# The Finite Element Method

## General Meshing Guidelines and Accuracy

ANSYS
Computational Mechanics, AAU, Esbjerg

# General Considerations in Meshing

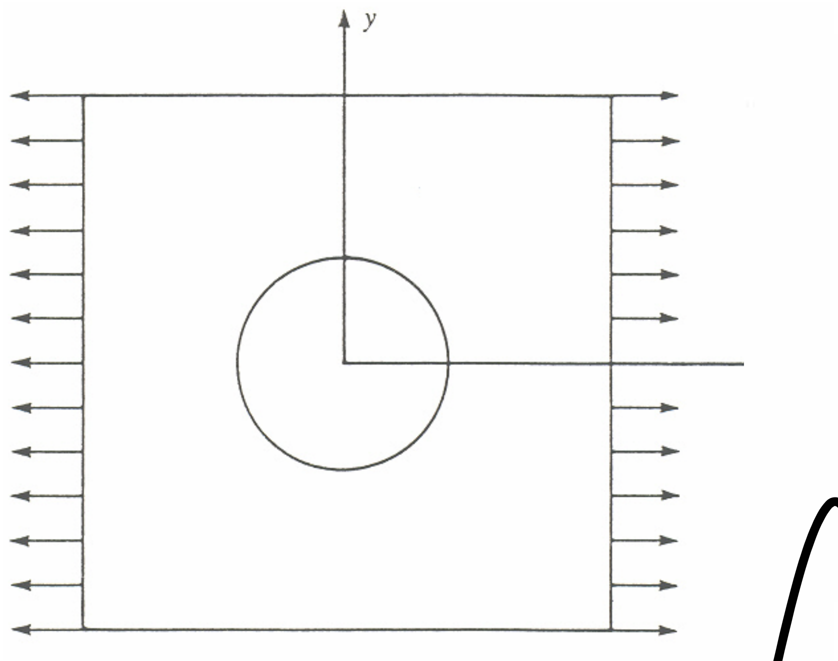
- When choosing elements and creating meshes for FEA problems users must make sure that
  - Chosen mesh size and density are optimal for the problem (to save computational time)
  - Chosen element types are appropriate for the analysis type performed (for accuracy)
  - Element shapes do not result in near singular stiffness matrices
  - Chosen elements and meshes can represent force distributions properly

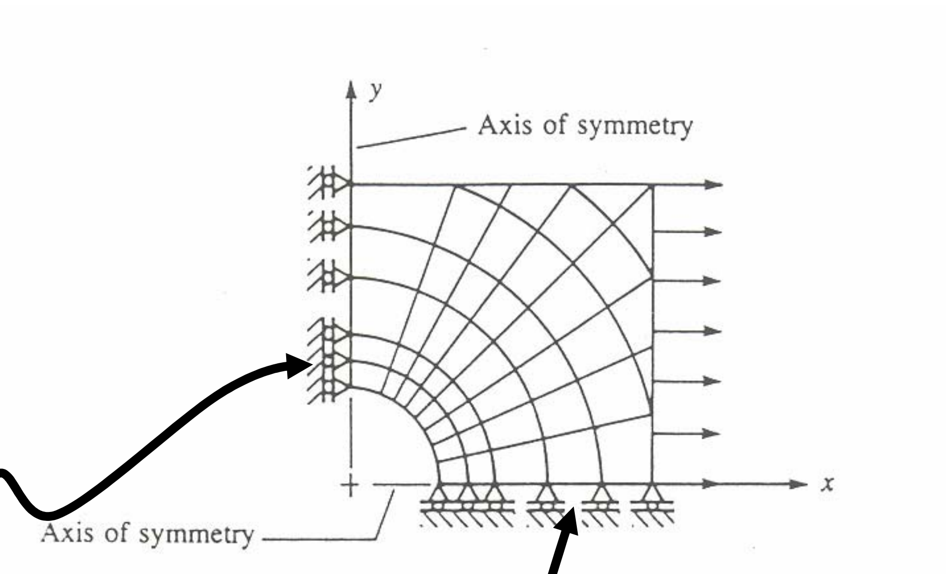
# Symmetry

- One of the most powerful means of reducing the size of a FEA problem is the exploitation of *symmetry*
- Symmetry is said to exist if there is a complete symmetry of geometry, loads and constraints about a line or plane of symmetry
- When exploiting symmetry model needs to be modified to replace the line or plane of symmetry without affecting the results

# Symmetry (cont'd)

- An simple case of complete symmetry

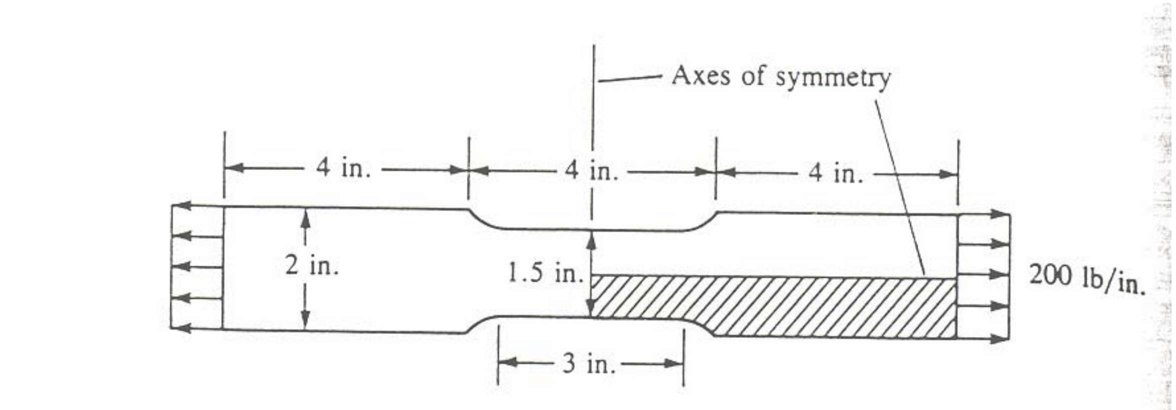


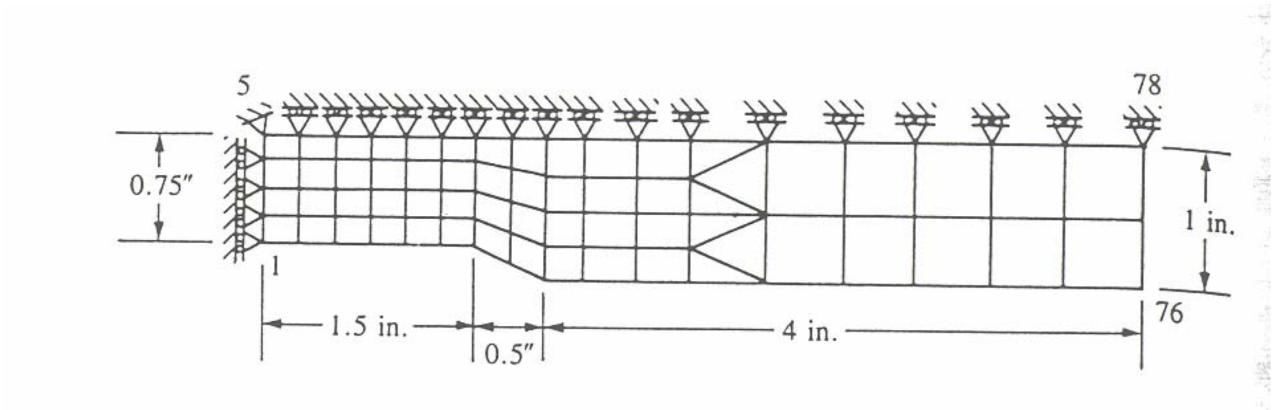




Constraints corresponding to lines of symmetry (LOS) do not allow displacements perpendicular to the LOS

# Symmetry (cont'd)

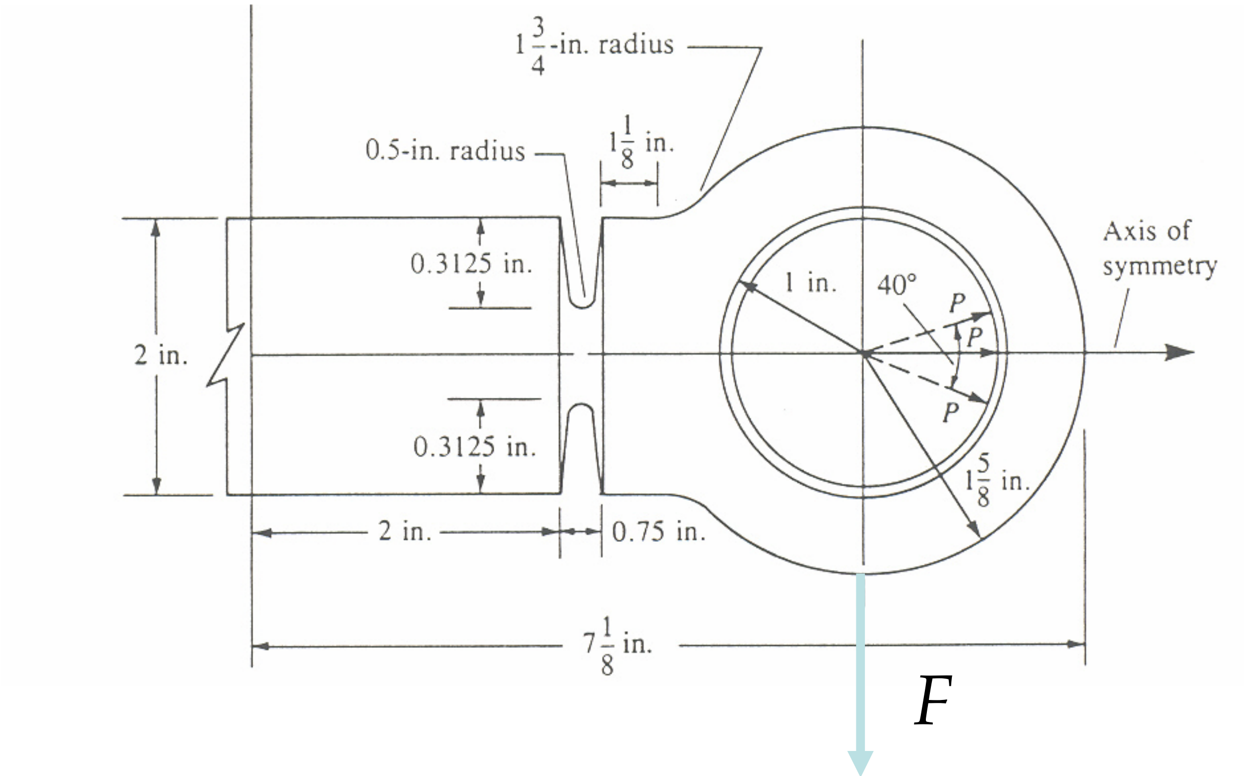
- Similarly



# Symmetry (cont'd)

- There is no symmetry in this case

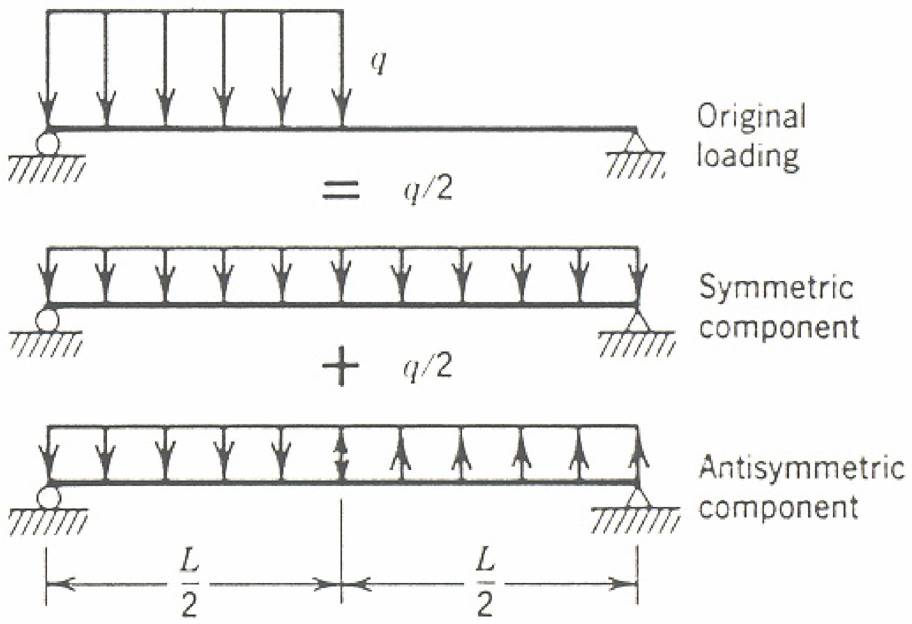


# Symmetry Meshing Rules

- Nodes must be placed on lines or planes of symmetry
- In 2D nodes on lines of symmetry (LOS) must be constrained to have zero displacements perpendicular to LOS; no rotational constraints on LOS (in-plane)
- In 3D nodes on the plane of symmetry (POS) must be constrained to have zero displacements out of the POS; no in-plane rotational constraints on POS

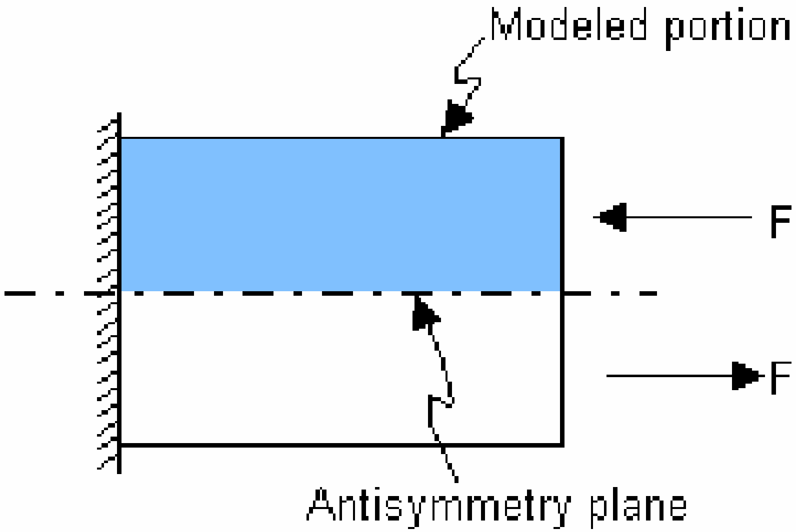
# Antisymmetry

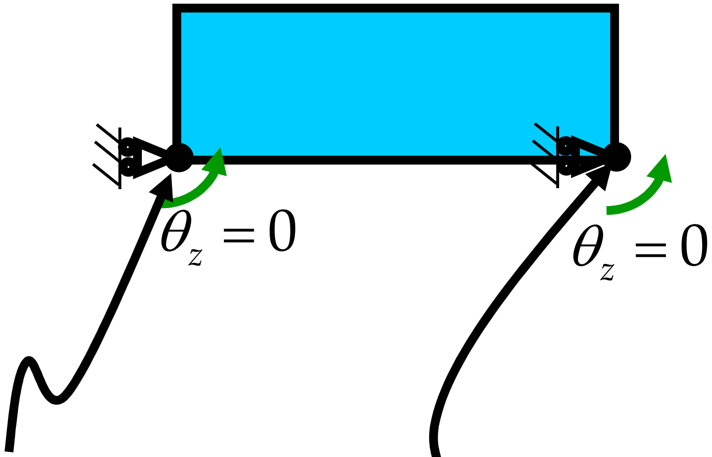
- Sometimes the loading or boundary conditions may be such that *antisymmetry* exists



# Antisymmetry (cont'd)

- Consider the simple antisymmetry case



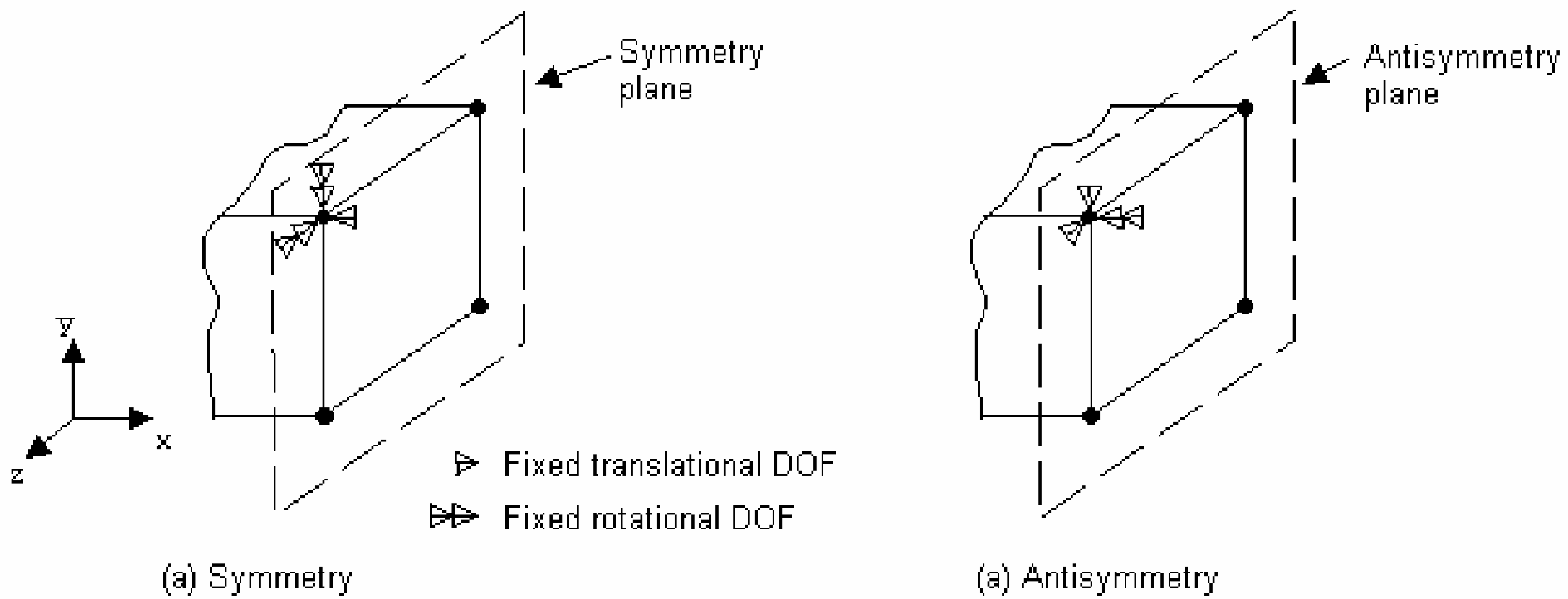

Constraints corresponding to lines of antisymmetry (LOAS) do not allow displacements along the LOAS or any rotational displacements

# Antisymmetry Meshing Rules

- Nodes must be placed on lines or planes of antisymmetry
- In 2D nodes on lines of antisymmetry (LOAS) must be constrained to have zero translational and rotational displacements along (in-plane) LOAS
- In 3D nodes on the plane of antisymmetry (POAS) must be constrained to have zero in-plane translational and rotational displacements

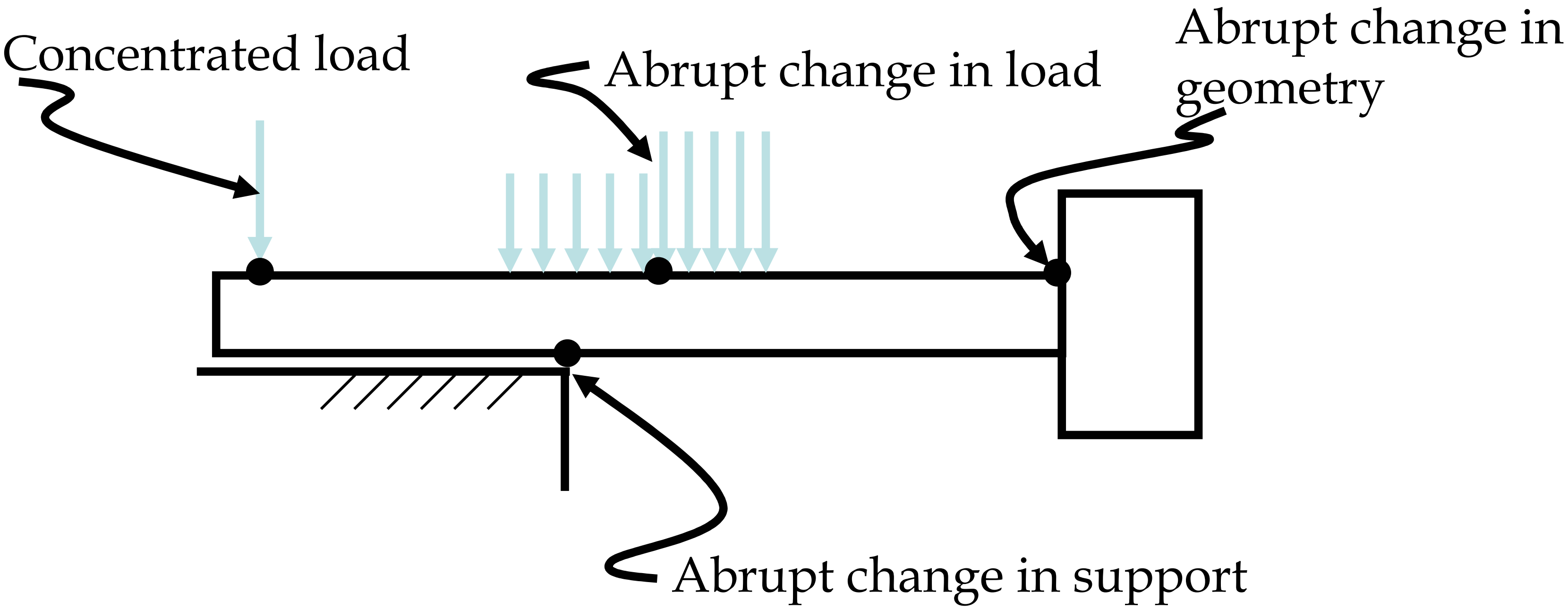
# Symmetry/Antisymmetry in ANSYS

- ANSYS supports symmetry and antisymmetry constraint sets



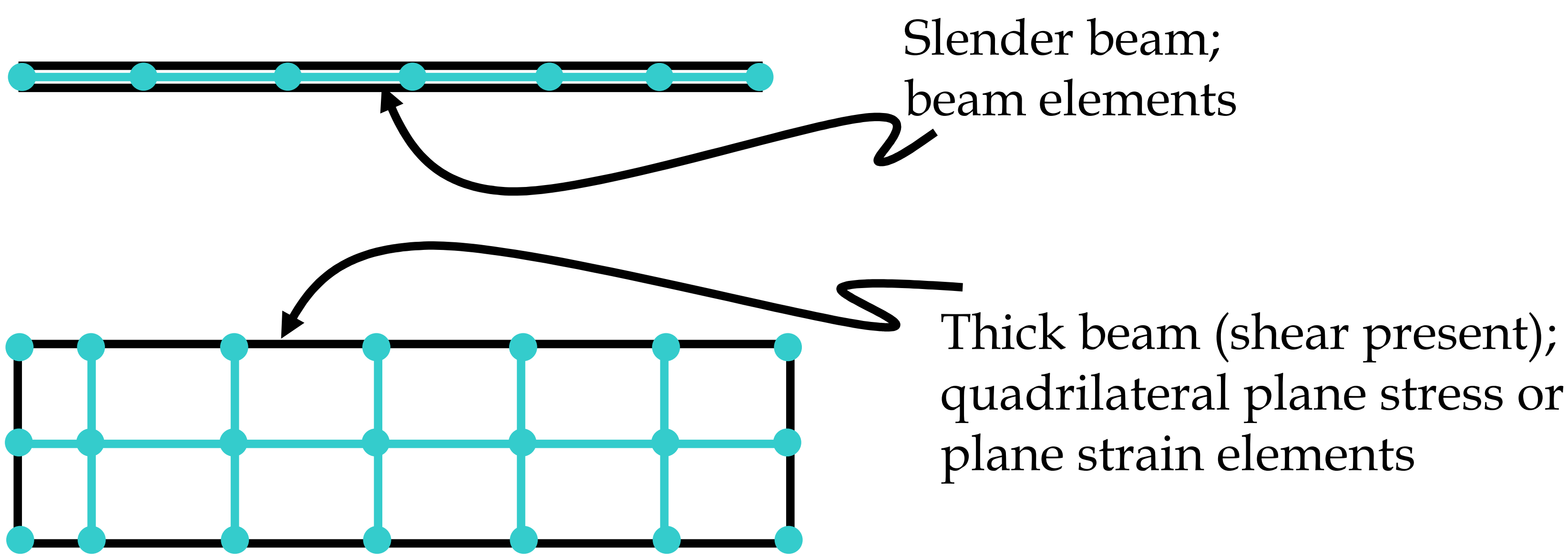
# Discontinuities

- Nodes must always be placed at locations where geometry, loads, or boundary conditions change abruptly (discontinuities)



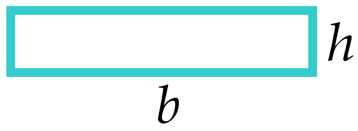
# Correct Choice of Elements

- Choose element types that are appropriate for the loading and stress conditions of the problem
- Make sure that the elements chosen capture all possible significant stresses that may result from the given loading, geometry, and boundary conditions



# Aspect Ratio

- For a good mesh all elements must have a low aspect ratio



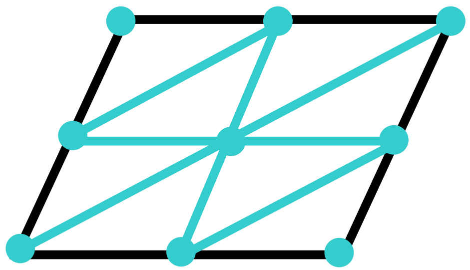

- Specifically

$$\frac{b}{h} \leq 2-4$$

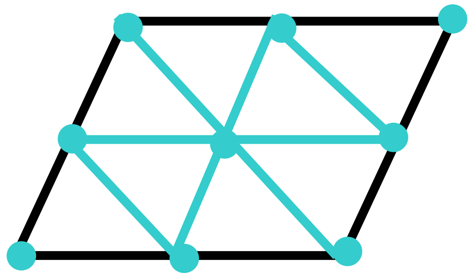
- where *b* and *h* are the longest and the shortest sides of an element, respectively

# Element Shape

- Angles between element sides must not approach 0° or 180°



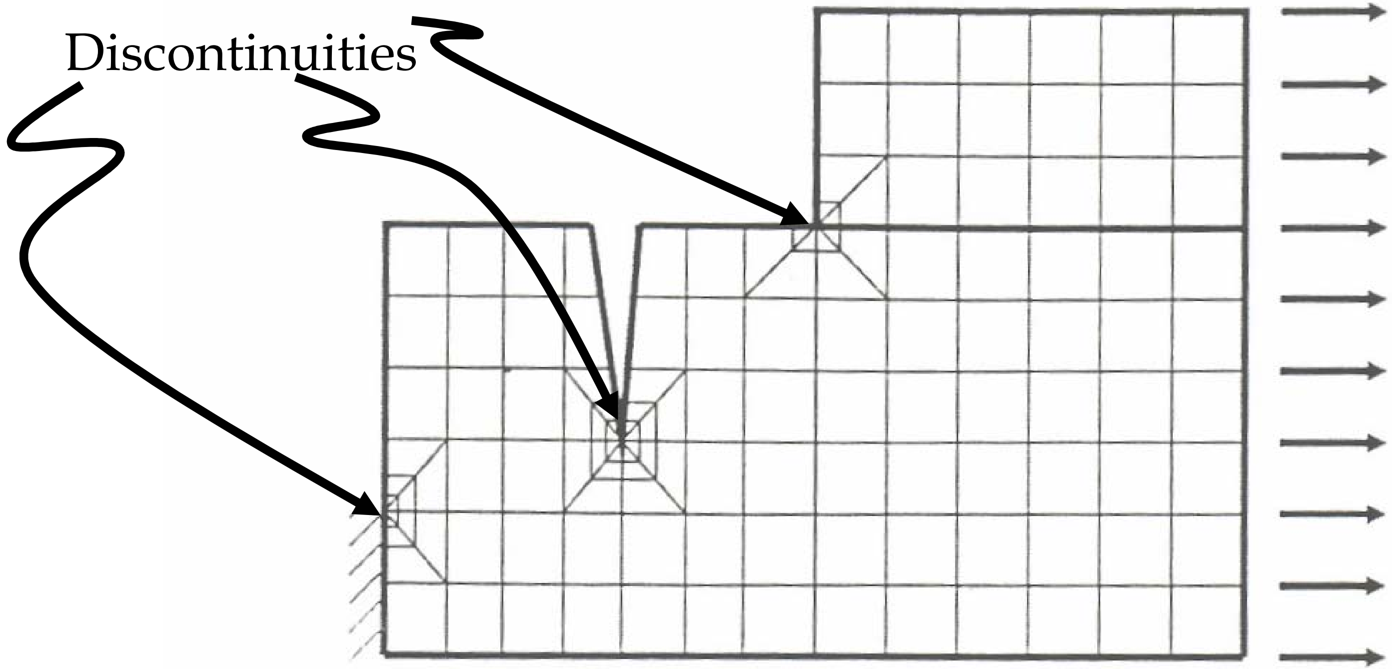
Worse



Better

# Mesh Refinement

- Finer meshing must be used in regions of expected high stress gradients (usually occur at discontinuities)



# Mesh Refinement (cont'd)

- Mesh refinement must be gradual with adjacent elements of not too dissimilar size
- Mesh refinement must balance accuracy with problem size
- ANSYS provides various tools for mesh refinement such as refinement at nodes, elements, lines, and volumes

# Dissimilar Element Types

- In general different types of elements with different DOF at their nodes should not share global DOF (for example do not use a 3D beam element in conjunction with plane stress elements)
- ANSYS allows certain classes of different element types to share nodes (e.g. spar and beam elements) but element and meshing guidelines must always be consulted before attempting to combine dissimilar element types

# Equilibrium and Compatibility

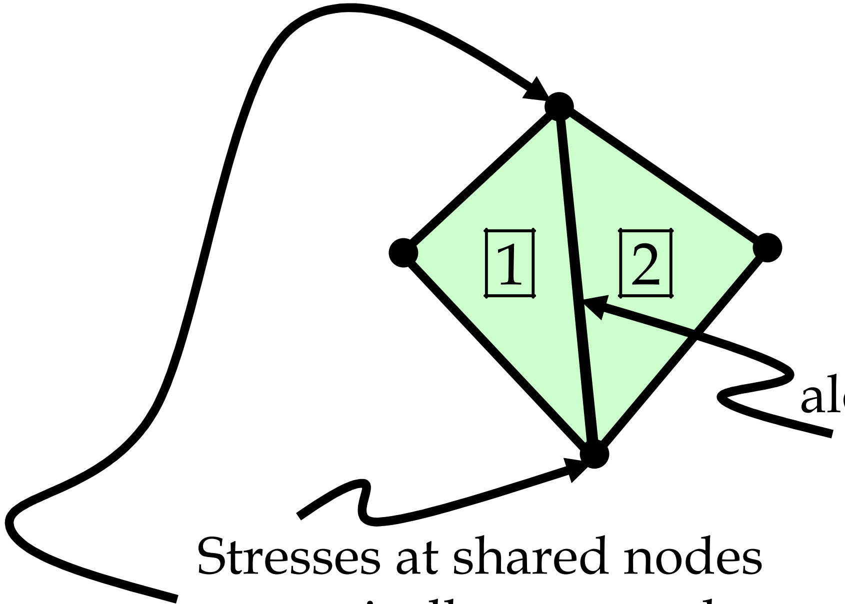
- The approximations and discretizations generated by the FE method enforce some equilibrium and compatibility conditions but not others
  - Equilibrium of nodal forces and moments is always satisfied because of

$$\mathbf{KU} = \mathbf{F}$$

  - Compatibility is guaranteed at the nodes because of the way **K** is formed; i.e. the displacements of shared nodes on two elements are the same in the global frame in which the elements are assembled

# Equilibrium-Compatibility (cont'd)

– Equilibrium is usually not satisfied across interelement boundaries; however discrepancies decline with mesh refinement




$$\sigma_x^{(1)} \neq \sigma_x^{(2)}$$

$$\sigma_y^{(1)} \neq \sigma_y^{(2)}$$

$$\tau_{xy}^{(1)} \neq \tau_{xy}^{(2)}$$

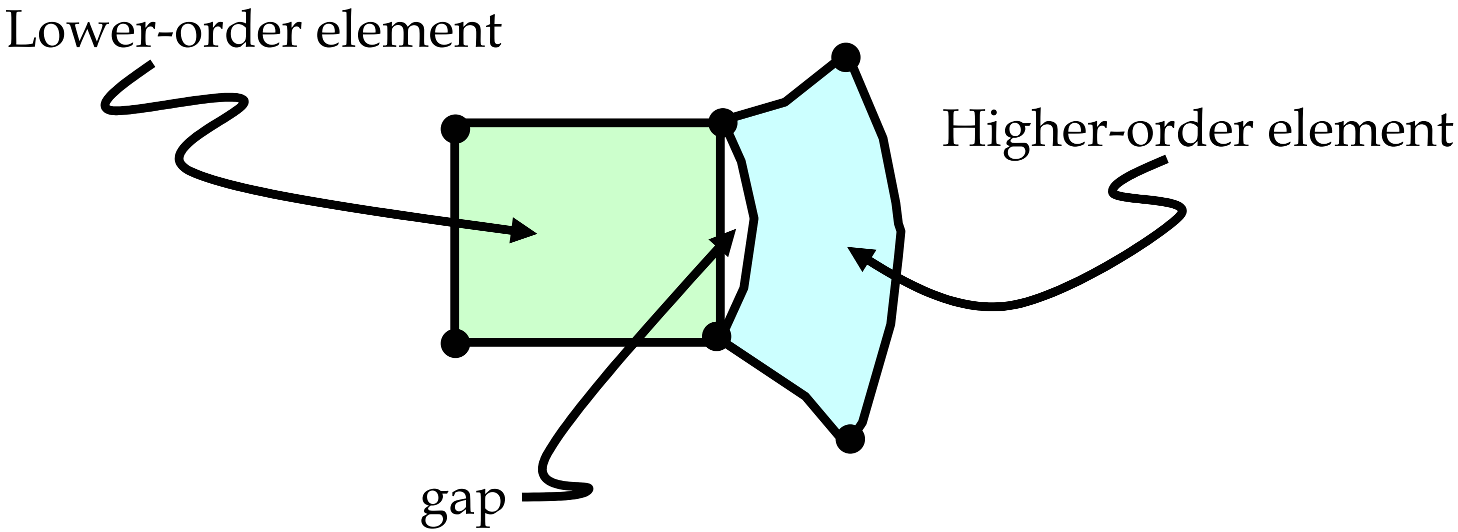
along this boundary

Stresses at shared nodes are typically averaged over the elements sharing the node

ANSYS uses stress disparities at nodes as a measure of discretization error

# Equilibrium-Compatibility (cont'd)

- Stresses are most reliable near the centers of elements and least reliable near their edges
- Compatibility may not be satisfied across interelement boundaries (happens with certain types of higher-order elements and junctures of dissimilar elements); incompatibilities decline with mesh refinement

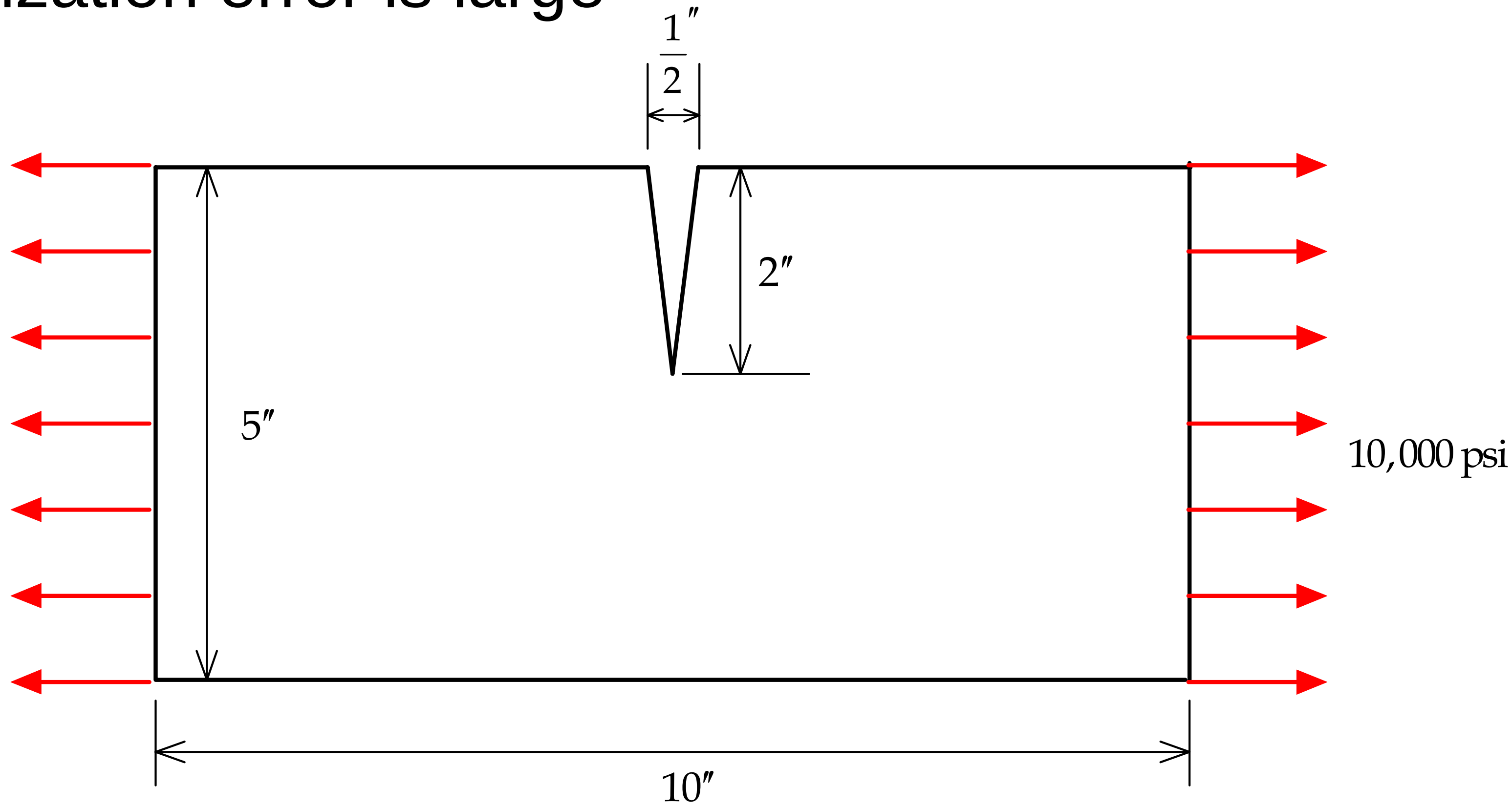


# Equilibrium-Compatibility (cont’d)

- Equilibrium is usually not satisfied within elements because **KU** = **F** does not enforce the relations produced by the partial differential equations that define equilibrium at infinitesimal levels; the assumed displacement functions that led to **KU** = **F** only satisfy <u>kinematic boundary conditions</u>, <u>not the differential equations themselves</u>
- Compatibility is satisfied within elements (guaranteed by the choice of continuous and single valued displacement functions)

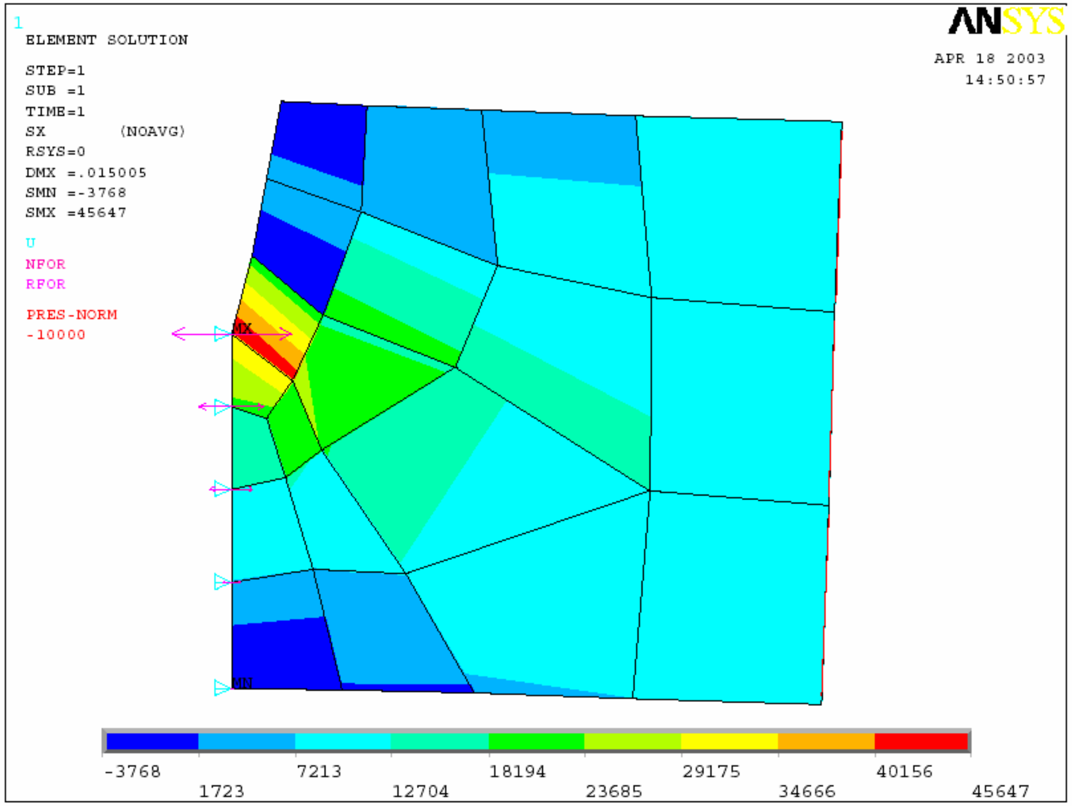
# Example: Plate with Crack

- Model the thin aluminum plate shown below using symmetry and refine mesh in regions where the discretization error is large



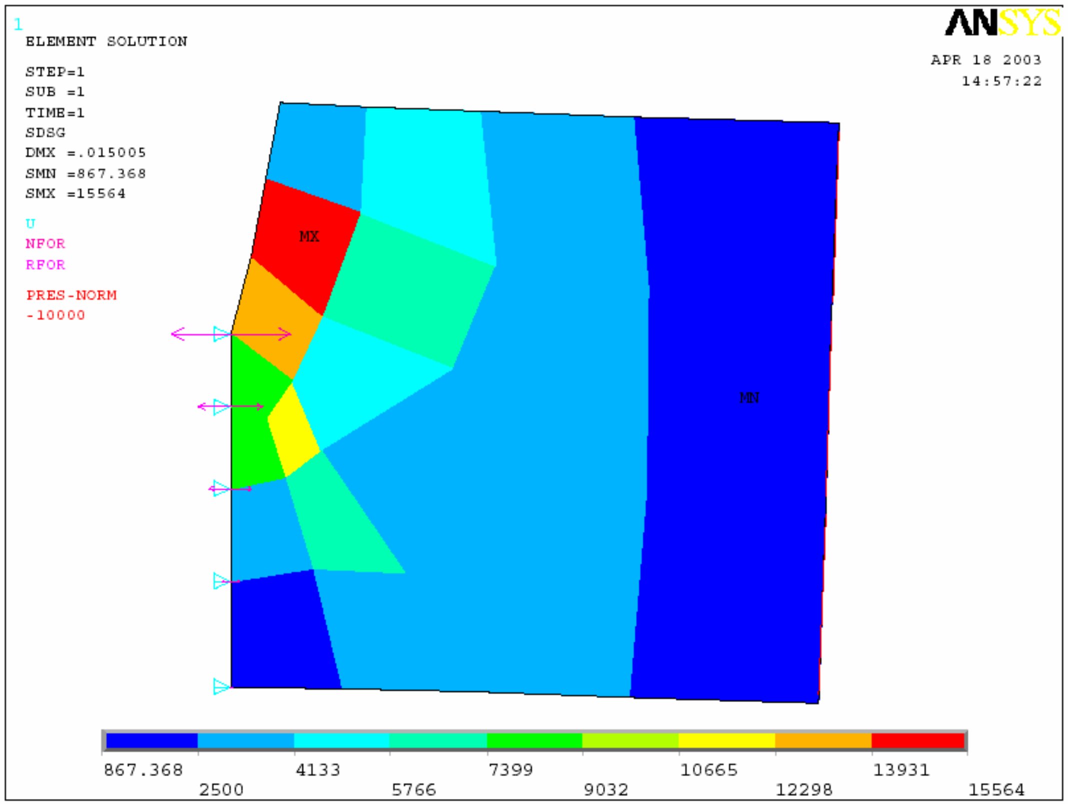
# Example (cont'd)

- Modeling only the right half of the plate, using PLANE42 elements and applying symmetry boundary conditions we obtain the following stress ( $\sigma_x$ ) distribution in ANSYS



# Example (cont'd)

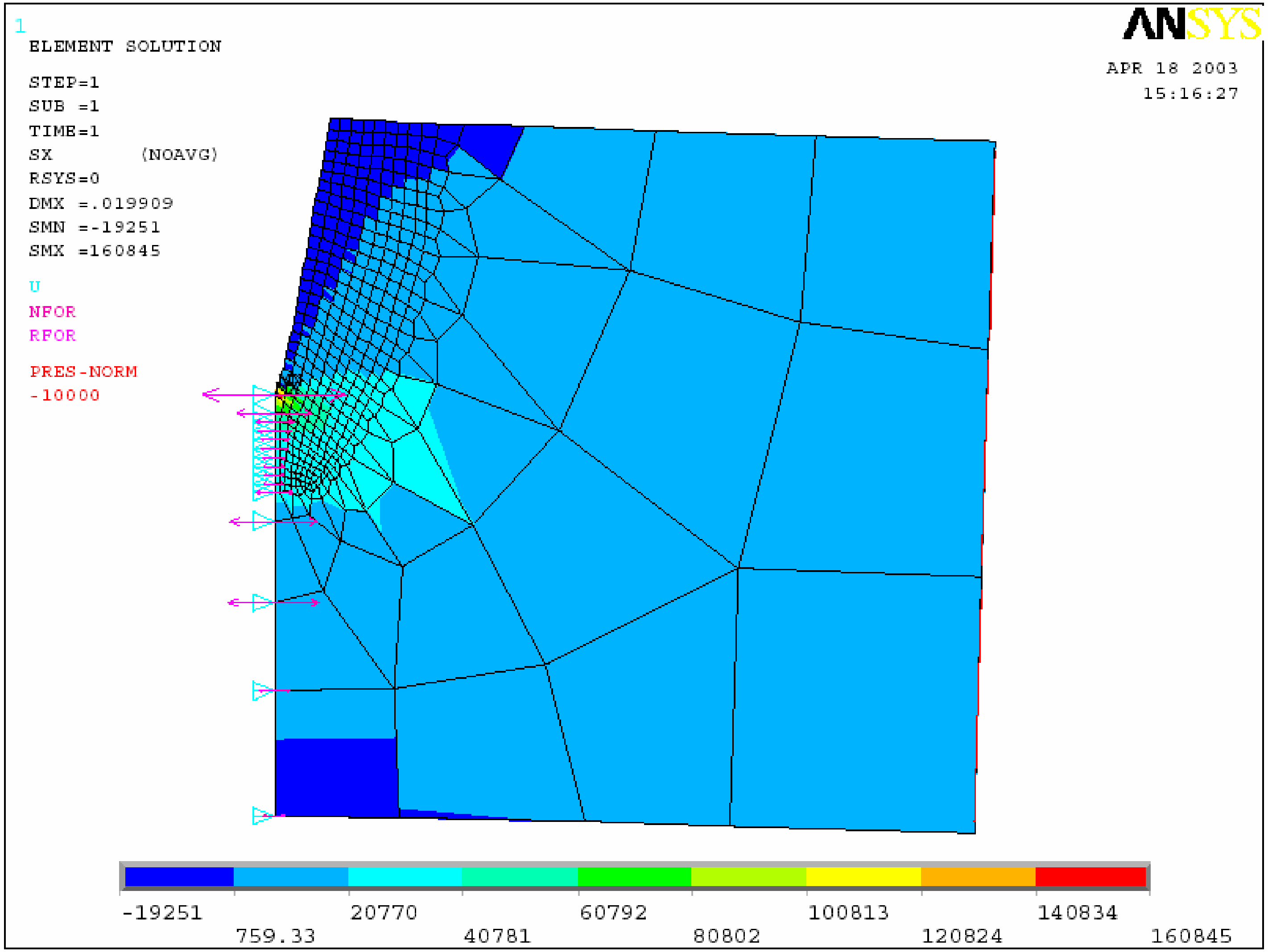
- To see the discretization error in ANSYS we plot the variable SDSG (from "Error Estimation" in "Element Solution")



# Example (cont'd)

- Clearly (and as expected) the worst error occurs around the crack meaning that the elements in that region need to be modified
- Contour plots of the same stress distribution and discretization error estimate are shown in the next page with a model that includes (Level 3) refined elements around the crack
- The refined models exhibit smoother distribution of stress with lower error estimates

# Example (cont'd)



# Example (cont'd)