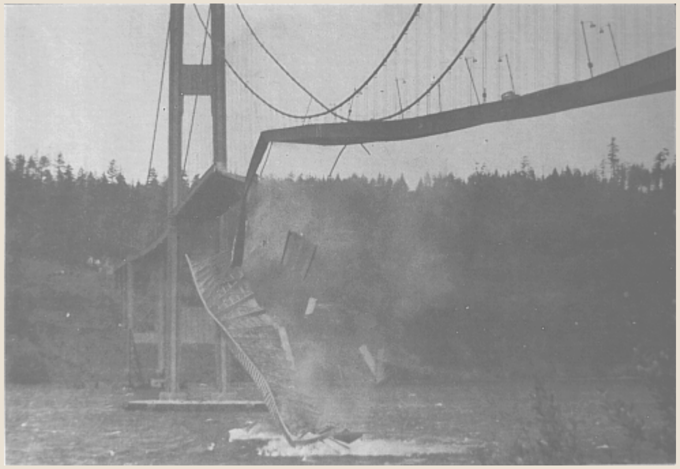
- FEA is not a black box; it is a tool that can be abused.
- It is an approximation
- Careful modeling and interpretation of results
- Garbage in, garbage out!

Example: Sleipner A Oil Platform

- Concrete structure with 24 cells, sank in 1991 during prep for deck installation
- Extensive FEA was performed, but inaccurate modeling of the tricells, the frame where three cells met, led to an underconservative design
- Shear stresses were underestimated by **47%**

How can this be prevented?

- Thorough planning
- Careful modeling
- Accurate loading and modeling of supports
- Thorough verification of results



Accurate Results Depend on:

- Understanding the physics of the problem
- Understanding the behavior of the elements
- Selecting the correct element, the number of elements, and their distribution
- Critically evaluating the results and making modification in the conceptual model to improve their accuracy
- Understanding the effects of the simplifications and assumptions used

Using FEA wisely requires using best practices: this seminar presents many of these, and discusses how to develop best practices for your process.

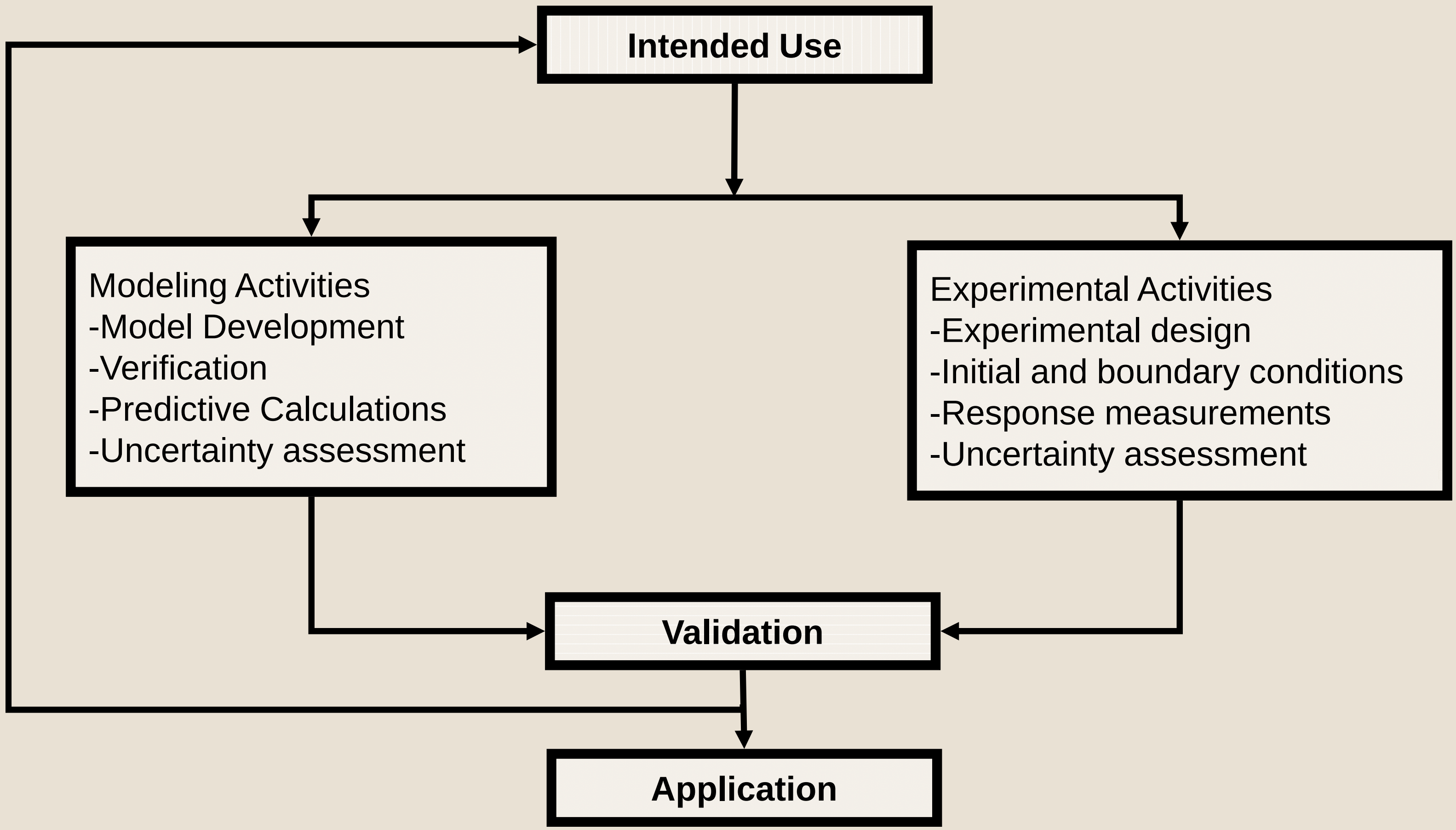
Guidelines for Best Practices

- Department of Defense’s Defense Modeling and Simulation Office
  - Recommended practices for large-scale simulations
  - Does not focus on first-principles directly
- American Institute of Aeronautics and Astronautics (AIAA) has the *Guide for the Verification and Validation of Computational Fluid Dynamics Simulations*
- FEA now has the recently formed ASME PTC 60 committee on Verification and Validation
  - Recently published *Guide for Verification and Validation in Computational Mechanics*

V&V 10-2006

- Provides structural mechanics community with:
  - Common language
  - Conceptual framework
  - General guidance for implementing V&V
- NOT a step-by-step guide!
  - Glossary of terms
  - Figures illustrating a recommended overall approach
  - Discussions of factors that should be considered
- Written as guidance to developing V & V processes

V&V 10-2006: Elements of V&V



V & V

- Model is validated for a specified use
  - e.g., for a certain range of loads
- Modeler should quantify the uncertainties
  - Due to inherent variability of parameters
  - Lack of knowledge of the parameters
  - Model form
- Combine with calculation verification to get overall uncertainty estimate
- Not always possible to test the full range of interest
  - Should still develop a plan of V&V

Best Practices Approach

- Plan your analysis
- Materials
- Model geometry
  - Element choice
  - Meshing
  - Simplifications
- Supports and Loads
- Model Calibration
- Verification

Plan your analysis

- What are the design objectives?
  - What do you need to know?
  - Why are you doing FEA?
- What is the design criteria?
  - What engineering criteria will be used to evaluate the design?

Plan your analysis

- What are you trying to find out?
- How much of the structure needs to be modeled?
- What are the boundary conditions and loads?
- Do you need to know stresses, displacements, frequency, buckling or temperature?
- Get ballpark figures through hand-calculations or test data, so you have an idea of how the structure will behave and what numbers are reasonable.

Analysis Decisions

- Analysis type
- How to *idealize* material properties
- Geometry details/simplifications
- Element type/options
- What are the supports or constraints
- What are the loads

Type of analysis

- Is it static or dynamic?
  - Are the loads applied gradually, or quickly?
  - Vibrations? Seismic?
- Linear or nonlinear?
  - Are there large deflections?
  - Nonlinear materials?
  - Contact?

Is it really static?

- Static analysis assumes that inertial and damping effects are negligible
- You can use time-dependency of loads as a way to choose between static and dynamic analysis.
  - If the loading is constant over a relatively long period of time, choose a static analysis.

Is it really static?

- In general, if the excitation frequency is less than 1/3 of the structure’s lowest natural frequency, a static analysis may be acceptable
- Cyclic loads can be modeled by a harmonic analysis rather than full transient

Linear vs. Nonlinear

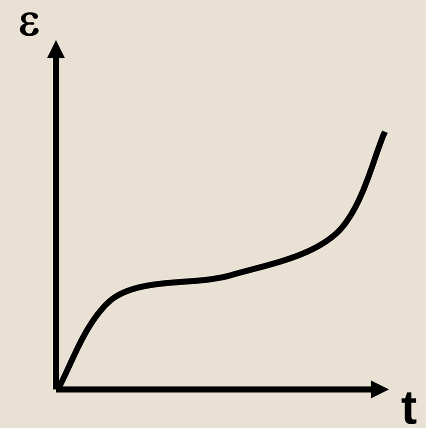
- Nonlinear structural behavior is a changing structural stiffness
- Several types of nonlinearities:
  - Geometric
  - Material (e.g., plasticity, hyperelasticity)
  - Changing Status (e.g., contact)

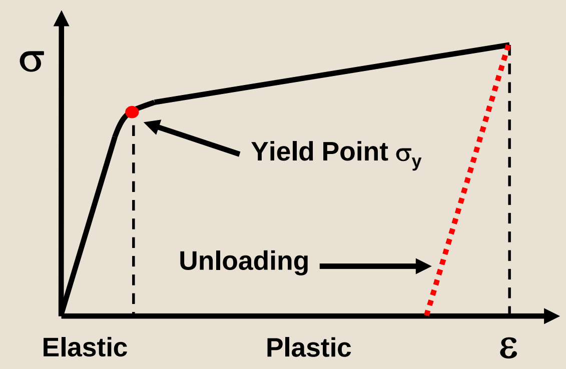
Geometric Nonlinearities

- Large deflections
- Large rotation
- Stress stiffening
  - Cables
  - Membranes
    - Membrane under deformation picks up bending stiffness
  - Spinning structures

Material Nonlinearities

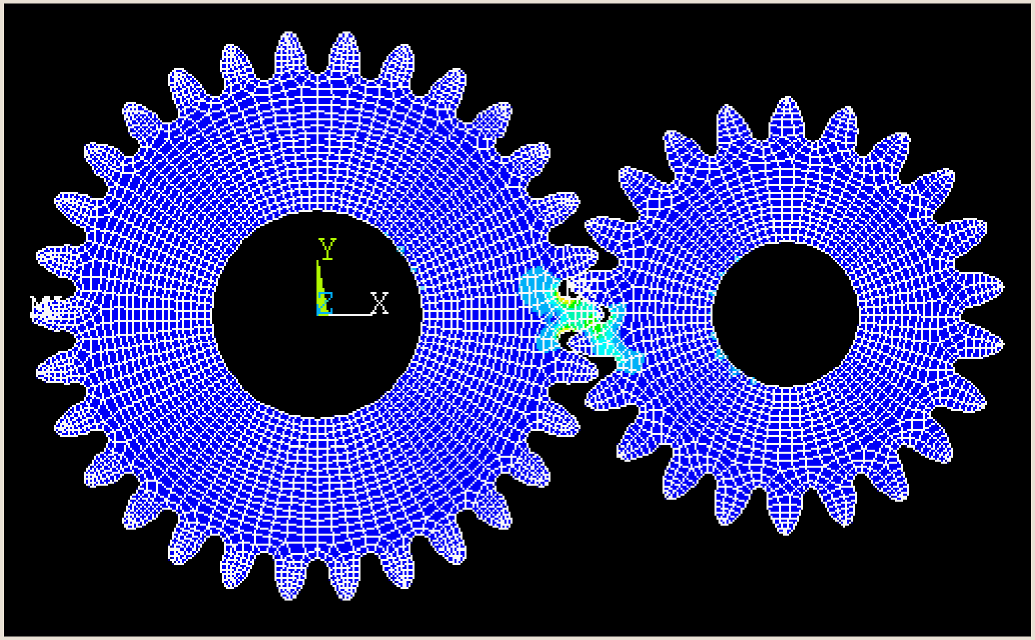
- Plasticity
- Creep/Viscoelasticity
  - Rate dependence
- Viscoplasticity
  - Time dependent
- Hyperelasticity





Changing Status Nonlinearities

- Contact



Contact

- Bonded vs. nonlinear contact
  - Welded/glued parts
  - Gaps in model
  - Will parts separate from each other?
  - Is delamination possible?
- Large vs. small sliding
  - Determines type of element to use
  - Determines type of contact

Contact

- Contact stiffness
  - Is the contact hard, or is there some softening?
  - Is contact pressure an important value?
- Does friction need to be modeled?
  - What value for the coefficient?
  - May need to run model with different values

Materials

- Material properties used will be approximate!
- Is the material homogenous (the same throughout)?
- Is it isotropic, orthotropic or anisotropic?
- Is temperature dependence important to the analysis?
- Is there rate or time dependence?
- Are composites used?

Material Information

- For linear isotropic material, need modulus of elasticity and Poisson's ratio for a static analysis
- Need density for inertial loads
- For thermal analysis, need thermal conductivity
- Also need Coefficient of Thermal Expansion for thermal stress
- Need test data for nonlinear materials

Material Data Sources

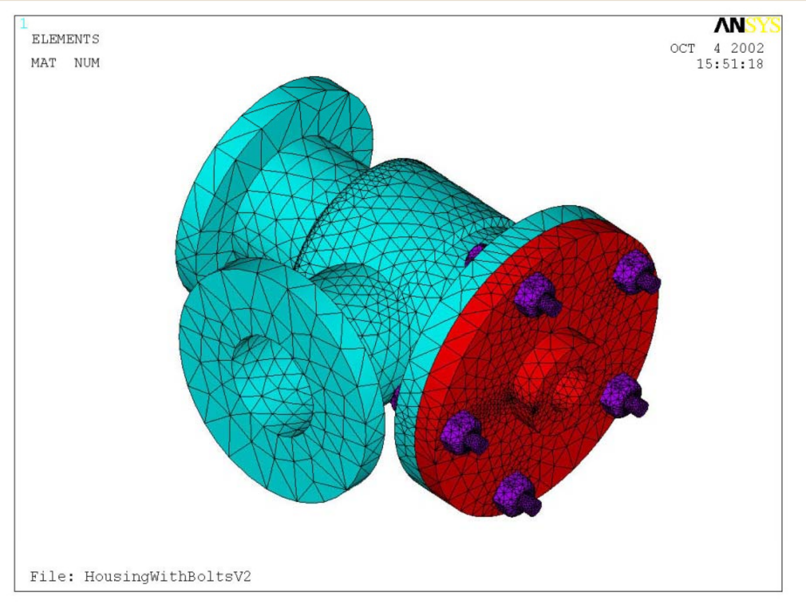
- Testing:
  - Datapoint Labs: http://www.datapointlabs.com/
  - Axel Products: http://www.axelproducts.com/
- Online:
  - Matweb: http://www.matweb.com
  - Material Data Network: http://matdata.net/index.jsp

Poisson's Ratio

- Used to calculate shear modulus
- Controls expansion/contraction in direction perpendicular to load direction
- If using $\nu = 0.5$, need to use element with hyperelastic ability
- For models that are constrained from expansion, the value of ν is very important!

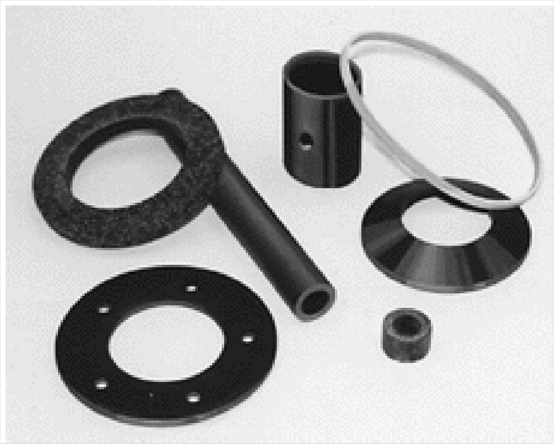
Multiple Materials

- Model a boundary wherever material properties change
- Make sure the appropriate material property is assigned to each part of the model
- Consider interaction between properties
  - Affects contact stiffness

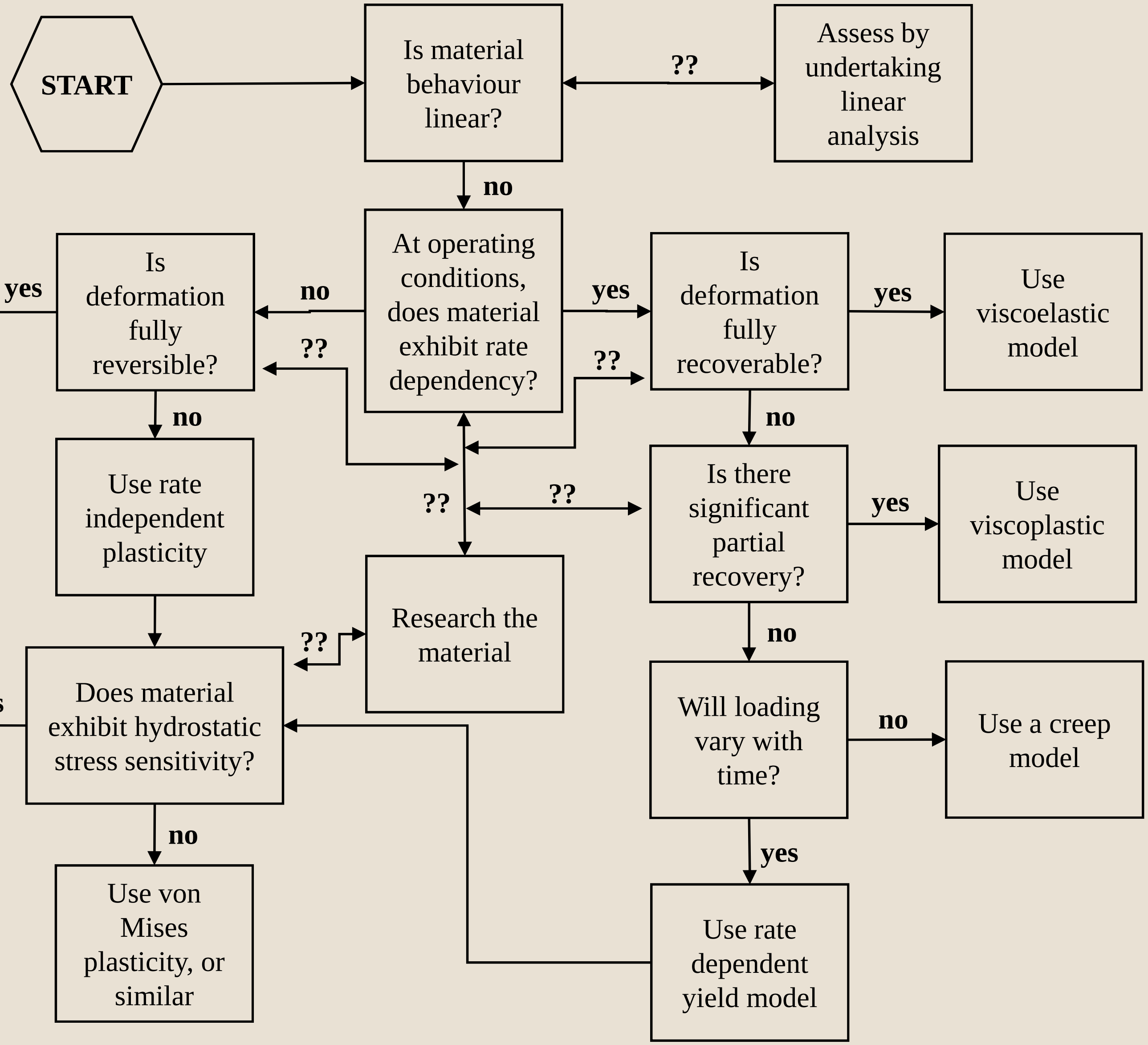


Linear vs. Non-Linear Materials

- Will the stresses stay in the elastic region?
- Are you using a non-linear material, e.g. concrete, soil, rubber?
- Do you need to consider creep or fatigue?

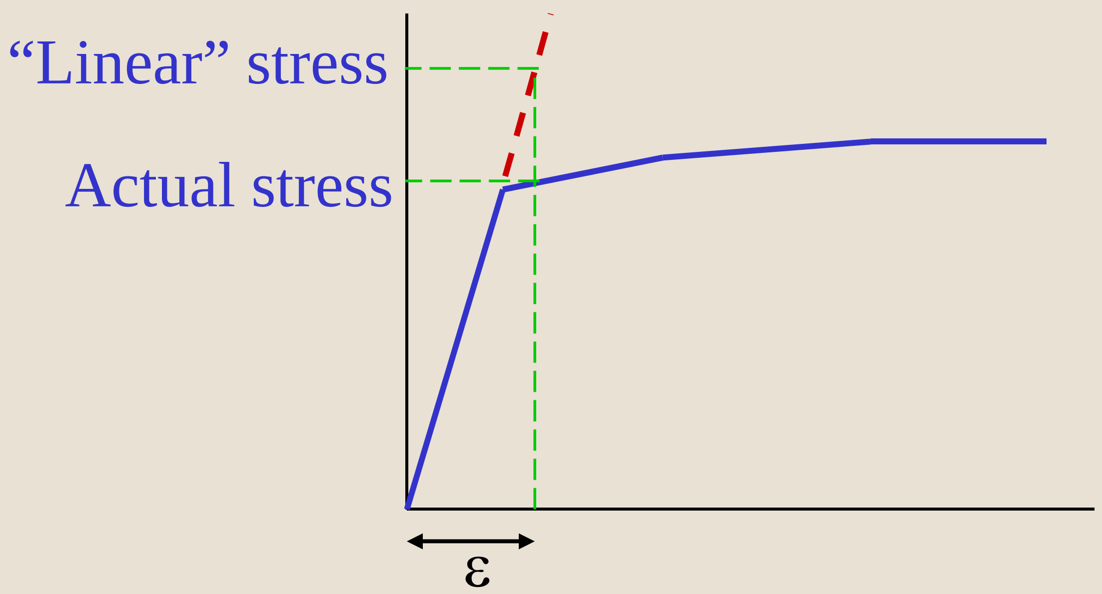


Material Non-linearity Flowchart, by Andrew Crocombe



Linear or Nonlinear?

If no stress-strain data is given, the program will assume the analysis is linear, and will use Young's Modulus even if the part yields. This gives erroneous results when the loads cause the model to exceed yield.



Non-Linear Materials

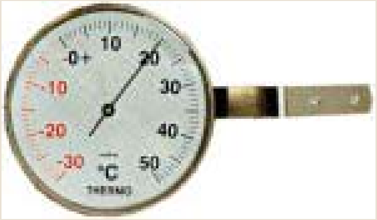
- Enter stress-strain curve using <u>true-stress</u> and <u>true-strain</u>

  true strain = ln(1 + engineering strain)

  true stress = eng. stress (1 + eng. strain)
- Graph stress-strain curve to check inputs
- Still need to enter Young's modulus
- Make sure Young's modulus matches yield stress and strain

Units

- Many general purpose FEA codes allow the user to enter a consistent unit set
- Make sure forces, displacements, material properties have same units– these determine the units of the results.
- Use mass = force/area to get proper mass units



Consistent Unit Systems

Mass unit	kg	kg	$lbf\text{-}s^2/in$	slug
Length unit	m	mm	in	ft
Time unit	s	s	s	s
Gravity const.	9.807	9807	386	32.2
Force unit	N	mN	lbf	lbf
Pressure/Modulus of Elasticity	Pa	kPa	psi	psf
Density Unit	kg/m^3	kg/mm^3	$lbf\text{-}s^2/in^4$	$slug/ft^3$
Mod. Elasticity Steel	0.2E12	0.2E9	30E6	4.32E9
Mod. Elasticity Concrete	30E9	30000	4.5E6	648E6
Density of Steel	7860	7.86E-6	7.5e-4	15.2
Density of Concrete	2380	2.38E-6	2.2e-4	4.61

How to Check Units

- Set loads to zero and run. Check mass and center of mass.
- Or, turn on gravity and check reactions.
- If using small dimensions (e.g. microns, millimeters) use smaller base unit to reduce round-off problems

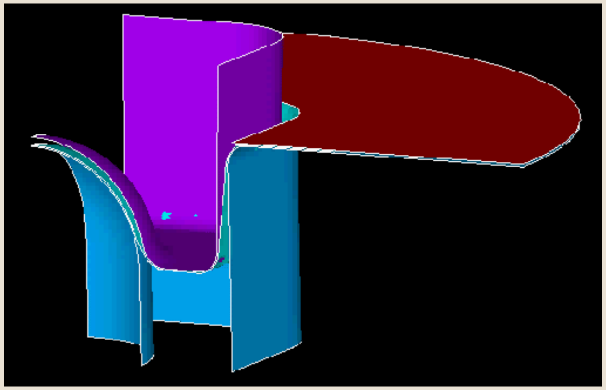
```
*** MASS SUMMARY BY ELEMENT TYPE ***

TYPE       MASS
   1   39.5720
```

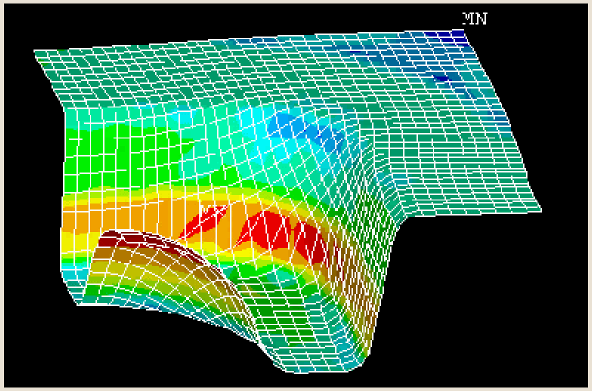
Create the Model-- What should I model?

- Do I model the entire structure, or only the part that is of interest?
- Should I model bolts? How?
- Do I model the welds?
- What details should be included?

How was the part made?

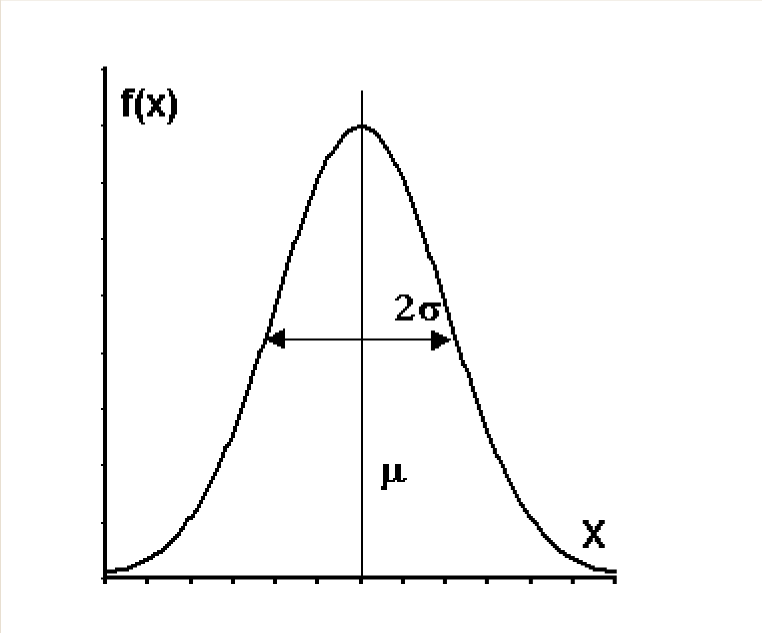
- Casting can create variation in material properties
- Forging affects the state of strain
- Formed sheet metal can have significant residual stresses in corners and bends





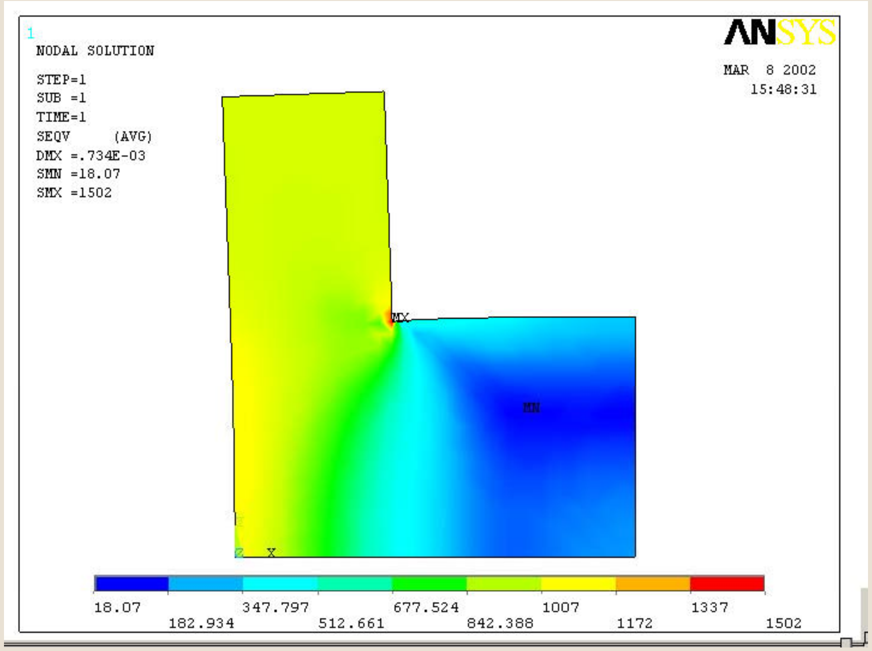
Variations and Tolerances

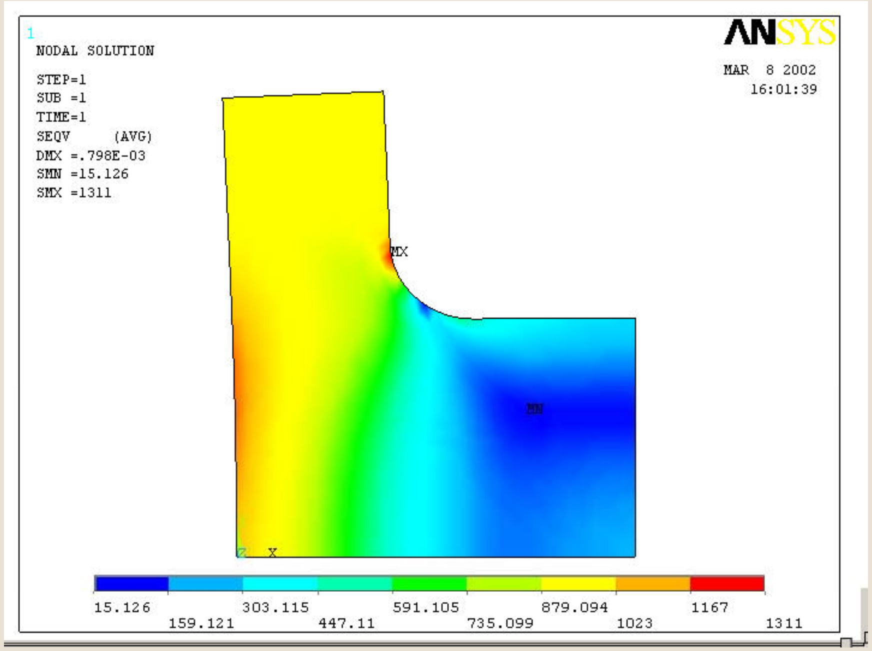
- Material variations and tolerances affect behavior– probabilistic analysis tackles this
- Consider effects of tolerances on key locations



Simplification—Be Careful!

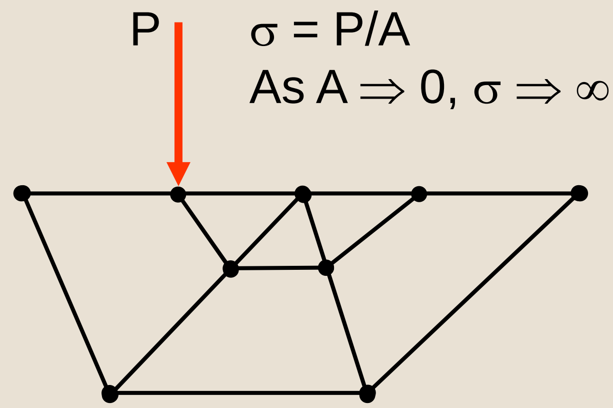
- Fillets can prevent singularities
- Holes, bosses, etc. in areas of high stress should be included





Singularities

- FEA uses the theory of elasticity: stress = force/area
- If the area=0, then stress=infinite
- Theory of FEA: as mesh is refined, the stresses approach the theoretical stress
- For a singularity, you would try to converge on infinity



$\sigma = P/A$

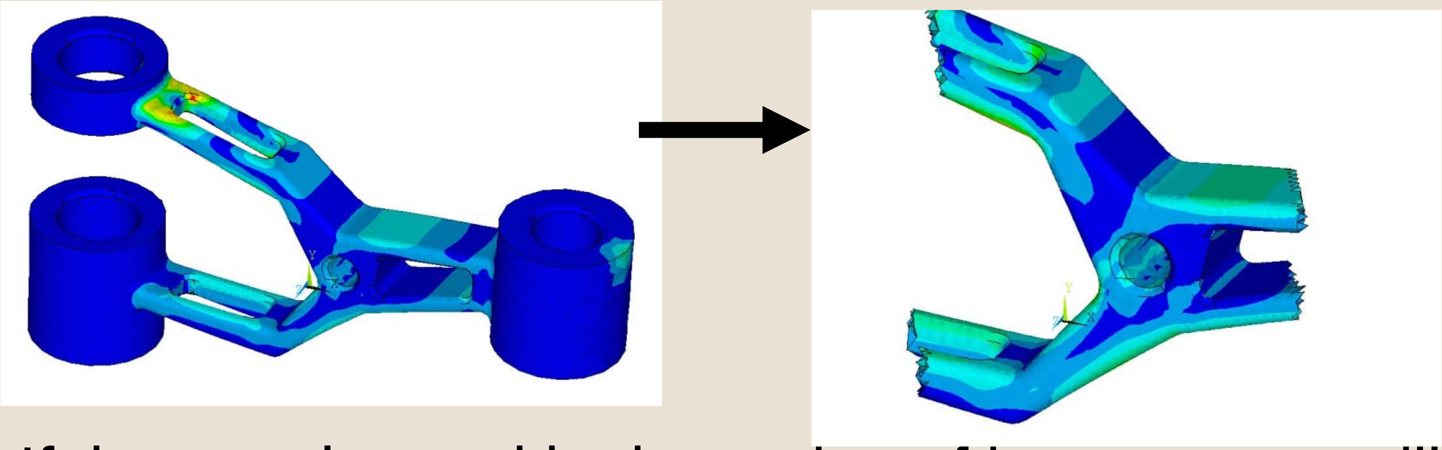
As $A \Rightarrow 0$, $\sigma \Rightarrow \infty$

Singularities

- A stress singularity is a location in a finite element model where the stress value is unbounded (infinite). Examples:
  - A point load, such as an applied force or moment
  - An isolated constraint point, where the reaction force behaves like a point load
  - A sharp re-entrant corner (with zero fillet radius)
- Real structures do not contain stress singularities. They are a fiction created by the simplifying assumptions of the model.
- Point loads are best used for line elements

What do you do with Singularities?

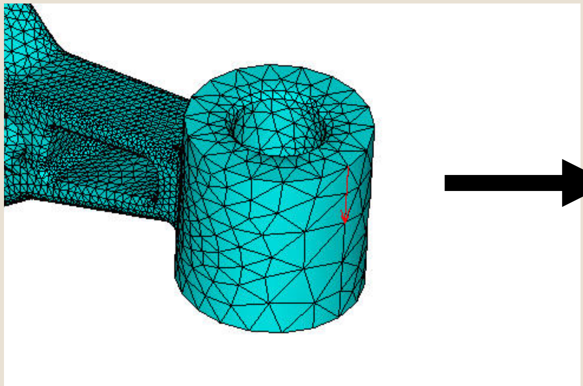
- If they are located far away from the region of interest, you can focus post-processing away from that part of the model

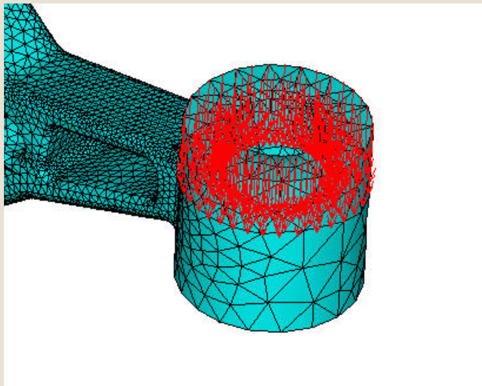


- If they *are* located in the region of interest, you will need to take corrective action

How do you Correct for Singularities?

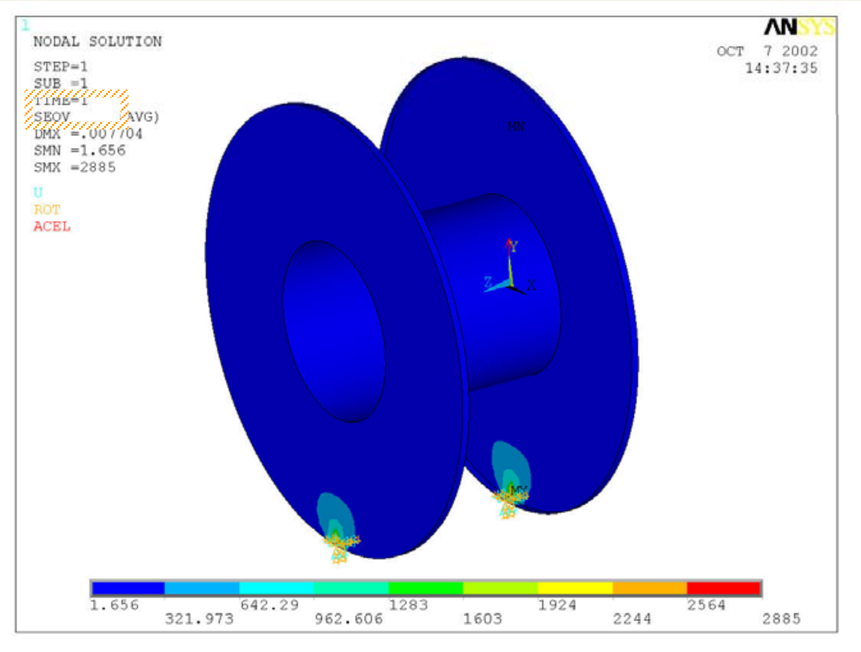
- Adding a fillet at re-entrant corners and re-runing the analysis.
- Replacing a point force with an equivalent pressure load.

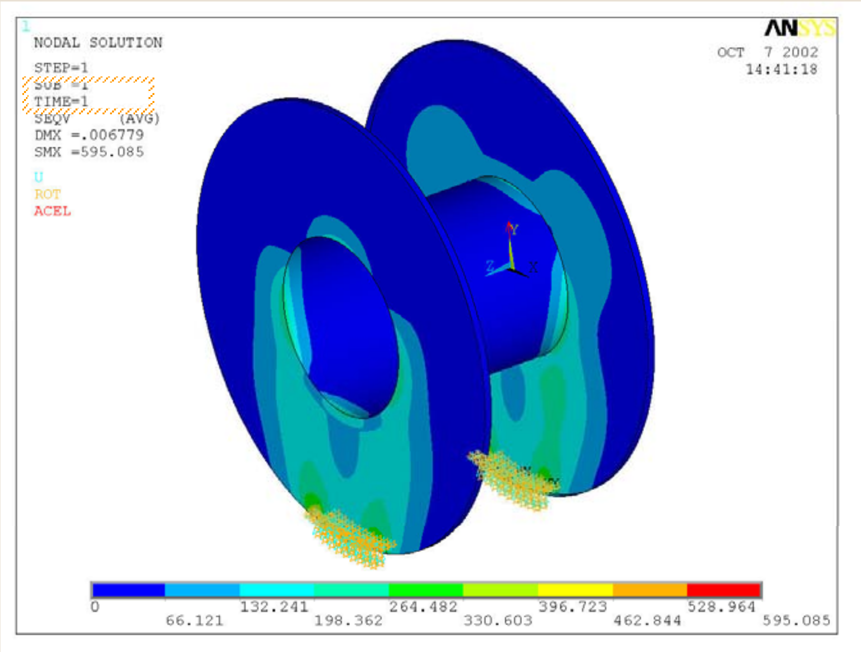




How do you Correct for Singularities?

- “Spreading out” displacement constraints over a set of nodes.
- Turning on material plasticity
- Using Stress Linearization



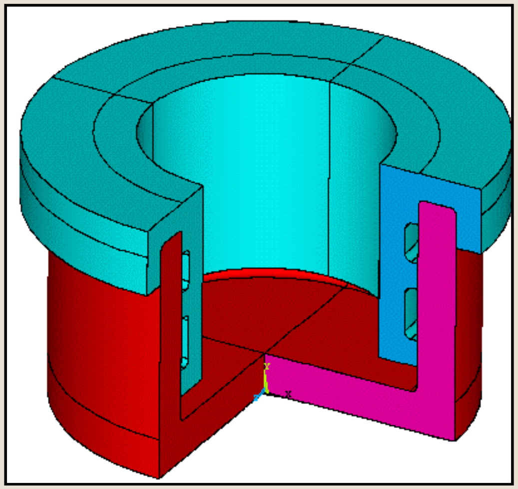


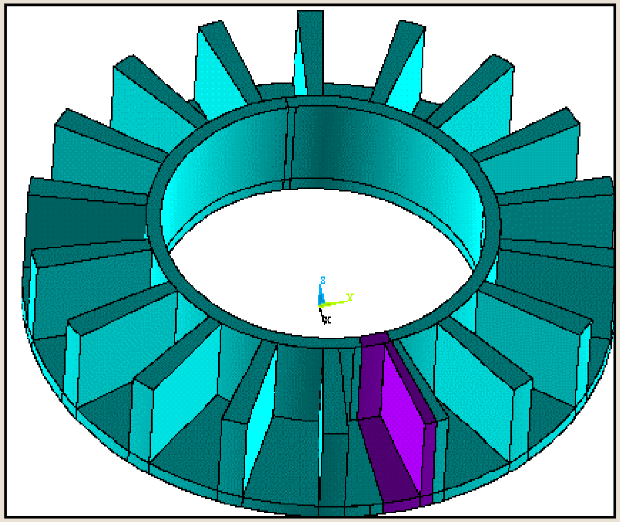
Can I Use Symmetry?

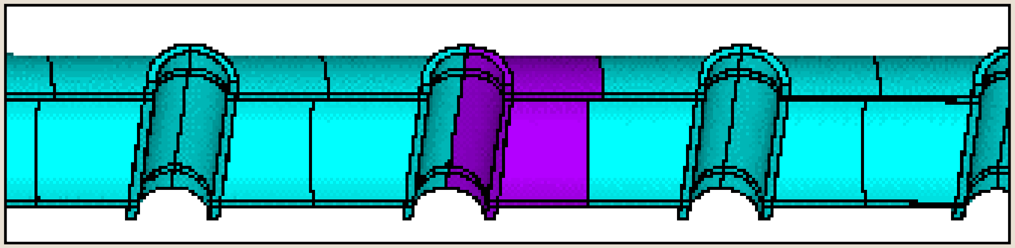
- Symmetric structures can be modeled by a smaller representative portion or cross-section
- Easier to create, can use a finer mesh
- Quicker run times– can run multiple load scenarios, multiple configurations much quicker
- It can be used when geometry, material behavior and loading are symmetric about the same plane(s).
- Generally can NOT use symmetry in modal analysis, as the mode shapes are not always symmetric.

Symmetry

- Types of symmetry:
  - Axisymmetry
  - Rotational
  - Planar or reflective
  - Repetitive or translational

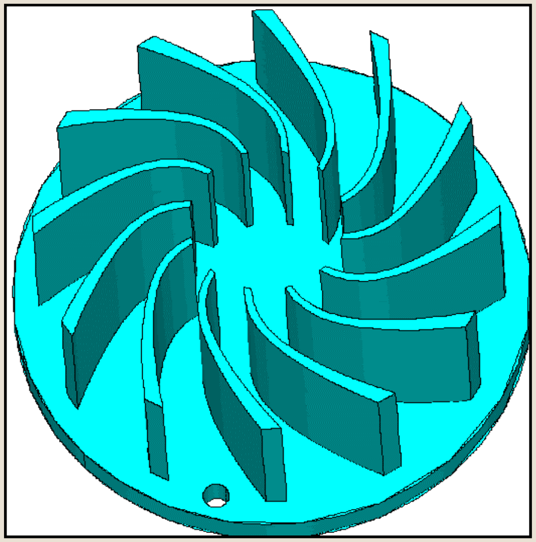






Symmetry, Interrupted

- Sometimes a small detail interrupts symmetry
- Can ignore it, or treat it as symmetric– best to do a small test case if unsure

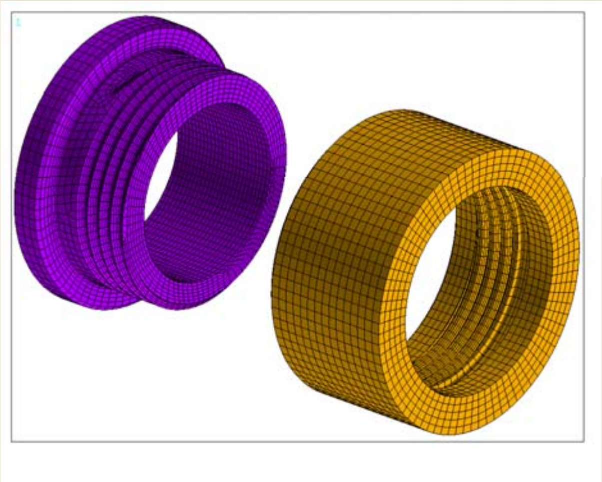


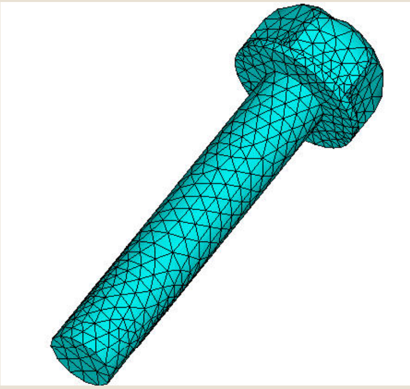
Superelement

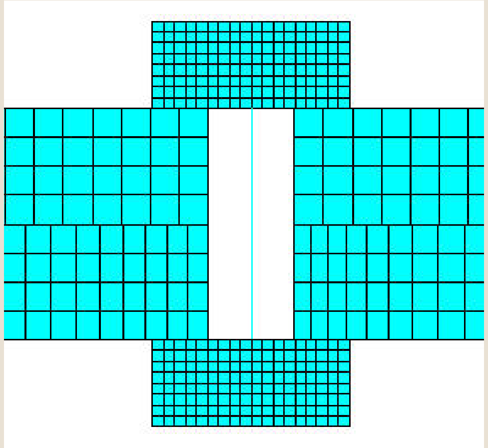
- A superelement is a single element that has the same stiffness as a large portion of a structure
- Can model in detail the area of interest, then use one or more superelements for the rest of the structure
- Can use superelements for repeating parts of a structure
- Also known as substructuring
- Requires more pre/post-processing
- The stiffness is exact, damping/mass is not

Modeling Bolts

- Solid model if you need stress in bolt
- Beam model if you don't
- Modeling threads increases model size– may be too much detail

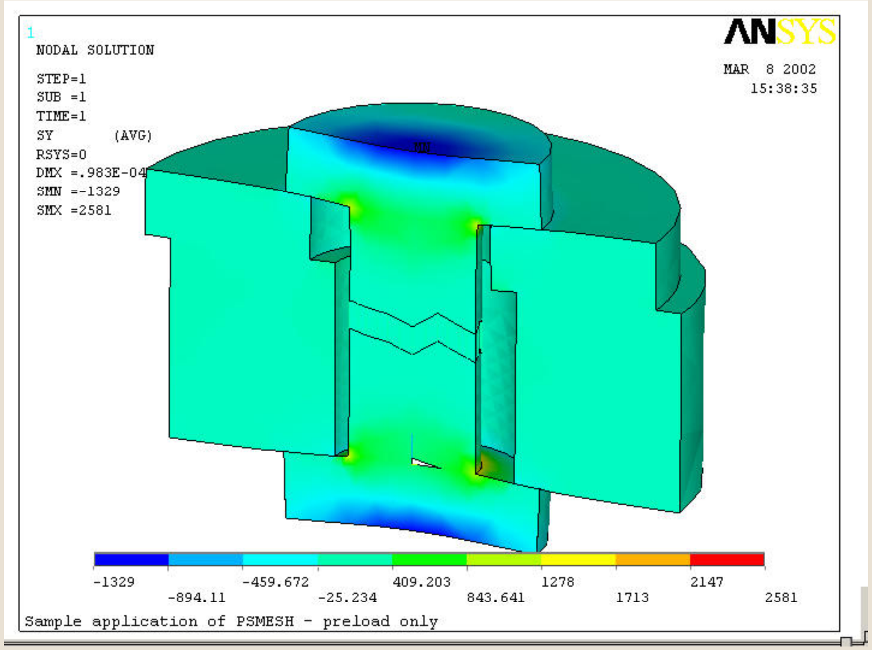






Bolt Pretension

- Affects stresses and deflections
- Traditionally, imposed a strain on the bolt equal to the pretension strain (requires multiple solutions)
- Can instead use pretension element.
- ANSYS can automate bolt pretension in either interface

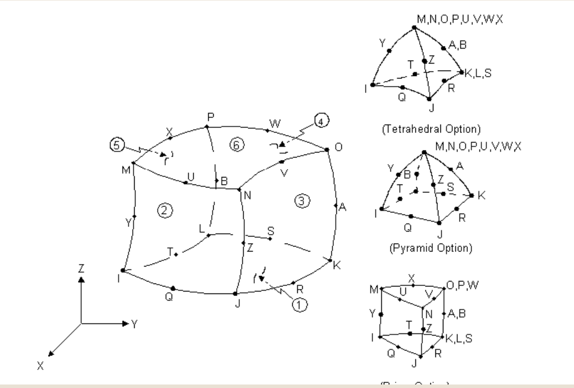


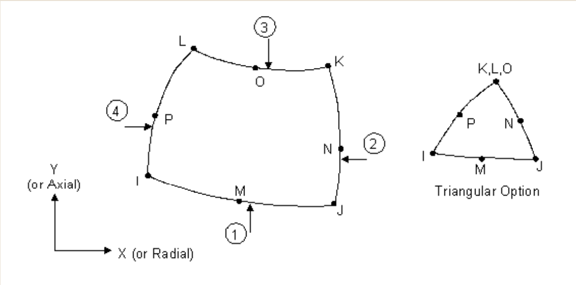
Check geometry

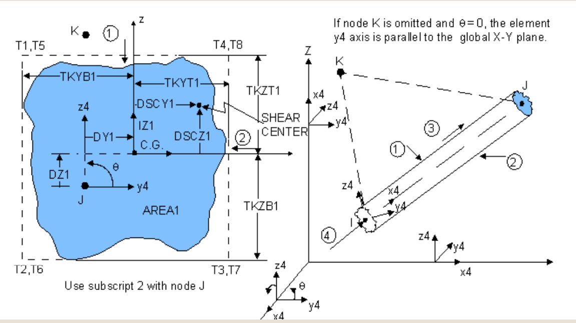
- If importing model, do some checks of the dimensions – don’t assume it’s right!
- Make sure the model is in the required units system
- If the model was created in a system different from the material data and loads, you need to scale the model by the proper conversion factor

Choice of elements

- 2D vs 3D vs line
  - 2D elements are *spatially* 3D, but in the model they are *geometrically* 2D
- Element Order: linear, quadratic, polynomial
- Specialized elements? (composites, concrete, acoustics, coupled field)
- Geometric dimensionality--how the geometry is

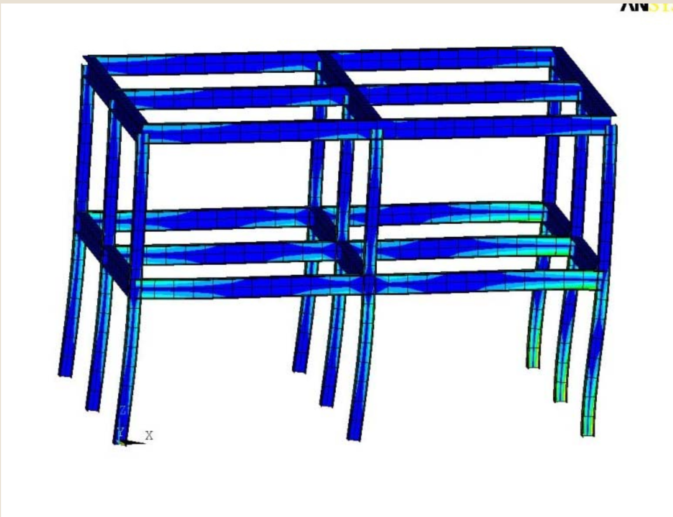






Line Elements

- *Beam* elements have bending and axial strength. They are used to model bolts, tubular members, C-sections, angle irons, etc.
- *Spar or Link* elements have axial strength. They are used to model springs, bolts, preloaded bolts, and truss members.



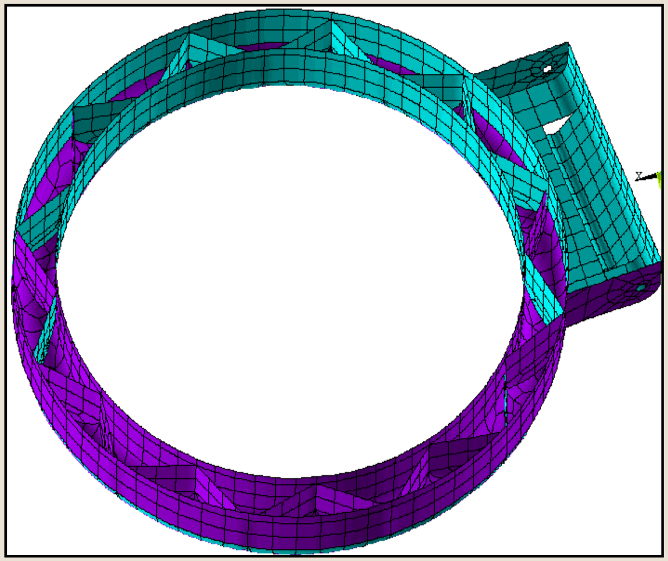
- *Spring or Combination* elements also have axial strength, but instead of specifying a cross-section and material data, a spring stiffness is entered. They are used to model springs, bolts, or long slender parts, or to replace complex parts by an equivalent stiffness.

Which Beam Element?

- Several types of beams are usually available
  - Different beam theories are used
    - Bending in BEAM188/189 is linear, unlike that in BEAM4; use several elements to model a member with 188 or 189
    - BEAM44 and BEAM188/189 include shear deformation
  - Some elements have special features:
    - Tapered beams (BEAM44)
    - Section offset (BEAM44, BEAM188/189)
    - Section visualization, including stresses (BEAM44, BEAM188/189)
    - Initial strain input (BEAM4, BEAM44)
    - Initial stress input (BEAM188/189)

Shell Elements

- Can use a shell when the maximum unsupported dimension of the structure is at least 10 times the thickness
- Use to model thin panels or tubular structures
- “Thick” shell elements include transverse shear, “thin” shell elements ignore this.
- Shell elements can be 2D or 3D; 2D shells are drawn as a line, 3D as an area

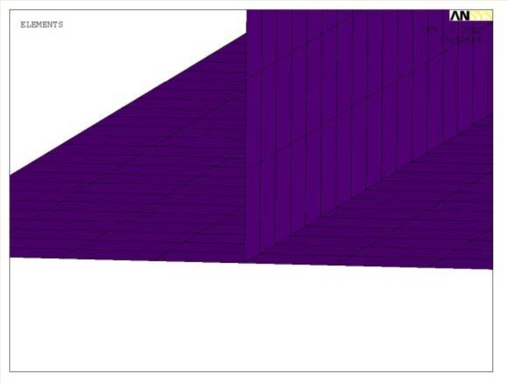


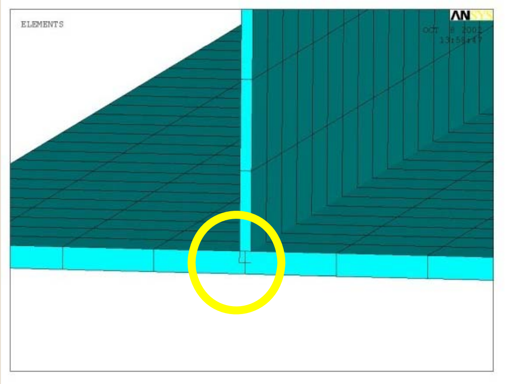
Shell Pitfalls

- Modeling tubes with straight-edged shells will result in a faceted model
  - The nodes are on the true surface, so the lengths of the elements are smaller than the circumference– inaccurate cross-section for axial stress
  - Pressure will produce spurious circumferential bending moments at the nodes
  - Use finer mesh, or use elements with midside nodes

Shell Pitfalls

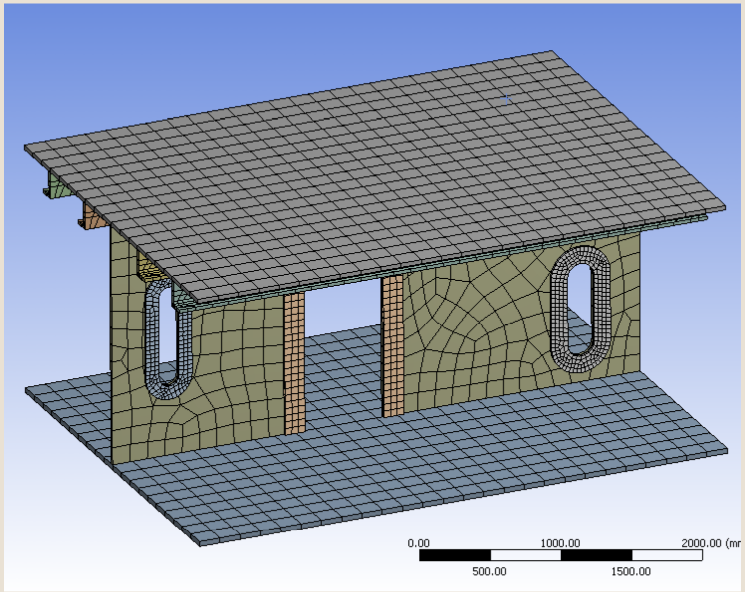
- When shell elements meet at a T, material is duplicated
- Can be easier to simply mesh the outer surface of a model, but the element does not correspond to the center of the actual plate
  - Local bending is now changed
  - Can use shell offset to accommodate this (SHELL91, SHELL99, SHELL181)
- Connecting shell and solid elements tricky
  - Different DOFs





Solid-Shell Element

- 3D Solid brick (or prism) element without bending locking
- Nodes have same DOFs as 3D elements– can connect thin and thick structures without constraint equations or MPCs
- Can model varying thickness bodies without using multiple real constants

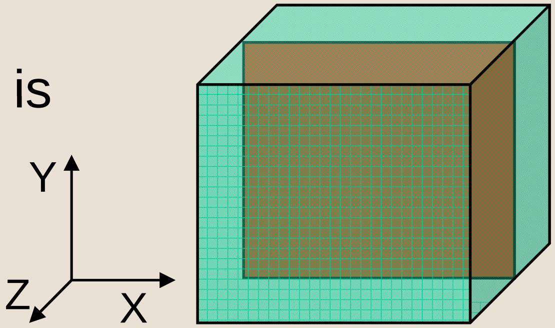


2D Solid Elements

- Used to model a cross-section of solid objects.
- Must be modeled in the global Cartesian X-Y plane.
- All loads are in the X-Y plane, and the response (displacements) are also in the X-Y plane.
- Element behavior may be one of the following:
  - plane stress
  - plane strain
  - generalized plane strain
  - axisymmetric
  - axisymmetric harmonic

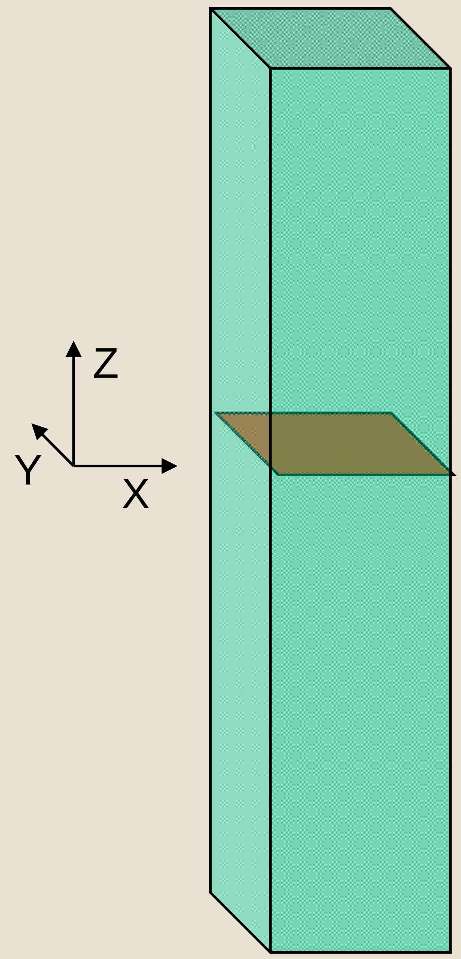
Plane Stress

- Assumes zero stress in the Z direction.
- Valid for components in which the Z dimension is smaller than the X and Y dimensions.
- Z-strain is non-zero.
- Optional thickness (Z direction) allowed.
- Used for structures such as flat plates subjected to in-plane loading, or thin disks under pressure or centrifugal loading.



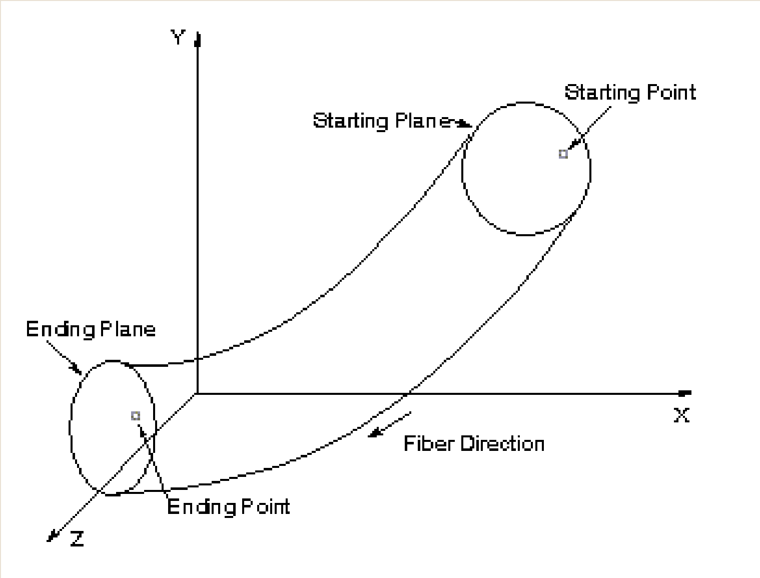
Plane Strain

- Assumes <u>zero strain</u> in the Z direction.
- Valid for components in which the Z dimension is much larger than the X and Y dimensions.
- Z-stress is non-zero.
- Used for long, constant cross-section structures such as structural beams.



Generalized Plane Strain

- Assumes a finite deformation domain length in Z direction instead of infinite
- Gives more practical results for when the Z direction is not long enough



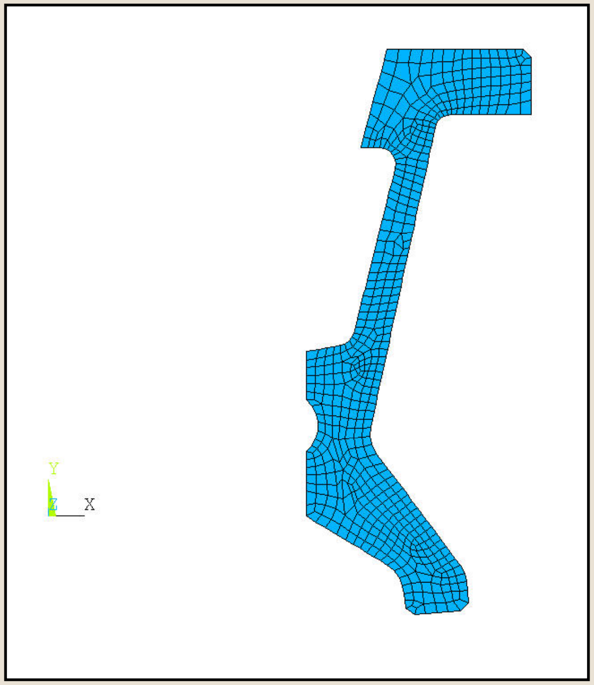
• The deformation domain or structure is formed by extruding a plane area along a curve with a constant curvature.

Axisymmetry

- Assumes that the 3-D model *and* its loading can be generated by revolving a 2-D section 360° about the Y axis.
- Axis of symmetry must coincide with the global Y axis.
- Y direction is axial, X direction is radial, and Z direction is circumferential (hoop) direction.
- Hoop displacement is zero; hoop strains and stresses are usually very significant.
- Used for pressure vessels, straight pipes, shafts, etc.

Axisymmetric pitfalls in ANSYS

- Check keyopt(3) setting– otherwise, model will be plane stress
- Pressures are input as force/area
- Point loads are the total force for the circumference
- All nodes must have +X coordinate
- Nodes at X=0 need to have UX constrained



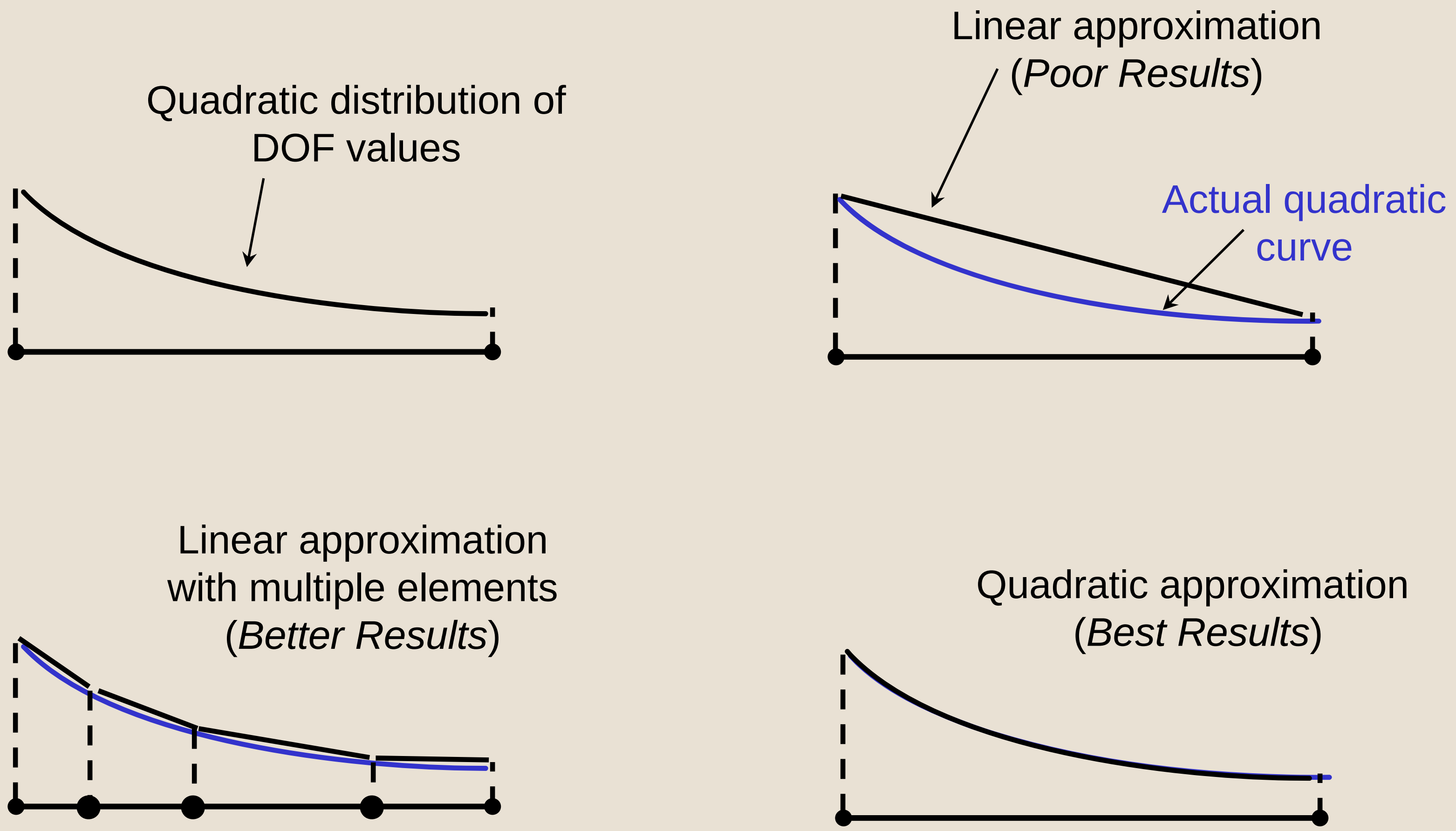
3D Solid Elements

- Used for structures which, because of geometry, materials, loading, or detail of required results, cannot be modeled with simpler elements.
- Also used when the model geometry is transferred from a 3-D CAD system, and a large amount of time and effort is required to convert it to a 2-D or shell form.

Element Order

- Element order refers to the polynomial order of the element's *shape functions*.
- What is a shape function?
  – It is a mathematical function that gives the "shape" of the results within the element. Since FEA solves for DOF values only at nodes, we need the shape function to map the nodal DOF values to points within the element.
  – The shape function represents <u>assumed</u> behavior for a given element.
  – How well each assumed element shape function matches the true behavior directly affects the accuracy of the solution, as shown on the next slide.

Comparing Element Order



Linear vs. Quadratic

Linear elements	Quadratic elements
• Can support only a linear variation of displacement and therefore (mostly) only a constant state of stress within a single element. • Highly sensitive to element distortion. • Acceptable if you are only interested in nominal stress results. • Need to use a large number of elements to resolve high stress gradients.	• Can support a quadratic variation of displacement and therefore a linear variation of stress within a single element. • Can represent curved edges and surfaces more accurately than linear elements. Not as sensitive to element distortion. • Recommended if you are interested in highly accurate stresses. • Give better results than linear elements, in many cases with fewer number of elements and total DOF.

Selecting Element Order

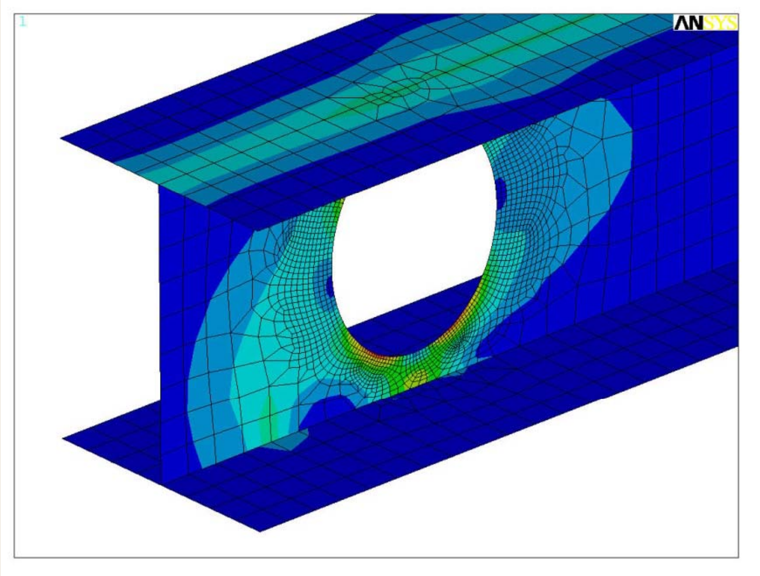
- When you choose an element type, you are implicitly choosing and accepting the element shape function assumed for that element type. Therefore, check the shape function information before you choose an element type.
- Typically, a linear element has only corner nodes, whereas a quadratic element also has midside nodes.

Selecting Element Order

- For shell models, the difference between linear and quadratic elements is not as dramatic as for solid models. Linear shells are therefore usually preferred.
- Besides linear and quadratic elements, a third kind is available, known as *p-elements*.   P-elements can support anywhere from a quadratic to an 8th-order variation of displacement within a single element *and* include automatic solution convergence controls.

Mesh Considerations

- For simple comparison, coarse mesh is OK
- For accurate stresses, finer mesh is needed
- Need finer mesh for fatigue
- Invest elements at locations of interest
- Avoid connecting quadratic and linear elements

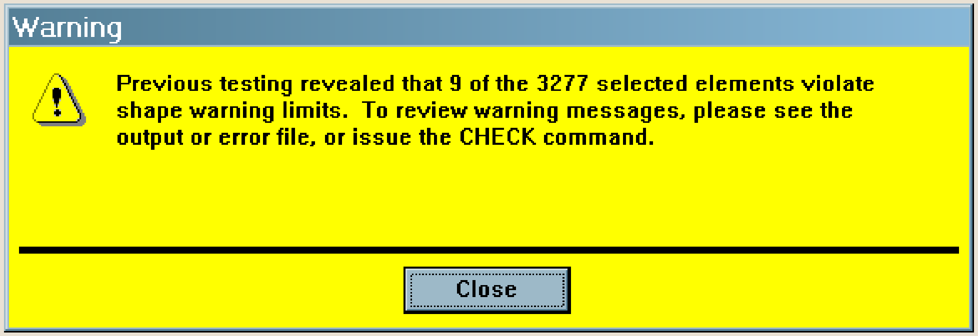


Mesh Distortion

- Elements distorted from their basic shape can be less accurate
- Greater the distortion, the greater the error
- Four types of distortion:
  - Aspect ratio (elongation)
  - Angular distortion (skew and taper)
  - Volumetric distortion
  - Mid node position distortion (higher order elements)

Mesh Distortion

- Most FEA packages have distortion checks, and will warn you if elements exceed those limits
- These limits are **subjective**, and a ‘bad’ element might not give erroneous results, and a ‘good’ element might not give accurate results!

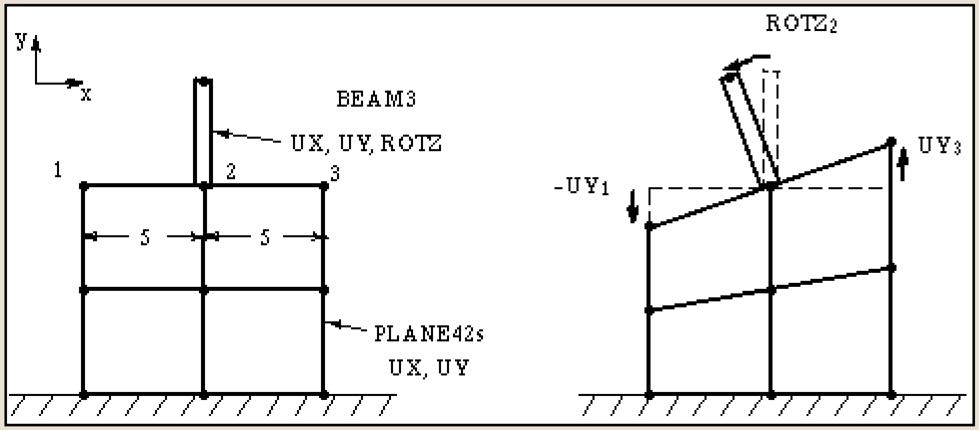


Connecting Different Element Types

- Meshing your entire structure is not always feasible– it’s nice to model some parts with simpler elements
- Can embed shells in solid elements to connect them, but be careful of doubling the stiffness– better to use MPC connection or Solid-Shell element
- Use constraint equation or MPC to connect shell to solid, beam to solid or beam to shell.

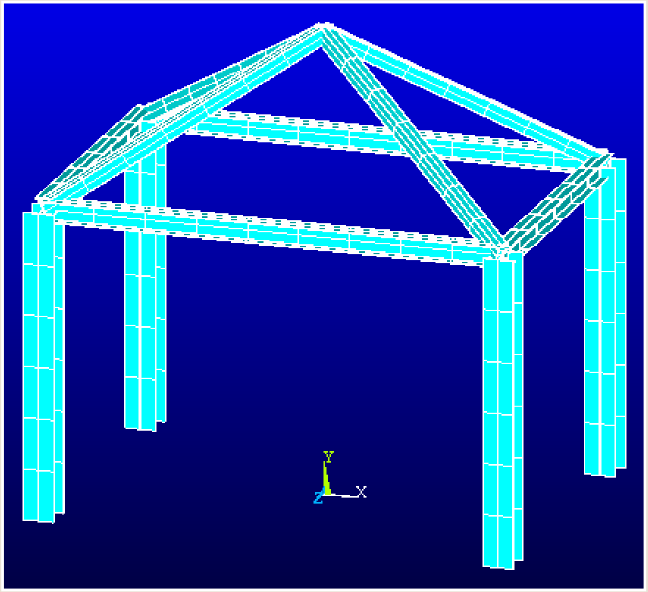
Connecting Different Element Types

- Constraint equation and MPCs allows single node to drive a set of nodes– so that moments are transferred properly



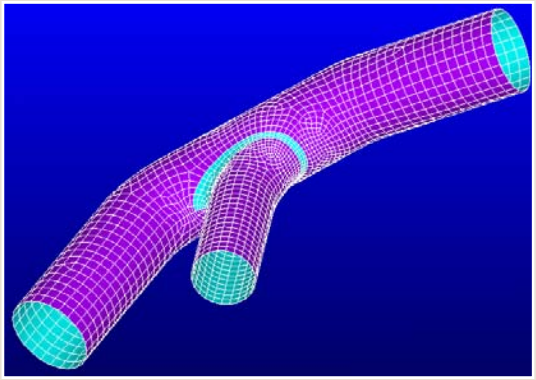
Check Real Constants

- Beam's often have a 'strong' and 'weak' axis– important to have proper stiffness resisting loads
- In ANSYS, use /eshape,on to turn on beam and shell cross-sections



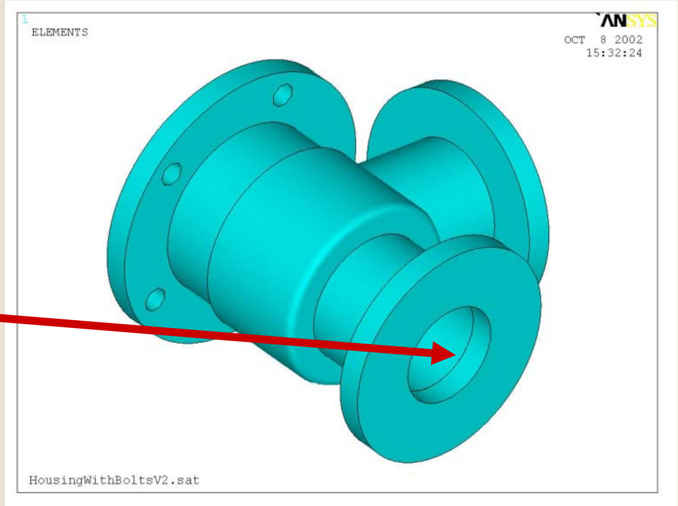
Check shell normals

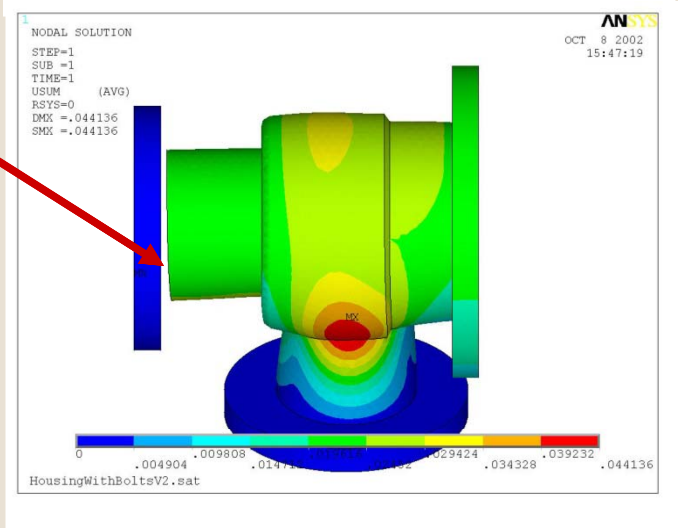
- Shells have a ‘bottom’, ‘middle’ and ‘top’
- Bending through thickness means stress on ‘top’ and ‘bottom’ will differ
- Want all ‘tops’ in the same direction, so that the stresses make sense
- Positive pressure is oriented opposite to the element normal (i.e., into the element)
- /PSYM,ADIR to view normals



Element connectivity

- Make sure there are no ‘cracks’ in the model
- Turn on edge plotting
- Apply a dummy load and solve, then view the displacements
- Can also use a shrink plot to check connectivity
- Can add density and do a modal analysis-- mode shapes will show cracks





Mesh Convergence

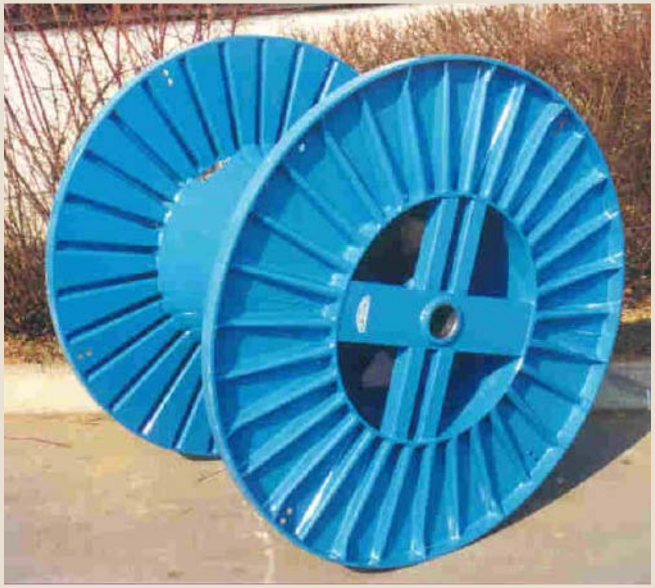
- FEA Theory: as mesh gets finer, it gets closer to real answer
- Mesh once, solve, mesh finer, solve again; if results change within a certain percentage, the mesh is converged, otherwise, repeat
- Perform a mesh convergence on a problem with a known answer to get a better understanding
- Displacement results converge faster than stress results

Boundary conditions

- Boundary conditions provide the known values for the matrix solution
- You always have to include them– even for a balanced pressure situation
- Can use symmetry to support a balanced load situation
- Can use weak springs to support a balanced structure

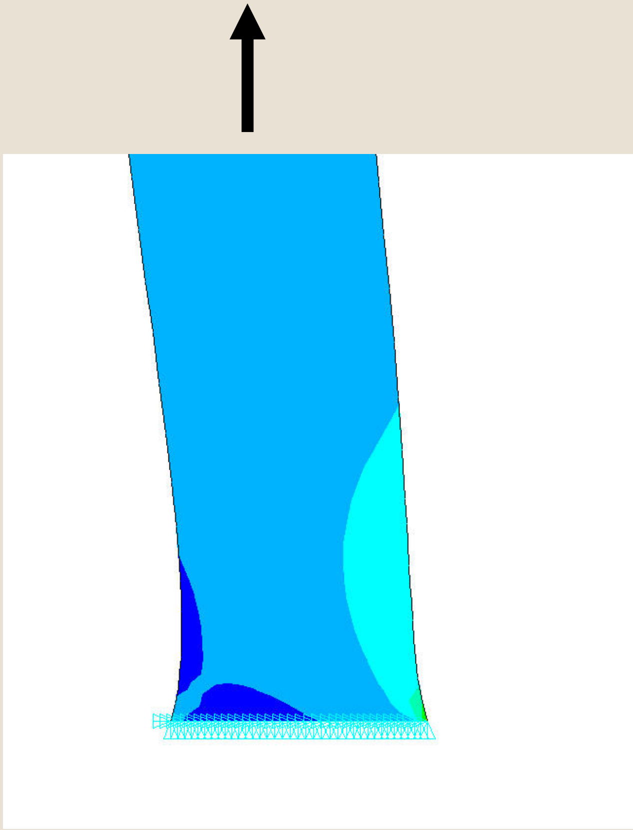
Boundary conditions

- Do the boundary conditions adequately reflect ‘real life’?
- There are no single-point or line supports; these are approximations we use. Real life has some small area
- Be wary of singularities
- If deformation of support isn't negligible, model support with coarse elements and use bonded contact to tie support to model



Be Careful about Fixed Supports

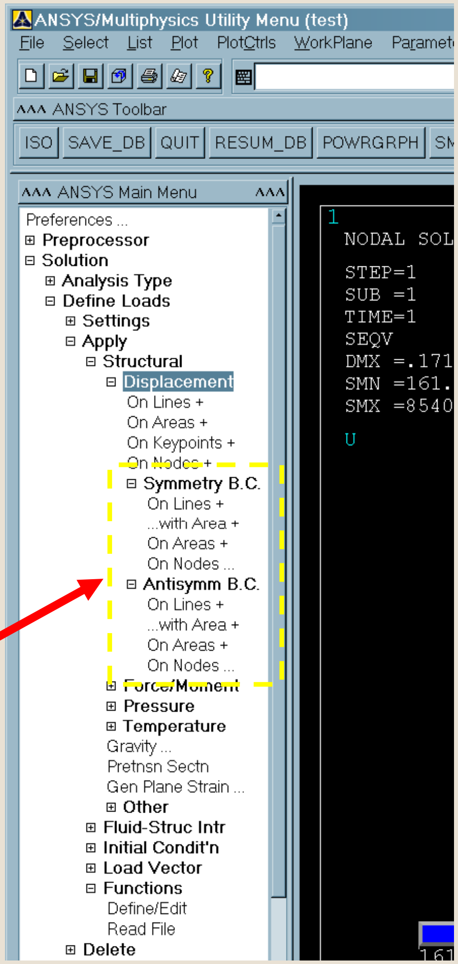
- The Poisson effect: when something is pulled in one direction, it shrinks in the perpendicular directions
- A fixed support will prevent this shrinking, which leads to a singularity
- Is there a potential of 'lift-off' at the support? Might need to model support and contact



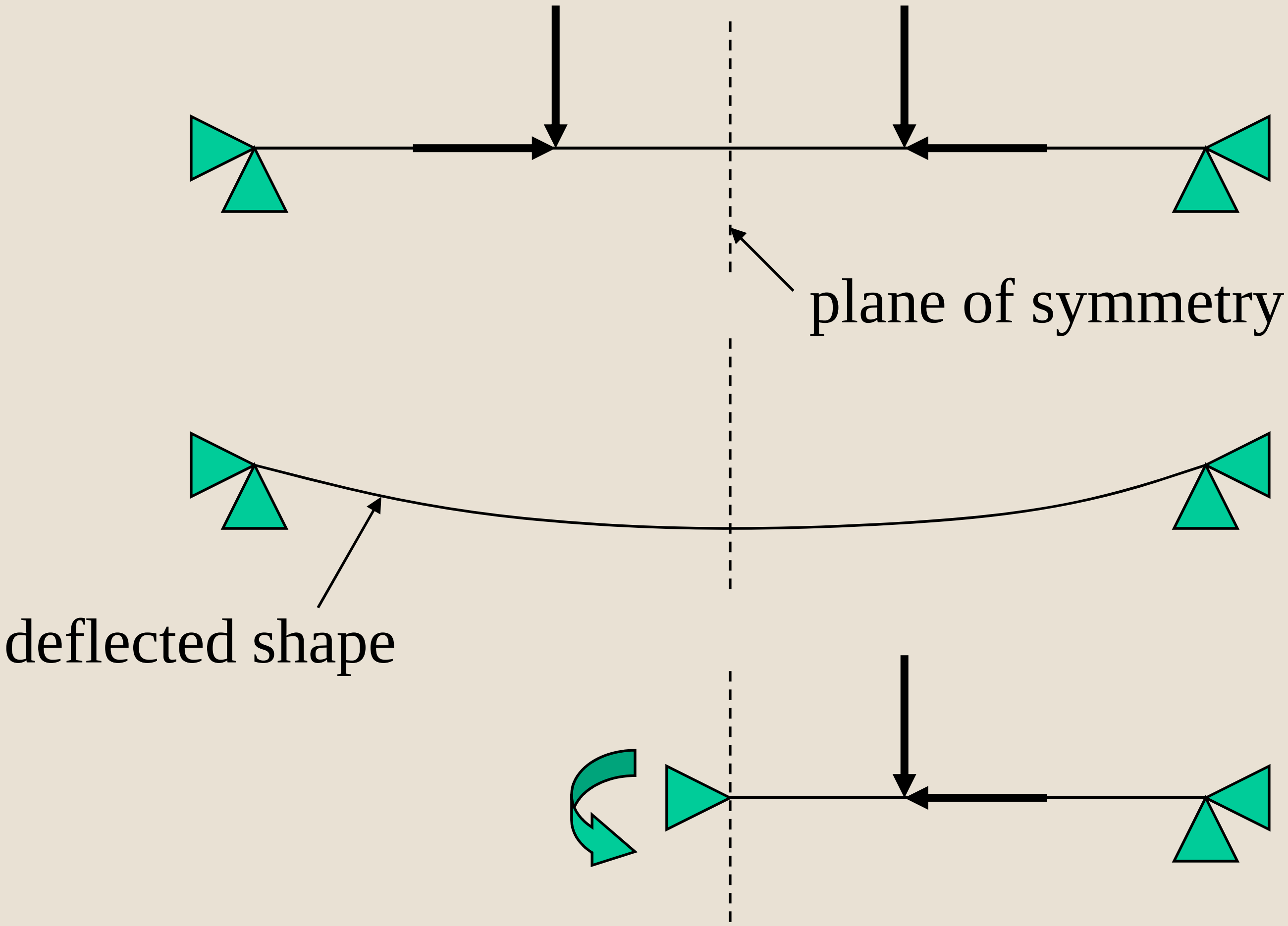
Symmetric and Antisymmetric Constraints

- Symmetry BC: Out-of-plane displacements and in-plane rotations are fixed.
- Antisymmetry BC: In-plane displacements and out-of-plane rotations are fixed.

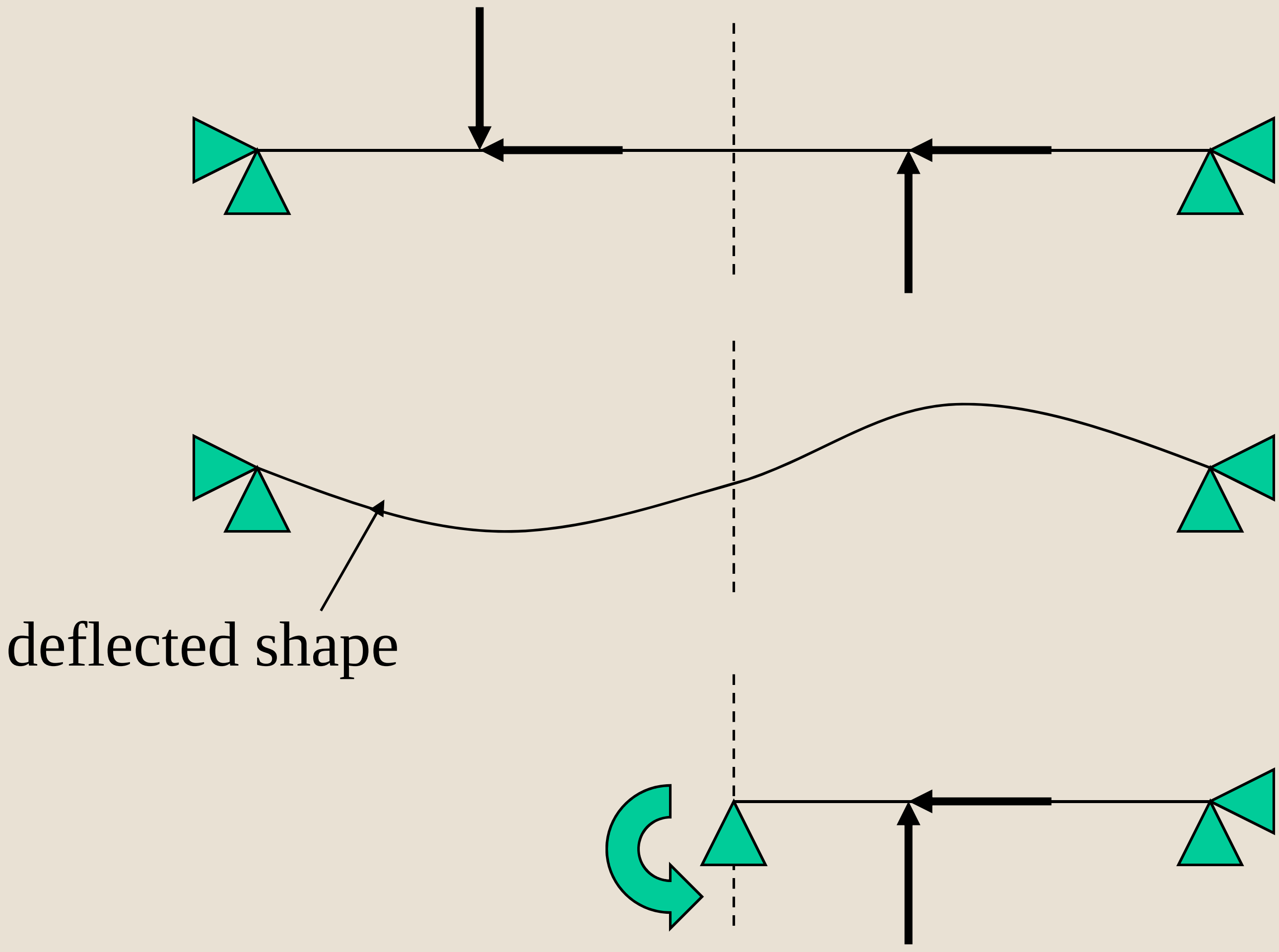
- Many packages have a single command to apply these constraints



Symmetric Constraints

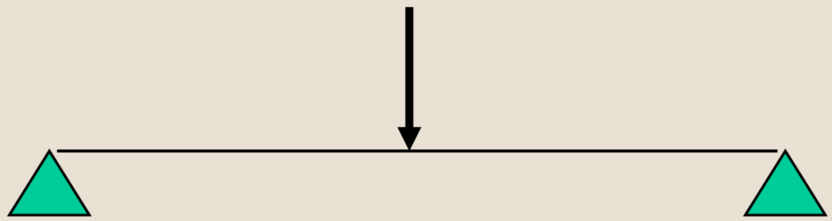


Antisymmetric Constraints



Prevent rigid body motion

- For structural static, need to constrain for translation and rotation.
- Even though the beam below is in equilibrium, you need a constraint in the horizontal direction, or you will get ‘pivot errors’



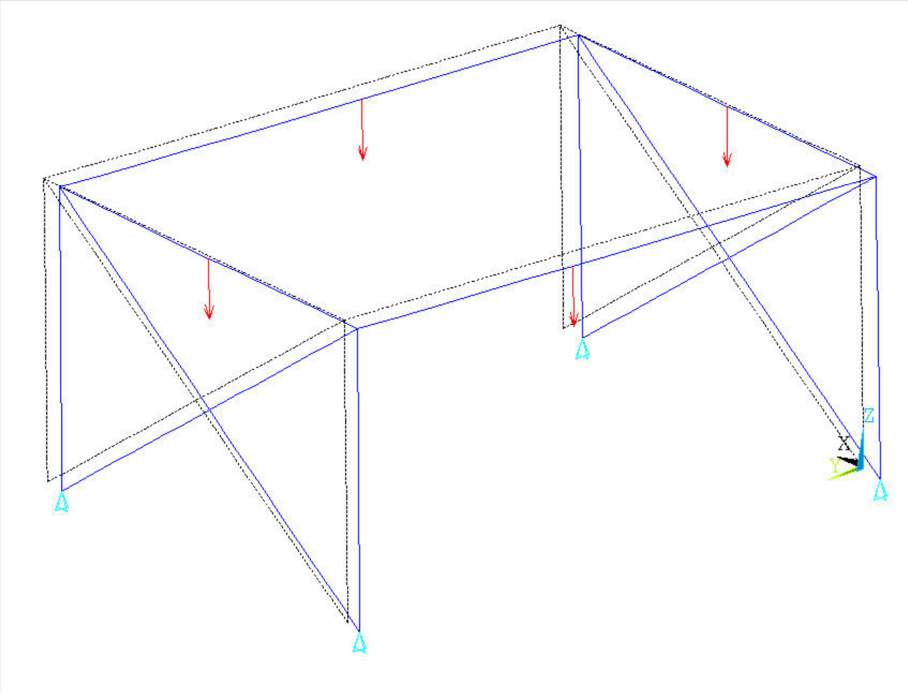
Pivot Errors?

- The program doesn’t have any known values for displacements in that direction, so it can’t solve the matrix

```
Small negative pivot equal to -9.953975677E-06 is found.  Check for
 unconstrained model.

*** WARNING ***                         CP=      90.110   TIME= 14:42:08
Small equation solver pivot term= 9.953975677E-06 encountered at UY DOF
 of node 822.  Check for an insufficiently constrained model.

*** ERROR ***                           CP=      90.170   TIME= 14:42:08
DOF (e.g.  Displacement) limit exceeded at time 1
 (load step 1 substep 1 equilibrium iteration 1)
 Max.absolute value= 168101384 (limit= 1000000) at UY of node 15 .
  May be due to an unrestrained or unstable model.
```

Prevent Rigid Body Motion

- Set loads to zero and run– if model doesn’t converge, or results look odd, you have rigid body motion.
- Or, add density and do modal analysis



Loading

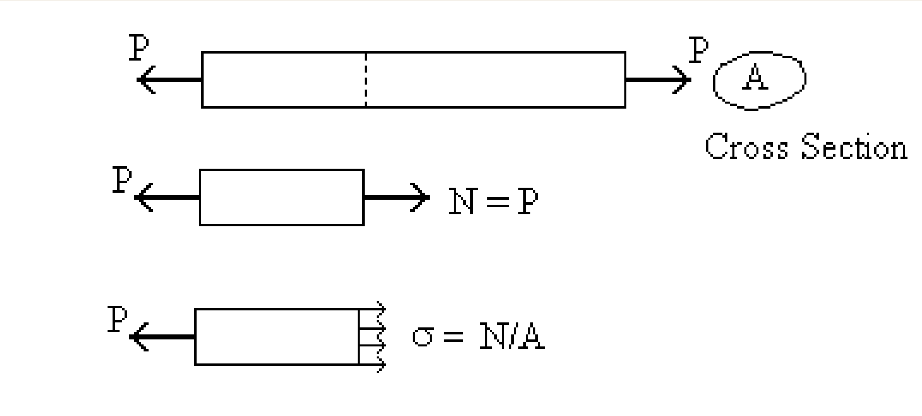
- Do the loads match real life?
- There are no point loads in real life, just really small areas with pressures on them
- Investigate all possible combinations of loads
- Make sure you have density entered if turning on gravity or doing modal analysis
- Might need to model load application device/structure

Loading

- Consider range of load values
  - Parametric analysis of different values
- Consider long-term vs. short-term loads
  - Long-term analyzed for creep and fatigue
  - Short-term analyzed for yielding
- Think about the load path through the structure

St. Venant's Principle

Two systems of loads which are statically equivalent will produce approximately the same effect at locations remote from the loads.



- This can be used to simplify loads or structures

Check Reactions

ANSYS®

- For statics, sum of forces = sum of reactions
- Can identify misplaced loads, incorrect units, geometry mistakes, typos
- Check reactions at contact pairs

PRRSOL Command

File

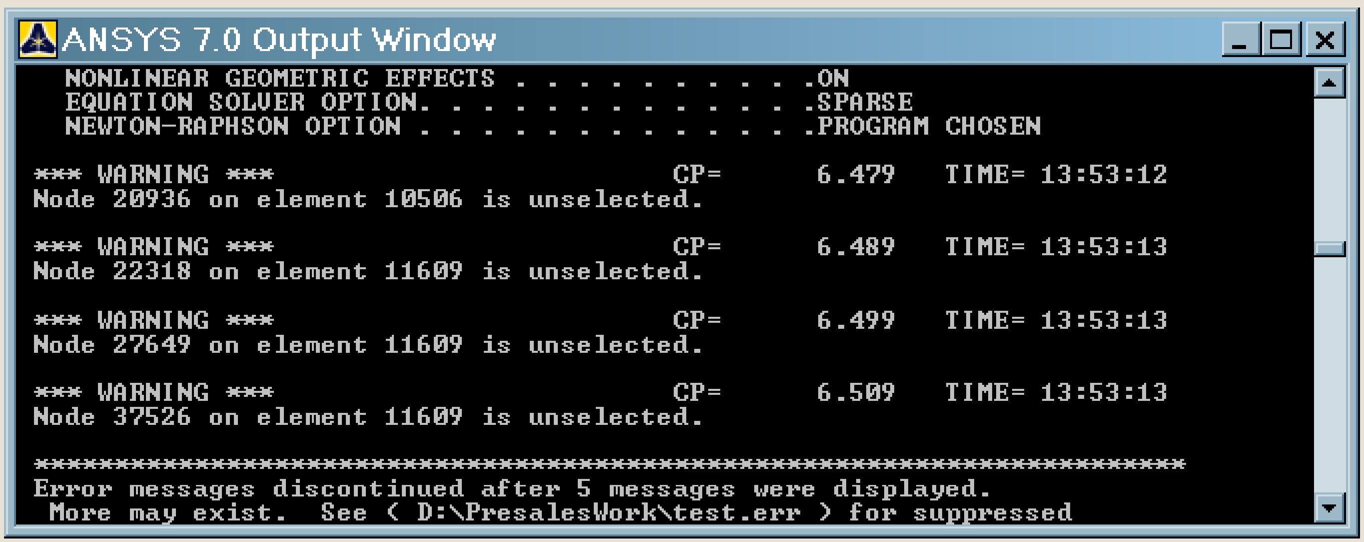
```
PRINT FY    REACTION SOLUTIONS PER NODE

 ***** POST1 TOTAL REACTION SOLUTION LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0

 THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

   NODE       FY
      1    500.00
      2    500.00

TOTAL VALUES
VALUE    1000.0
```

Check for buckling

- Occurs when part of structure is slender, and under compressive stress (will not happen under tension).
- Linear buckling (eigenvalue) is <u>UNCONSERVATIVE</u>!
- Nonlinear buckling is more accurate, but also more difficult (need to use full Newton-Raphson method)

Watch Your Errors and Warnings

- Warnings about loads or constraints not applied because the node or element are nonexistent
- Undefined material properties



Large differences in stiffness

- Possible source of problem when the ratio of the maximum and minimum element stiffness coefficients should be less than 1e8
- Otherwise, the stiffer part of the model will ‘crash through’ the less stiff part
- Can indicate problem with inputs

```
Range of element maximum matrix coefficients in global coordinates
Maximum= 287420016 at element 978.
Minimum= 4.418766981E-03 at element 1677.

*** WARNING ***                          CP=       4.727   TIME= 15:02:27
Coefficient ratio exceeds 1.0e8 - Check results.
```

Evaluating Results

- Stress criteria
- Factor of safety
- Is stress greater than yield?
- Don't assume the results are correct!
- Are the displacements in the expected range?
- Compare to tests or theory, when possible
- Does the displaced shape make sense?
- Check out stress hotspots

Evaluating Results

- Use linearization, if needed
- Check the whole model– don’t focus so much on one spot, you miss a problem elsewhere
- Check reactions against applied loads
- Check contact pairs for penetration
- Check element error to assess mesh
- Plot unaveraged stresses

Results Verification

- Use deformed animation to check loads and look for cracks in model
- Combined load behavior is sometimes difficult to predict– consider separating each load into its own load case to check

Do Test Cases

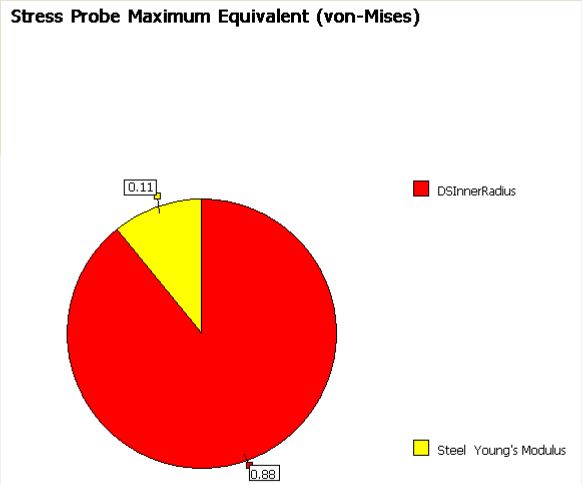
- Do a simplified case to learn about behavior
  - Do 2D instead of 3D, beam instead of 3D
  - Bonded contact instead of frictional
  - Elastic instead of plastic or hyperelastic
  - Hand calcs!
- Find pitfalls BEFORE running a model for several days…

Sensitivity Analysis

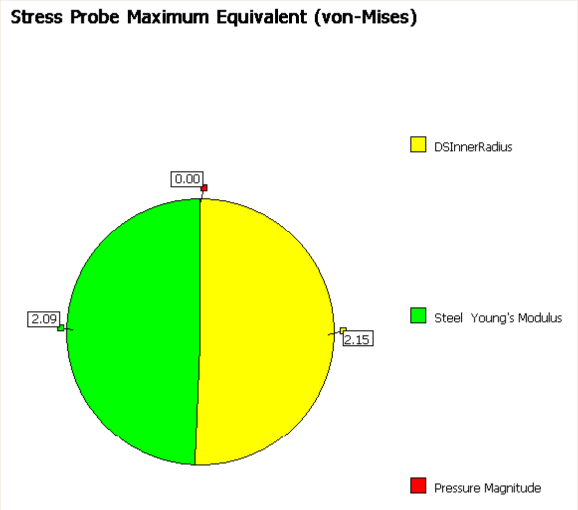
- Process of discovering the effects of model input parameters on response
- Can provide insight into model characteristics
- Can assist in design of experiments
- Should be subject to same scrutiny as all V&V

Sensitivity Analysis

- A probabilistic analysis using your model and statistical data of input parameters to see how much variation there is in output
- Generally requires several analyses– very time consuming
- Positive sensitivity indicates that increasing the value of the uncertainty variable increases the value of the result parameter



Global Sensitivity



Local Sensitivity

Calibration

- Also known as “model updating”, “model tuning”, “parameter calibration”
- Adjust for
  - Compliance in joints
  - Damping
  - Unmeasured excitations
  - Uncertain boundary conditions
  - Material variations

Model Calibration

- Determines model’s fitting ability, NOT predictive capability
- Use tests or hand calcs to tweak the model
  - Tests used for calibration CAN’T be used for validation
- Mirror the test behavior as close as possible when calibrating

Calibration

- Calibrate against
  - Test data
  - Field data
  - Engineering experience
  - Hand calculations
  - Simplified analyses

Validation

- Making sure the FE model will be accurate for a specified range of loads
- Use experimental data (different from calibration data)
- Engineering experience
- Hand calculations
- Don’t just assume the model is correct!!

Document Everything!

- Detail all decisions made
- Explain simplifications
- Detail loads and supports
- Document material data
- Document test data
- Document as much results data as possible
  - List reaction forces
  - Stresses
  - Displacements