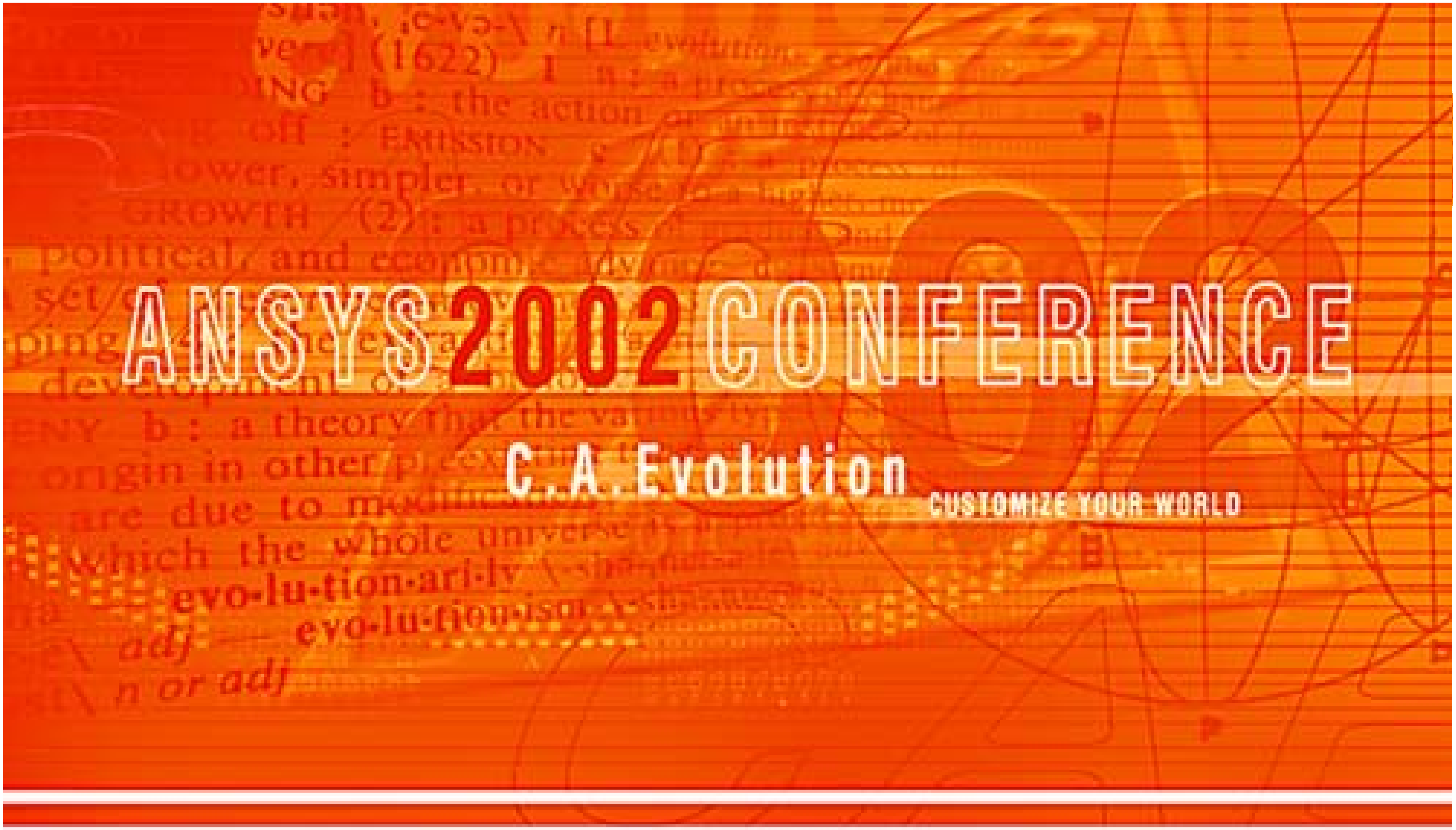


The Building Blocks of Simulation

A New Family of Elements for Stress Analyses

Mechanics Development Group

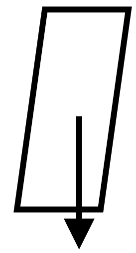
Contents

- Information provided here pertain to
  - 18x Solid Elements and
  - 18x Structural Elements
- Due to time constraints, we will focus only on solid elements
  - You are welcome to contact us with questions on structural elements and discuss

FE Model Creation

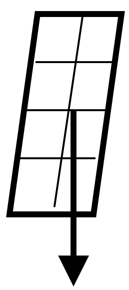
TARGET: accuracy and efficiency

BC, Geometry, Material



Real life problem

Mechanical Problem



Contact & parameters
Step length, convergence,
Solver

ELEMENTS

Finite element model

Overview

- Solid elements in general
- ANSYS 18x solid elements
- Technologies to alleviate volumetric locking
- Methods to avoid shear locking
- Generalized plane strain modeling (✷ new)
- Elements selection guidelines
- Convergence and some tips

Solid Elements; General Information

- Element orders and stress states
  - Order of interpolation functions
    - Lower order: linear, insensitive to mesh distortion
    - Higher order: quadratic, no shear locking, accurate for curved boundary
  - Stress States
    - 2D
      - Plane stress
      - Plane strain
      - Axisymmetric
      - Generalized plane strain
    - 3D

Solid Elements; General *(cont.)*

- Element primary variables (commonly used)
  - Displacements only
  - Displacements and hydrostatic pressures
- Element technologies
  - Full integration (regular/conventional)
  - Selective reduced integration
  - Uniform reduced integration
  - Enhanced strain formulations

18X Solid Elements

- Very rich functionality

18x Solid Elements	Numbers of Nodes	Dimensions	Element Shapes	Element Order	Interpolation	Stress States					Element Technologies			Formulation Options		
						Plane Stress	Plane Strain	Axisymmetric	Generalized Plane Strain	3D	Selective Resuced Integration/B-Bar	Uniform Reduced Integration	Enhanced Strain	Displacement	Mixed u/P (nearly)	Mixed u/P (fully)
PLANE182	4	2D	Quad.	Low/Linear	Bilinear	•	•	•	•		•	•	•	•	•	•
PLANE183	8	2D	Quad.	High/Quad	Seren.	•	•	•	•			•		•	•	•
SOLID185	8	3D	Brick	Low/Linear	Trilinear					•	•	•	•	•	•	•
SOLID186	20	3D	Brick	High/Quad	Seren.					•		•		•	•	•
SOLID187	10	3D	Tet.	High/Quad	Tet					•		•		•	•	•
						keyopt(3)					keyopt(1) or keyopt(2)			keyopt(6)		

- Element technologies: keyopt(1) for 182, keyopt(2) for 185
- Stress states : keyopt(3)
- Formulation options: keyopt(6), ANSYS automatically chooses between nearly and fully incompressible materials
- Total combinations: 72 !

Why So Many Choices ? *(72 Combinations)*

- General solution is often the most expensive (especially in nonlinear analysis)
- Factors that influence element selection are
  - Material behavior
    - Elastic, Plastic, Hyperelastic
      - Compressible, nearly incompressible and fully incompressible
  - Structural behavior
    - Bulk or bending dominated deformation
      - Shear locking
  - Geometry and mesh quality
    - Element order and technologies

18X Solid Elements *(cont.)*

- Flexible architecture
  - Accepts all material models
    - Elastic, hyperelastic, viscoelastic
    - Plastic, viscoplastic, creep, cast iron
  - Supports all features & infrastructure
    - Initial stress, thermal loading, element death/birth
    - Static, transient, modal, buckling and restart
- Consistency has been a major theme
  - Consistent within the element family and constitutive algorithms
  - Accounts for follower pressure effects
  - Lagrange multipliers for constraints
  - Have minimum assumptions

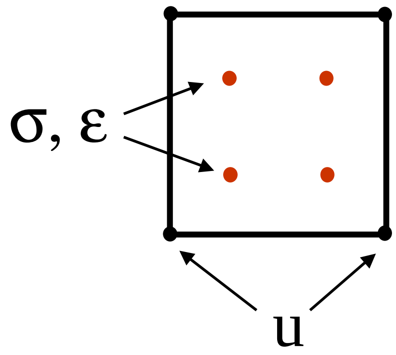
Volumetric Locking *(why?)*

- FE solutions:
  - For any element, DOF solution {u} is solved at nodes, usually accurate (based on mesh density)
  - Strains (and stress) are calculated at integration points by the derivative of DOF(s), less accurate

$$\{\Delta e\} = [B]\{\Delta u\}$$

$$\Delta e_V = \Delta e_x + \Delta e_y + \Delta e_z$$

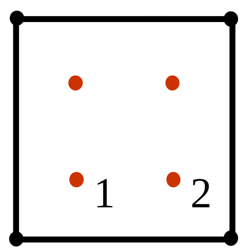
$$\Delta \sigma_m = K \Delta e_V, \qquad K = \frac{E}{3(1-2\nu)}$$



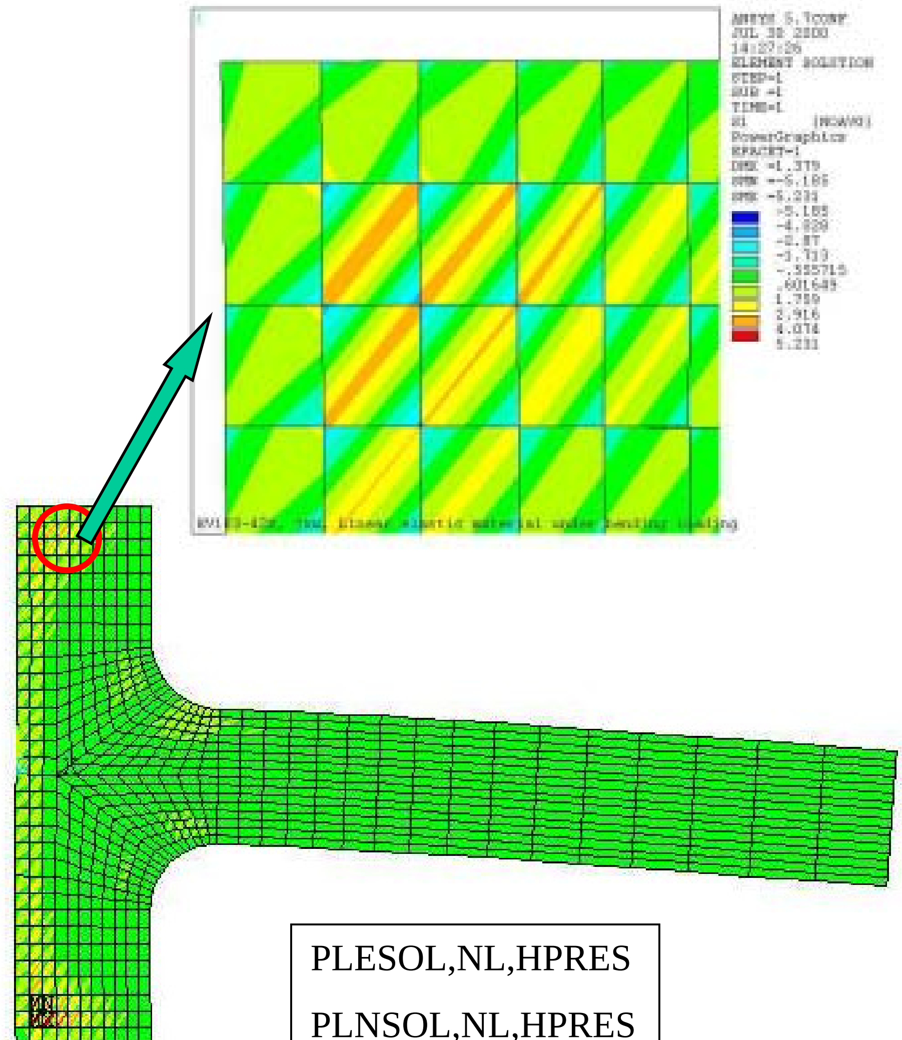
- Nearly incompressible material

$$\nu \approx 0.5, \qquad e.g. \quad \nu = 0.4999, \qquad if \quad E = 1.0 \times 10^6$$
$$K = 1.7 \times 10^9$$

Spurious Hydrostatic Stress



- Strain is less accurate:
  - At integration point 1,
    $$\Delta e_V = 1.0\times10^{-6}$$
  - At integration point 2 ,
    $$\Delta e_V = -1.0\times10^{-6}$$
- The hydrostatic pressure
  - At point 1:
    $$\Delta\sigma_m = 1700$$
  - At point 2:
    $$\Delta\sigma_m = -1700$$



PLESOL,NL,HPRES

PLNSOL,NL,HPRES

Conventional Formulations

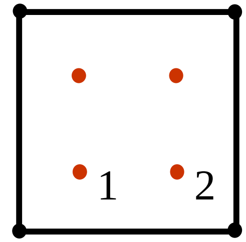
- Nearly incompressible material
  - Checkerboard pattern
  - Volumetric locking, deformation is less than it should be
  - Convergence problems in nonlinear analysis
- Fully incompressible material
  - Stress not solvable ! $\nu = 0.5, \quad K = \frac{E}{3(1-2\nu)} = \infty$
- Observations on volumetric locking
  - Full integration and pure displacement formulations is used
  - Checkerboard pattern is reduced if plotting nodal solutions, a averaged value
  - Not an issue in plane stress analysis, even for fully incompressible materials

Volumetric Locking *(how to detect ?)*

- The following may indicate volumetric locking
  - Checkerboard pattern of elemental hydrostatic stress plot, even normal stress (some times)
  - Deformation is much smaller than expected, especially in nonlinear analyses
  - Elemental direct stress plot is “sort of homogeneous” in a inhomogeneous deformation, usually with high values
  - Elemental direct stress plot is quite different from nodal direct stress plot; minimum or maximum stress values are quite different
  - Nonlinear analysis doesn’t converge

Selective Reduced Integration

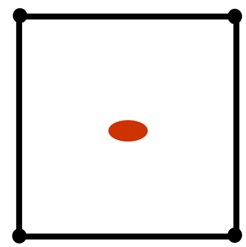
- Use averaged Δe_V in stress calculation
  - Stress calculation is split into volumetric and deviatoric parts by:



$$\sigma_{ij} = \sigma_m + \sigma'_{ij}$$
$$\sigma_m = \frac{E}{3(1-2\nu)} e_V = -p$$
$$\sigma'_{ij} = 2Ge'_{ij}$$

  - Deviatoric strain component e'_{ij} is evaluated at each Gauss integration point, as in the conventional formulation
  - Volumetric strain e_V is evaluated by: (mean dilation)

$$\overline{e_V} = \frac{\int_V e_V dV}{V}$$



  - It can be understood as integrating the volumetric part at one integration point

Outcome of Selective Reduced Integration

- Integration rule is one order less
    - The integration rule for volumetric term is one order less than the rule for deviatoric terms---selective reduced integration
- Volumetric constraint is reduced
    - Volumetric constraint equation is only satisfied on averaged sense in element domain; a weaker constraint
    - Equations of deviatoric terms are satisfied at each integration pints
    - It makes structure softer and overcome volumetric locking

An Alternative Name: B-Bar

- When B matrix is formed

$$[B]=[B_V]+[B_d]$$

$$\left[\overline{B_V}\right]=\frac{\int [B_V]dV}{V}$$

$$\left[\overline{B}\right]=\left[\overline{B_V}\right]+[B_d]$$

$$\{e\}=\left[\overline{B}\right]\{u\}$$

  - Called B-Bar method

- Availability in 18x elements
  - Lower order element 182 and 185, currently the default technology
  - For plane strain, axisymmetric, generalized plane strain and 3D stress states
  - Since volumetric locking is not an issue for plane stress, B-Bar is not needed

Summary of Selective Reduced Integration

- Advantage
  - Can alleviate volumetric locking
  - Is good to nearly incompressible material
    – Any material, e.g. plastic, hyperelastic..
  - No hourglass modes
  - No additional degree of freedom required
  - Can be used with mixed u/P formulations
- Limitation
  - Parasitic shear strains still exist, so it may suffer from shear locking since deviatoric term is same as the conventional formulation
    – Should not be used in bending dominated problems
  - Doesn't work for fully incompressible material

Uniform Reduced Integration

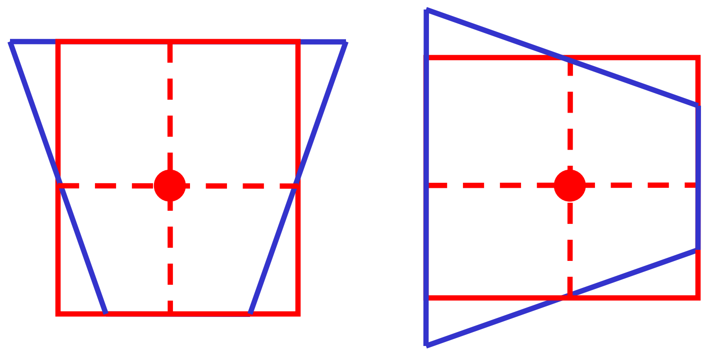
- Lower order integration rule than required for numerically exact integration used

Element Type	Full Integration Order	Reduced Integration Order
4 Node Quad	2x2	1x1
8 Node Quad	3x3	2x2
8 Node Hex	2x2x2	1x1x1
20 Node Hex	3x3x3	2x2x2

  - To reduce the volumetric constraint equations so that the volumetric locking can be alleviated
  - Also use lower order to integrate the deviatoric terms so that shear locking in low order elements can be avoided
  - Less CPU for element calculations (especially with material nonlinearities)

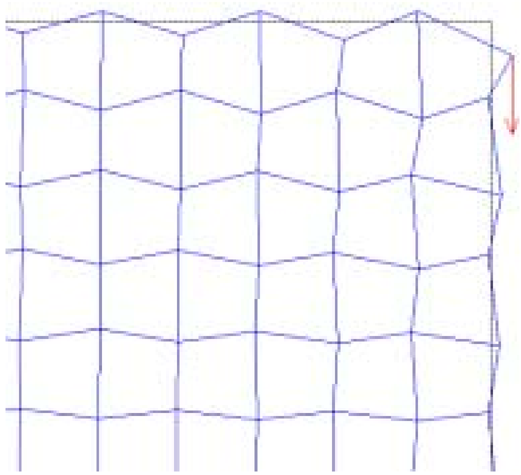
Hourglass Modes (*what is ?*)

- Zero energy mode
  - Unfortunately, the reduced integration of deviatoric terms causes modes of deformation which have zero strain energy, called *zero energy or hourglass modes*.
- An issue in lower order elements
  - With one integration point shown below, two modes of deformation are illustrated where the single integration point does not capture any strain energy in the element. By themselves, these are uncontrollable modes of deformation which lead to physically unrealistic behavior.
- Not an issue in higher order element
  - Will not propagate if more than one element in each direction



Hourglass Modes (*how to detect ?*)

– The following may indicate hourglass modes exist
  - Plotted deformed mesh has hourglass/“zig-zig’’ shape
  - Absolute values of displacements are very high, but strain and stress values are OK
  - The averaged displacement of neighbor nodes seem to give reasonable/better results
  - Excessive displacements for the loading than expected
  - Displacements don’t converge In nonlinear analysis



Hourglass Control

- Hourglass stiffness
  - Is automatically added in lower order elements (182 and 185)
  - Is small and used to control zero energy
  - Is an artificial energy and doesn't have physical significance
  - The default calculated value is usually sufficient for most analyses, but user can scale the stiffness by changing the scaling factor (default to 1.0)
  - If the scaling factor is too small, Hourglass mode may not be avoided; if the scaling factor is too large, the solution may not be accurate since too much artificial energy is introduced. The artificial energy content should not be larger than 5% of total energy. (command:ETABLE, Item: SENE and AENE)

Hourglass Control *(cont.)*

- The other ways user can try
  - Use refined mesh, which will prevent hourglass mode from propagating
  - Try to apply distributed loads instead of point loads since they tend to excite hourglass modes
  - Try to use “distributed” displacement constraints instead of single point constraints for the same reason as above
  - Switch to other element formulations

Summary of Uniform Reduced Integration

- Availability in 18x elements
  - Default for all higher order elements: 183, 186, and 187
    - Barlow points to calculate stress, optimal point for accuracy
    - *Hourglass propagation is NOT an issue with these elements*
  - Specified by keyopt (1)=1 in element 182 and keyopt(2)=1 in 185
  - Available for all deformations: plane stress, plane strain, axisymmetric, generalized plane strain and 3D
- Advantages
  - Can be used in nearly incompressible problems to overcome volumetric locking
  - Can be used in bending problems without worrying about shear locking
  - No additional DOF are required
  - Less CPU time is required for element calculations
  - File sizes (e.g., *.esav) are reduced. This provides efficient solutions, especially for nonlinear problems.
  - Elements are compatible with ANSYS/LS-DYNA explicit-dynamics elements

Summary of URI *(cont.)*

  - Higher-order URI elements have no hourglass modes, as long as there is more than one element in any direction
  - Can be used with mixed u/P formulations
- Limitations
  - Lower-order URI elements are susceptible to hourglassing, and this needs to be checked
  - Lower order URI elements may be too flexible, especially in bending-dominated problems, so a finer mesh may be required such that displacements are not over-predicted
  - URI elements have an integration rule which is one order lower than full integration. Hence, more elements may be required to capture stress gradients.
  - URI cannot be used in fully incompressible analyses.
  - Spurious modes may arise in modal analysis

Volumetric Locking *(alternative solution)*

- Increase element DOFs
  - Add internal degree of freedoms at element level and enhance strain calculation—Enhanced strain formulation; (instead of reducing constraint equations as in selective and uniform reduced integration)
  - Is useful and practical only when used in conjunction with strain enhancement
  - “Extra Shapes” formulation available in several elements is targeted towards avoiding shear locking only. Enhanced Strain Formulation, a generalized form of “extra shapes”, is aimed at avoiding both shear and volumetric locking.
    - Will be discussed when we cover Enhanced Strain Formulations

Mixed u/P Formulations *(why ?)*

- Limitations of pure displacement formulation
  - Accuracy of solution is dependent on Poisson's ratio
  - Nearly incompressible materials
    - Spurious pressures if no other algorithm is employed
      - Since volumetric strains are calculated from derivatives of displacements, these values are not as accurate as displacements. Any small error in volumetric strain will appear as large error in hydrostatic pressure
    - Volumetric locking
  - Fully incompressible materials
    - Not solvable

Mixed u/P Formulations *(what is ?)*

- Hydrostatic pressure $\overline{p}$ is interpolated and solved independently as primary unknowns, in addition to displacements

$$\begin{bmatrix} K_{uu} & K_{up} \\ K_{pu} & K_{pp} \end{bmatrix} \begin{Bmatrix} \Delta u \\ \Delta \overline{p} \end{Bmatrix} = \begin{Bmatrix} \Delta F \\ 0 \end{Bmatrix}$$

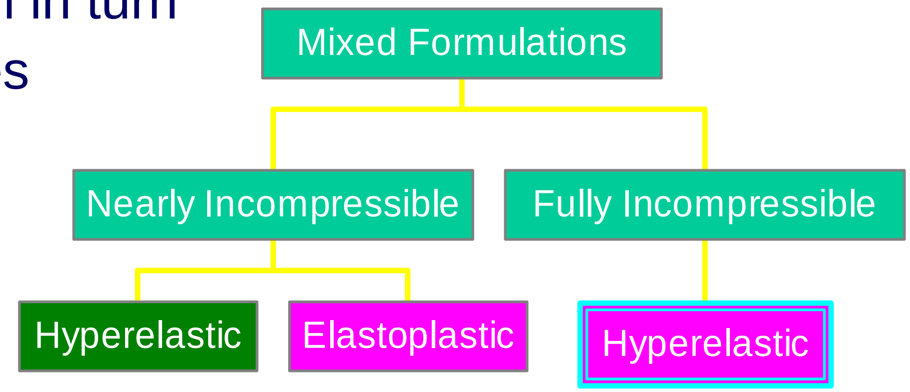
  - $\Delta \overline{p}$ is introduced by constraint equations. It is approximation of and usually different from Δp, which is from material constitutive law $\Delta p = -K \Delta e_V \quad \Delta e_V \Leftarrow \Delta u$
  - The stresses component is updated by $\Delta \overline{p}$ instead of Δp

$$\sigma_{ij} = \sigma_m + \sigma_{ij}^{'} = \sigma_{ij}^{'} - \overline{p}$$

  - The accuracy of solutions are independent from bulk modulus, Poisson's ratio and accuracy of volumetric strain

Mixed u/P Formulations in 18x

- Introduce constraint equations by Lagrange multiplier
- Incorporate finite strain effects
- $\Delta \bar{p}$ is kept to the global level, not condensed out at element level
- The interpolation function is one order lower than the one for volumetric strain
- Are for fully AND nearly incompressible material
  - Final formulation should be different according to different constraint equations, which in turn is decided by material types
  - Initiated by keyopt(6)=1 (or 2 for 187)
  - ANSYS automatically switches between different formulations



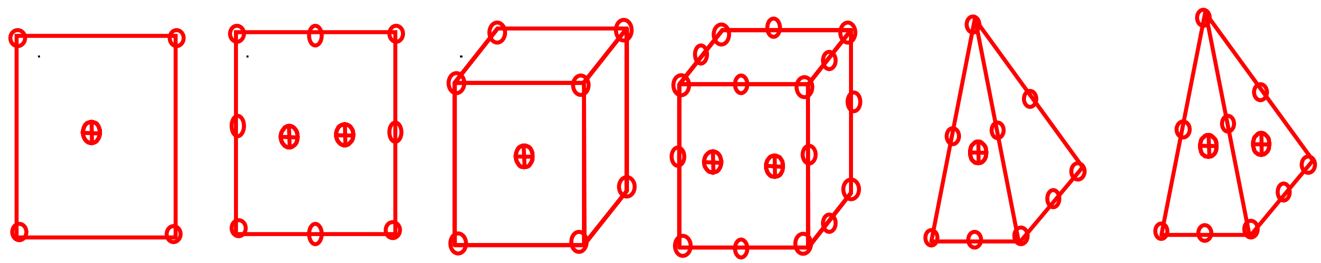
Interpolation of Hydrostatic Pressure

- Determined by displacement interpolation functions
- One order lower than the one for volumetric strain

Element	Keyopt(6)	# of $\bar{p}$	Interpolation Functions
182	1	1	$\bar{p} = \bar{p}_1$
182 Enh.	1	3	$\bar{p} = \bar{p}_1 + r\bar{p}_2 + s\bar{p}_3$
183	1	3	$\bar{p} = \bar{p}_1$
185	1	1	$\bar{p} = \bar{p}_1 + r\bar{p}_2 + s\bar{p}_3$
185 Enh.	1	4	$\bar{p} = \bar{p}_1 + r\bar{p}_2 + s\bar{p}_3 + t\bar{p}_4$
186	1	4	$\bar{p} = \bar{p}_1 + r\bar{p}_2 + s\bar{p}_3 + t\bar{p}_4$
187	1	1	$\bar{p} = \bar{p}_1$
187	2	4	$\bar{p} = \bar{p}_1 + r\bar{p}_2 + s\bar{p}_3 + t\bar{p}_4$

Internal Nodes for Mixed u/P

- To bring the pressure DOFS to global level
- Determined by the interpolation functions of $\bar{p}$

Element	182 (ENH)	183	185 (ENH)	186	187	187
Mixed						
Num. Of pres.	1 (3)	3	1 (4)	4	1	4
Internal Nodes	1 (2)	2	1 (2)	2	1	2

- Associated with only one element
- Created by ANSYS automatically
- Not accessible by users

Constraint Equations

- For all nearly incompressible material other than hyperelastic (elastoplastic)

$$\frac{p-\bar{p}}{K}=0 \qquad p=-\sigma_m=-\frac{1}{3}\sigma_{ii}$$

  – p is hydrostatic pressure from constitutive law;
  $\bar{p}$ is independently calculated hydrostatic pressure

  ⇒Volumetric compatibility

- For fully incompressible hyperelastic material

$$\frac{1-J}{J}=0 \qquad J=\left|F_{ij}\right|=\left|\frac{\partial x_i}{\partial X_j}\right|=\frac{dV}{dV^0}$$

  ⇒Volume is constant

- For nearly incompressible hyperelastic material

  (to be released soon)

Fundamental Equations

- Principle of virtual work

$$\int_V \sigma_{ij} \delta e_{ij} dV = \int_V f_i^{\,B} \delta u_i dV + \int_S f_i^{\,S} \delta u_i dS$$

$$e_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

- Impose constraint equations by Lagrange multiplier
- Consistent stiffness matrices
  - slope of non-linear force-displacement: differentiation of principle of virtual work, with minimal assumptions
  - Consider the effect of finite deformation

Nearly Incompressible *(Elastoplastic)*

- PVW with imposed Constraints

$$\delta\overline{W}_{\text{int}} = \int_V \sigma_{ij}\delta e_{ij}dV + \int_V (p-\overline{p})\delta e_V dV + \int_V \frac{p-\overline{p}}{K}\delta\overline{p}dV$$

- Element stiffness

Material stiffness (Part of Kuu)

$$D\delta\overline{W}_{\text{int}} = \int_V \delta e_{ij}C_{ijkl}De_{kl}dV - \int_V KDe_V\delta e_V dV$$

$$+\int_V \overline{\sigma}_{ij}\left(\frac{\partial\delta u_k}{\partial x_i}\frac{\partial Du_k}{\partial x_j} - 2\delta e_{ik}De_{kj}\right)dV$$

Stress stiffness

(part of Kuu)

(used with nlgeom,on)

$$\underbrace{-\int_V (D\overline{p}\delta e_V + De_V\delta\overline{p})dV}_{\text{Material stiffness (Kup and Kpu)}} - \boxed{\frac{1}{K}\int_V D\overline{p}\delta\overline{p}dV}$$

Material stiffness (Kup and Kpu)

Material stiffness (Kpp)

Nearly Incompressible *(Elastoplastic)*

- Selective reduced integration—a particular case

$$\delta\overline{W}_{\text{int}} = \int_V \sigma_{ij}\delta e_{ij}dV + \int_V (p-\overline{p})\delta e_V dV + \int_V \frac{p-\overline{p}}{K}\delta\overline{p}dV$$

  - Assume $p = -Ke_V$ $\quad \overline{p} = -K\overline{e_V}$ and K is homogeneous

$$\delta\overline{W}_{\text{int}} = \int_V \sigma_{ij}{}'\delta e_{ij}{}'dV + \int_V K\overline{e_V}\delta e_V dV + \int_V K(\overline{e_V} - e_V)\delta\overline{e_V}dV$$

  - The last term is the constraint. Since $\delta\overline{e_V}$ is variation, if further assume that $\overline{e_V}$ is constant in an element:

$$\int_V K(\overline{e_V} - e_V)dV = 0 \Rightarrow \overline{e_V} = \frac{\int_V e_V dV}{V}$$

  - This is the selective reduced integration ! For elastoplastic material, if $\overline{p}$ is assumed constant in an element and eliminated by static condensation, mixed u/P is equivalent to SRI.

Additional Convergence Check

- Fully incompressible $$\left|\frac{\int_V \frac{J-1}{J} dV}{V}\right| = \left|\frac{V-V^0}{V}\right| \le tol_V$$

- Nearly incompressible $$\left|\frac{\int_V \frac{p-\bar{p}}{K} dV}{V}\right| \le tol_V$$

- Checked at each DOF of pressures
  - tol_V: tolerance for the compatibility
    - Default value: $tol_V = 10^{-5}$
    - Can be modified by: SOLC, , , tol_v

Additional Convergence Check *(cont.)*

- ANSYS reports the number of elements which don't satisfy constraints (volumetric compatibility)

```
*** LOAD STEP     1   SUBSTEP     3  COMPLETED.    CUM ITER =
*** TIME =  0.750000         TIME INC =  0.250000
*** AUTO STEP TIME:  NEXT TIME INC = 0.25000      UNCHANG

   1058  U-P ELEMENTS DO NOT SATISFY THE VOLUMETRIC COMPATIBILI
  FORCE CONVERGENCE VALUE  = 0.1220E+05  CRITERION=  76
  DISP CONVERGENCE VALUE   = 0.2559E-01  CRITERION= 0.2485E
 EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.2298E
   1054  U-P ELEMENTS DO NOT SATISFY THE VOLUMETRIC COMPATIBILI
  FORCE CONVERGENCE VALUE  =  5767.      CRITERION=  79
  DISP CONVERGENCE VALUE   = 0.3083E-02  CRITERION= 0.2530E
 EQUIL ITER   2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.3003E
    613  U-P ELEMENTS DO NOT SATISFY THE VOLUMETRIC COMPATIBILI
  FORCE CONVERGENCE VALUE  =  508.6      CRITERION=  800.3     <<< CONVER
  DISP CONVERGENCE VALUE   = 0.3223E-03  CRITERION= 0.2530E
 EQUIL ITER   3 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.4143E
    114  U-P ELEMENTS DO NOT SATISFY THE VOLUMETRIC COMPATIBILI
  FORCE CONVERGENCE VALUE  =  65.89      CRITERION=  816.4     <<< CONVER
  DISP CONVERGENCE VALUE   = 0.4432E-04  CRITERION= 0.2530E-03 <<< CONVER
 EQUIL ITER   4 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.8295E
 >>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATIO
```

FE Model of Mixed u/P Elements

- No. of constraint equations
  - Equations of fully incompressible materials

$$\boxed{(K_{uu})_{ij}\Delta u_j + (K_{up})_{il}\Delta \bar{p}_l = \Delta F_i \quad i,\, j = 1,2....Nd}$$ Equilibrium

$$\boxed{(K_{pu})_{kj}\Delta u_j = 0 \qquad\qquad k,l = 1,2....Np}$$ Constraint

  - Equations of nearly incompressible materials

$$\boxed{(K_{uu})_{ij}\Delta u_j + (K_{up})_{il}\Delta \bar{p}_l = \Delta F_i \quad i,\, j = 1,2....Nd}$$ Equilibrium

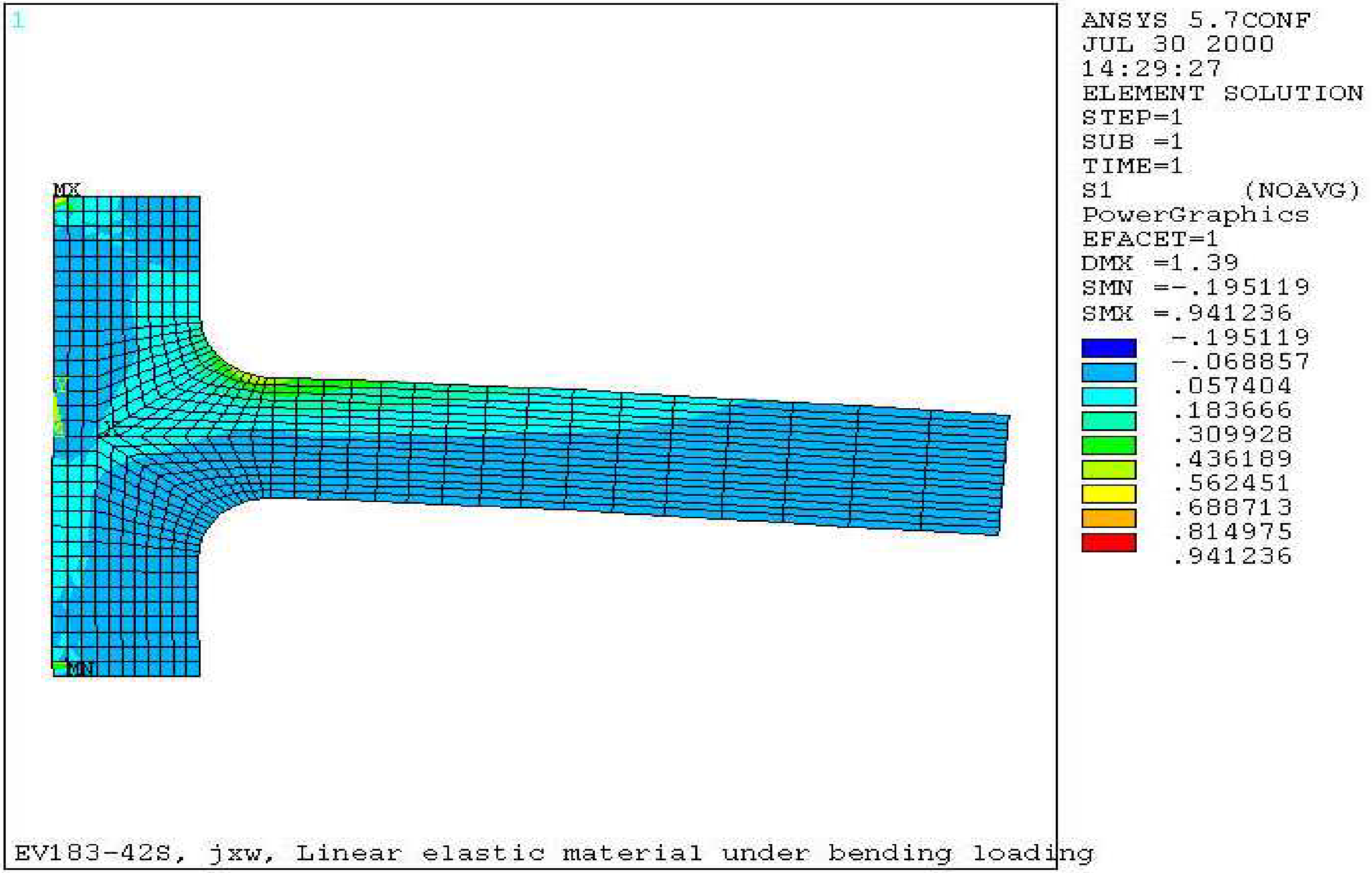
$$\boxed{(K_{pu})_{kj}\Delta u_j + (K_{pp})_{kl}\Delta \bar{p}_l = 0 \quad k,l = 1,2....Np}$$ Constraint

  - Different types of DOFs
    - Np: No. of constraint equations=No. of $\bar{p}$ DOFs
    - Nd: No. of displacement DOFs WITHOUT any constraint

FE Model of Mixed u/P Elements

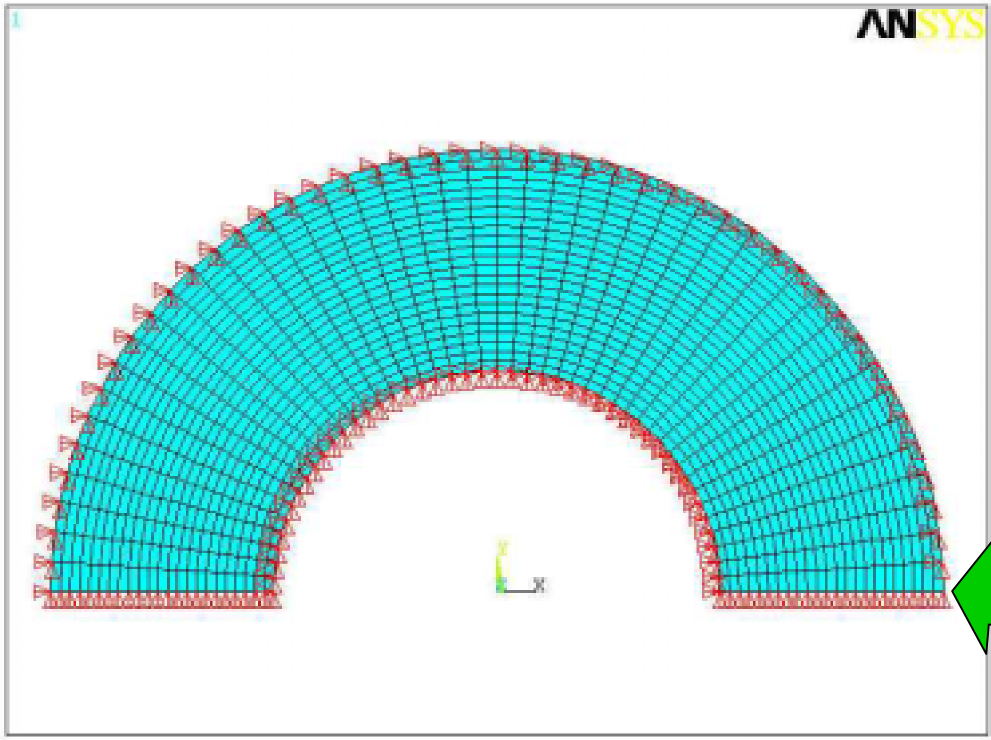
- Over-constrained model: Np>Nd
  - Fully incompressible: NO solution! Has to be avoided !
  - Nearly compressible: Very ill-conditioned system. Should be avoided !
- Best ratio of Nd/Np:
  - 2 for 2D problems
  - 3 for 3D problems
- No unique solution
  - If fully incompressible and all nodes on boundary have prescribed displacements (see the example)

Pressure with mixed u/P



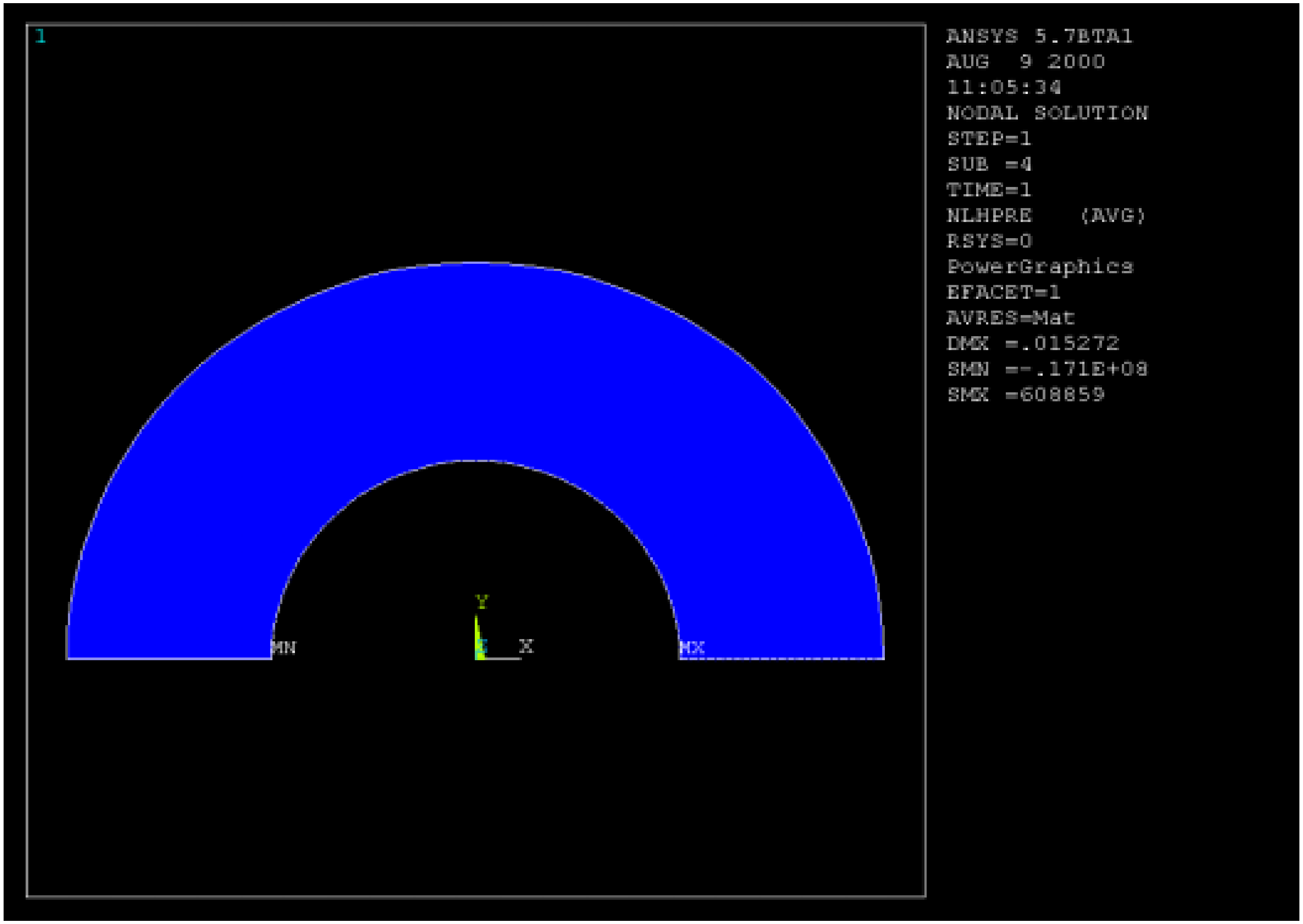
A Rubber Bushing *(between a frame and shaft)*

- Geometry
  - Plane strain
  - R_o=30mm, R_i=15mm
- Elements
  - Plane 182 with u/P
  - (Hyper 56)
- Mooney-Rivlin material
  - c1=0.177 N/mm^2
  - c2=0.045 N/mm^2
  - d=0.0 (Plane 182)
  - (μ=0.4998 (hyper 56))
- BC
  - frame fixed
  - shaft moves 9 mm to left (volume conservative)



To avoid no unique solution, keep at least ONE DOF on boundary free

Results of Plane 182

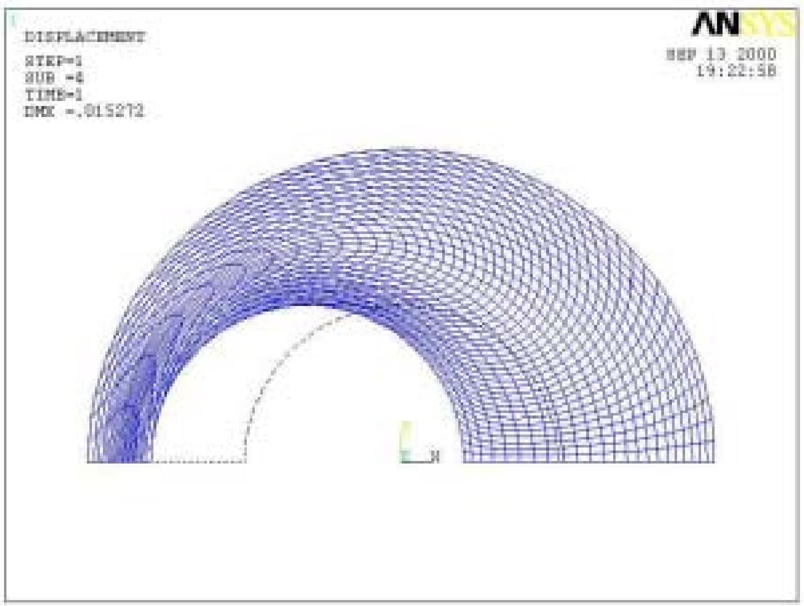


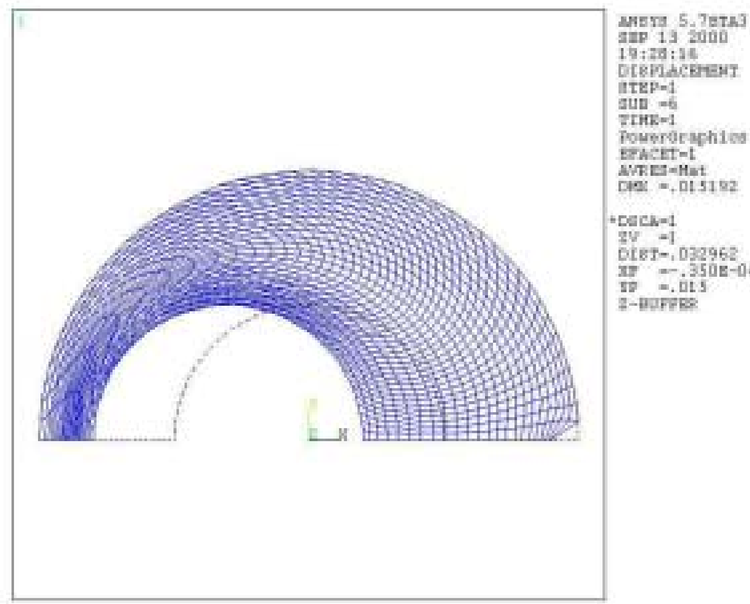
Results of Plane182 and Hyper 56

- Difference because of incompressibility

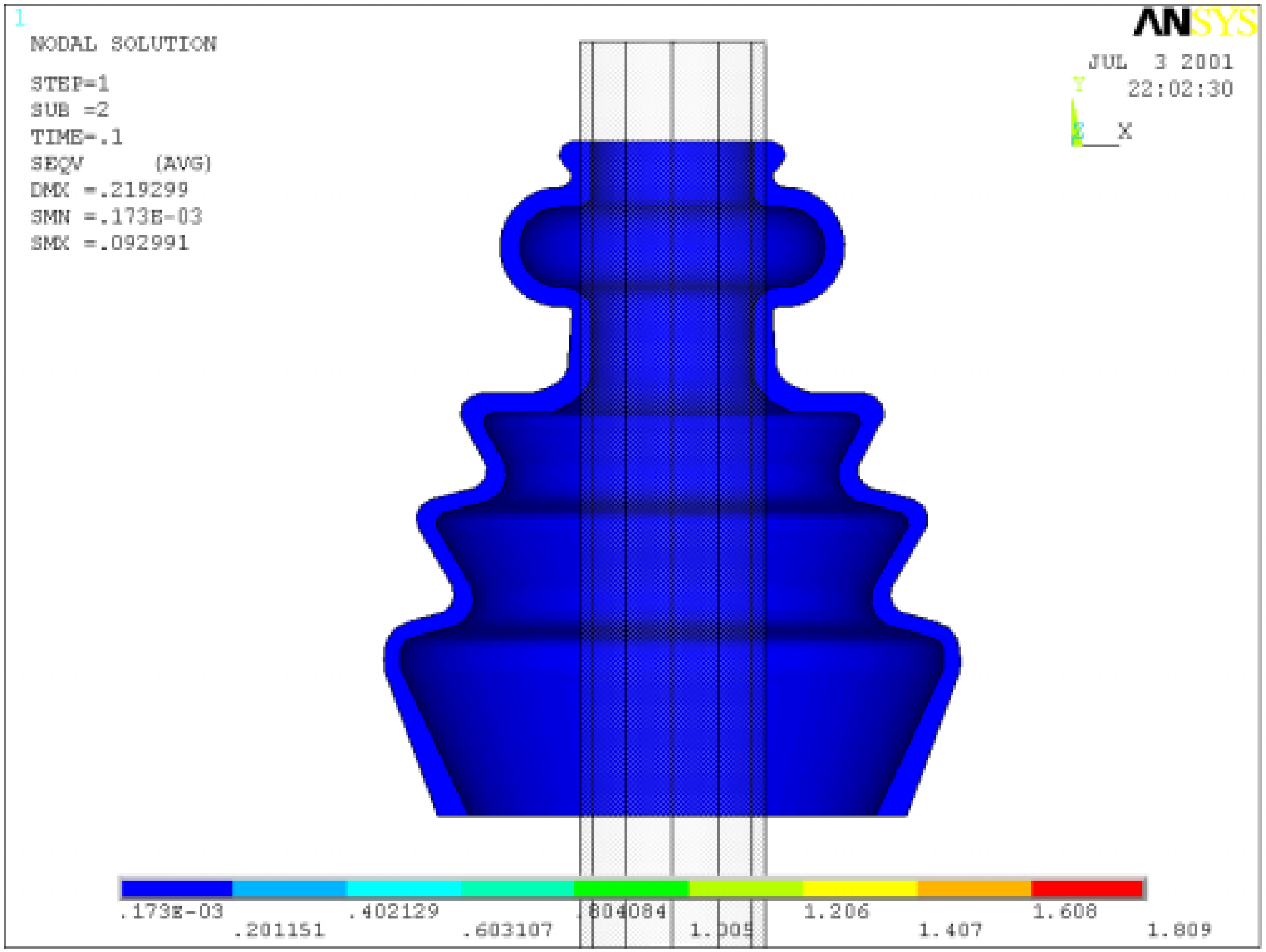
Fully (Plane 182 with U/P formulation)

Nearly (Hyper 56)



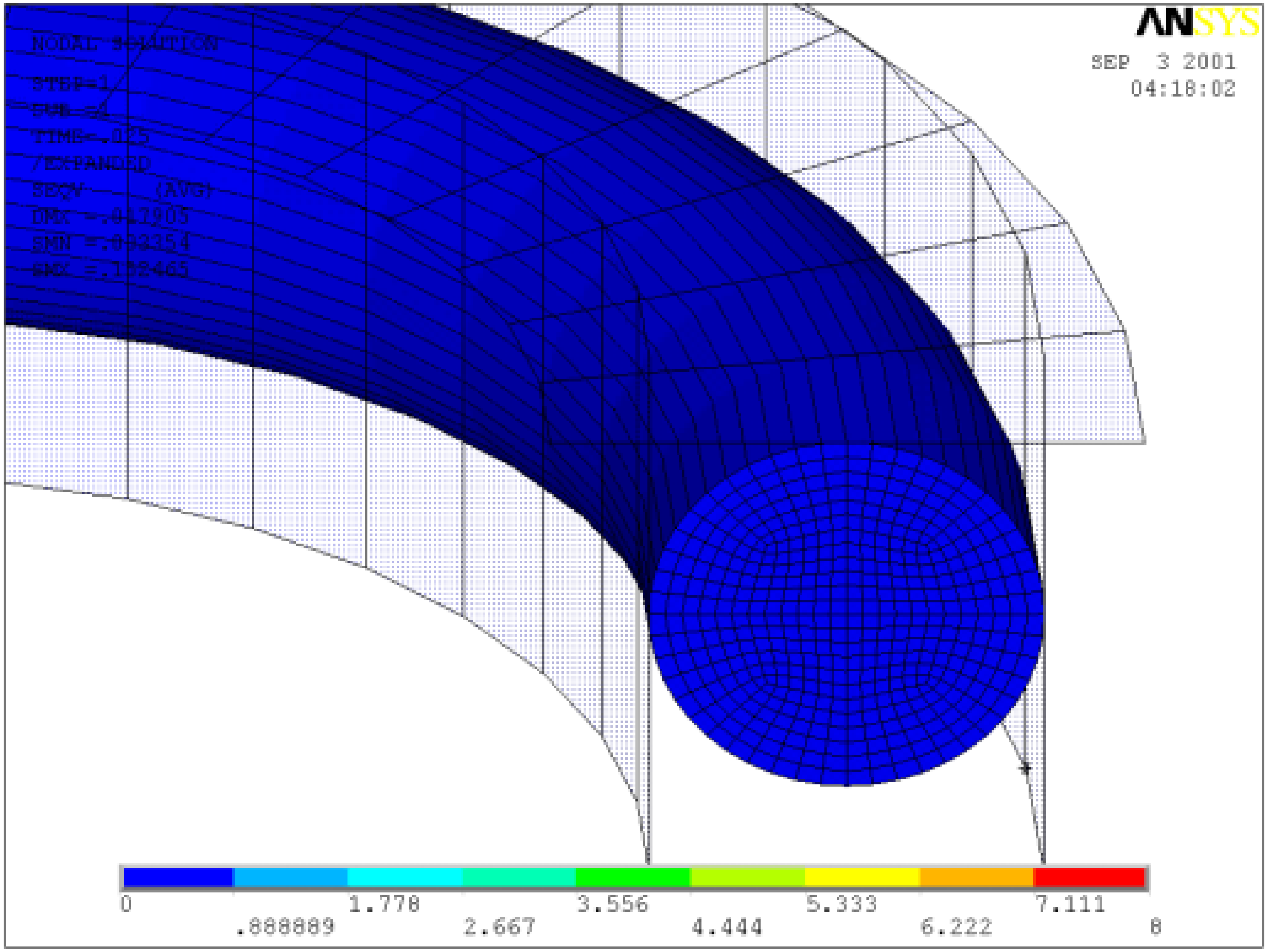


Example of a Rubber Boot



185 B-bar, mixed u/P with Neo-Hookean

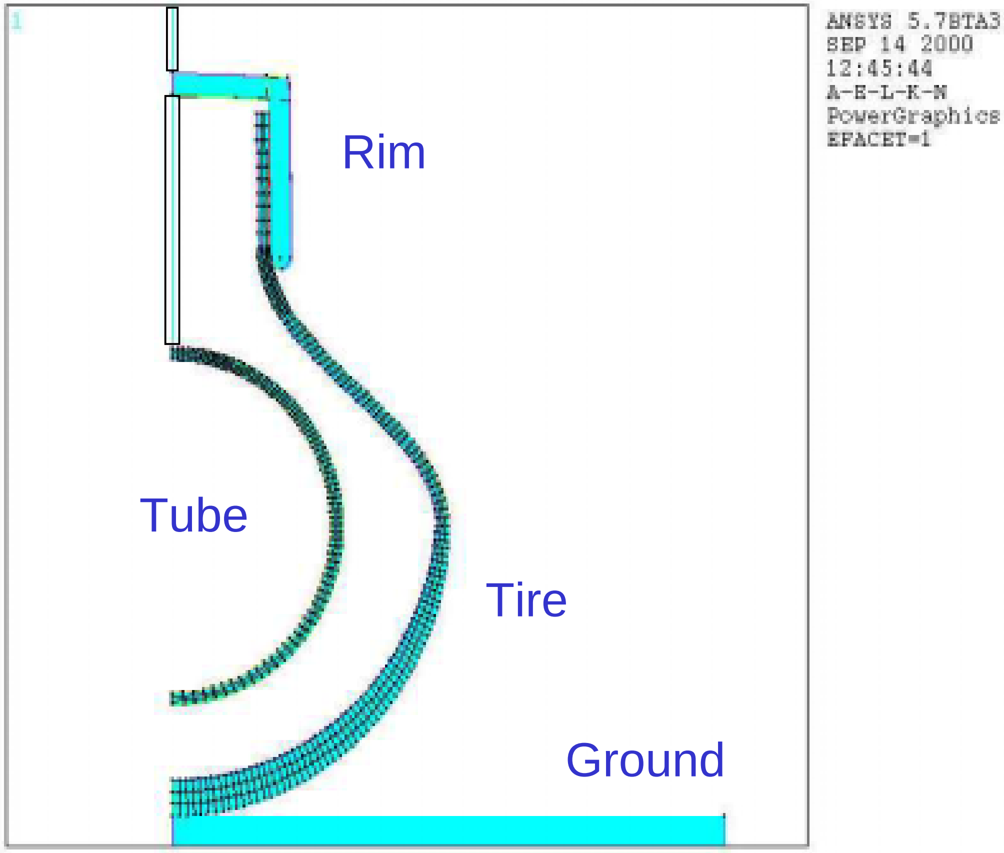
Example of O-ring Compression



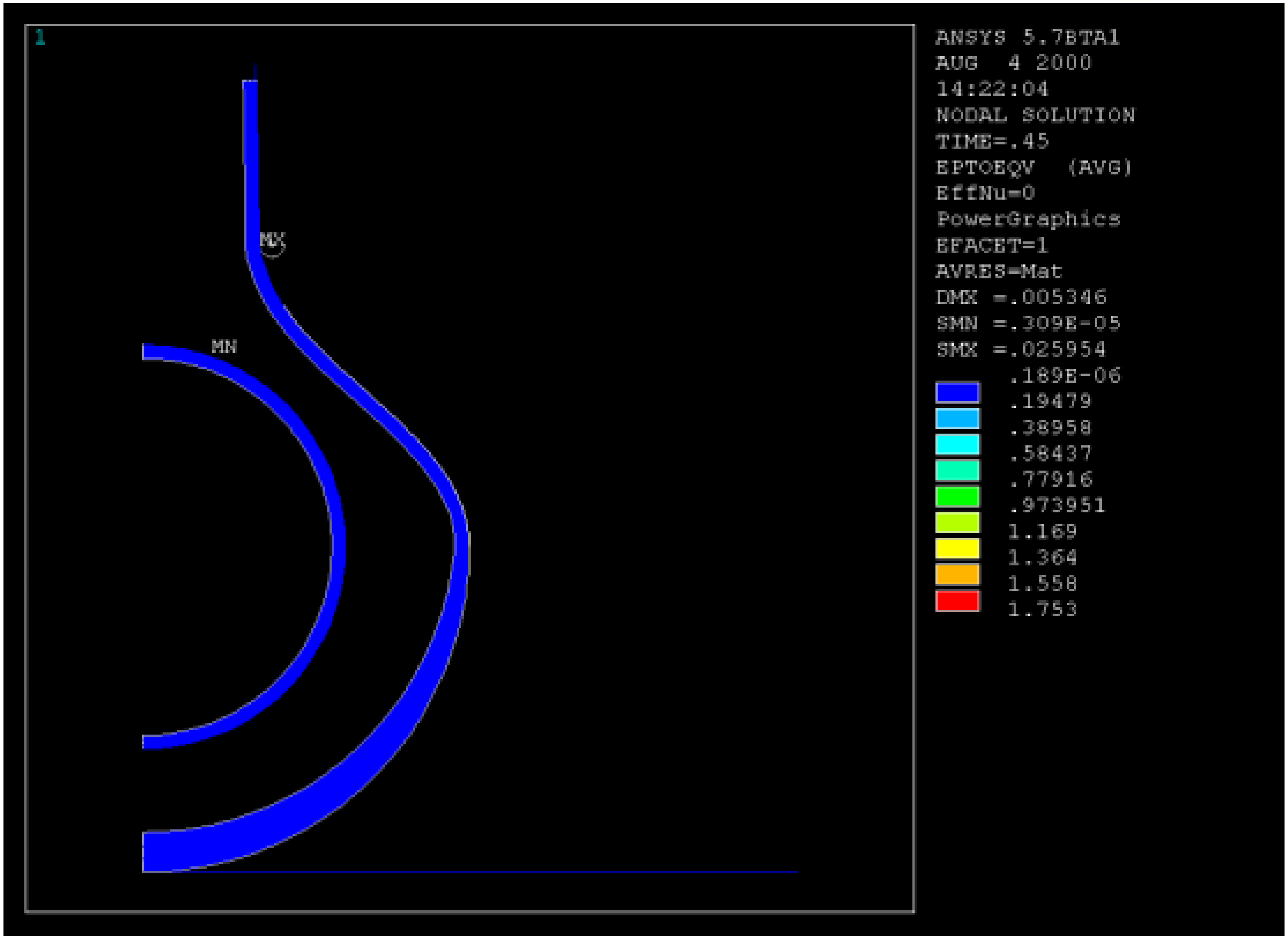
PLANE 182 with Ogden material model

Bike Tire Inflation and Pinch Flat

- Fully incompressible hyperelastic material
- Element 182 with mixed u/P formulation
- Surface to surface contact
- Self-contact
- Load step one: Inflation
- Load step two: pinch flat



Bike Tire Inflation and Pinch Flat *(cont.)*



Summary of Mixed u/P

- Availability in 18x elements
  - Available for all 18x solid elements
  - Available for axisymmetric, plane strain, generalized plane strain and 3D stress state but not needed for plane stress
  - Available for fully hyperelastic and nearly incompressible elastoplastic materials
- Advantages
  - The only formulation for fully incompressible materials
  - The best technology in handling incompressibility
  - Solution independent from bulk modulus
  - Can be combined with other element technologies
- Limitations
  - Additional DOFS introduced, not as efficient as pure displacement formulations

Notes on Mixed u/P

- For fully incompressible material
  - No unique solution if all nodes on BC have prescribed displacements
  - Not solvable if FE model is over constrained, i.e., nd<np
- For nearly incompressible material
  - Should try other element formulations first for efficiency
  - Stress and strain are consistent with constitutive law on average sense

$$\sigma_{ij} = \sigma_m + \sigma_{ij}^{'} = \sigma_{ij}^{'} - \overline{p} \qquad \overline{p} \neq K e_V$$

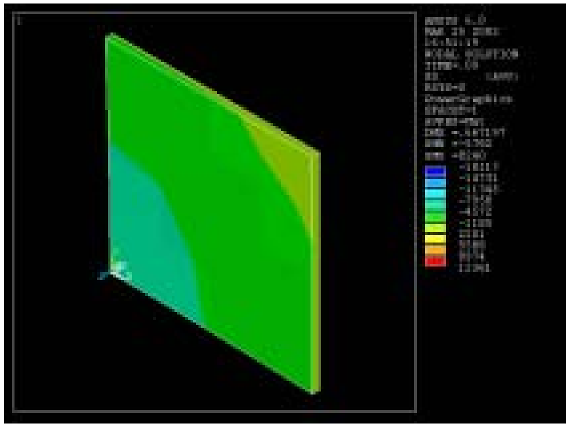
  - May be hard to solve if the model is over constrained, i.e., nd<np

Notes on Mixed u/P *(cont.)*

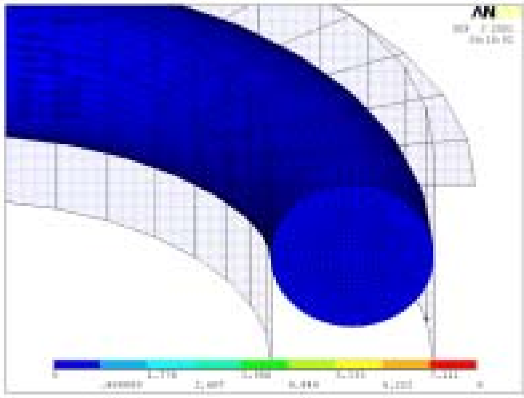
- For all materials
  - Try to keep the optimal ration of Nd/Np; 2 for 2D and 3 for 3D analysis
  - Use direct solver, sparse or front; sparse is the default at 6.1
- Tolerance for the volumetric compatibility
  - Default value of tolerance for the compatibility: $tol_V = 10^{-5}$ matches the default value for displacement convergence check 5%; and the default value for force: 0.5%. If either of them changes, the value for volumetric compatibility should be changed in the same direction
  - If displacement convergence check is turned off, a very large value (close to 1.0) should be used for the tolerance of the volumetric compatibility

Bending Dominant or Not ?

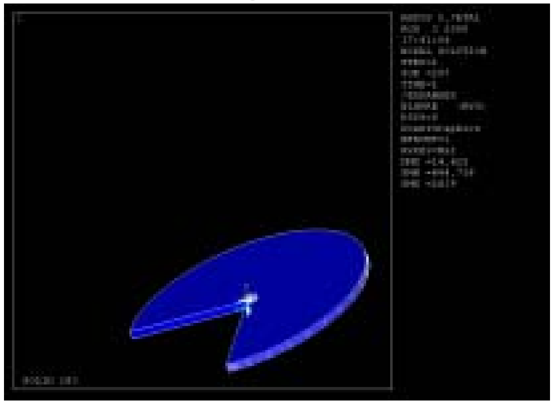
- Sometimes, it is not obvious

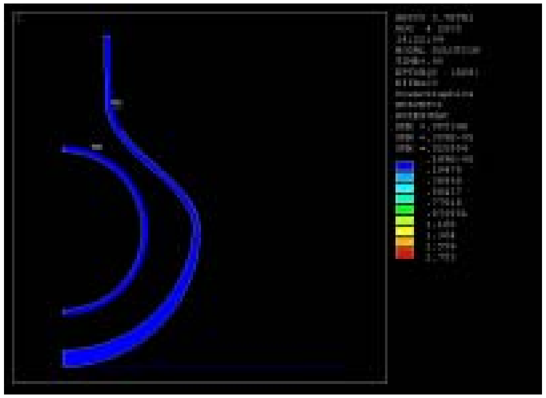


Bending dominant



Not bending dominant



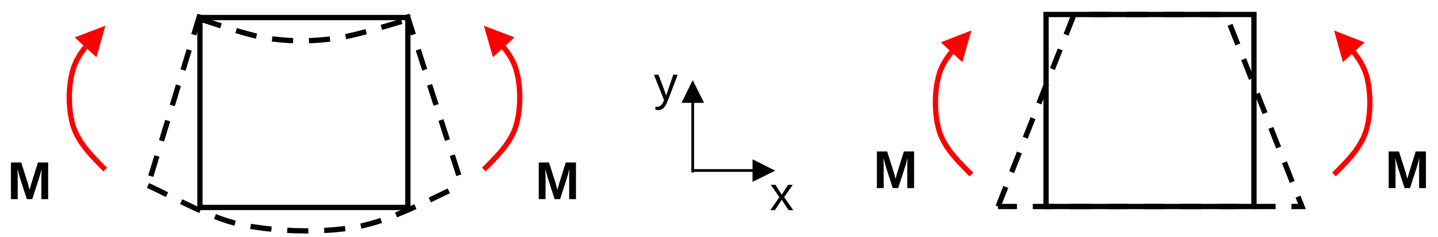


Looks bending dominant, but, actually not !

Shear Locking *(what is ?)*

Pure bending deformation for a differential volume, plane sections remain plane, top and bottom edges become arcs, $\gamma_{xy} = 0$.

Fully integrated lower order element deformation, top and bottom edges remain straight, right angles are not preserved, γ_{xy} is non zero. It causes parasitic stress.

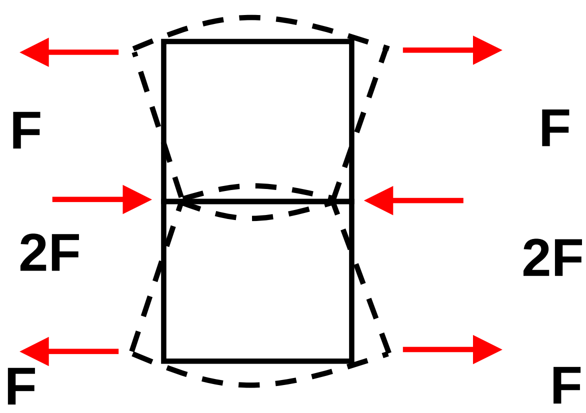


Shear Locking *(Cont.)*

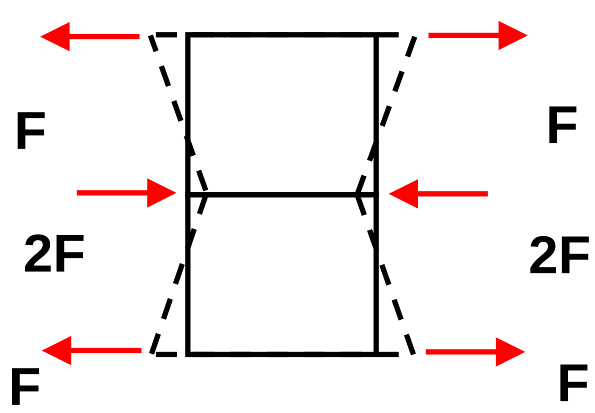
- Parasitic shear in fully integrated lower order elements causes stiffness overestimated, which in turn make the calculated displacements much less than they should be – Shear Locking.
- It happens in both geometric linear and non-linear analyses
- It is more obvious in thin/long structures subject to bending loads
- Error persists with mesh refinement

To Avoid Shear Locking

- Add additional DOFs
  - A simplified explanation is that the additional DOF augment the element's shape function to allow curvature (to cancel the parasitic shear in the regular lower order element)
  - They are also known as "incompatible modes" because they lead to gaps and overlaps in the mesh



Enhanced strain



No enhanced strain

To Avoid Shear Locking *(History)*

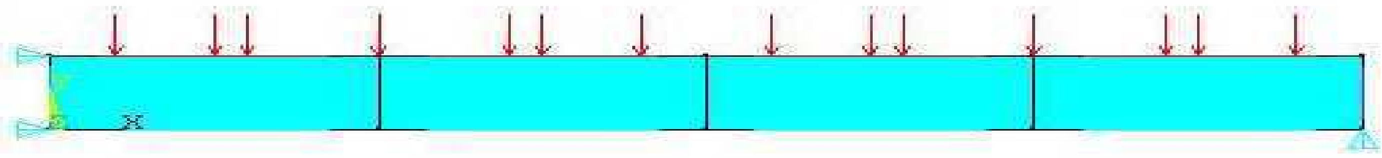
- Extra shape (bubble) functions
  - Wilson, Taylor, Doherty and Ghaboussi (1973)
  - Efficient for linear and regular element shapes
  - Elements have to be parallelogram for passing patch test
- Extra shape functions and modification in derivative of the extra shape functions wrt x, y and z (QM6)
  - Taylor, Beresford and Wilson (1976)
  - Pass linear patch test for all element shapes (ANSYS 42 and 45)
  - May be susceptible to volumetric locking
  - May not robust for finite strain analysis
- Modify strain calculation, enhanced strain formulations for geometric linear problems
  - Simo and Rifai (1990)
  - Passes patch test for all element shapes, degenerated to QM 6
  - Easy to be generalized to geometric nonlinear analysis

To Avoid Shear Locking

- Modify deformation gradient tensor; enhanced strain formulation for geometric nonlinear problems
  - Simo and Armero (1992)
  - A consistent formulation, reduced to linear case automatically
  - Passes linear and nonlinear patch tests for all element shapes
- Enhanced strain formulations
  - Based on Simo and Armero (1992)
  - Introduce additional enhanced terms for alleviating volumetric locking, based on:
    - Based on Andelfinger and Ramm (1993)
    - Simo, Armero and Taylor (1993)
  - Implemented in element 182 and 185

Enhanced Strain vs Extra Shapes

- Bending of a thin plate (R=10, h=1)
- Element 182 with enhanced strain formulation and 42 with extra shape function
- Axisymemtric deformation state
- Pure elastic material, different Poisson ratios (E=1875, nu=0.0, 0.25, 0.3, 0.49, 0.499, 0.4999)
- Linear analysis, under pressure (p=1)



Enhanced Strain vs Extra Shapes *(cont.)*

- Vertical displacements of central point

Results from Element 42

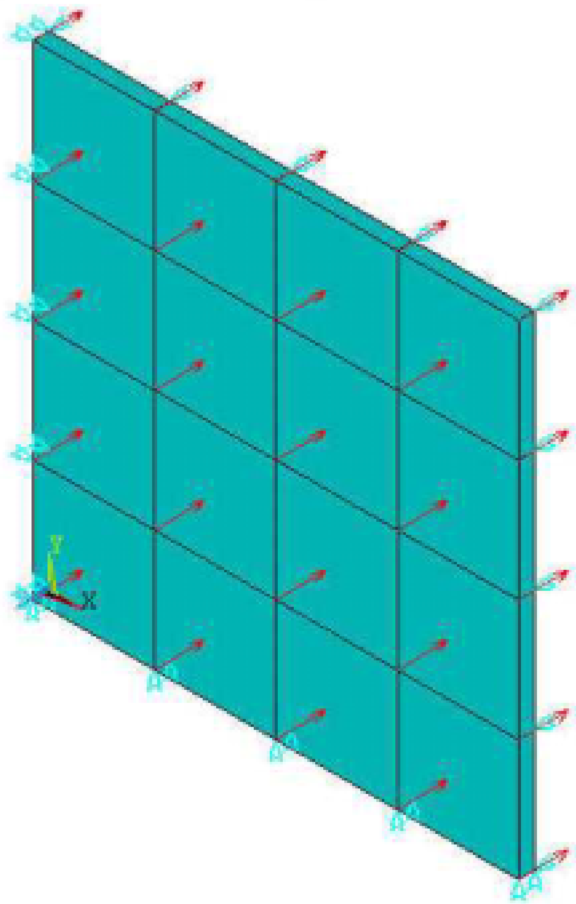
NU	Node	Theory	ANSYS	Error(%)
0	1	5.03200	4.95185	1.59273
	7	5.03200	4.95220	1.58579
0.25	1	3.97070	3.91587	1.38088
	7	3.97070	3.91623	1.37170
0.3	1	3.74360	3.69292	1.35370
	7	3.74360	3.69329	1.34380
0.49	1	2.82480	2.72489	3.53689
	7	2.82480	2.72529	3.52272
0.499	1	2.79000	2.36694	15.16353
	7	2.79000	2.36734	15.14912
0.4999	1	2.78550	2.06928	25.71246
	7	2.78550	2.06968	25.69802

Results from Element 182

NU	Node	Theory	ANSYS	Error(%)
0	1	5.03200	5.15370	2.41859
	7	5.03200	5.15406	2.42564
0.25	1	3.97070	4.04281	1.81599
	7	3.97070	4.04317	1.82514
0.3	1	3.74360	3.79787	1.44975
	7	3.74360	3.79824	1.45955
0.49	1	2.82480	2.79481	1.06156
	7	2.82480	2.79520	1.04796
0.499	1	2.79000	2.74437	1.63538
	7	2.79000	2.74476	1.62158
0.4999	1	2.78550	2.73931	1.65809
	7	2.78550	2.73970	1.64426

Finite Deformation Analysis *(185 vs 45)*

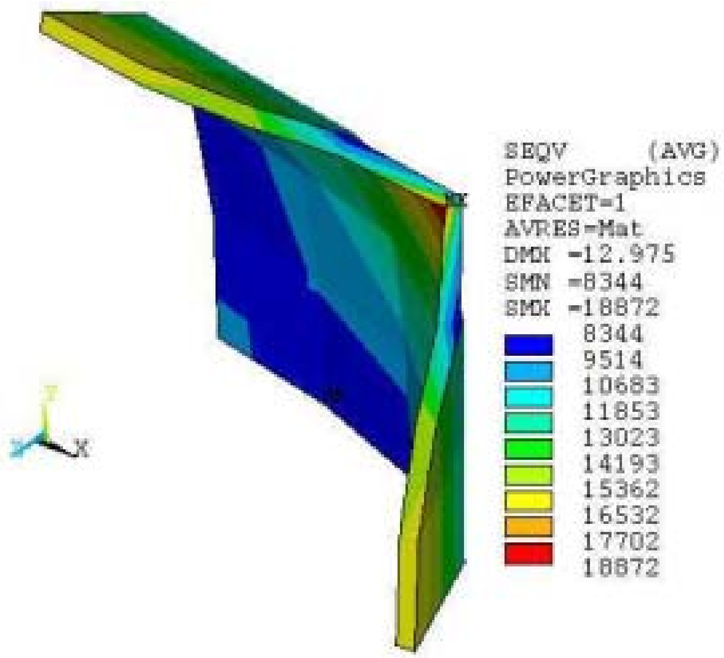
- Square plate (60X60X1)
- Simply supported
- Under distributed loading in z direction (out of plane)
- Nlgeom=on
- To compare the performances of 45 and 185, focus on volumetric locking



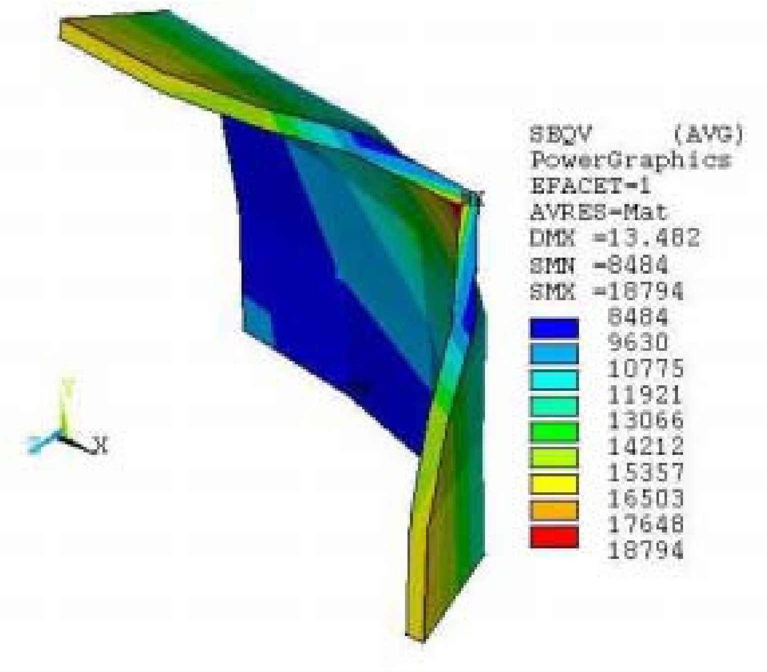
Enhanced Strain vs Extra Shapes

- Seqv close…

Element 45



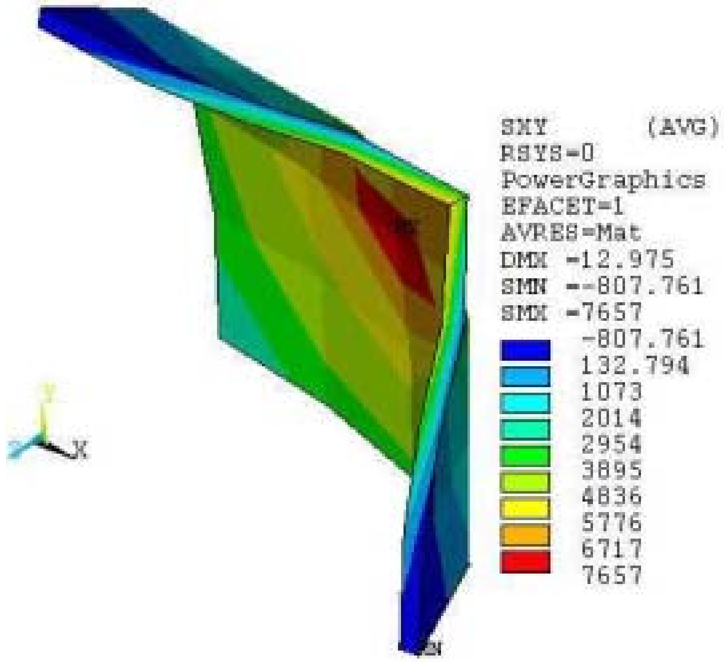
Element 185



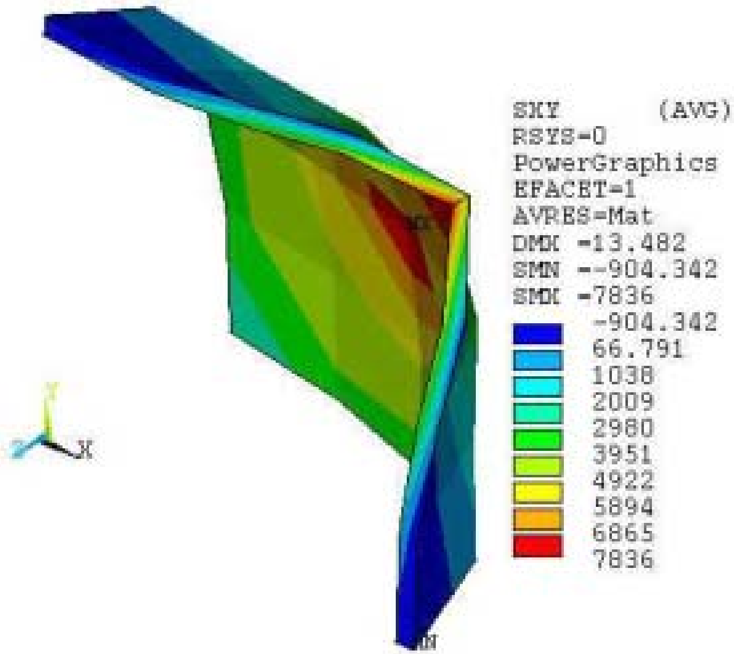
Enhanced Strain vs Extra Shapes*(cont.)*

- Sxy close…

Element 45



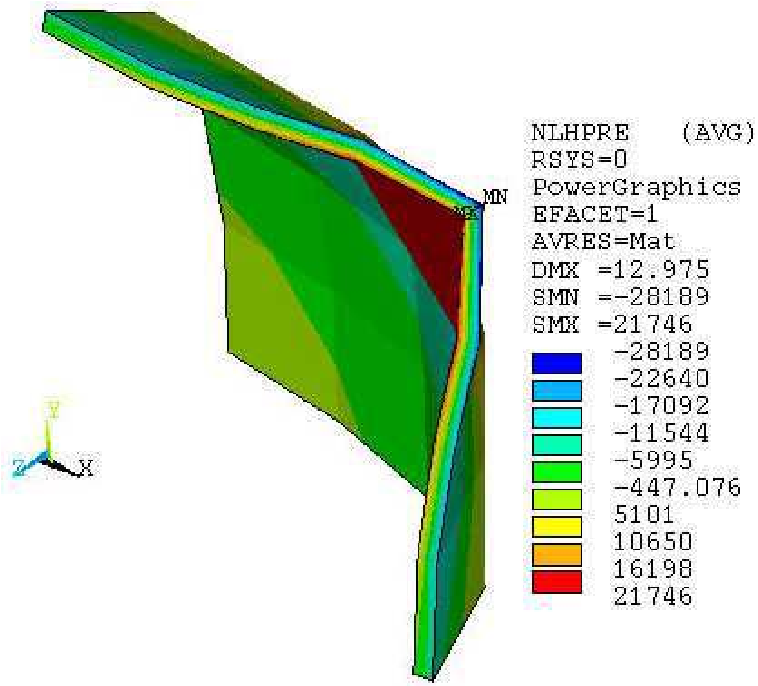
Element 185



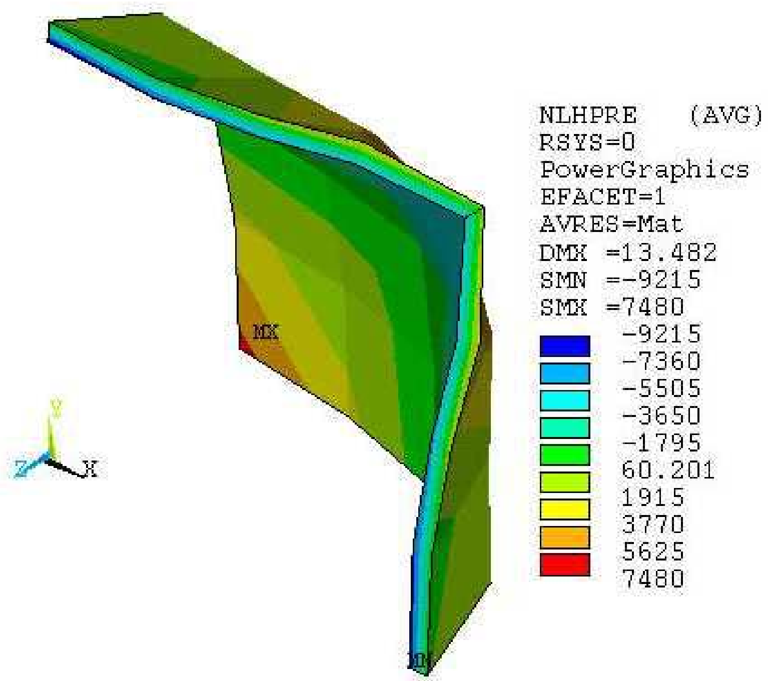
Enhanced Strain vs Extra Shapes*(cont.)*

- Sm different; it shows locking in 45…

Element 45



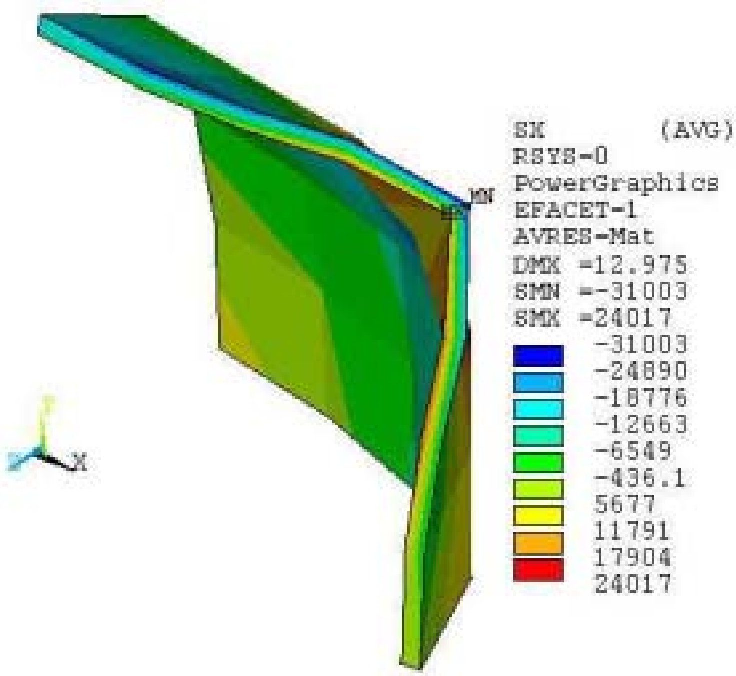
Element 185



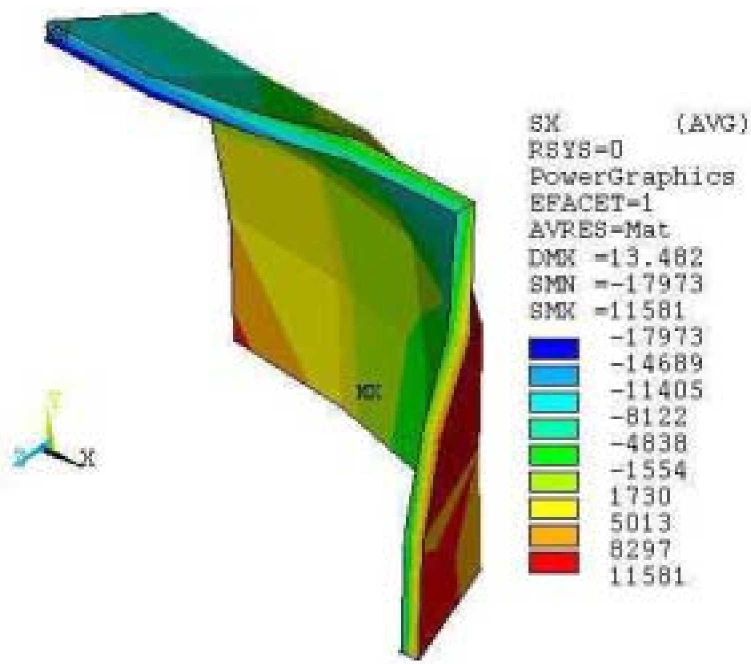
Enhanced Strain vs Extra Shapes*(cont.)*

- Sx different…

Element 45



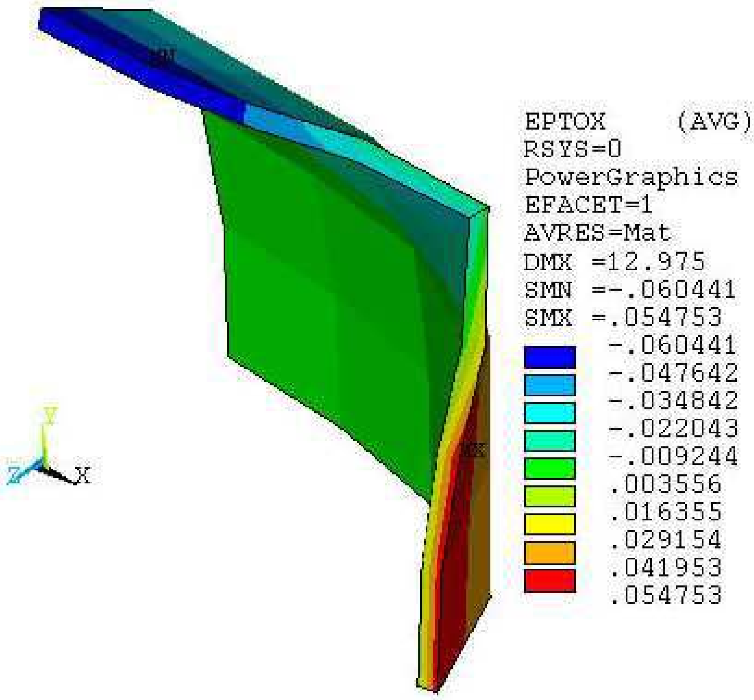
Element 185



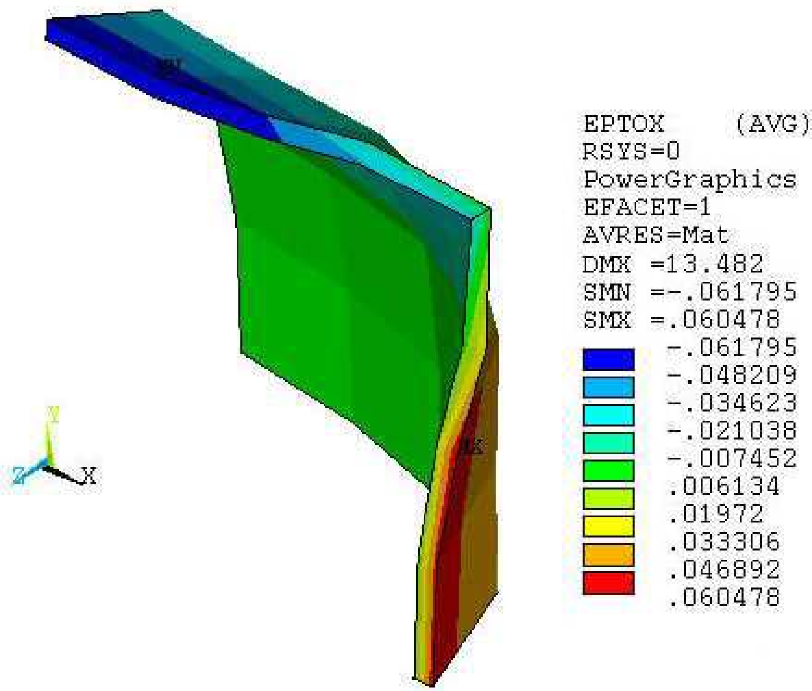
Enhanced Strain vs Extra Shapes *(cont.)*

- EPTOx different slightly

Element 45



Element 185

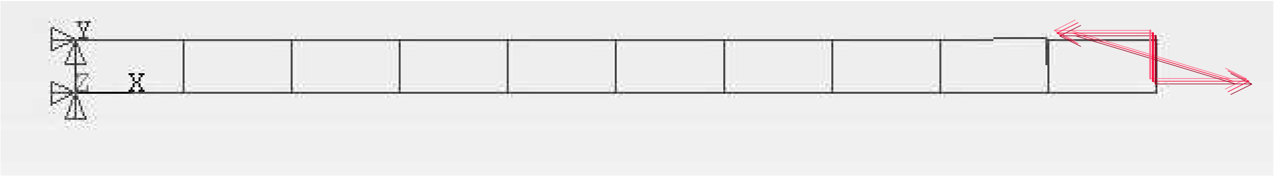


Observations *(185 vs 45)*

- Displacements are close to each other
- All deviatoric components are close and so the quantities based on them, such as Seqv
- Hydrostatic pressures are significantly different; it shows volumetric locking
- Due to the errors of hydrostatic pressures, stress and strain are not accurate, especially stress
- Volumetric locking may cause convergence problem in nonlinear analysis
- Element 45 is a proven workhorse element for a broad of problems, but clearly there are situations where enhanced strain formulation helps

Example of a Rubber Beam *(182 vs 56)*

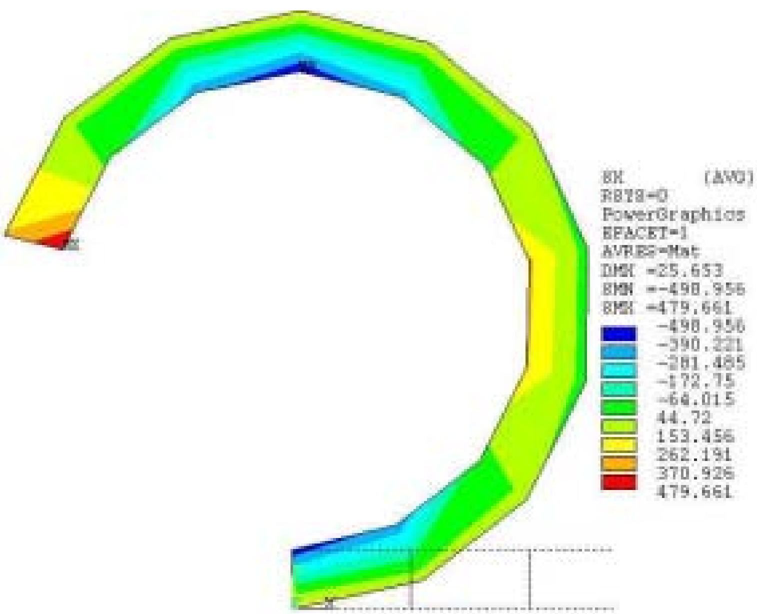
– A Mooney-Rivilin beam (20X1)
– Solid 182 with enhanced strain and mixed u/P (fully incompressible)
– Hyper 56 (nu=0.4999)
– Plane strain, Nlgeom=on
– Pressure loading at one end to simulate pure bending



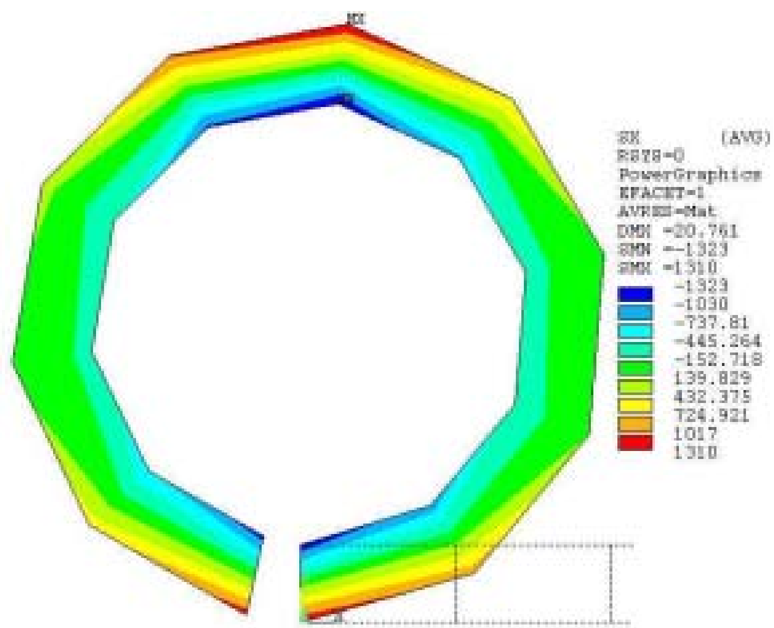
Difference between 182 and 56

- Shear locking in Hyper 56

Hyper 56

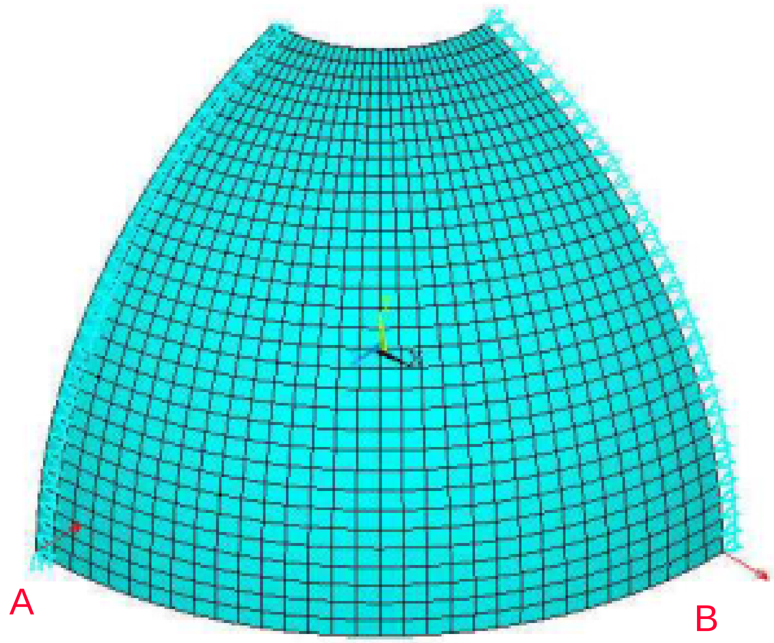


Solid 182

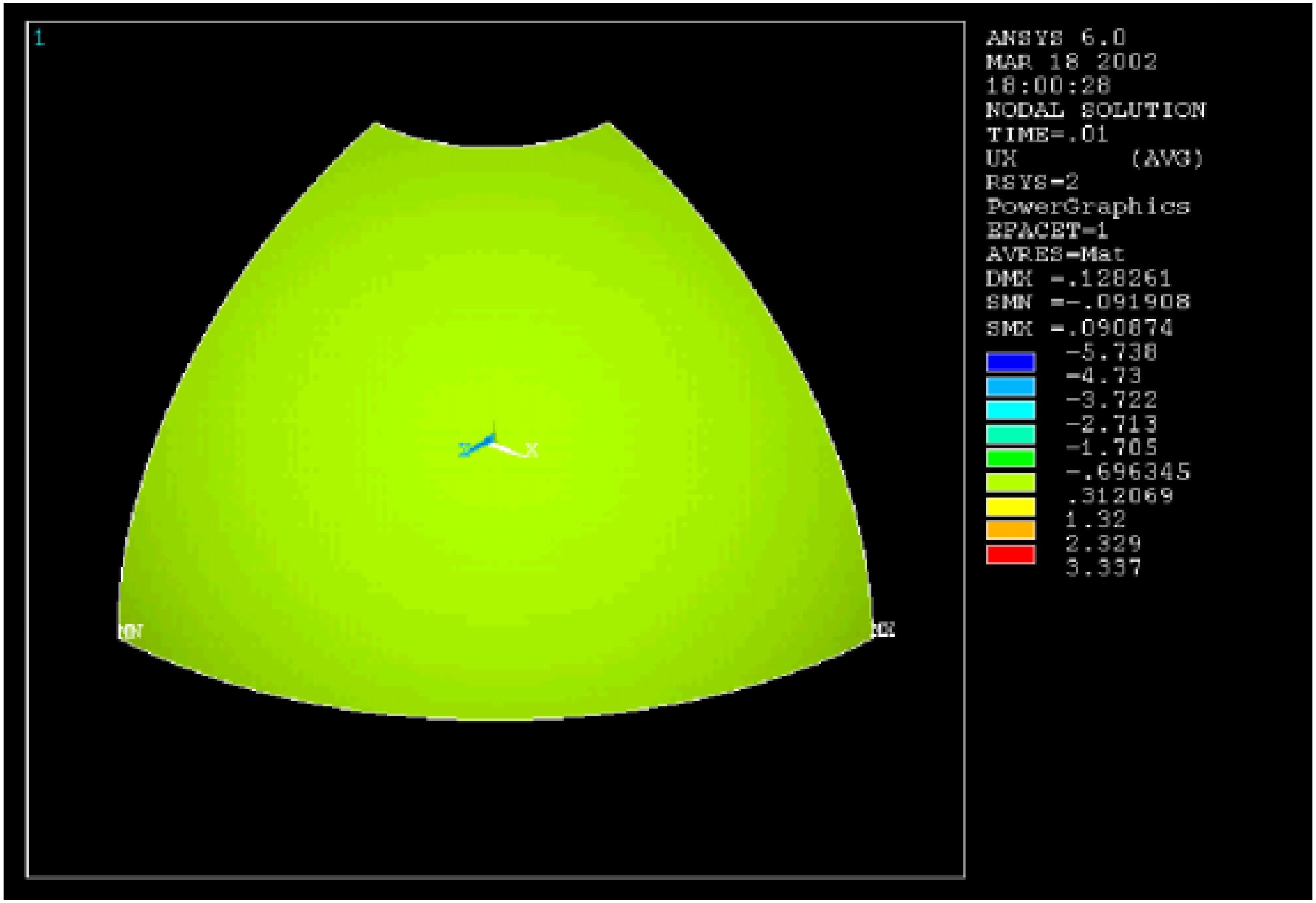


Pinched Hemisphere Shell

- A Standard NAFEMS example
- R=10 and t=0.04
- Elastic material
  $E=6.825X10^7$ v=0.3
- Nlgeom=on
- Concentrated force
  at A and B
- Simulated by 185
  with enhanced strain
  and B-bar
  formulation



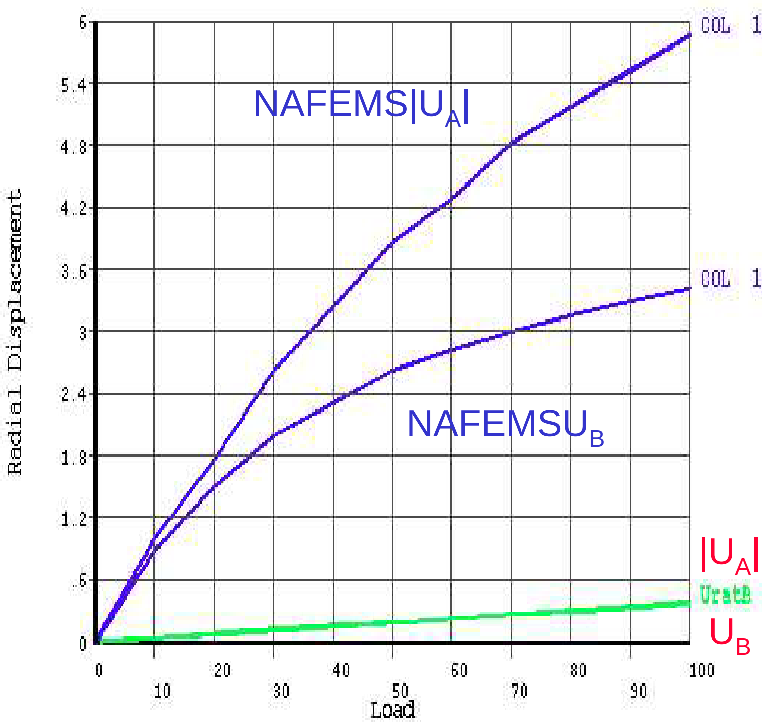
Deformed Shape from Enh. Form.



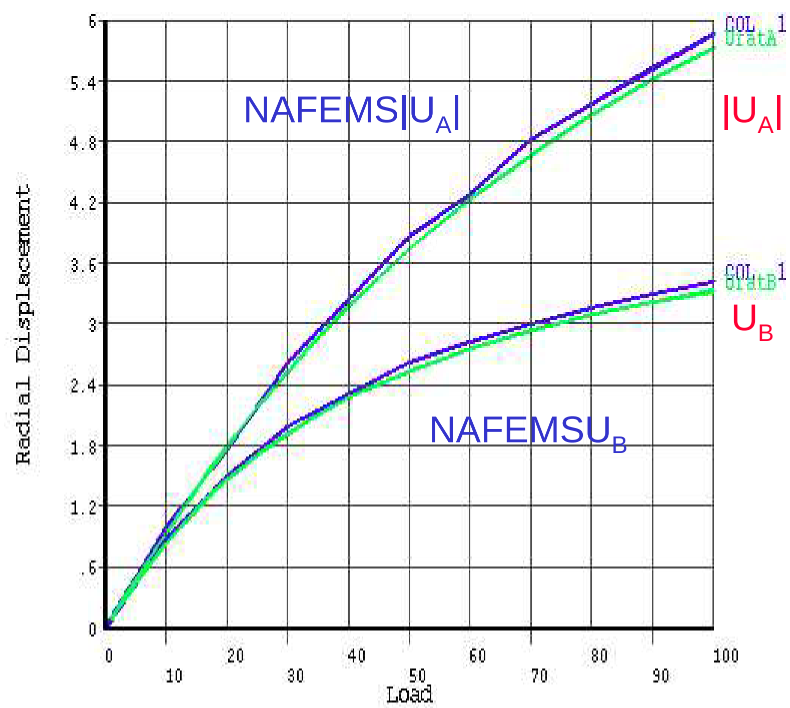
Shear Locking in B-Bar

- Radial displacements at A and B
  - Enhanced strain is much more accurate

B-Bar Formulation



Enhanced Strain Formulation



Summary *(of Enhanced Strain Formulation)*

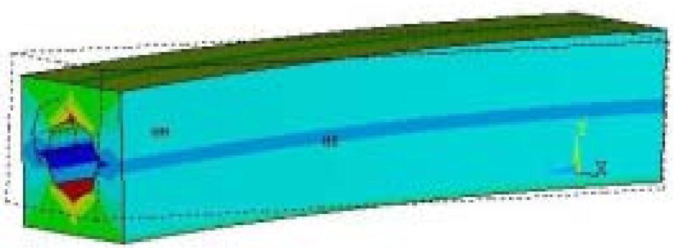
- Availability
  - Available for all deformation states: plane stress, plane strain, axisymmetric, generalized plane strain and 3D
  - No terms needed for avoiding volumetric locking in plane strain and all mixed u/P formulations (ANSYS chooses automatically)
  - Specified by keyopt(1)=2 in element 182 and keyopt(2)=2 in 185
- Advantages
  - No shear locking and volumetric locking is also alleviated
  - Good for all kinds of deformation problem; very useful when it is hard to judge deformation types in advance
  - Can be combined with both fully and nearly incompressible mixed u/P formulations

Summary *(of Enhanced Strain Formulation)*

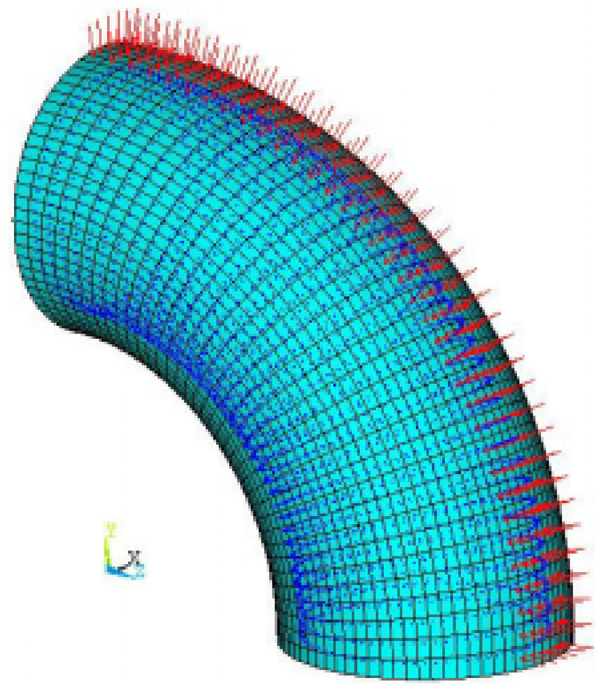
- Limitations
  - Not efficient as B-Bar and uniform reduced integration since additional DOFs are introduced and static condensation is performed
  - Large .ESAV file since the data has to be saved to retrieve the value of the additional internal DOFs
  - Enhanced strain effect is reduced when element shape is distorted

Generalized Plane Strain *(what is)*

- Purpose
  - To extend the functionalities of 2D elements to more general 3D deformations
    - To reduce engineering time
    - To reduce computer time

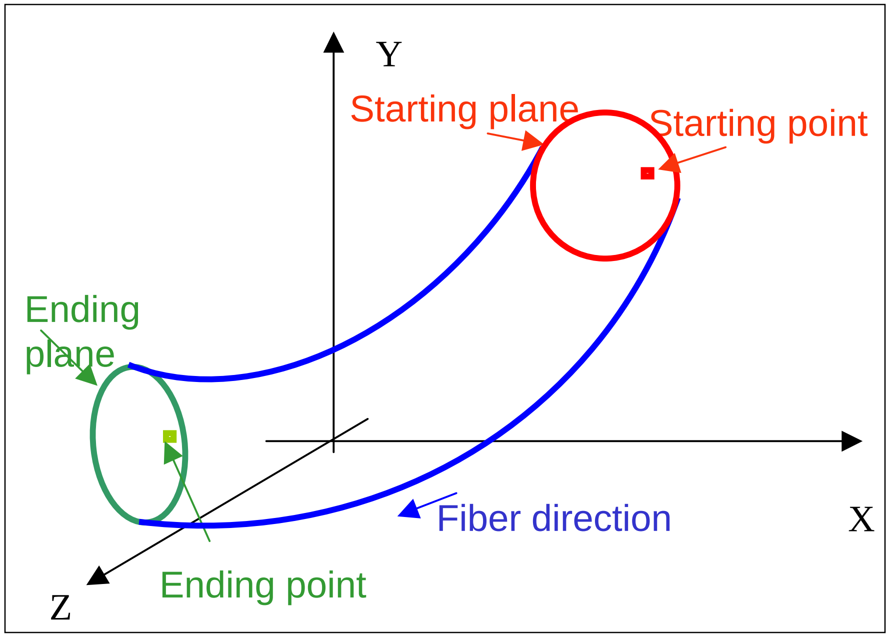


Rotated about X and Y



Generalized Plane Strain *(what is)*

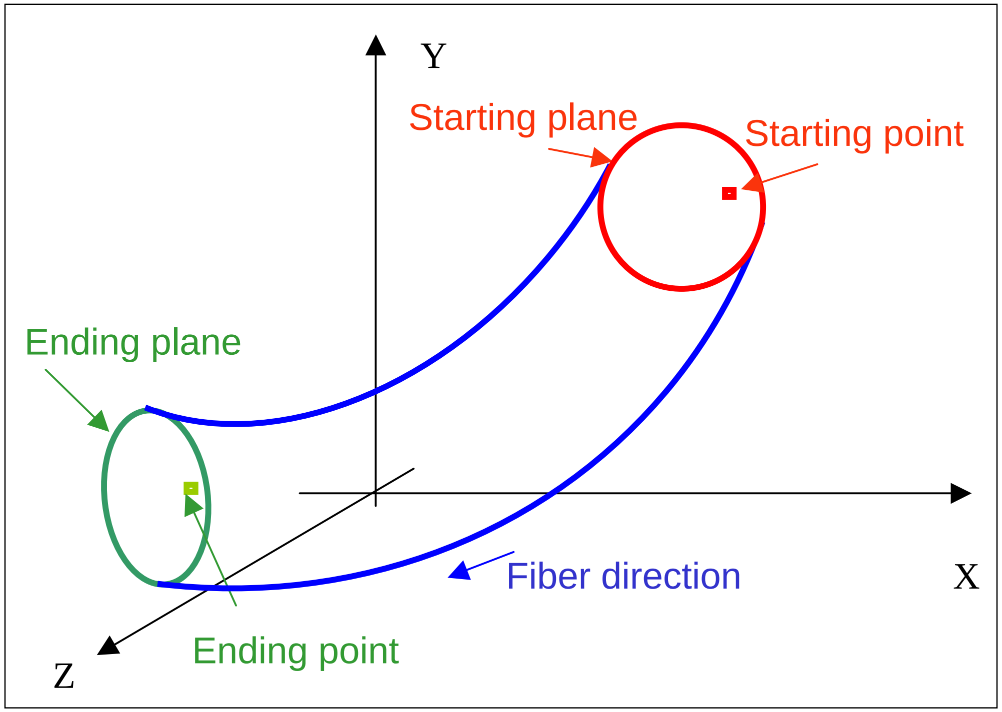
- Geometry
  - "formed" by extruding a plane area along a curve (fiber direction) with constant curvature so that a 3D "segment" is defined by two planes; "starting plane" and "ending plane"
- BC/Load conditions
  - Boundary conditions and loads have to be homogeneous in fiber/Z direction
  - Starting and ending plane are kept as planes during deformations



- Homogeneity
  - Any deformation can be represented by deformation of starting plane

Generalized Plane Strain *(how to define)*

- Geometry: shapes and positions of starting (reference) and ending planes
  - Fiber length: curve length of the fiber passing through the starting (reference) point and ending points
  - Starting (reference) point is specified by users
  - Rotation angles: ending plane about x and about y; starting plane has to be on XY plane
- BC/Load
  - Specified at the ending point



Assumptions *(of Generalized plane strain)*

- Geometry
  - Curvature is constant in fiber direction for any point (x,y) on reference plane, independent from the position in fiber direction
  - Each cross section is identical to others. They are perpendicular to the fiber direction and kept as planes during deformation.
  - Reference point is fixed in space. It doesn't have to be a nodal point. It is used to specify the geometry and apply BC/loads in the fiber direction
- BC and loads
  - The loads applied on reference plane have to be constant in fiber direction
  - The BCs at ending points are fiber length change, rotation angle changes about x and y
  - The loads at ending points are resultant force in fiber direction and the moment about x and y applied on ending plane

Implementation *(of generalized plane strain)*

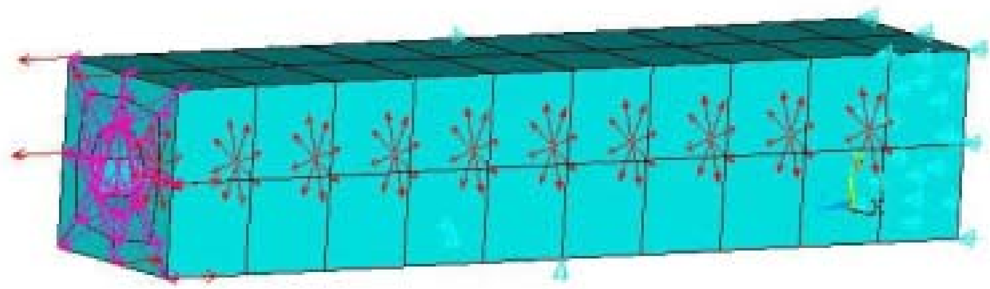
- Three new primary variables:

$$dl_r, drot_x, drot_y$$

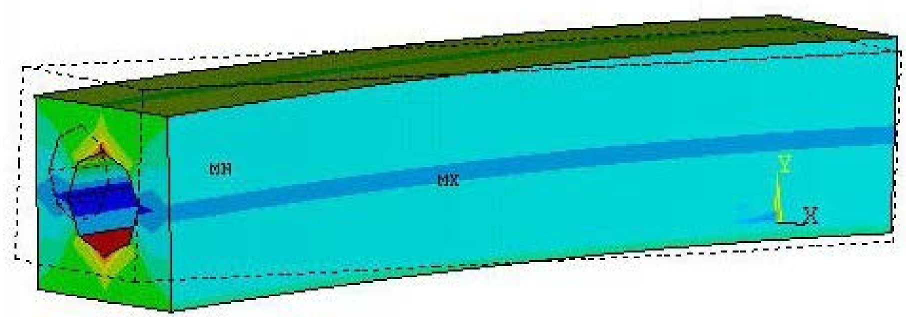
- Introduced by 2 internal nodes shared by all elements
  - Still 2D 4-node (182) or 8-node element (183)
  - The 2 additional internal nodes are created automatically at solution stage
  - User is not able to access the 2 nodes
  - Apply BC/load by new commands
  - Check the results by new commands
- Initiated by
  - keyopt(3)=5 in both element 182 and 183

An Example *(how to use)*

- A straight structure with a hole in middle
  - Geometry: 2x2x10 cross section with a hole of radius=1.0 at the center
  - Load: internal pressure 5000 and bending moment about x and y of 1100
  - Material: $E=2.5\times10^5$, nu=0.35, yield stress=800 and h=1000
  - Nlgeom=off
  - Element 185 for 3D and 182 for 2D; enhanced strain formulation



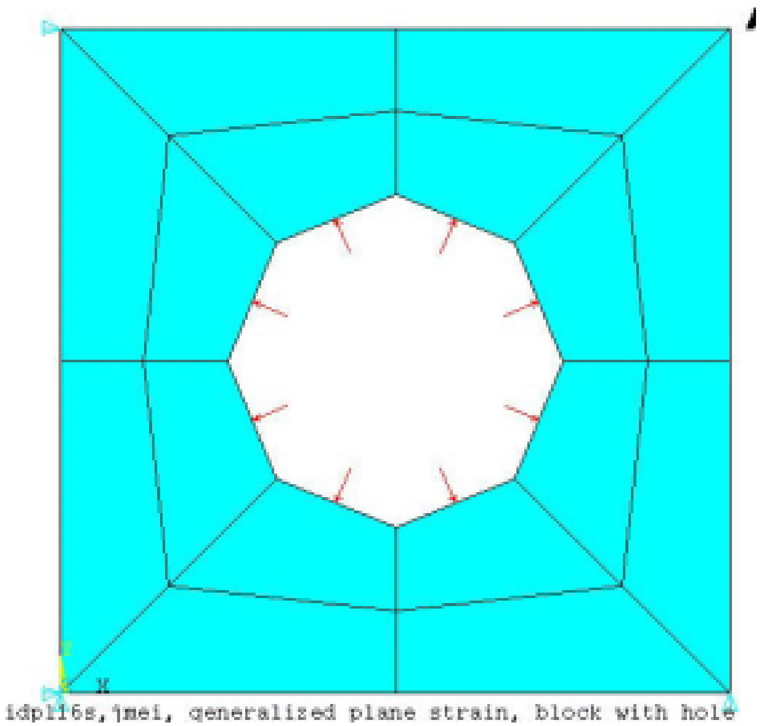
Initial geometry and BCs



Deformed Shape and Sx,
Rotated about X and Y

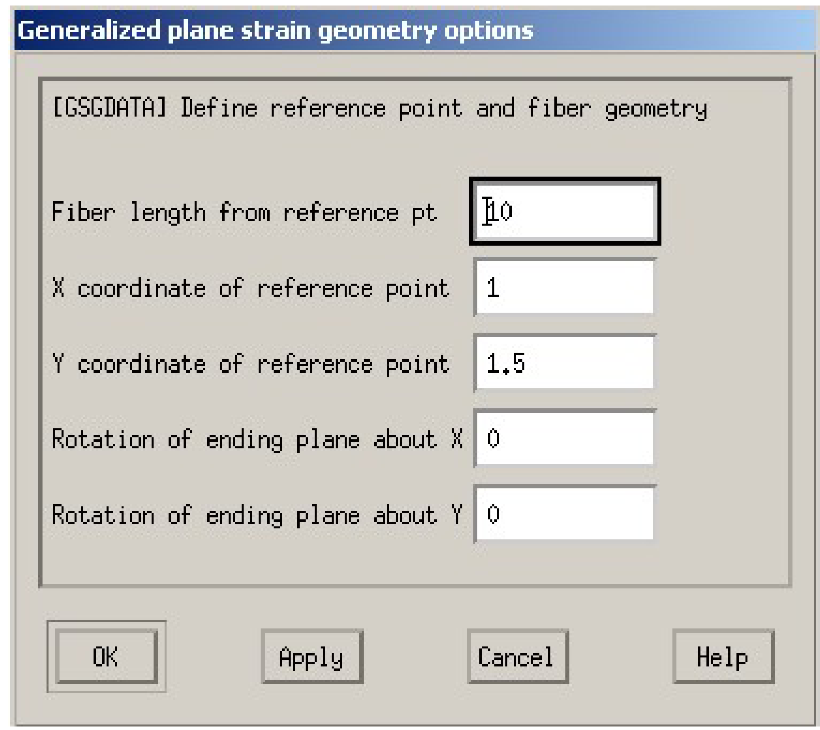
The Input Data *(for generalized plane Strain)*

– Create 2D model as usual



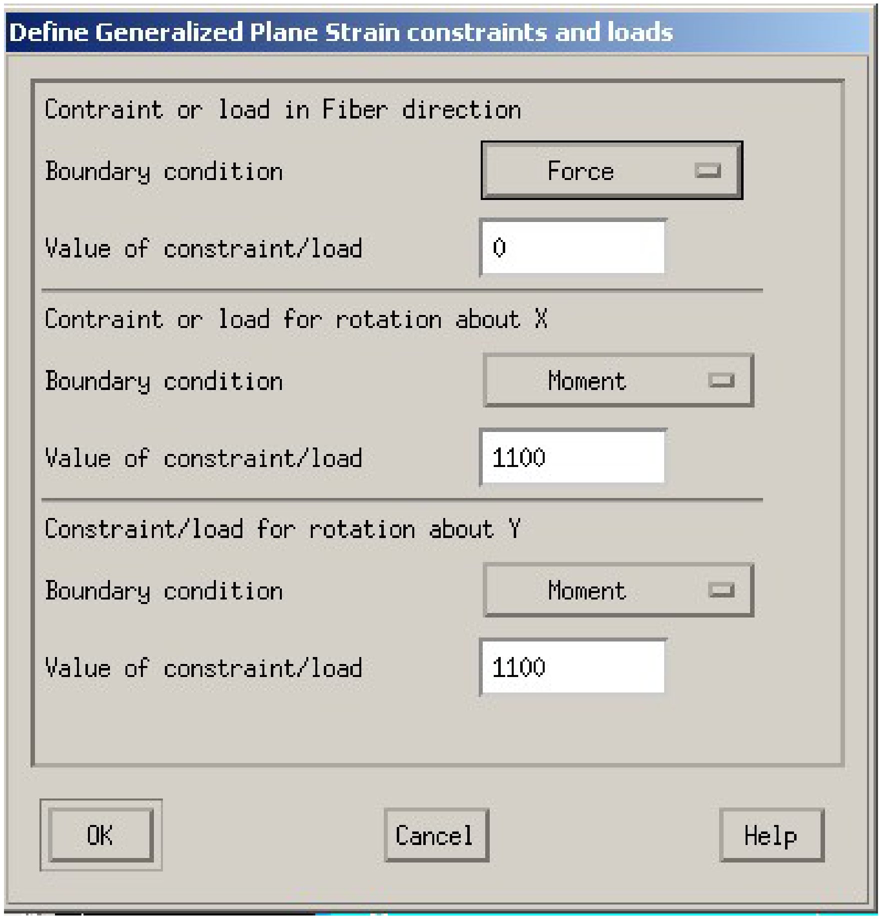
– GSGDATA…

- Create the geometry in fiber direction
- GSGDATA,10.0,1,1.5,0.0,0.0



The Input Data *(for generalized plane Strain)*

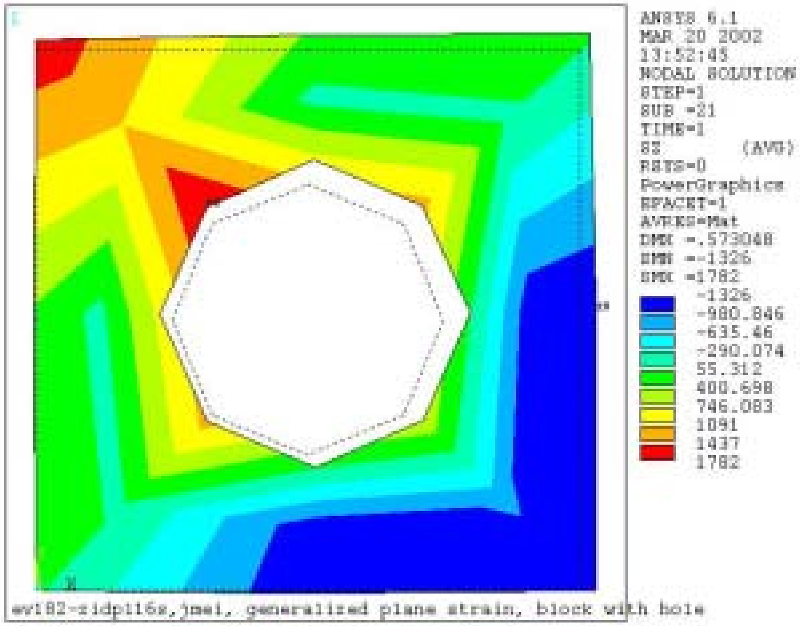
- GSBDATA…
  - To specify the loads and BCs in fiber direction
  - GSBDATA,f,0.0,mx,1100,my, 1100
- GSLIST,geometry
  - To check the input geometry
- GSLIST,bc
  - To check the applied BCs and loads in fiber direction
- GSLIST,results
  - To list the results about ending plane
- GSLIST,reactions
  - To list the reaction forces/moments on ending plane
- GSLIST, (all)
  - List all above



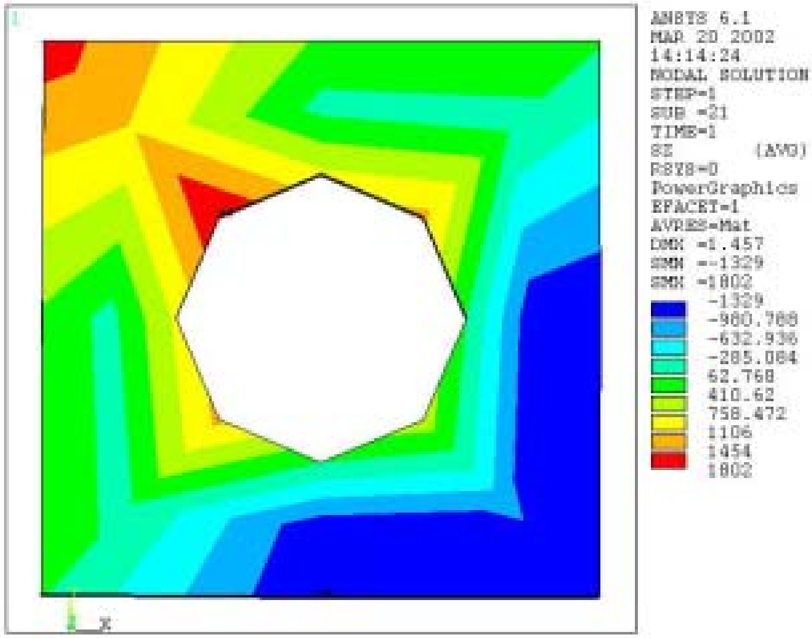
Comparison of 2D and 3D

- Nodal stress results, σ_z

Element182



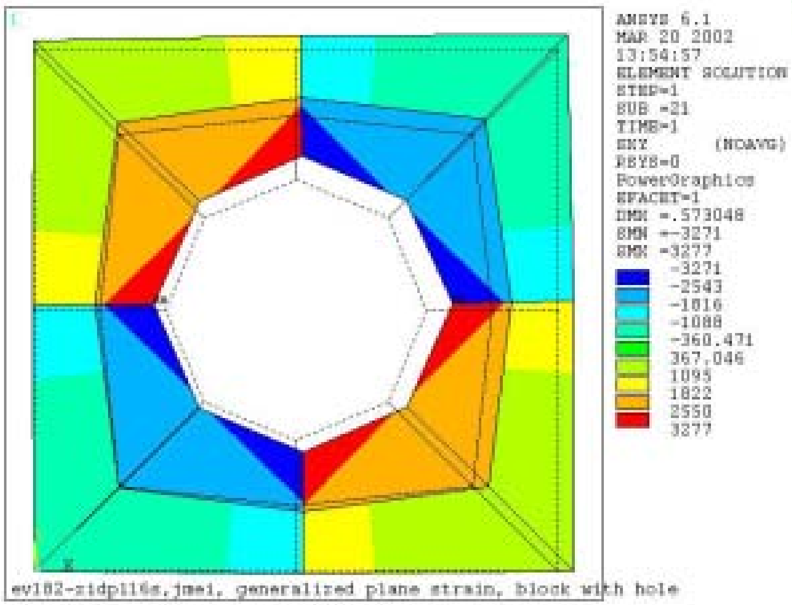
Element185



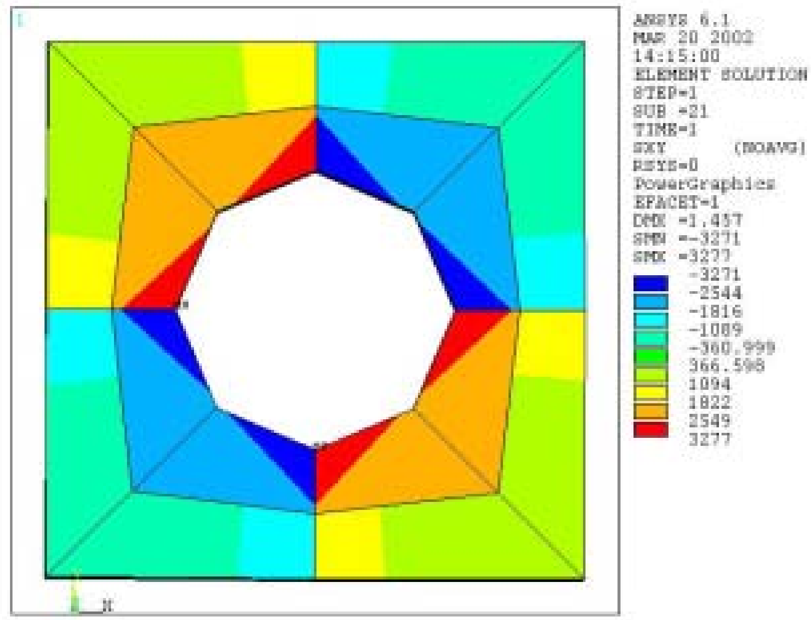
Comparison of 2D and 3D

- Elemental stress results, σ_{XY}

Element182

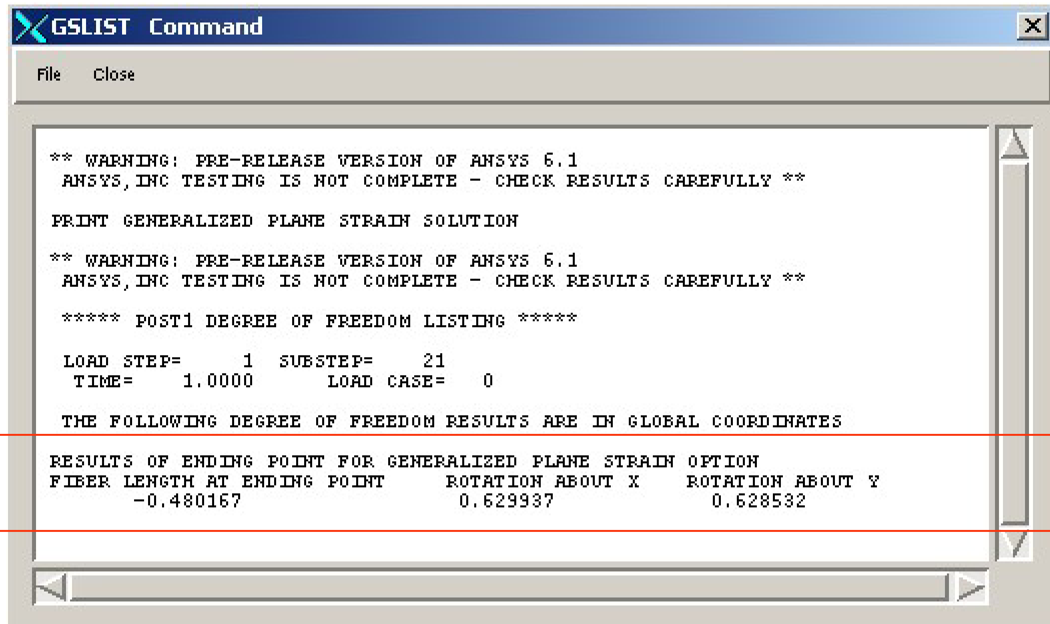


Element185



Comparison of 2D and 3D

- The results reported by "GSLIST,results"



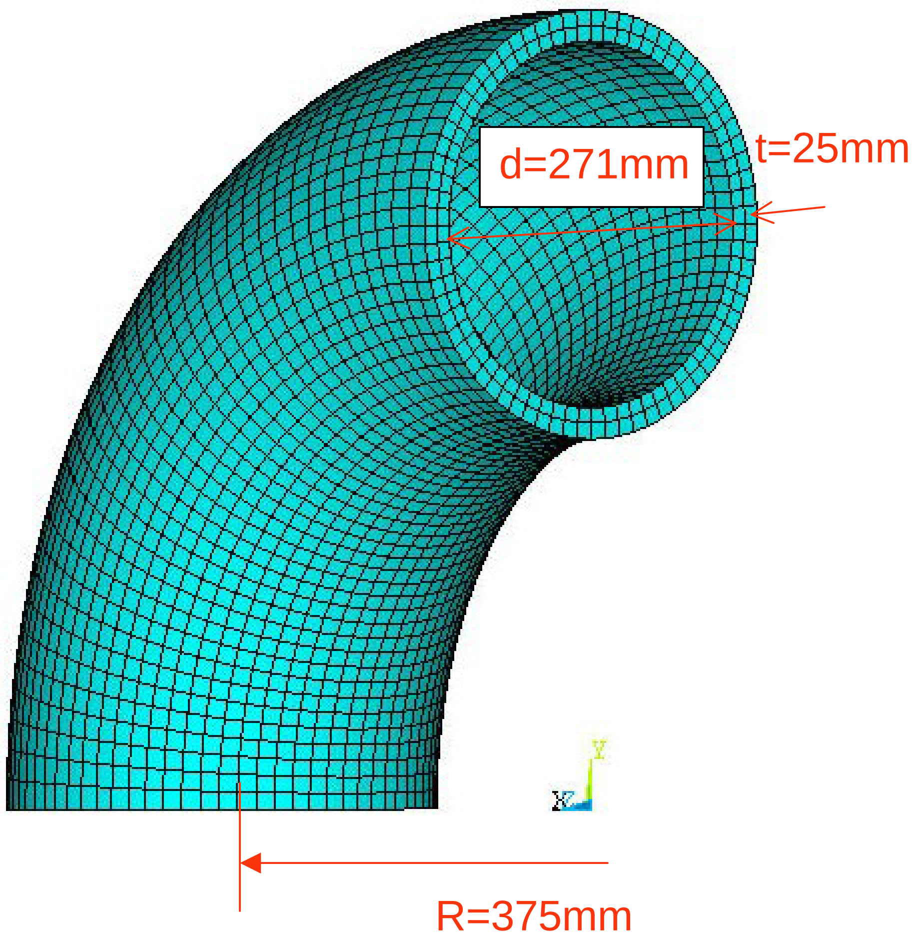
The results about ending plane

- Generalized plane strain is more efficient

Element	# of nodes	# of Elements	# of substeps	# of iteratins	CPU time (sec.)
182	24	16	21	30	4.3
183	264	176	21	40	36.73

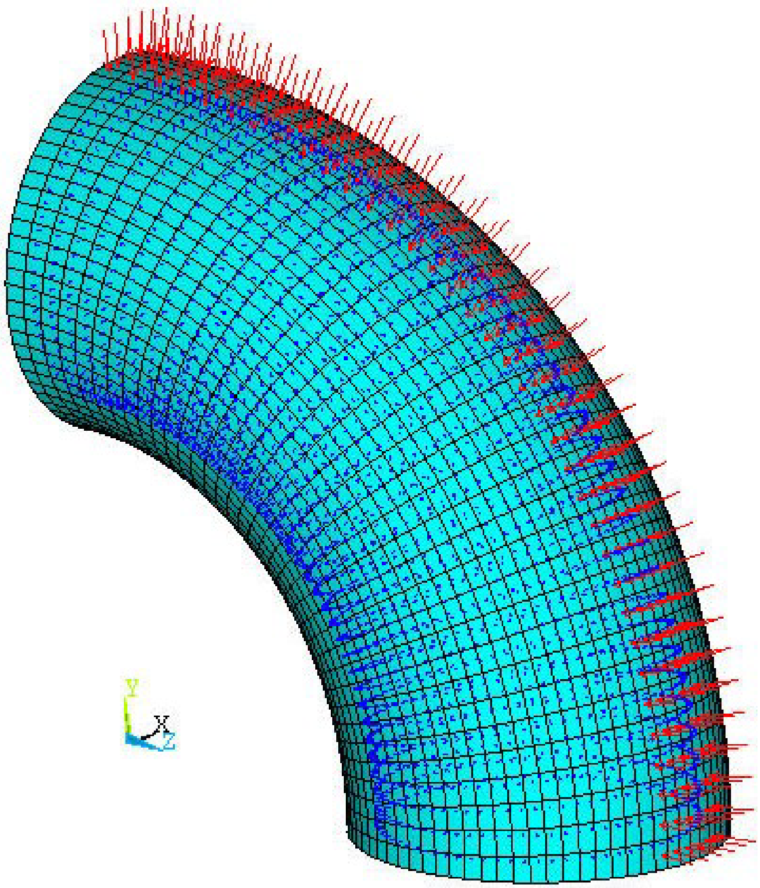
Elbow Pipe Under Pressures

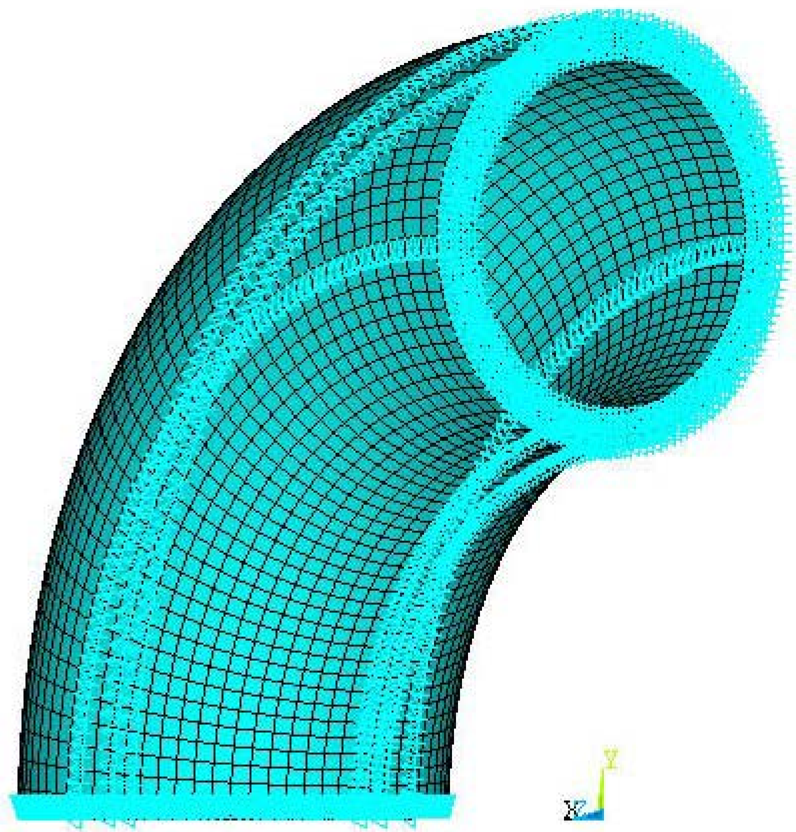
- Geometry
  - As shown in the figure
  - The two end planes have an angle of 90 degrees
- Material
  - E=200 Gpa
  - Nu=0.28
- Load
  - Inner pressure: 150 Mpa
  - External pressure: 1200 Mpa
- FE model
  - 2D 183 generalized plane strain
  - 3D 186



Elbow Pipe Under Pressures *(cont.)*

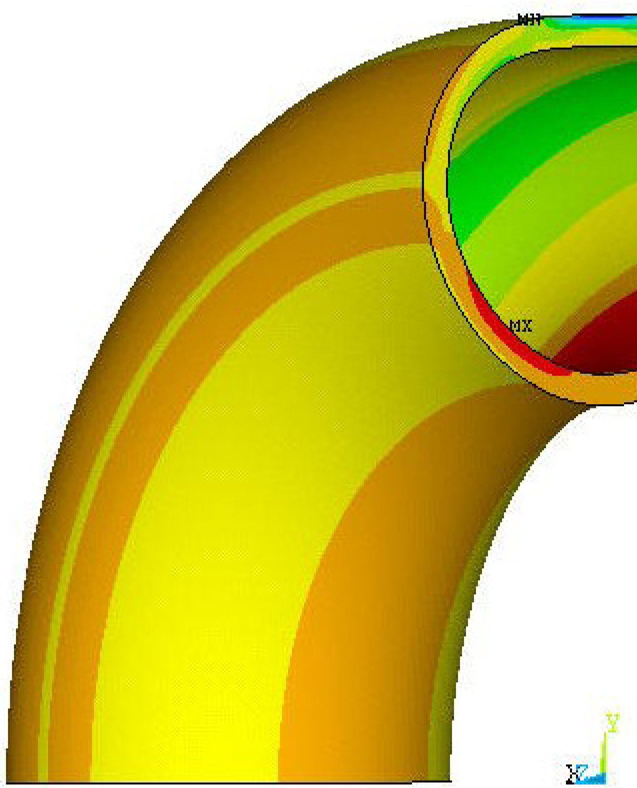
- 3d Model: time consuming and easy to make mistakes



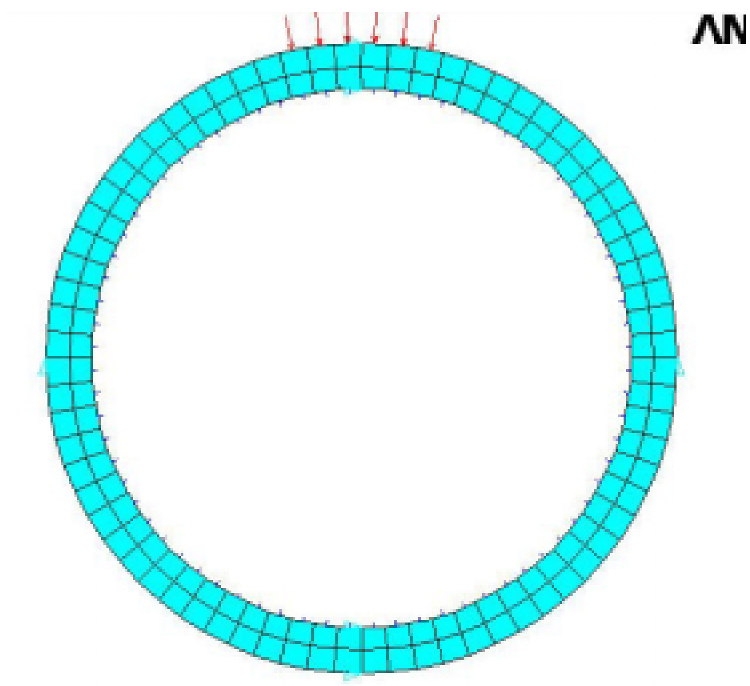


Elbow Pipe Under Pressures *(cont.)*

- Deformed shape and σ_2 homogeneous along "fiber" direction



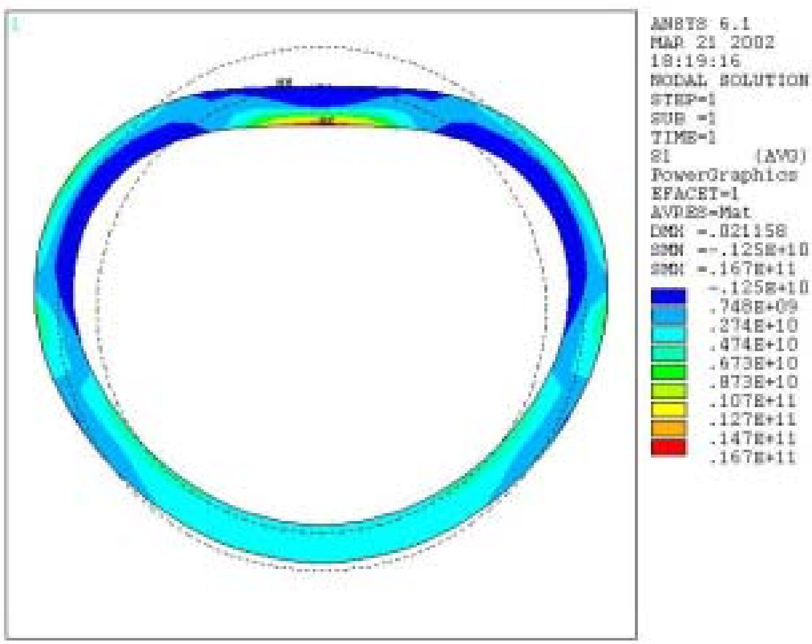
- The simulation can be done by 2D model !
  - Model is so simple !



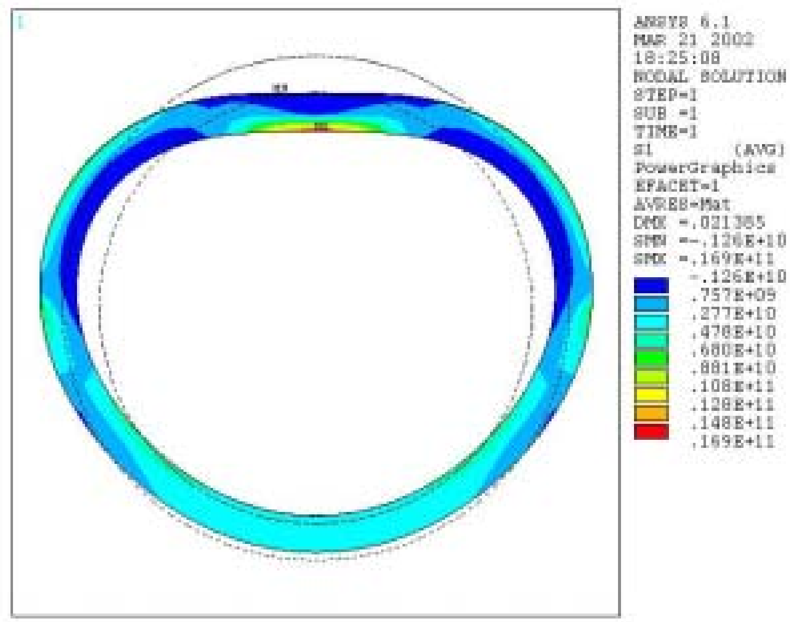
Elbow Pipe Under Pressures *(cont.)*

- The solutions are almost identical
  - Nodal solution: σ_1

Element183



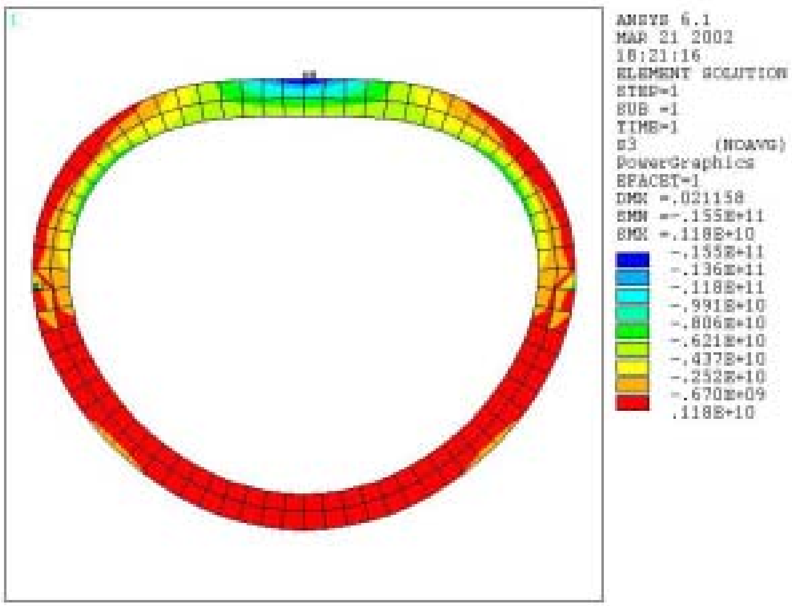
Element186



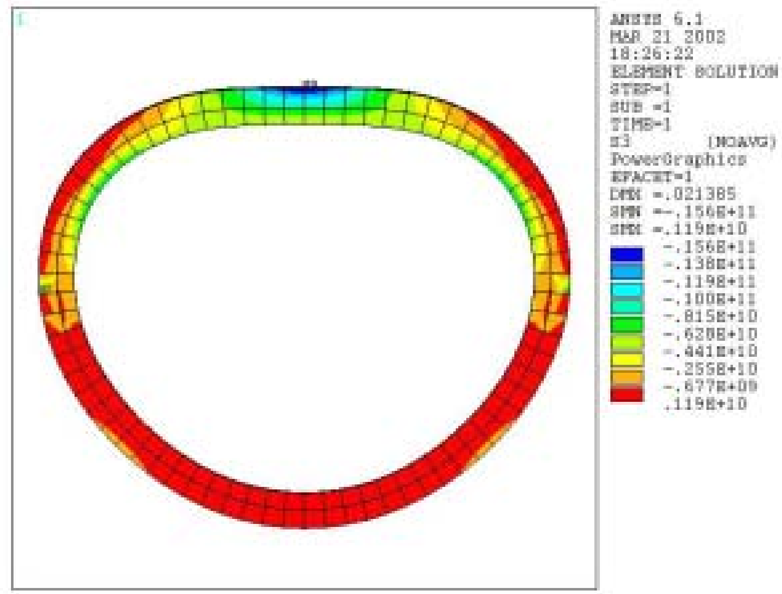
Elbow Pipe Under Pressures *(cont.)*

- Elemental solution: σ_3

Element183



Element186



- 2D is MUCH more efficient, about 100 times faster !
  - CPU time: 2D, 2.8 sec and 3D, 220.03 sec.

Summary *(of generalized plane strain)*

- Availability
  - Available for element 182 and 183 with all element technologies, such as B-Bar, uniform reduced integration and enhanced strain formulations
  - Can be combined with mixed u/P formulations
  - Can be used with all other features, such static, initial stress, thermal loading
    - (modal, buckling analysis and geometric nonlinearity will be at 7.0)
- Special cases
  - Plane strain and axisymmetric are special cases of generalized plane strain with little computational penalty
  - Plane strain:
    - With GSGDATA,1.0,,,,,
    - And GSBDATA lfiber,0.0,rotx,0.0,roty,0.0
  - Axisymmetric;
    - With GSGDATA, lfiber, xref, yref,0.0,roty0
    - And GSBDATA, lfiber, 0.0,rotx,0.0,roty,0.0

Summary *(of generalized plane strain)*

- The reference point
    - Usually a nodal point but it doesn't have to be
    - Used to specify the geometry and BC/Load in fiber directions so that if same geometry and equivalent loads are specified at different reference points, the solutions are same. However, the values of fiber length change and force in fiber direction at ending point may be different.
    - Is fixed in space, doesn't move with nodes on 2D domain (reference plane)
- Things to remember
    - The load in the fiber direction should not destroy the homogeneity of load in the fiber direction
    - Geometric nonlinearity and so on is a undocumented feature at 6.1 and will be officially supported at 7.0

Guidelines on Selection *(of 18x Solid Ele.)*

- Element order
  - Lower elements are preferred
    - To obtain a higher efficiency with less number of DOFs
    - When boundary doesn't have too much curved shape
    - When mesh could be seriously distorted in geometric nonlinear analysis
  - Higher order elements are preferred
    - To match the boundary of deformation domain accurately
    - To capture stress gradients, or get higher accuracy for a given problem size
    - Mesh shape is not good and the deformation is bending dominant
    - When mesh distortion is not serious in geometric nonlinear analysis

Guidelines on Selection *(of 18x Solid Ele.)*

- Lower order elements:
  - Enhanced strain formulation
    - Is needed if bending is dominant in deformation
    - Is preferred when it is hard to decide how the deformation would be, a safer choice
    - Is good even when material is nearly incompressible
    - has the following limitations/disadvantages
      - Extra DOFs, not efficicent and large esav file
      - Enhanced strain effect is reduced if mesh is distorted
  - Selective reduced integration (B-Bar)
    - should be used as long as bending is not dominant in deformation
    - Is preferred when material is nearly incompressible

Guidelines on Selection *(of 18x Solid Ele.)*

- Uniform reduced integration
  - Is not susceptible to both shear and volumetric locking
  - Is very efficient (especially with complex nonlinear material models)
  - Use for compatibility with LS-Dyna
  - NOT recommended because of hourglass mode

- Mixed u/P or pure displacement Formulations ?
  - Material is compressible
    - Always use pure displacement formulations
  - Material is FULLY incompressible
    - Mixed u/P is the only choice !
  - Material is NEARLY incompressible
    - displacement formulation is preferred for efficiency
    - use mixed u/P formulation if efficiency is not a concern or Poison's ration is very high (e.g. >0.4999)

Convergence *(in nonlinear analysis)*

- The best convergence you can expect:
  - quadratic
    - The correction to displacements/forces reduced quadratically

```
DISP CONVERGENCE VALUE   = 0.6603
EQUIL ITER   1 COMPLETED.
FORCE CONVERGENCE VALUE  = 0.5511E-02
DISP CONVERGENCE VALUE   = 0.1229E-01
EQUIL ITER   2 COMPLETED.
FORCE CONVERGENCE VALUE  = 0.3125E-04
DISP CONVERGENCE VALUE   = 0.2391E-04
EQUIL ITER   3 COMPLETED.
FORCE CONVERGENCE VALUE  = 0.1583E-08
DISP CONVERGENCE VALUE   = 0.6626E-09
EQUIL ITER   4 COMPLETED.
>>> SOLUTION CONVERGED
```

- Quadratic rate can be obtained only if
  - Displacement/force curve is continuous & monotonic

Convergence *(cont.)*

- No abrupt change of stiffness in the neighborhood of correct solution (contact, element birth and death usually cause abrupt stiffness changes)
- Stiffness matrices are updated at each iteration and updated accurately (full Newton-Raphson method should be used, material should be stable, no buckling etc.)
- Mesh is not distorted too much
- Incremental step is small enough

- If not converged, you should check/try
  - NOTE:
    - the message "error in element formulation" just means no error detected in solution but it does not converge. It is most often, a symptom of other difficulties. It could be any of the following;
  - Deformation limit
    - Is the deformation limit met?
    - Is the deformation practical?
      - (e.g. sharp corner contacts with a surface ?)
    - Is the material properties defined at the current strain level?

Convergence *(cont.)*

- Element technologies/formulations
  - Is volumetric locking a possibility?
    - If yes, is appropriate element technology used
  - Is shear locking possible?
    - If yes, is enhanced strain formulation used or uniform reduced integration used in lower order element (with refined meshes)
  - Is mixed u/P used in the model?
    - If yes, is correct solver (sparse or front) used? Is model over-constrained?
  - Is material fully incompressible?
    - If yes, mixed u/P has to be used. Is solution of the problem not unique?
- Convergence criteria
  - Are the tolerances unreasonably tight?
    - Relaxing tolerance often makes convergence worse ! (moving away from equilibrium)
  - Are different criteria consistent with each other? (displacement, force and volumetric compatibility if mixed u/P)

Convergence *(cont.)*

- Step length
  - Is the allowed minimum step length still too big? check the displacement/strain/force changes in the sub-step
- Solution tools
  - Is problem stable? If no, is arc-length method used?
- Other control parameters
  - Maximum number of iterations
  - Maximum value of equivalent plastic strain, creep ratio..
  - Parameters related to contact elements

Shell Analysis

18x Shell Element

- ANSYS Offers a variety of shell elements
- SHELL181 provides comprehensive set of nonlinear capabilities:

	SHELL63	**SHELL91, 93**	**SHELL181**
Shell Theory	Kirchhoff	Reissner/Mindlin	Reissner/Mindlin
Transverse Shear	Neglected	Included (constant transverse shear)	Included (constant transverse shear)
Element Order	Lower-order	Higher-order	Lower-order
Kinematics	Small strain	Finite strain	Finite strain; includes change of thickness effects
Materials	Linear; Homogenous	Plasticity; Composite for SHELL91	Plasticity, Creep; Hyperelasticity; Viscoelastic, Composite
Section Definition (SECxxx commands)	No, input via real constants only	No, input via real constants only	Yes (can also use real constants for homogenous material)
Bending Response	Cubic	Quadratic	Linear
Thru-plane Integration Points	3 integration points	SHELL93: 2 for linear, 5 for nonlinear materials; SHELL91: 3 for each layer	User-definable number of integration points (default: 5).
In-plane Behavior	Extra displacement shapes	URI	URI (default); Extra displacement shapes
Drilling DOF	Artificial spring, Allman-type DOF	Artificial spring	Penalty method

Characteristics of Shell181

- Finite Strain
- Large Rotations
- Consistent Stress stiffness terms
- Many Materials
- Includes shear flexibility
- Compatibility with LS-Dyna
- Reduced Integration
  - Performance
  - No hourglass modes in bending/shear
  - Projected Mass terms
- Full integration
  - Consistent mass terms
  - Incompatible modes enhance inplane deformation

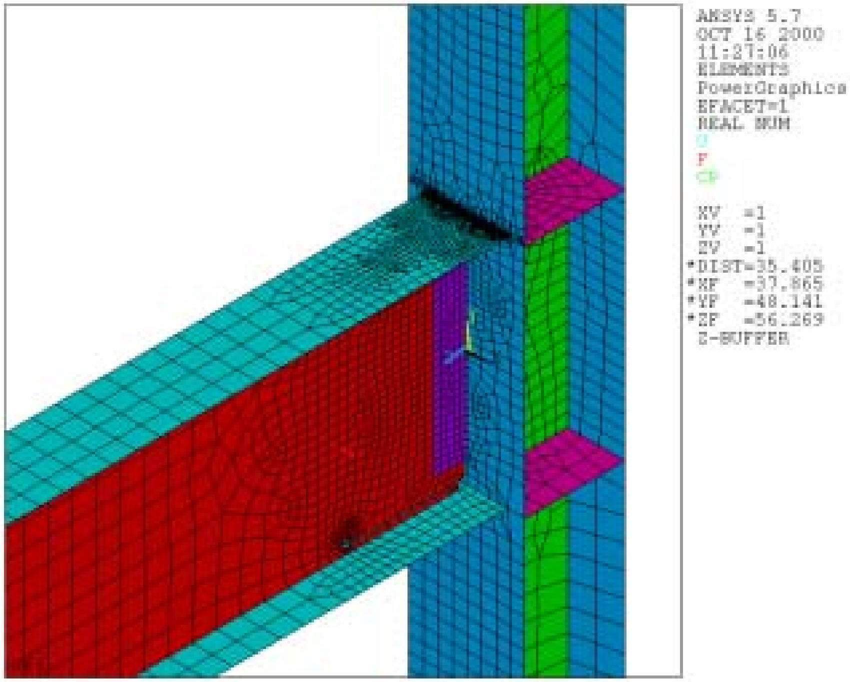
181 Features

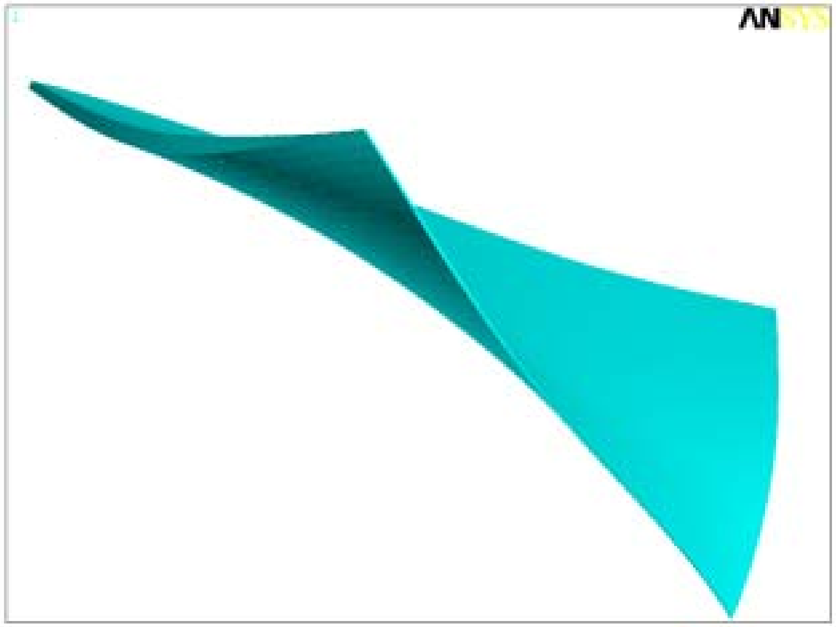
ANSYS Element Library	Type of Element	Interpolation	Formulation				Elasticity		Plasticity								Viscoplasticity		Hyperelasticity				Viscoelasticity	
			B-Bar	URI/Standard	Enhanced Strain	Mixed U-P	Isotropic, Orthotropic (MP)	ANEL	BISO	MISO	NLISO	BKIN	KINH/MKIN	CHAB	CAST/UNIAXIAL	HILL	RATE (PEIRCE, PERZYNA)	CREEP (Implicit)	Mooney-Rivlin	Polynomial Form	Ogden	Arruda-Boyce	Viscoelasticity (hypoelasticity)	Viscoelasticity (hyperelasticity)
SHELL181	Shell	Bilinear		•	•		•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•

- HILL (anisotropic Hill potential) can be used with any plasticity model (including CREEP, RATE).
- CHAB (Chaboche nonlinear kinematic hardening) can be combined with any isotropic hardening.
- RATE is combined with any isotropic hardening law
- All 18x elements support USERMAT user-defined material as well as USERCREEP user-defined implicit creep law. Elements 181-187 support USERHYPER user-defined hyperelasticity model.
- SHELL181 supports composite definition.

What can it do for me?

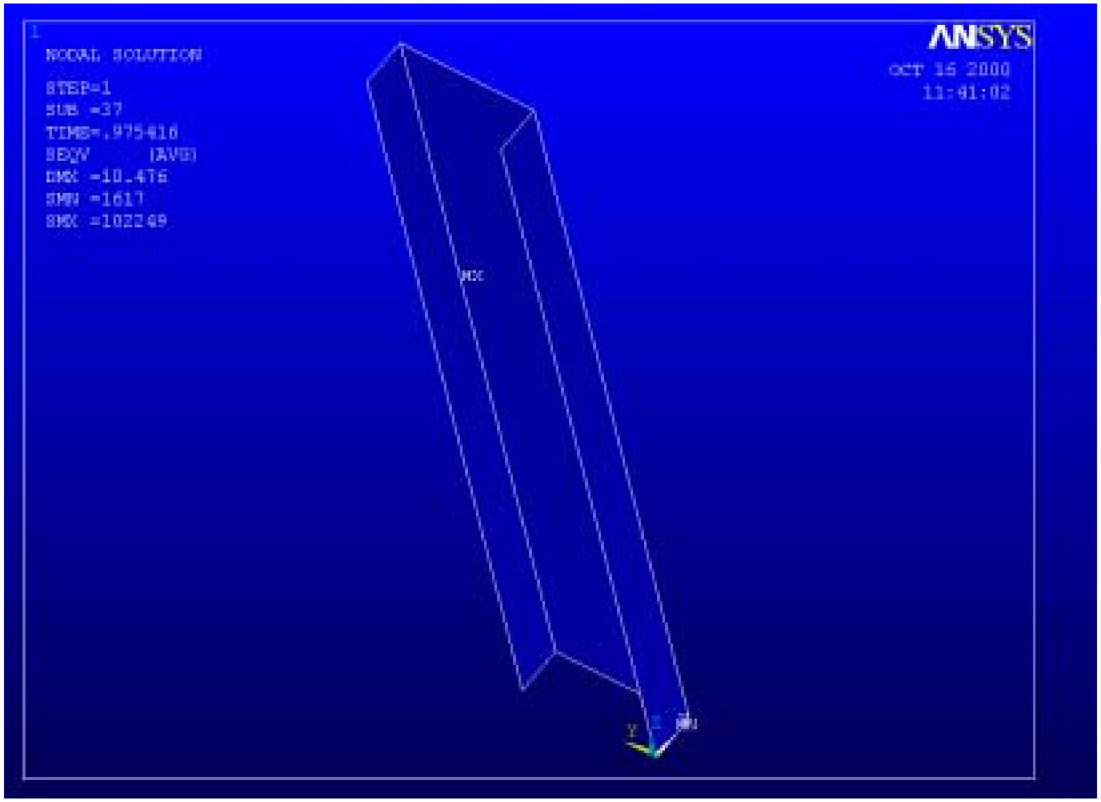
Shell/Elasto-plastic
frame analysis

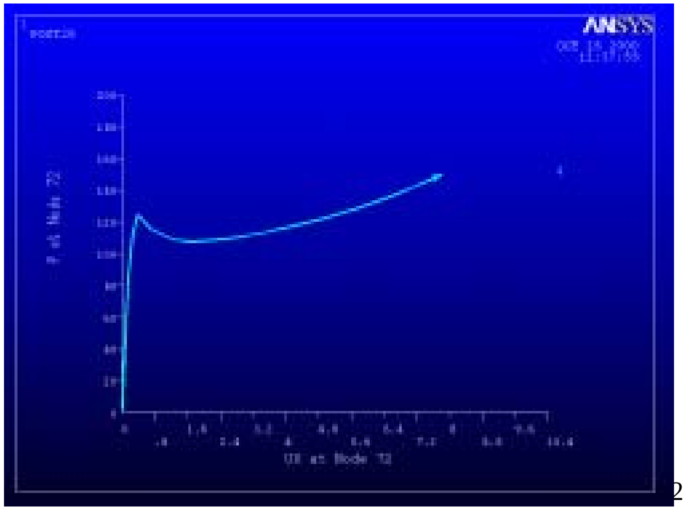




Shell -
Turbine blade
model

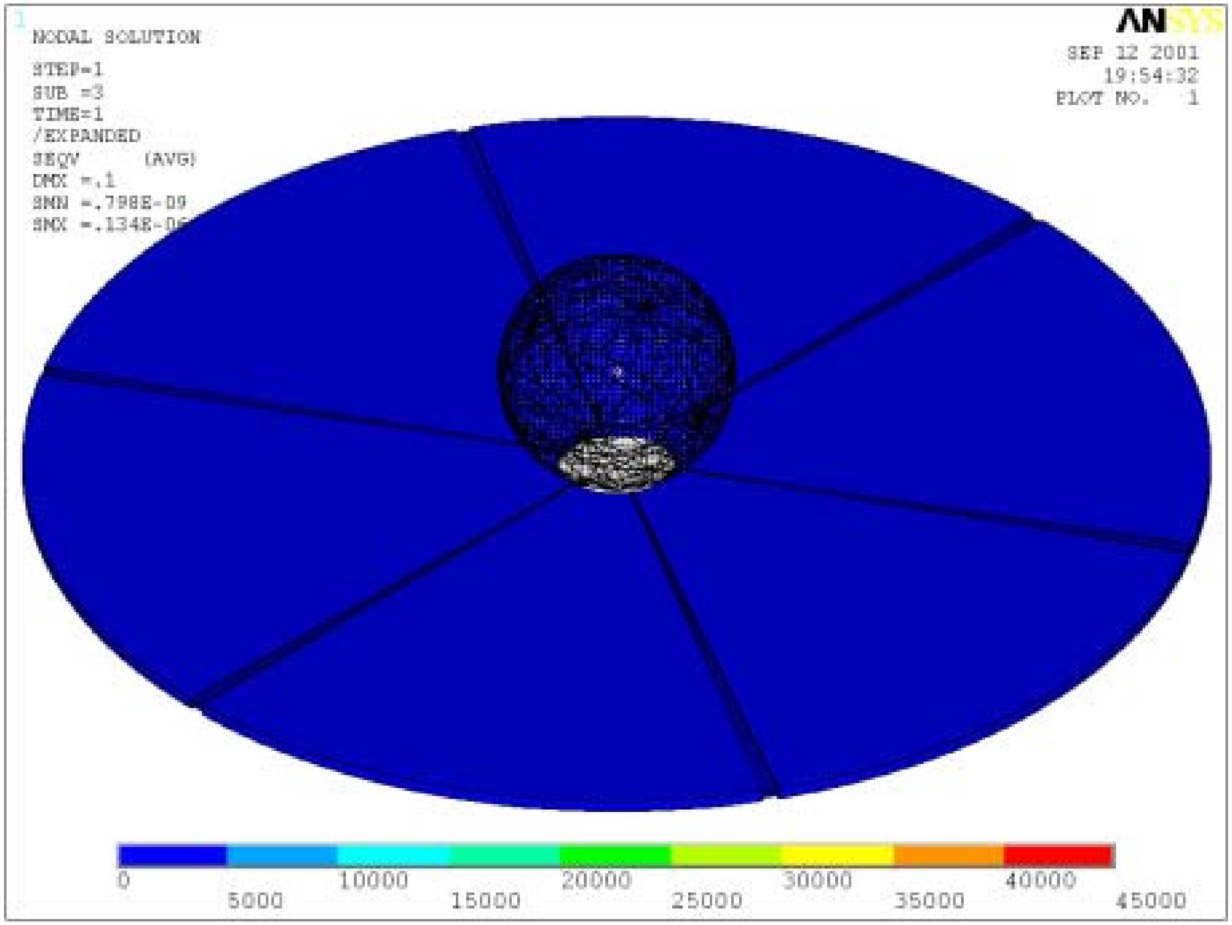
Shell/Collapse analysis





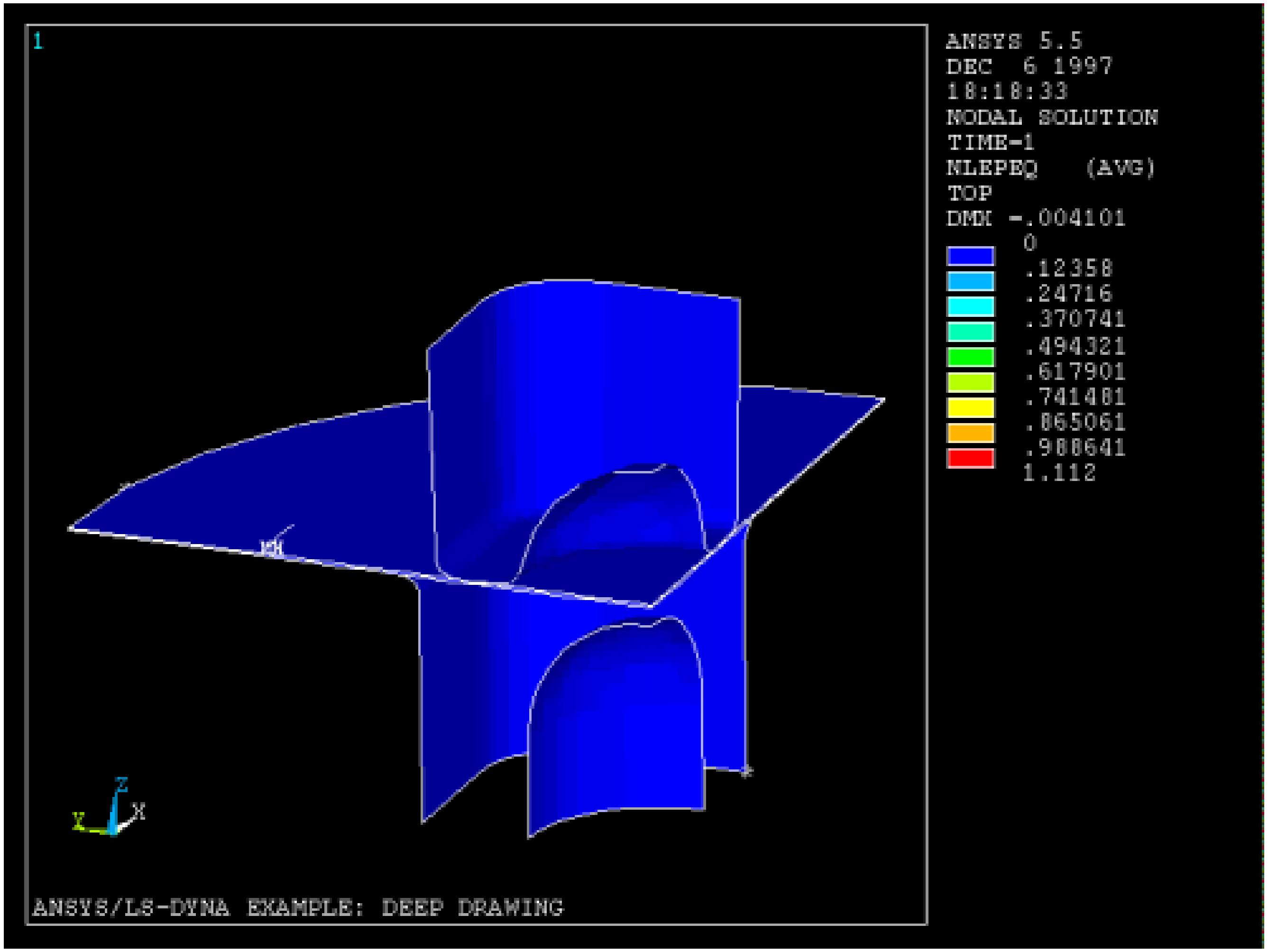
Analyze stability and post-buckling

Example of Rigid Sphere Penetration



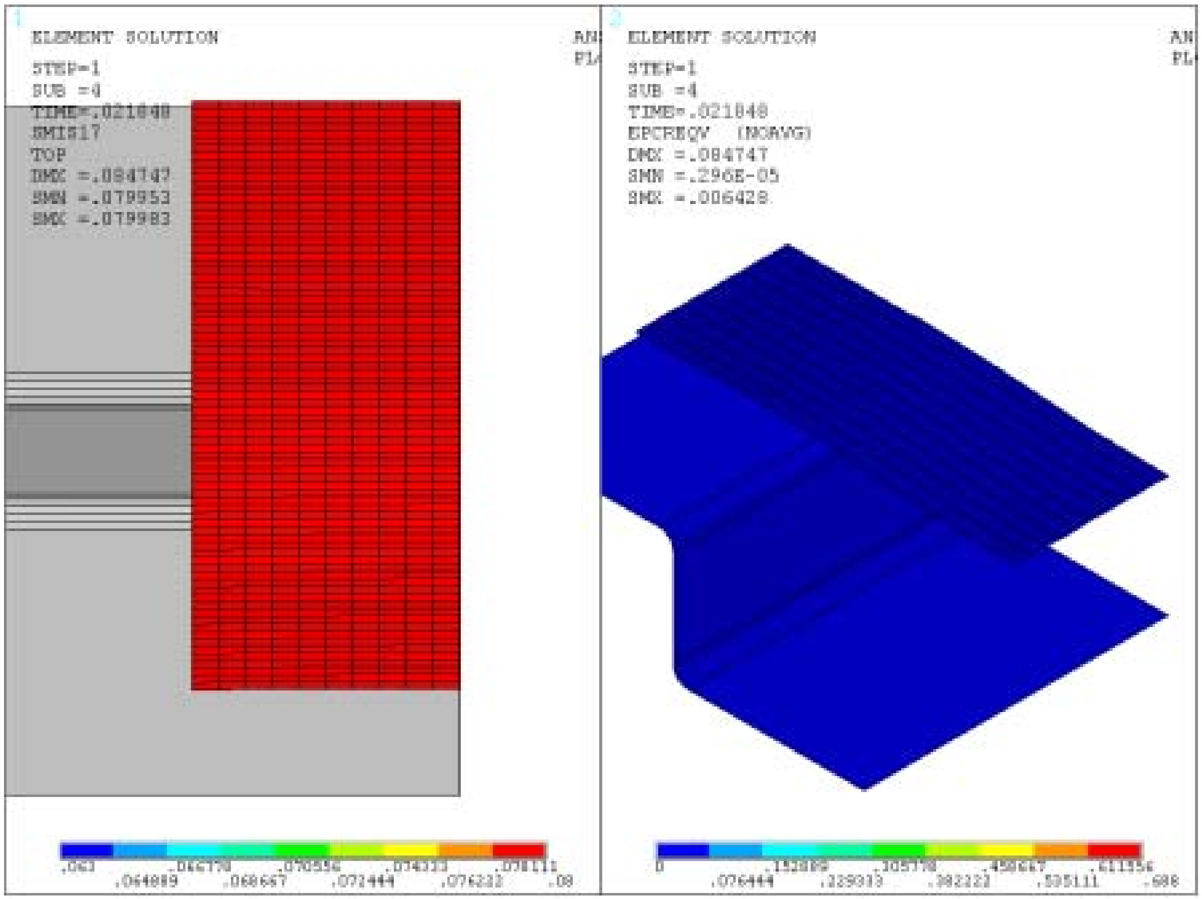
SHELL181, cyclic display expanded with thickness shown. Plasticity, birth & death, and rigid-deformable contact.

Shell-Deep Drawing example



Large strain, Multiple Contact Pairs, Material Nonlinearity

Example of Sheet Forming

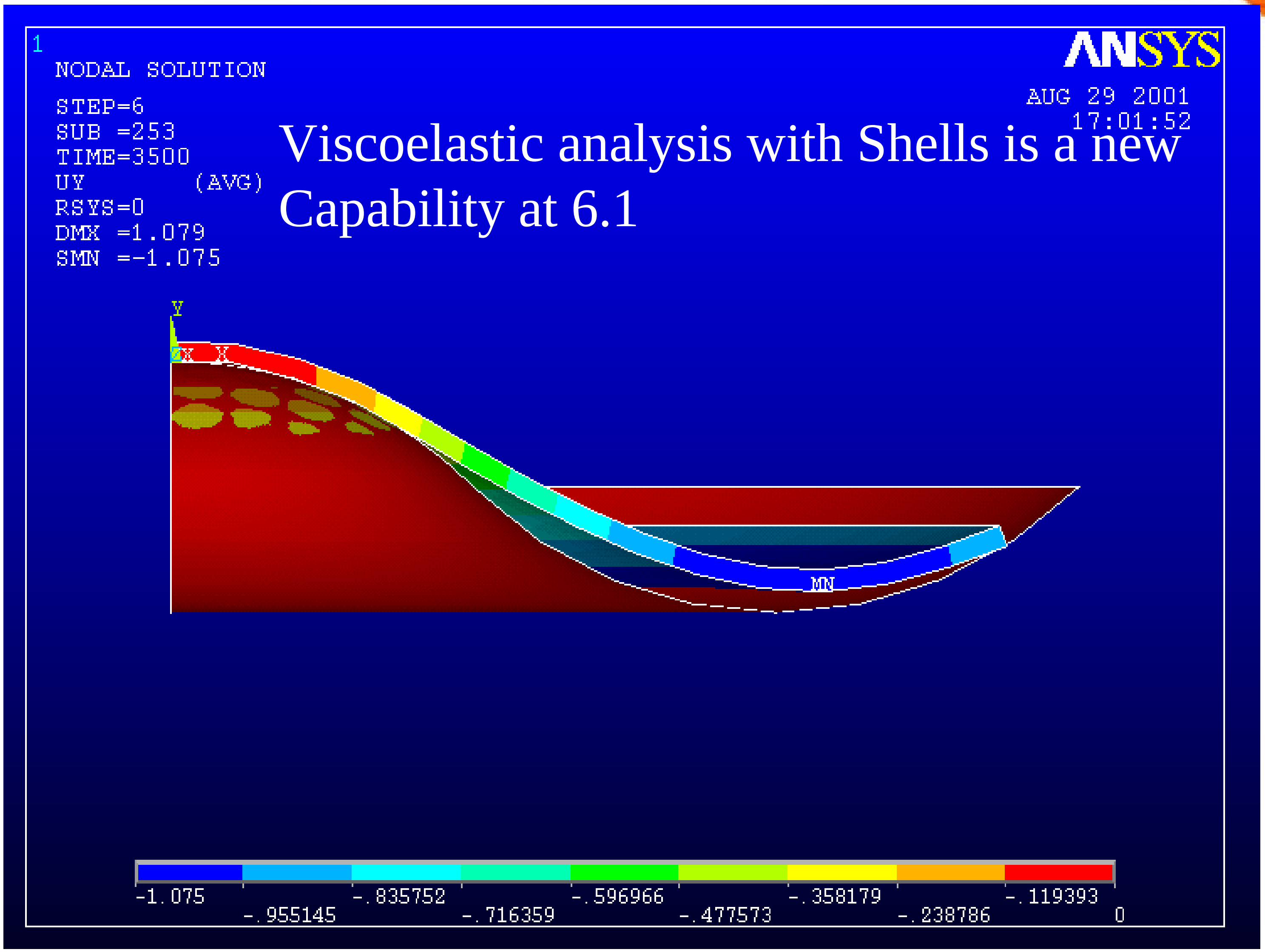


Animation of shell thickness and equivalent creep strain contours. SHELL181 with rigid-deformable contact using power law creep.

Shell-Springback study

ANSYS/LS-Dyna
Used for forming,
ANSYS implicit
Springback soln.

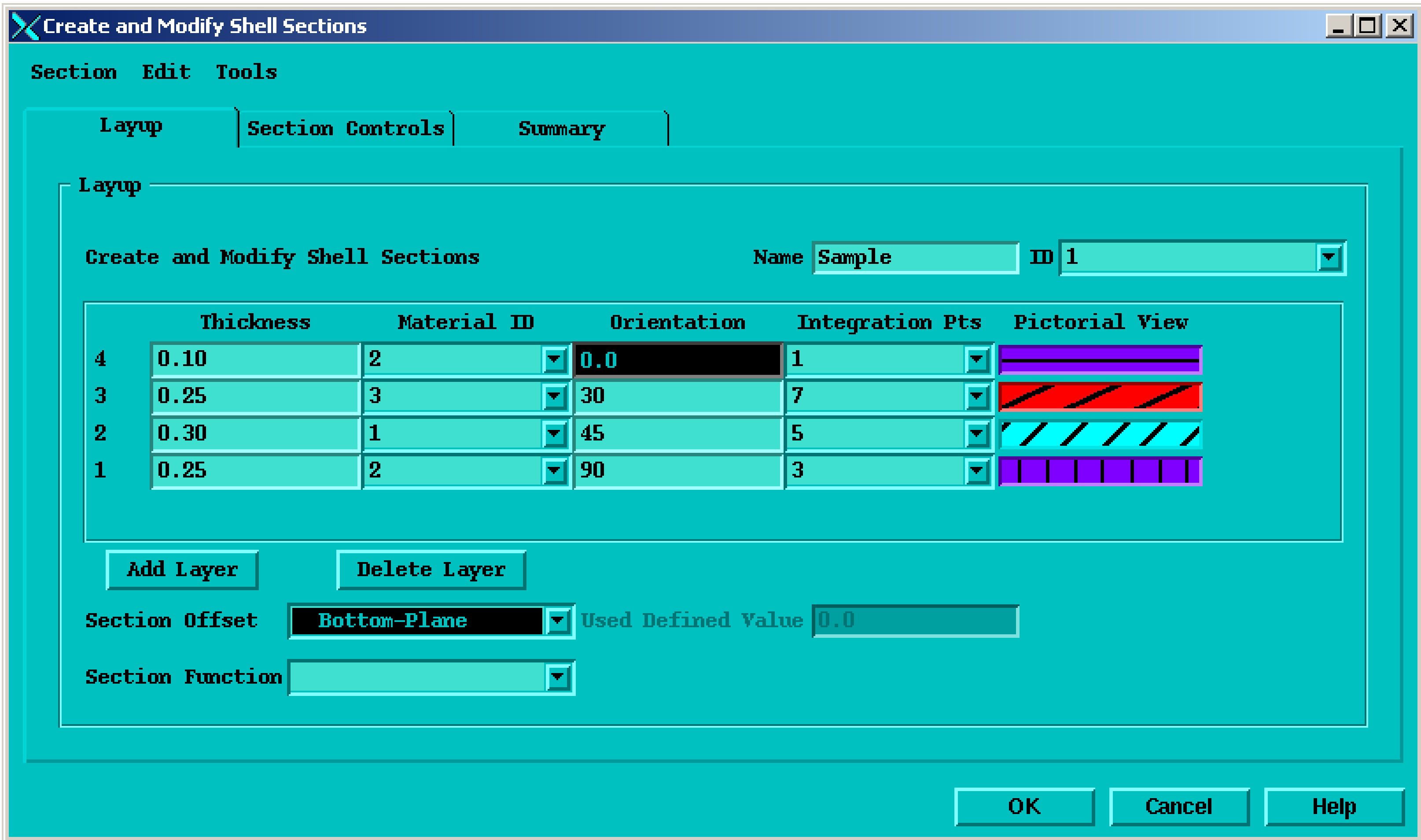
Deformation of glass under gravity



Shell181 is a layered shell

- Abundant choice of material models
- Consistent temp. dep. Matl properties
- Appealing approach to tapered shell analysis
- Usability, functionality enhancement
- In the beginning of each load step, ANSYS will evaluate
  - Interlaminar shear stress distribution coefficients
  - Consistent energy equivalent transverse shear correction factors
- Robust nonlinear analysis

Shell Section



Frequencies of a sandwich beam

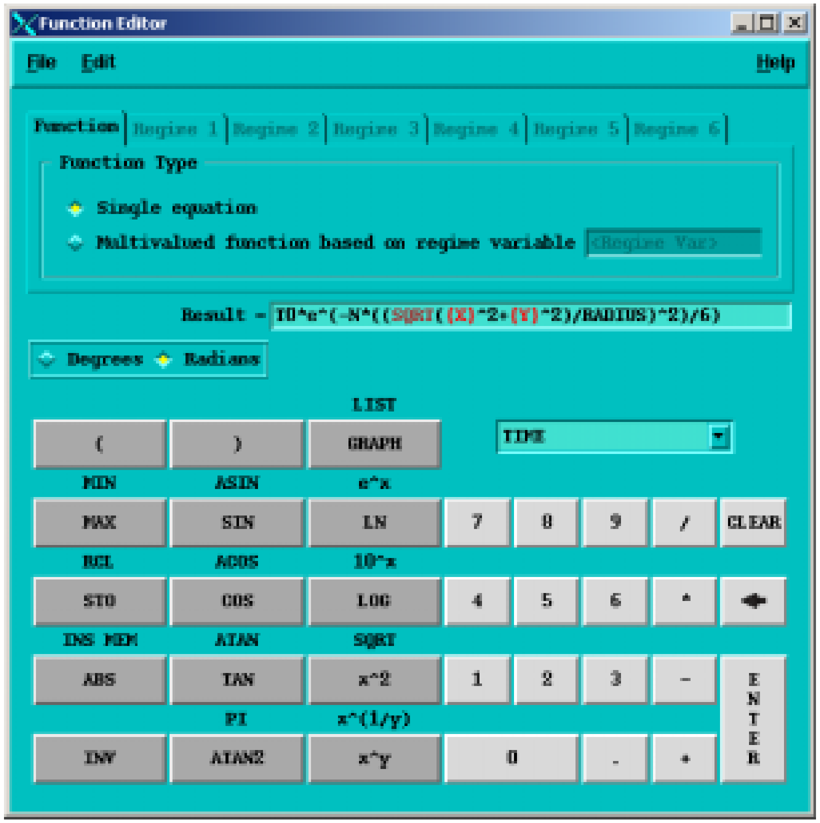
	45	91	91SW	181
1	35.887	36.918	35.297	35.640
2	184.73	183.29	182.52	183.61
3	192.85	229.91	183.28	191.18
4	211.71	339.61	194.19	208.56
5	457.71	639.23	419.96	452.45
6	625.67	988.07	568.07	614.20
7	759.96	1035.8	678.47	749.01
8	1001.3	1240.4	902.05	987.64
9	1014.0	1462.0	943.71	997.84
10	1082.4	1779.8	988.06	1063.4

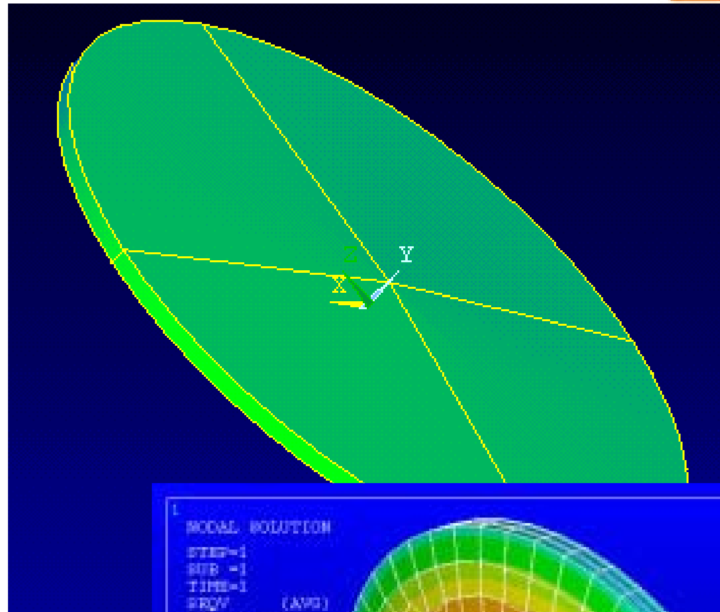
Tapered Shells

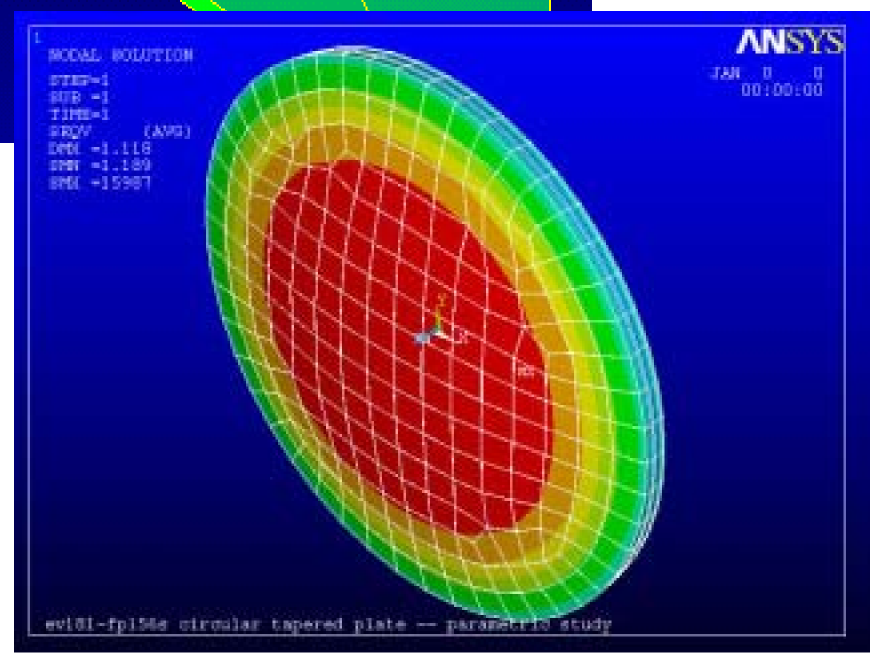
- Other options
  - Define real constants for elements
  - Use RTHICK
- High level of mesh dependence
- Not a frequent need in layered composites

※ Shell181 employs ANSYS Function capability

Tapered circular plate using Function Builder







Secfun and multi-layer shells

- The total thickness of laminate is indicated by the table
- Individual layer thickness are treated as “normalized layup configuration” which are scaled to actual thickness to match total
- Further improvements may be necessary for complex composite layups

Section & Real ?

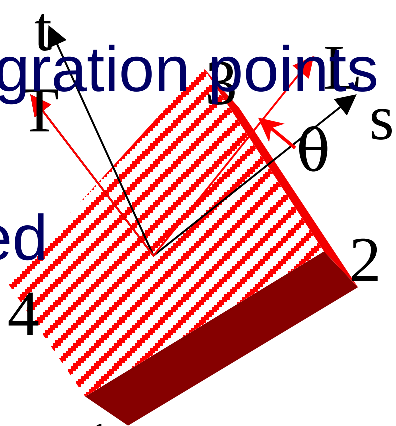
- SHELL181 will continue to accept Real constant data as before => legacy support
- When a valid “shell section” is associated with the element, real constant is ignored
- Section definition is more flexible and feature packed than real constants
  - Recommended input option
- Rebars and other features will be supported in a future release!

Section Commands

- SecType, SecId, SHELL, Name
- SecData, Thick, MatID, Ori, numSectPt

Thick	Layer Thickness
MatID	Material ID for layer
Ori	Layer Orientation
numSectPt	Number of integration points thru layer thickness

Repeat as many times as needed



"s-t" = Element Coordinate System (default or ESYS)

SecData parameters

- Thickness of layer
  $\geq$ 0.0 ; support ply drop-offs
  Sum of all layer thickness > 0.0
- MatId
  - If it is a homogeneous shell, blank is acceptable
    - We then use element attribute MAT
  - Required for all composite lay-ups
    - Element attribute is not used (except for TREF)
  - Can be any material supported in 18x family including UserMat

SecData parameters … (3)

- Layer Integration Points
  - Defaults to 3 (numerically exact for linear prob)
    - Slightly faster & smaller .esav file
  - ≥ 1, Must be an ODD number (if specified)
  - Integration points are equally spaced thru layer thickness
    - Except when 5 points per layer is specified.
  - Control layer output to .rst file

Section Offset

SecOffset, Position, Offy

- Position is one of TOP, BOT, MID, USER
- Offy is valid only when POSITION=USER

- SHELL181 will ignore rotary inertia effects for unbalanced/unsymmetrical lay-ups
  - Other shell elements in ANSYS ignore rotary inertia effects in all circumstances.

Usage: Full integration keyopt(3)=2

- Advantages
  - Higher accuracy in stresses
  - No spurious modes
  - No limitations in support of material models, applicability
  - Ability to model “cantilever” type bending with a single element thru thickness (stiffeners, coarse models)
- Disadvantages
  - Higher cost (more dof, constitutive calculation at a larger number of points)
  - Not so accurate in coarse models of doubly curved shells

Usage: Reduced Integration (default)

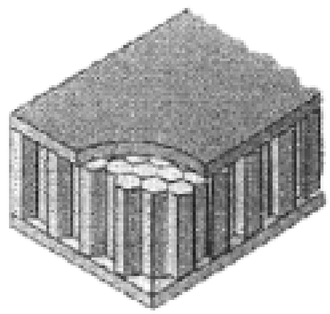
- Advantages
  - Under integration is only for inplane strains, robust hourglass control
  - Approximates curved shells better
  - Highly efficient in nonlinear applications (esp. when complex material models are employed)
  - Experience indicates that this is a better option with hyperelasticity
  - Superior convegence w.r.t mesh refinement
- Disadvantages
  - Not adequate stress accuracy, unless refined meshes are used

Beam Elements: State of the art capabilities

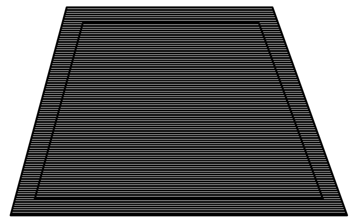
Accurate Analysis of Beams

	Classical	**188/189**
Shear Def.	Neglect	Included
Torsion	Approx.	Poisson Equation
Warping	Unrestrained	Restrained Unrestrained
Kinematic	Small Strain	Finite Strain
Material	Linear	Plasticity, creep... Multiple materials

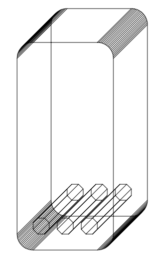
Applicability



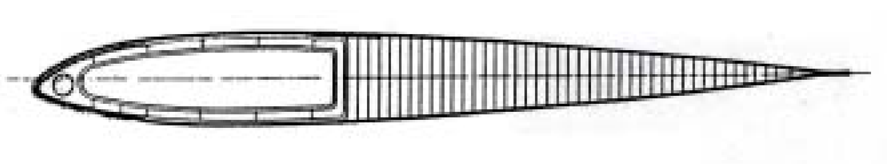
Sandwich



MEMS



Reinforced Beams



Optimized section

● ● ●

188/189 Beam Elements

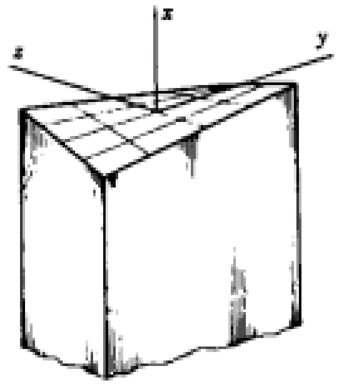
ANSYS Element Library	Type of Element	Interpolation	Formulation				Elasticity		Plasticity								Viscoplasticity		Hyperelasticity				Viscoelasticity	
			B-Bar	URI/Standard	Enhanced	Mixed U-P	Isotropic, Orthotropic	ANEL	BISO	MISO	NLISO	BKIN	KINH/MKIN	CHAB	CAST/UNIAXIAL	HILL	RATE (PEIRCE, PERZYNA)	CREEP (Implicit)	Mooney-Rivlin	Polynomial Form	Ogden	Arruda-Boyce	Viscoelasticity (hypoelasticity)	Viscoelasticity (hyperelasticity)
BEAM188	Beam	Linear		•			•		•	•	•	•	•	•	•		•	•						
BEAM189	Beam	Quadratic		•			•		•	•	•	•	•	•	•		•	•						

Unrestrained warping

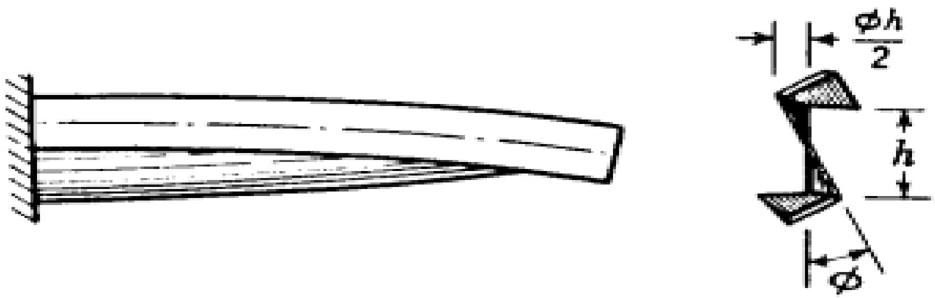
- Cross sectional warping is assumed to be small, such that the axial strains caused by warping may be neglected
  - Good approximation for solid sections (Circular, Rectangular…)

$$\mathbf{x} = \mathbf{x}^0 + H^\alpha \mathbf{n}^\alpha$$

$$\varepsilon = \ln(\lambda) + H^\alpha \varepsilon_\alpha^\beta \kappa^\beta + O(h^2)$$



Open Sections

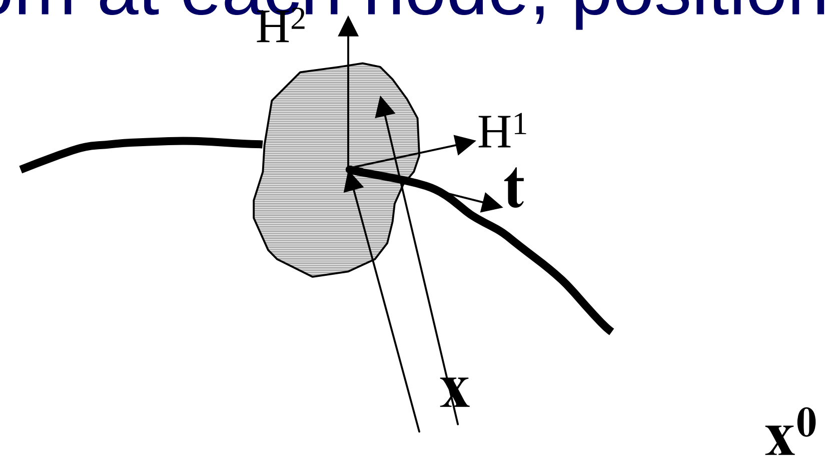


Thin walled open sections exhibit substantial warping. Torsional stiffness of such sections is negligible, and restraining of warping provides resistance against twist.

Inclusion of warping

- Degrees of freedom at each node, position vector

$$\{\mathbf{u}, \boldsymbol{\varphi}, \omega\}$$



$$\mathbf{x} = \mathbf{x}^0 + H^\alpha \mathbf{n}^\alpha + \omega \psi \mathbf{t}$$

$$\varepsilon = \ln(\lambda) + H^\alpha \varepsilon_\alpha^\beta \kappa^\beta + \frac{d\omega}{dS} \psi \mathbf{t} + O(h^2)$$

Basic formulation

$$D\delta W = \int_{\ell} (DF_A \delta\varepsilon + DS_\alpha \delta\gamma_\alpha + DM_\alpha \delta\kappa_\alpha + DT\delta\kappa_t + DB\delta\chi) d\ell +$$

$$\underbrace{(F_A D\delta\varepsilon + S_\alpha D\delta\gamma_\alpha + M_\alpha D\delta\kappa_\alpha + TD\delta\kappa_t) d\ell}_{\text{Stress Stiffness}}$$

Section integration is employed for

- Nonlinear materials
- Temp. dependency
- Multi-materials

Shear force/Shear Strain relationship is assumed to be elastic

FEM solution for section

– Torsion problem

$$\delta W = \int_A \left(\frac{\partial \omega}{\partial y}\delta\frac{\partial \omega}{\partial y} + \frac{\partial \omega}{\partial z}\delta\frac{\partial \omega}{\partial z}\right)dA + \int_A \left(-z\delta\frac{\delta \omega}{\partial y} + y\delta\frac{\partial \omega}{\partial z}\right)dA$$

– Direct Shear

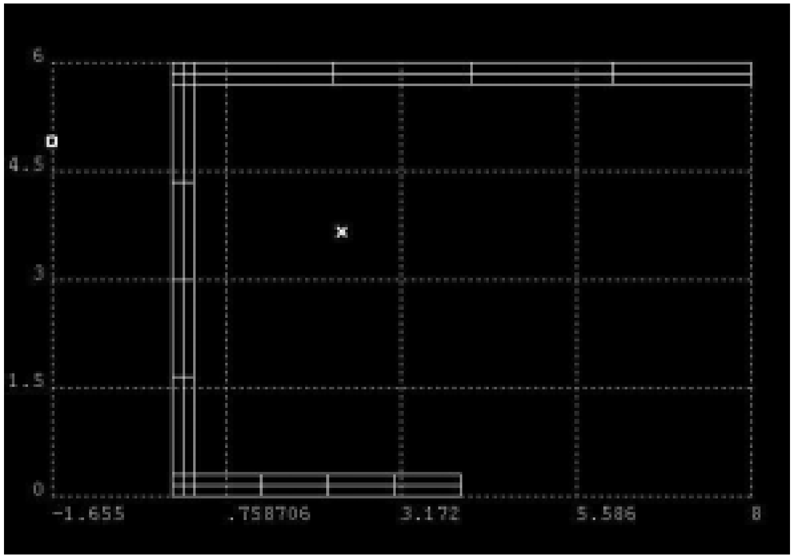
$$\delta W = \int_A \left[\left(\frac{\partial \psi}{\partial y} - S_1\right)\delta\frac{\partial \psi}{\partial y} + \left(\frac{\partial \psi}{\partial z} - S_2\right)\delta\frac{\partial \psi}{\partial z}\right]dA +$$

$$K_y \int_A y\delta\psi dA + K_z \int_A z\delta\psi dA$$

– Ref: Schramm, U., Kitis, L., Kang, W., and Pilkey, W.D., “On the Shear Deformation Coefficient in Beam Theory”, Finite Elem in Analysis and Design, Vol. 16, 1994

Section FEM

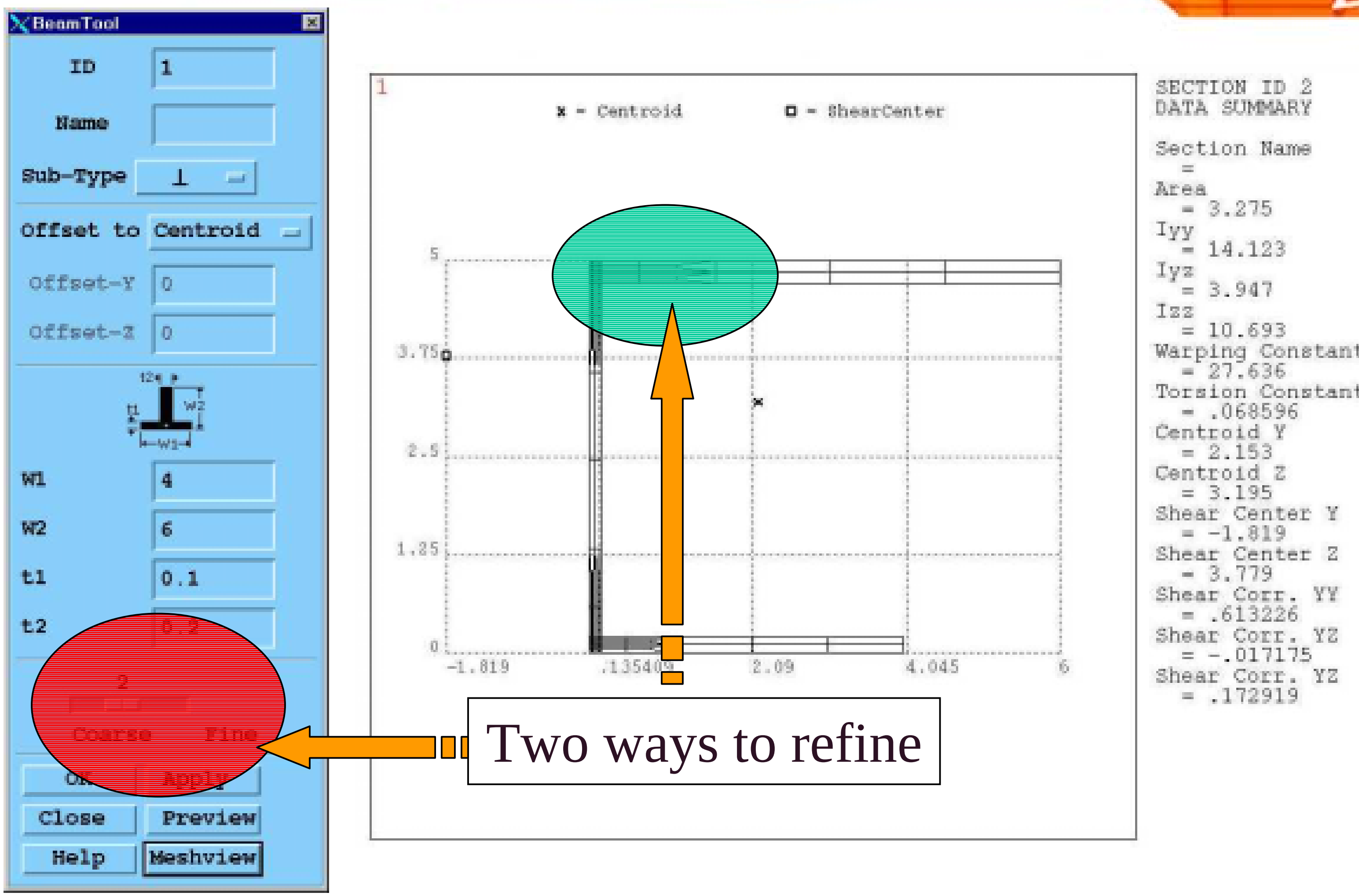
All Cross Sections are discretized using a second order element



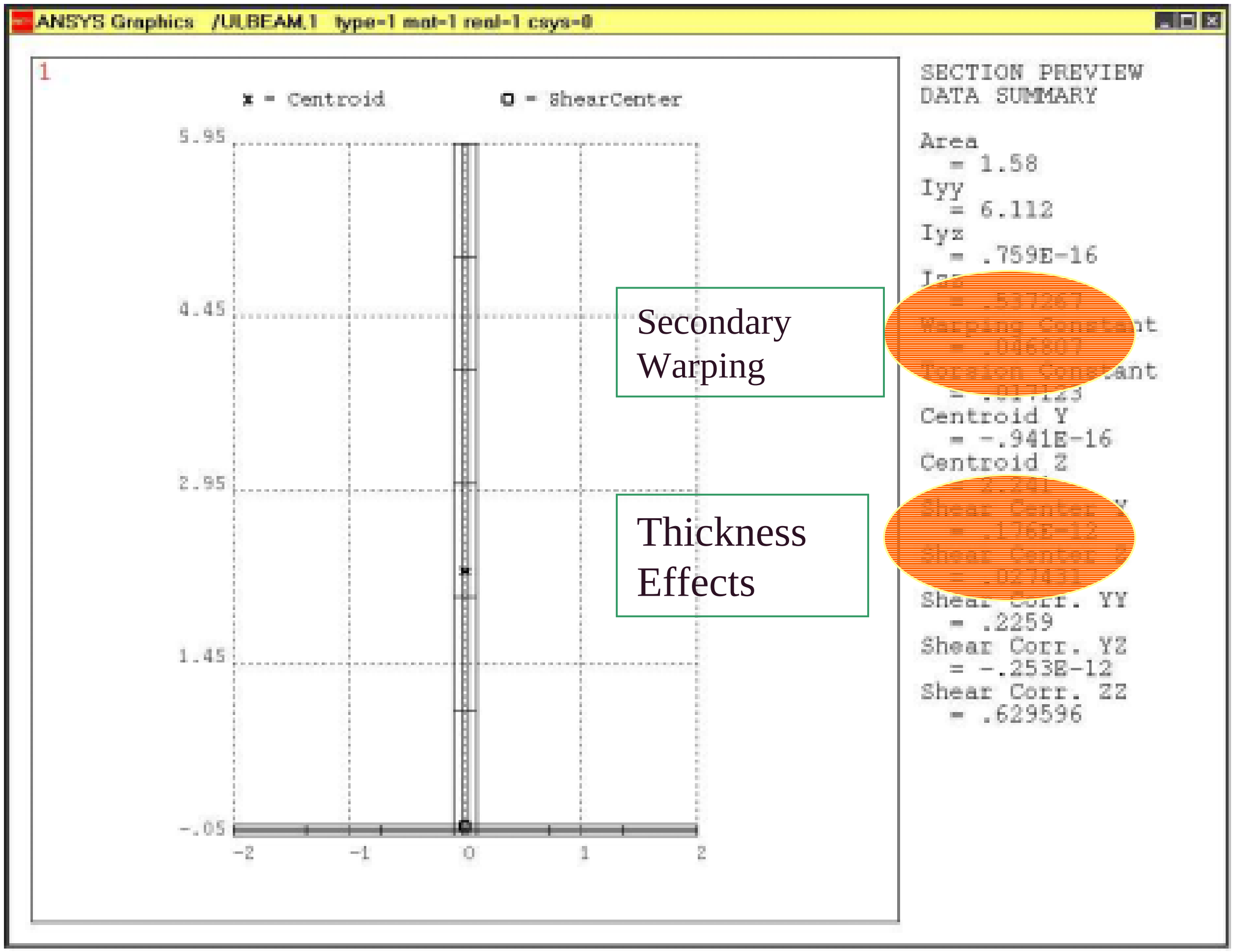
Section Solver

- Shear center, warping rigidity and shear correction factors are calculated for all sections, including user mesh
  – Unified treatment,
- Transformed section method is used for multi-material cross sections
- No hard limits on section size
  - Employs ANSYS sparse solver for efficient section solution

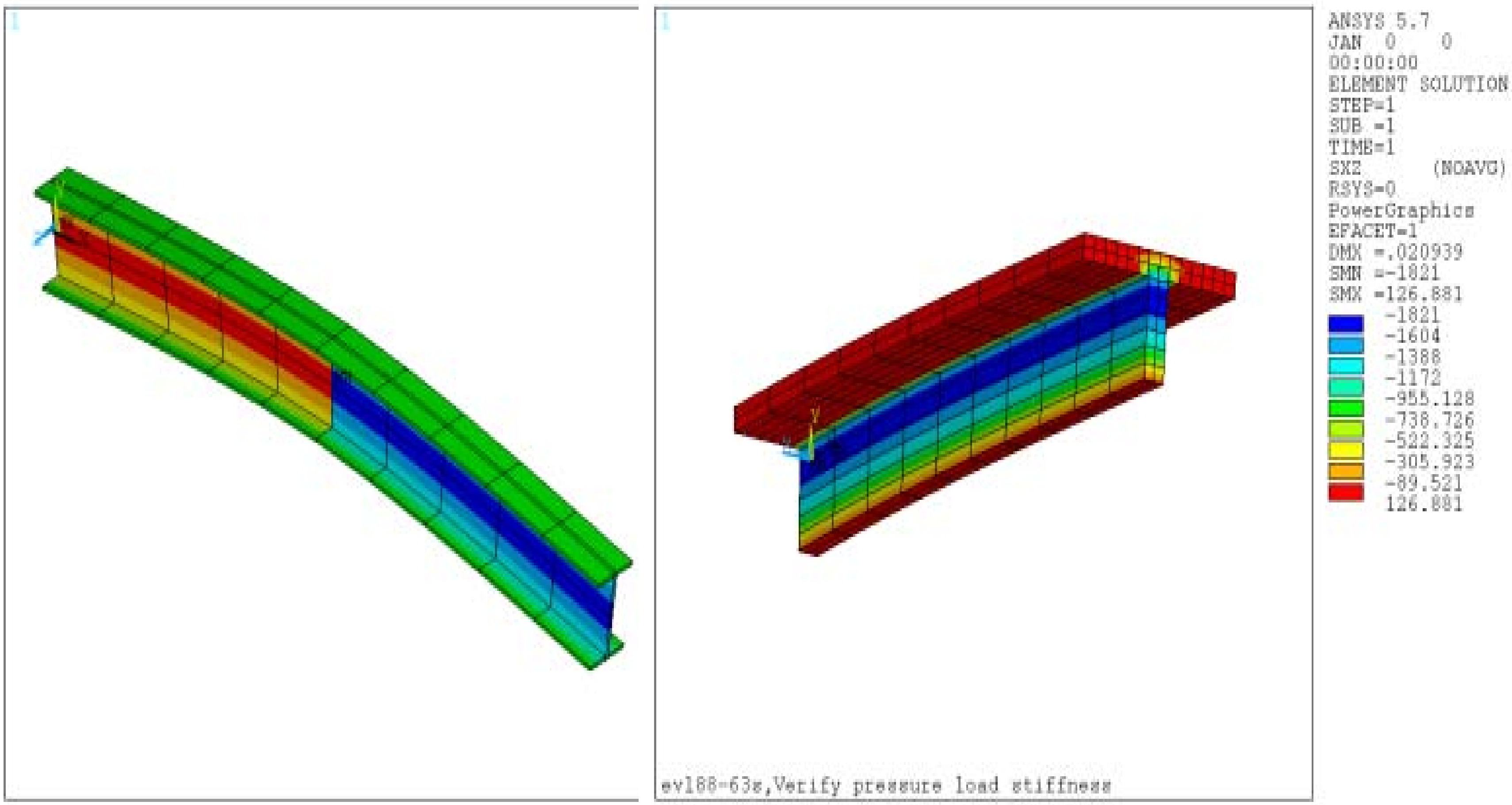
Refinement Tools



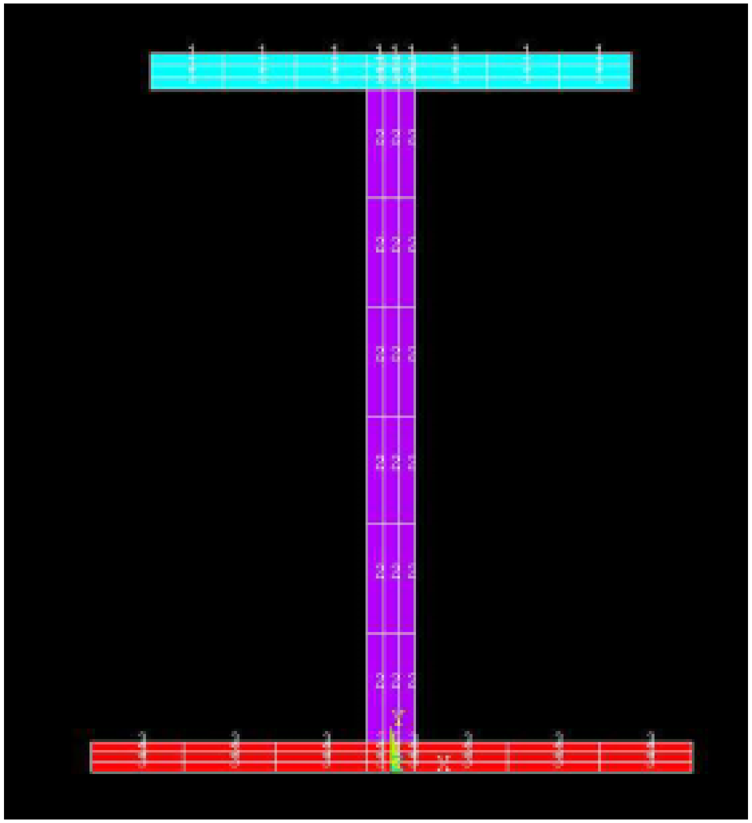
Accurate Section Props.



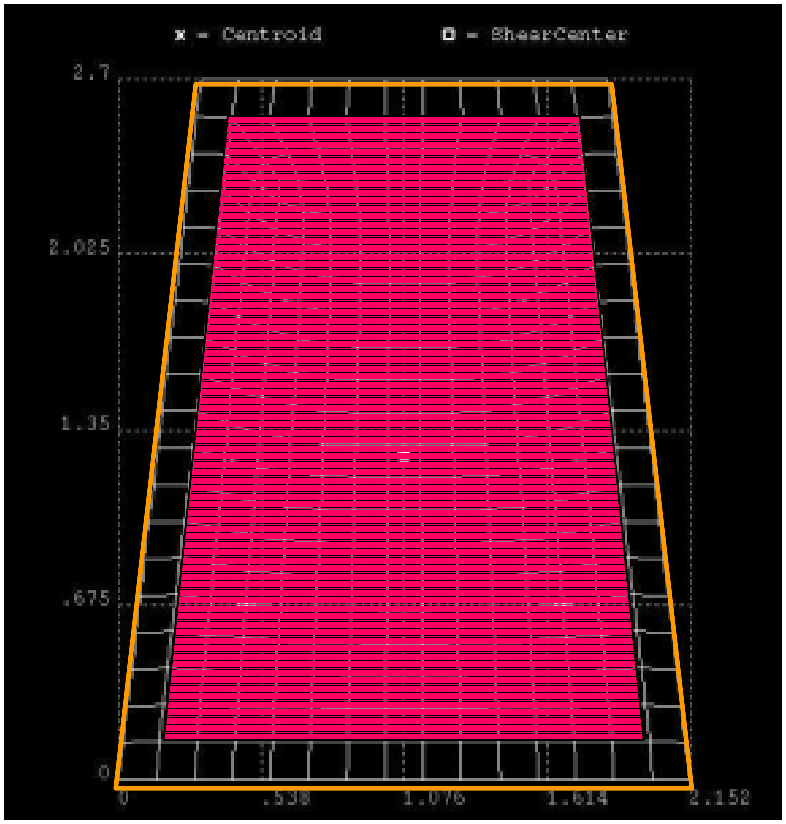
Direct Shear Stress Visualization



Built-up Section Tools

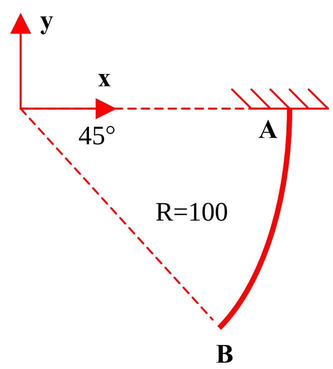


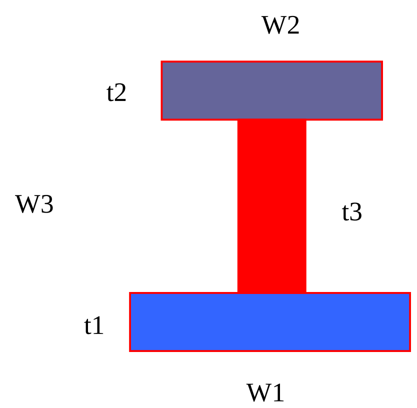
Define Materials for
a Standard Section



Define Materials for a Custom Section

curved composite cantilever beam



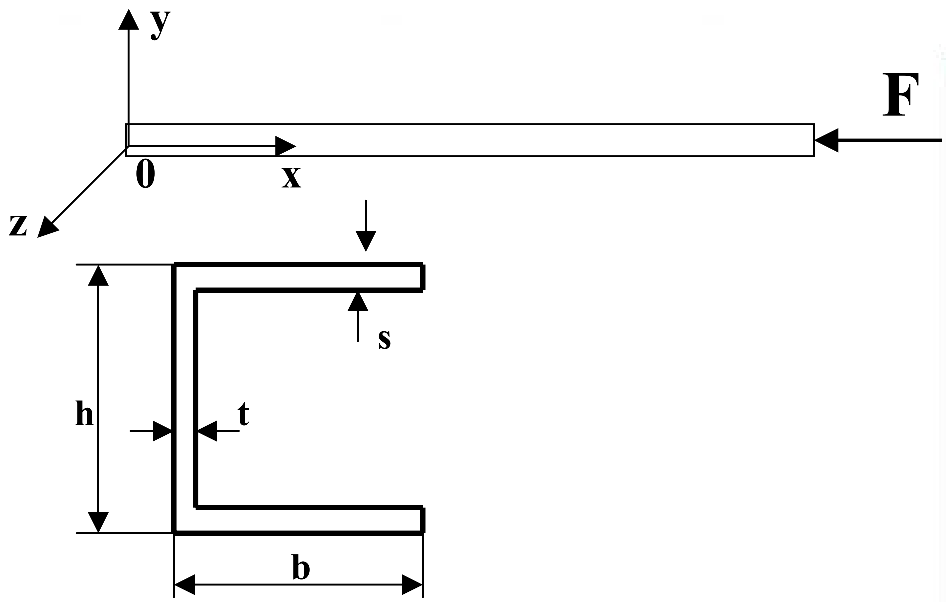


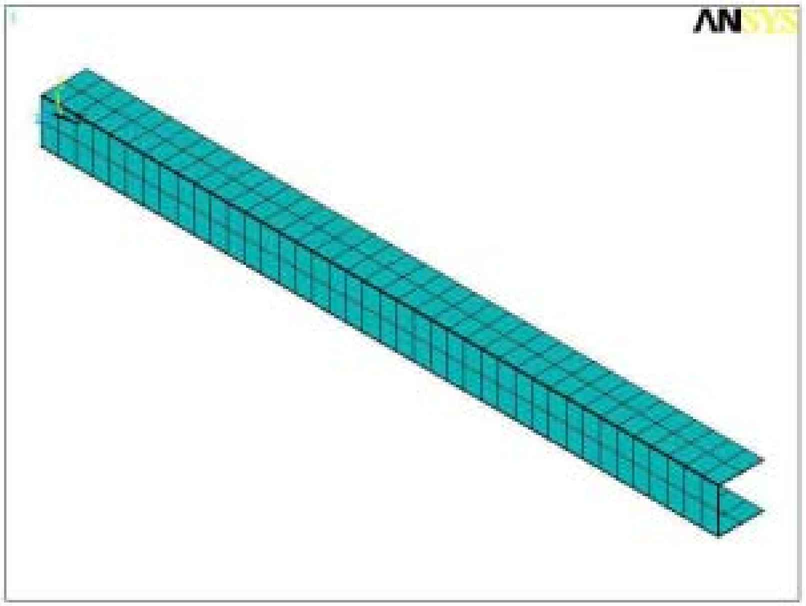
	BEAM189 (NDOF=96)		BEAM189 (NDOF=192)		SOLID186 (NDOF=18900)
Max. displacement	Value	% diff.	Value	% diff.	Reference value
Ux	19.664	0.2	19.666	0.2	19.625
Uy	24.819	1.9	24.822	1.9	25.310
Uz	54.486	0.5	54.490	0.5	54.769
CPU Time	82.610		115.460		4587.850

Parametric Study (Sandwich Beam)

Normalized Results (Beam189/SOLID45)									
E-Face/E-Core	Length/Thickness	UZ	UY	UX	ROTX	Frequencies			
1	40	1.0025	1.0025	1.0000	1.0162	0.9987	0.9989	0.9757	0.9768
2	40	1.0025	1.0025	1.0000	1.0148	0.9987	0.9989	0.9758	0.9770
20	40	1.0025	1.0025	1.0030	1.0281	0.9988	0.9989	0.9758	0.9323
200	40	1.0026	1.0025	1.0050	1.0704	0.9988	0.9987	0.9759	1.0402
2000	40	1.0062	1.0025	1.0050	1.0659	0.9988	0.9930	0.9759	0.9625
20000	40	1.0510	1.0025	1.0040	0.9252	0.9987	0.9436	0.8883	0.9761
1	20	1.0025	1.0023	0.9930	1.0186	0.9987	0.9993	0.9757	0.9799
2	20	1.0025	1.0025	0.9940	1.0187	0.9988	0.9994	0.9760	0.9807
20	20	1.0025	1.0025	1.0030	1.0425	0.9988	0.9995	0.9762	1.1674
200	20	1.0035	1.0025	1.0190	1.0654	0.9988	0.9987	0.9762	0.9757
2000	20	1.0232	1.0025	1.0080	1.0537	0.9988	0.9930	0.9726	0.9763
20000	20	1.1332	1.0070	1.0090	0.7943	0.8676	0.9988	0.7716	0.6789

Lateral Torsional Buckling





Geometry

h = 10 cm, b = 10 cm, s = t = 0.2 cm,

L =150 cm (length)

Material Properties

E = 21000 kN/cm^2, G = 8077 kN/cm^2

Boundary Conditions

x = 0: $u_x = u_y = u_z = 0$

$\theta_x = 0$

x = L: $u_y = u_z = 0$

$\theta_x = 0$

Warping restraint is important!

Critical buckling loads (Theory and numerics of three-dimensional beams with elastoplastic material behavior by F.Gruttmann et al.):

$$F_1 = n^2 \frac{\pi^2 EI_{22}}{L^2} \qquad F_2 = n^2 \frac{\pi^2 EI_{33}}{L^2} \qquad F_3 = \frac{1}{(i_M)^2}\left(GI_T + n^2 \frac{\pi^2 EI_w}{L^2}\right)$$

The theoretical critical buckling load:

$$F_{cr} = 2\left[\left(\frac{1}{F_2}+\frac{1}{F_3}\right)+\sqrt{\left(\frac{1}{F_2}-\frac{1}{F_3}\right)^2+\frac{4}{F_2F_3}\left(\frac{m_2}{i_M}\right)^2}\,\right]^{-1}$$

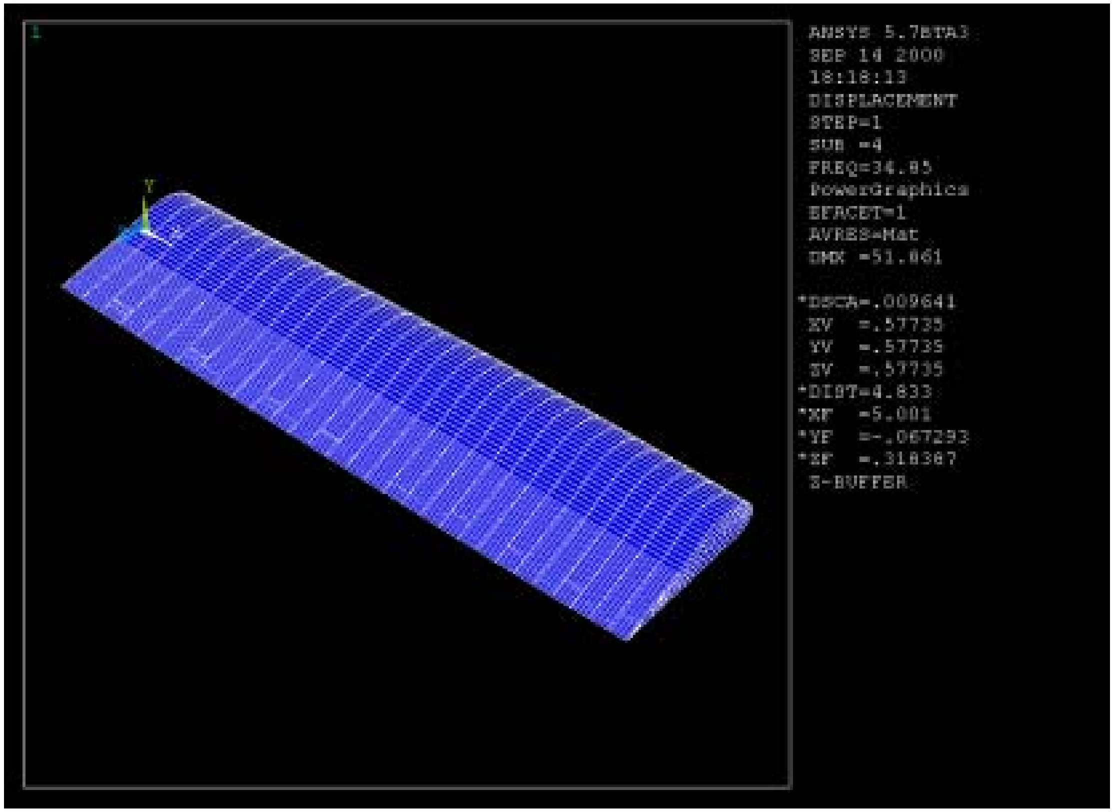
where $(i_M)^2 = (i_p)^2 + (m_2)^2$

Mode Number	Value of the Buckling Load (kN) FEM (w/o warping)	FEM (w/ warping)	Reference
1	7.371	113.9	115.5
2	7.398	419.6	443.3

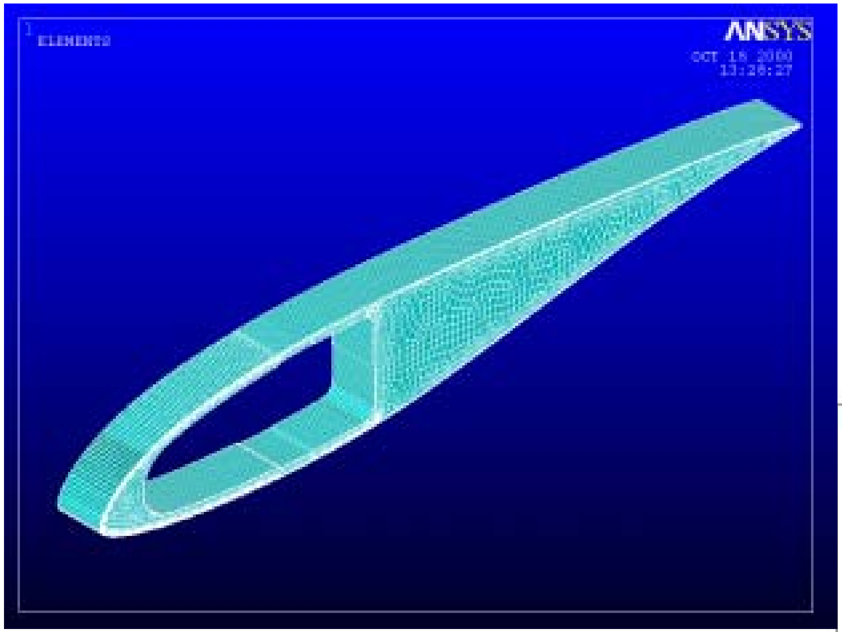
Pure torsional reference critical buckling load w/o warping = 7.386 (kN)

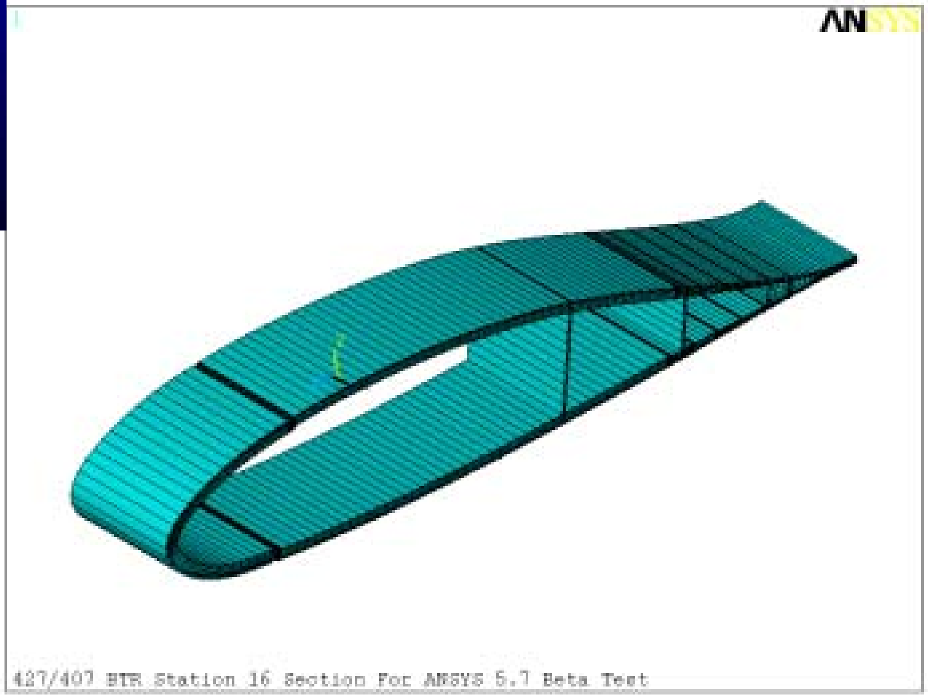
Modal Analysis of a Airplane Wing

Mode	*SOLID45*	*BEAM189*
1	3.56799	3.5306
2	17.2689	17.174
3	22.0382	21.830
4	35.6358	34.850
5	60.4129	60.019



Composite Rotor Section





Other Infrastructure

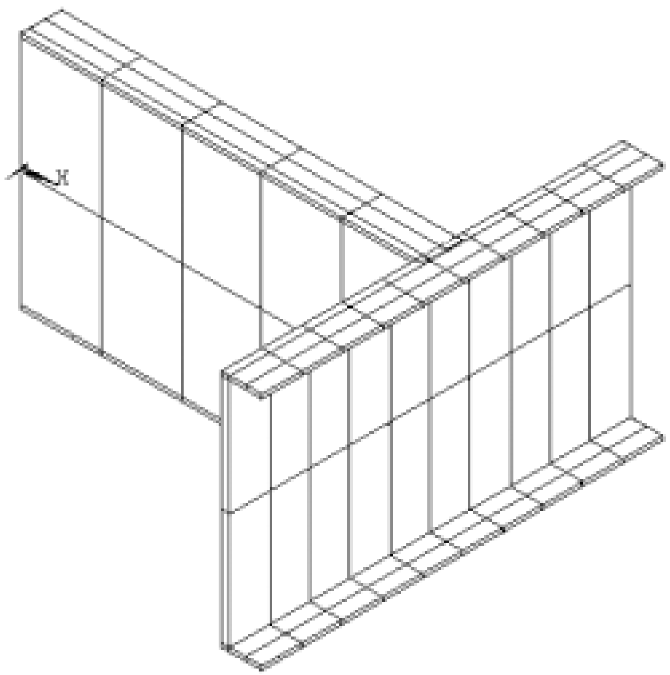
- Extensive nonlinear material library
  - Orthotropic elasticity,
  - BISO, MISO, BKIN, MKIN, KINH, NLISO, Creep, Chaboche, Rate Dependent plasticity
  - User material
- Distributed load, Follower Load Effects
- Initial Stresses
- 3D Post processing (/eshape,/efacet)
- Tapered capability is planned

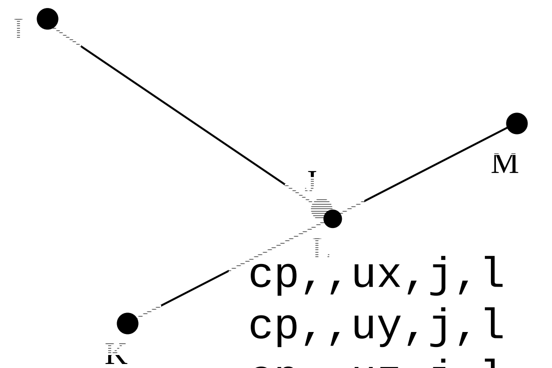
Usage tips:

- We recommend BEAM189 “almost always” for beam models
  - BEAM189 has linear bending moment variation, and hence requires fewer elements
- BEAM188 should be used for stiffeners, when SHELL181 (or SHELL43) is used for the shells
- Activate warping d.o.f. “almost always”
  - Computational overheads are minimal
  - ANSYS deals automatically with cases when you don’t need them

Usage Tips: Frame intersections

- Need independent warping d.o.f. for each member
- Use multiple nodes if $\omega_J \neq \omega_L \neq 0$






```
cp,,ux,j,l
cp,,uy,j,l
cp,,uz,j,l
cp,,rotx,j,l
cp,,roty,j,l
cp,,rotz,j,l
```

Bear in mind...

- Beam elements 188/189 are based on a first order shear deformable theory
  - No warping is considered due to direct shear stresses
- Applicability to non-homogeneous cross section is limited by the approximations of beam theory
- Validate by solid/shell models first