General modeling guidelines

Some quotes from industry FEA experts:

"Finite element analysis is a very powerful tool with which to design products of superior quality. Like all tools, it can be used properly, or it can be misused. The keys to using this tool successfully are to understand the nature of the calculations that the computer is doing and to pay attention to the physics."

"Some engineers and managers look upon commercially available FEA programs as automated tools for design. In fact, nothing could be further from reality than that simplistic view of today's powerful programs. The engineer who plunges ahead, thinking that a few clicks of the left mouse button will solve all his problems, is certain to encounter some very nasty surprises."

"With the exception of a very few trivial cases, all finite element solutions are wrong, and they are likely to be more wrong than you think. One experienced analyst estimates that 80% of all finite element solutions are gravely wrong, because the engineers doing the analyses make serious modeling mistakes."

How to make sure that your analysis is reliable

Understand physics

Always assume your analysis is wrong until proven otherwise. Perform simple "back of the envelope" calcultions using principles learned in statics and deformable bodies.

Always check reaction forces. Do they match the applied loads? For models with both structural and gravity loads, turn off gravity and check your reaction forces. Do they match the applied load? Turn on gravity. Is the increase in reaction force consistent with the gravity load? Is the direction correct?

Study the deformed shape. Does it look correct? Displacement contours should be smooth. Is there separation or gaps in elements?

Know theory behind the elements

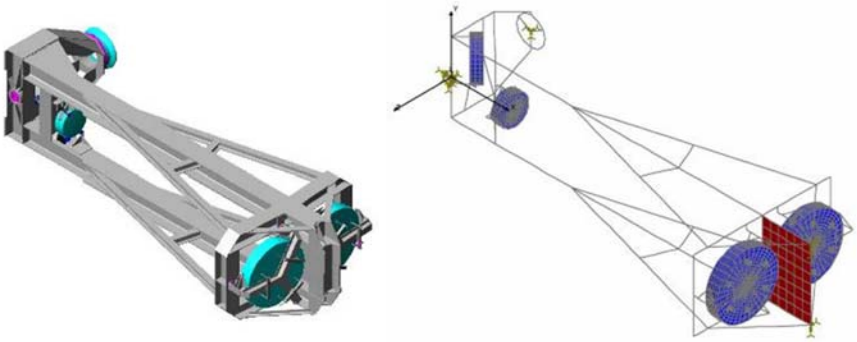
All finite element equations are derived using a large number of assumptions. Do not use an element unless you understand these assumptions. Read all relevant documentation first. For any new element that you have not used before first always solve a problem for which you know the analytical solution or have reliable finite element results available.

FEA results in general are NOT conservative

Most element formulations are overly stiff meaning deformations will be smaller than in the real structure. Thus finite results are NOT on the conservative side. As you increase the mesh density the deformations increase. Deformations converge rapidly. Typically a coarse model will yield good deformation results. Stresses converge slowly. In most cases, a fine mesh model is required to capture accurate stresses. Check stresses in surrounding elements sharing a common node/edge. Large differences in stresses across common boundaries typically indicate the mesh is too coarse.

CAD models are not same as FEA models

Model geometry is different from the physical geometry. Good FEA models DO NOT need the detail of CAD models. Decide which aspects of the geometry are important to the analysis. An accurate FEA models may have little or no resemblance to the CAD model.



Accurately modeling of load and boundary conditions is very difficult.

Complex joints are difficult to model. In FEA loads and boundary conditions are eventually transferred to nodes. Single node loads/constraints rarely represent reality.

Keep it simple

Start with a simple model and coarse mesh to “debug” your model.

Simple models have many advantages:

- Easy to develop and easy to change.
- Fast solution time.
- Easy to optimize or perform “What If” studies.

Complex models are time consuming to develop:

- Require precise modeling and care.
- Difficult to change.
- May require long run times.

Use consistent units

Understand the units and make sure all data is in consistent units.

Specific Modeling Tips

Make sure you have no unconnected nodes.

An element is a mathematical relation that defines how nodal unknowns (DOFs) are related to each other. Many element types can not be used together because their DOFs are incompatible.

Defining real constants requires thought and planning. Real constants define:

- Cross sectional areas, perimeter.
- Moment of inertia, torsional constant, shear factor, width, depth.
- Geometry constants, Neutral Axis offsets.

End release for beams.

Make a list and refer to your sketch.

Real constants must be defined prior to meshing.

Defining material properties requires thought and planning. Common material properties are:

Elastic modulus, Poisson's Ratio, Shear Modulus, Mass density.

Thermal conductivity, coefficient of thermal expansion, specific heat.

Pick a material from the library and check the units for understanding.

Make a materials list and refer to it.

Materials must be defined prior to meshing.

May define dummy materials.

Mesh density must be suitable for the analysis type and required accuracy.

Before meshing, activate appropriate material properties and real constants.

Define average element size or element density.

Knowing that the sides of the element remain straight ask yourself are there enough elements to accurately display the deformed shape?

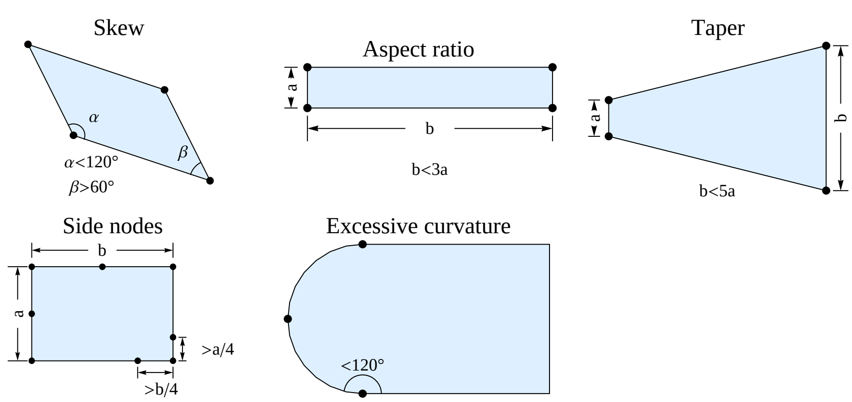
Stress gradients require a very fine mesh. An accurate stress analysis requires more elements than an accurate displacement analysis.

Verify accuracy by performing a convergence study.

Rules of Thumb:

4 elements minimum through the thickness (plane of interest).

Make sure all elements in your mesh are within the "element shape" guidelines illustrated in the following figure.



Defining loading, boundary conditions, and constraints.

Are all six (6) rigid body motions (3 translations and 3 rotations) accounted for? A model with too few constraints causes a singular stiffness matrix. An over constrained model creates alternate load paths. When in doubt, release constraints and add soft springs.

Loads or constraints? Ask if you can you use loads instead of constraints (or vice versa) to better simulate the physics of the problem.

Think carefully about the physics of the real constraints. Does it resist displacement and/or rotation?

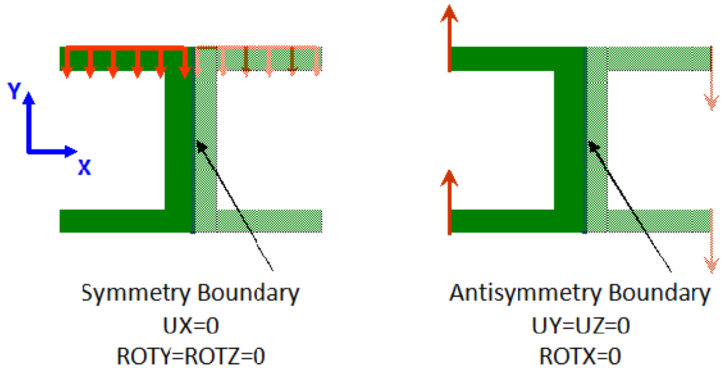
Is the gravity direction correct?

Use multiple load cases to better understand the performance and accuracy.

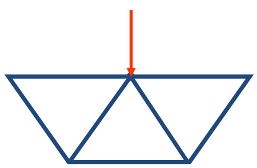
Displacement constraints are also used to enforce symmetry or antisymmetry boundary conditions.

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

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

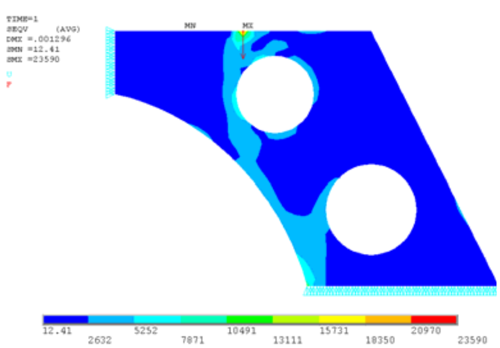


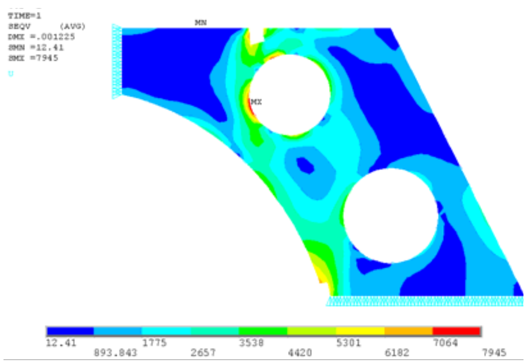
Point loads are appropriate for line element models such as beams, spars, and springs.



In solid and shell models, point loads usually cause a stress singularity, but are acceptable if you ignore stresses in their vicinity.

If point loads are used in a solid model make sure to look at results by ignoring the high stress values in the vicinity of the load. Note in the first figure the true stress concentration around a hole is masked by the artificial high stress due to a point load. The second figure, drawn by excluding the region around the point load, shows a much better picture of the actual stress distribution in the solid. The same comment applies to singularities at corners if they are not modeled with appropriate fillets.





Hierarchical Modeling - Summary and More Examples

Start simple and create higher fidelity models as necessary.

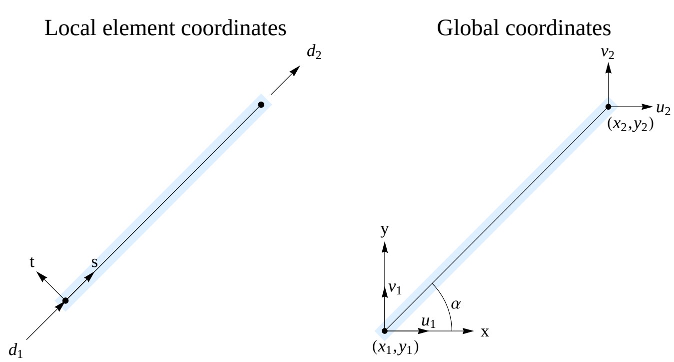
Use simple models to isolate regions that need further scrutiny.

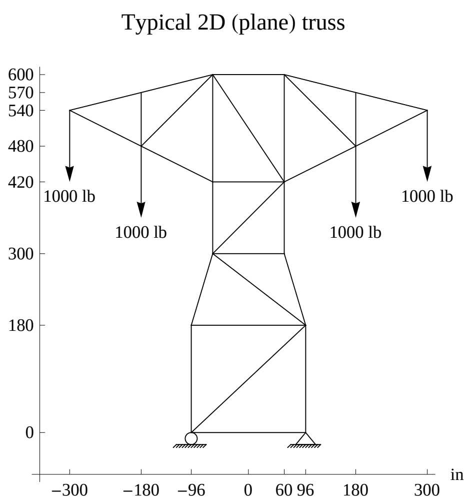
Use results from simple models to perform “sanity” checks for complex models.

Axial Deformation/Truss Model

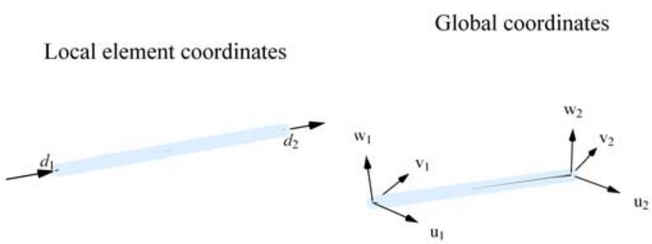
Long slender objects that are loaded in the axial direction only.

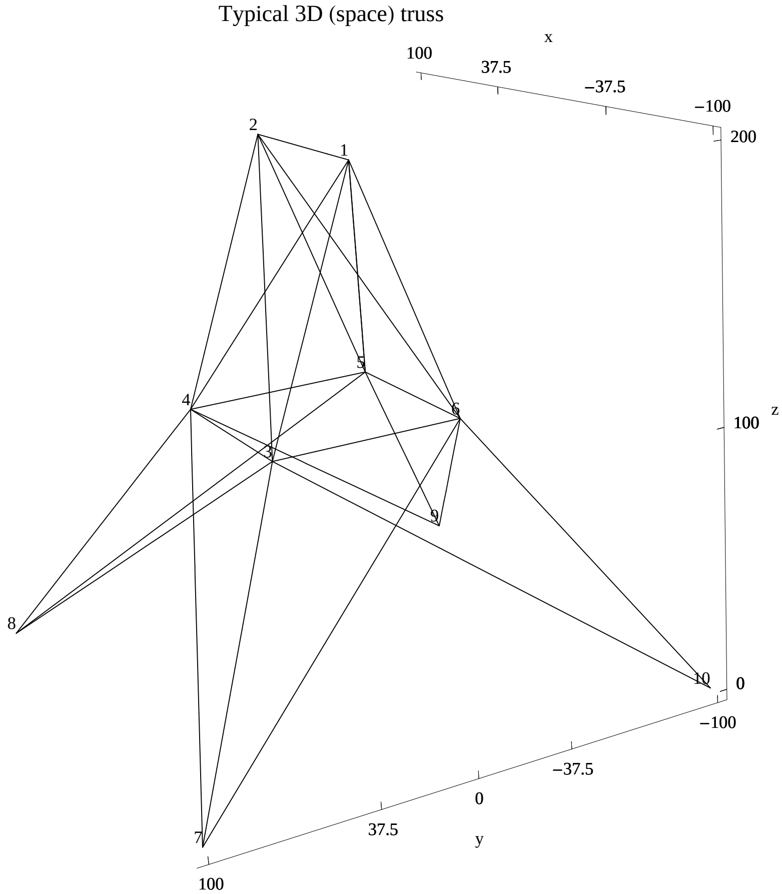
Use for modeling typical truss structures. Only axial deformations are allowed in the element. Cannot use this model if there are loads applied normal to the axis of elements.



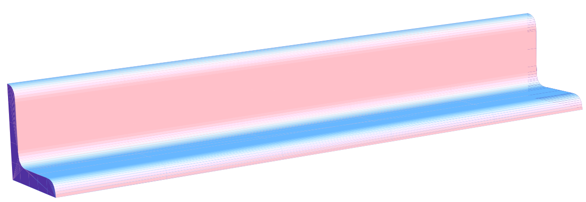


3D Truss Element





All emebers are structural steel angles $1\frac{1}{2}\,\text{in}\times1\frac{1}{2}\,\text{in}\times\frac{1}{4}\,\text{in}$



Nodes 7, 8, 9, and 10 are pinned to the ground. Concentrated loads are applied at nodes 1, 2, 3, and 6 as follows.

Node	F_x(lb)	F_y(lb)	F_z(lb)
1	1000	−10 000	−10 000
2	0	−10 000	−10 000
3	500	0	0
4	600	0	0

Ansys LINK180 element: Use for modeling both 2D and 3D trusses. When modeling 2D trusses with this element it is good practice to fix all nodes in the z directions (Select all nodes and set UZ = 0)

Beam/Frame Model

Long slender bodies that are loaded in ANY direction.

Length of the solid should be at least 5 times the cross section dimensions for the model to give good results.

σ_x is the only nonzero stress component. The stress strain law is simple Hook's law

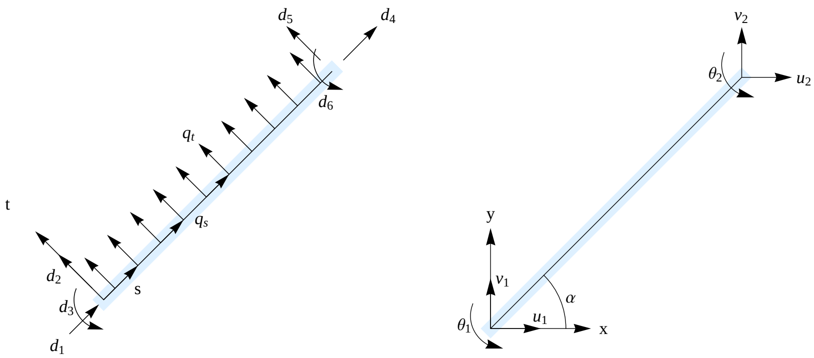
$$\sigma_x = E\,\epsilon_x$$

Shear stress may also be present. However since there is no corresponding differential equation it is not considered a primary unknown.

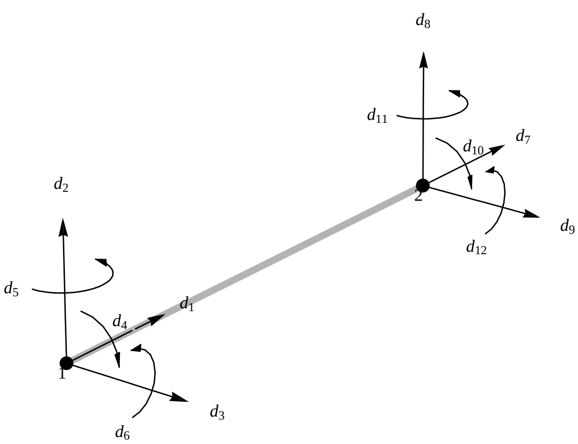
Applied distributed loads must be in the units of force per unit length.

Frame Element

Local element coordinates — Global coordinates

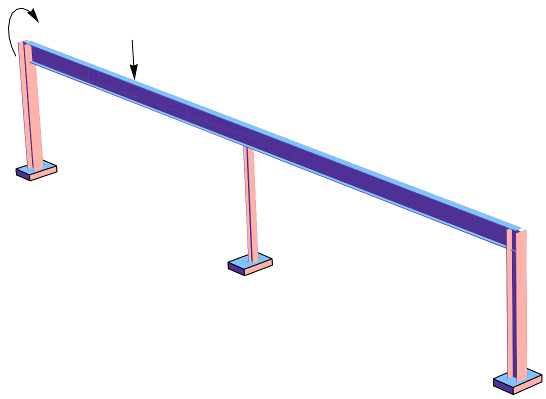


Three dimensional frame element



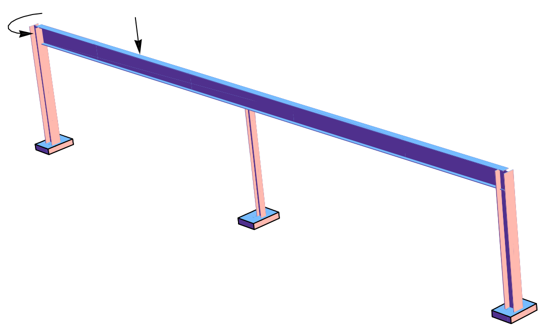
Ansys BEAM188 element: Use for modeling both 2D and 3D frames. The element is based on a generalized version beam theory that is typically discussed in deformable bodies courses. It can include shear deformations and warping of cross section under shear and torsional loading. There are several options provided in the element that can reduce it to the one based on the simple beam theory. Unless you have studied the advanced beam theory set these options to use the simple beam theory.

2D (plane) Frame

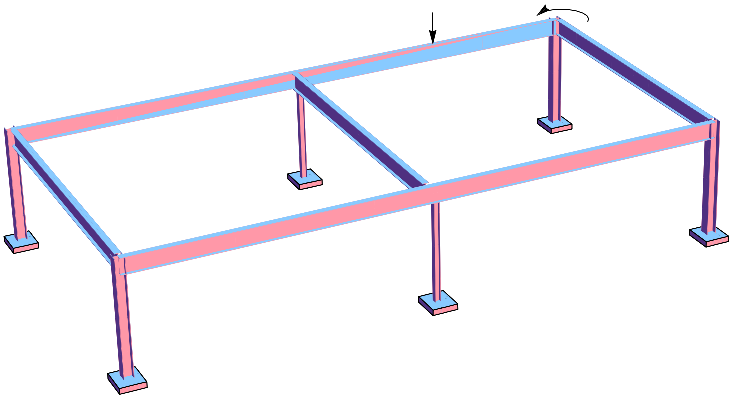


3D (space) Frame

(Because the applied moment will cause twisting
and out-of-plane bending)



3D (space) Frame



Plane Stress Models

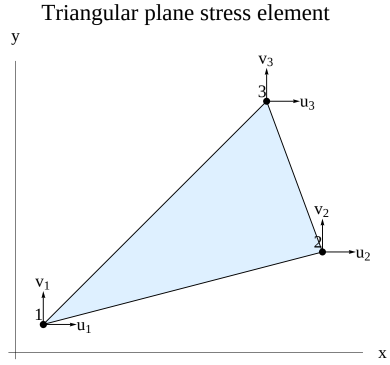
Components in which the z dimension (thickness h) is much smaller than the dimensions in the x, y plane L_x and L_y, say

$$h < \approx \frac{L_x}{10} \text{ and } \frac{L_y}{10}$$

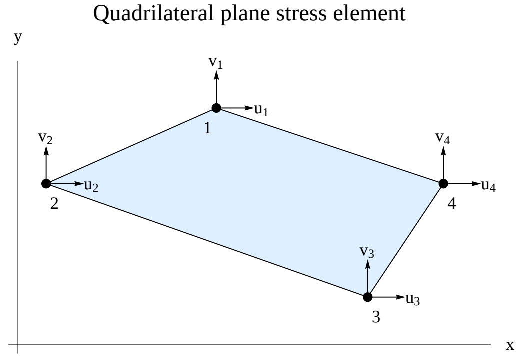
Applied loading is only in the $x - y$ plane.

Assumed zero stresses: $\{\sigma_z, \tau_{yz}, \tau_{zx}\}$

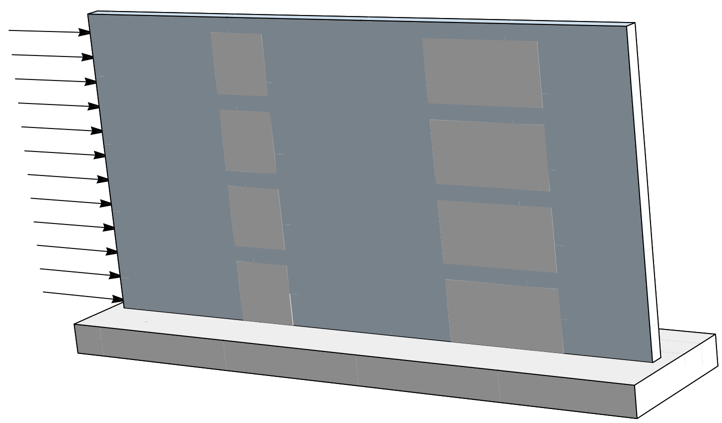
Triangular plane stress element



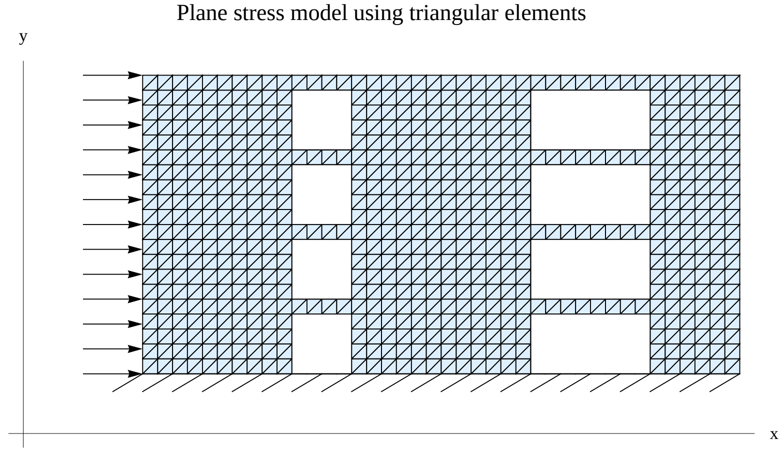
Quadrilateral plane stress element



Shear wall with openings subjected to in−plane loading



Plane stress model using triangular elements



Plane Strain Model

Consider a solid in which the z dimension is much larger than the dimensions in the *x*, *y* plane L_x and

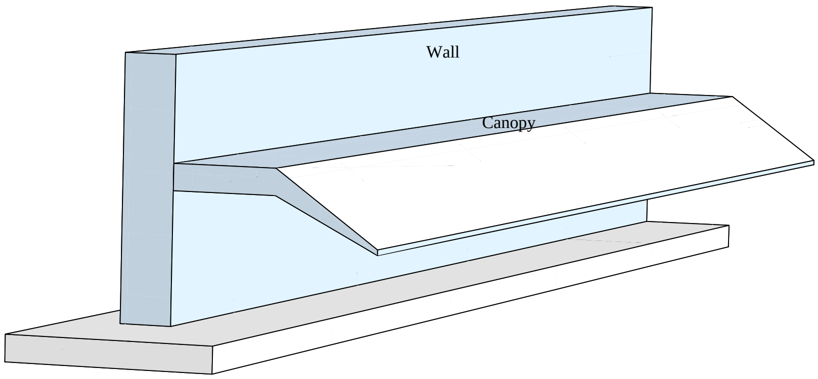
L_y, say

$L_z > \approx 10\,L_x$ and $10\,L_y$

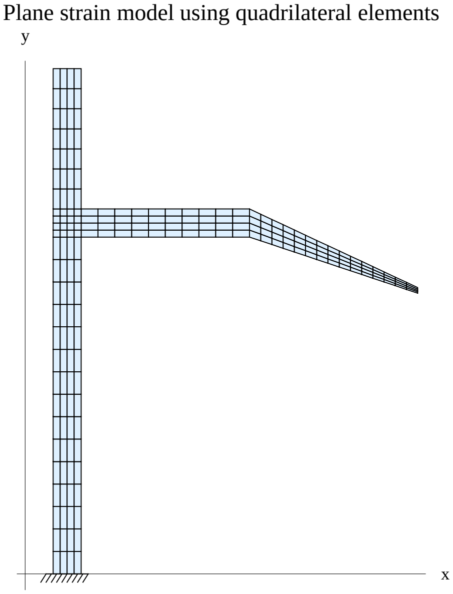
Furthermore assume that any loading or boundary conditions are applied only in the $x-y$ plane and are uniform along the z-axis. In this case it is reasonable to assume that the strains on the plane normal to z – axis are zero.

Assumed zero strains: $\{\epsilon_z, \gamma_{yz}, \gamma_{zx}\}$

Long canopy projecting from a wall
(Length much longer than shown)

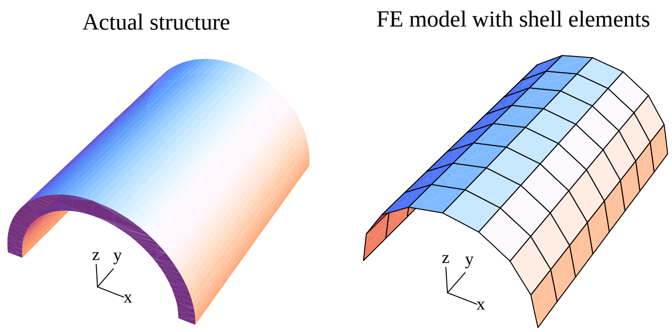


Plane strain model using quadrilateral elements



Plate/Shell Model

Plates are flat structures with thickness much smaller than the other two dimensions. Thus plates are same as “plane stress” structures. The difference is that the plates can support loads normal to the plane. Shell structures are essentially plates with non-flat geometry. Loading can be applied in any direction to shell elements.



Usual Shell Element Nodal Degrees of Freedom

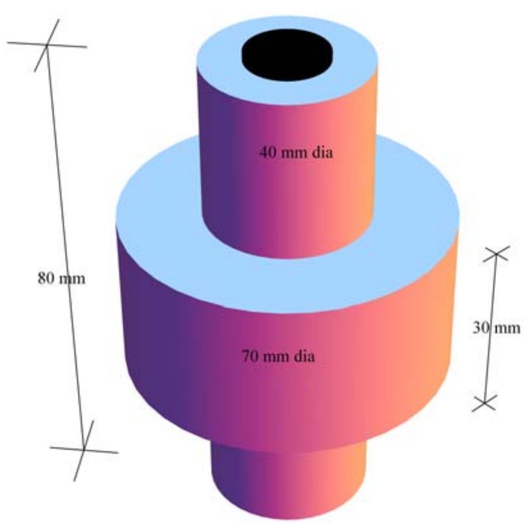
u, v, w = Displacements along three coordinate directions

θ_x, θ_y, θ_z = Rotations about three coordinate directions

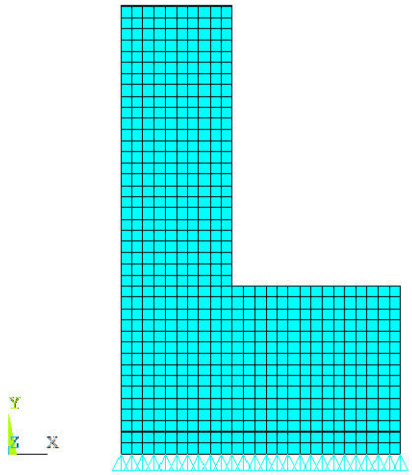
Shell thickness is entered as a section property (real constant).

Axisymmetric Model - Solids of Revolution

Three dimensional solids that can be generated by revolving a plane figure about an axis.

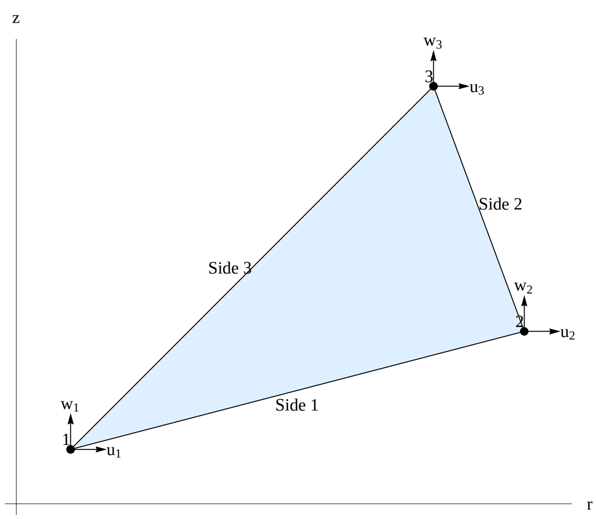


691 node and 625 element model



Important note: For plane stress/plane strain analysis the global $x - y$ axes can be placed at any arbitrary point in the plane. This is not the case for the axisymmetric analysis. The z-axis is the axis of axial symmetry and the coordinates in the finite element model must be described with respect to its location in the solid.

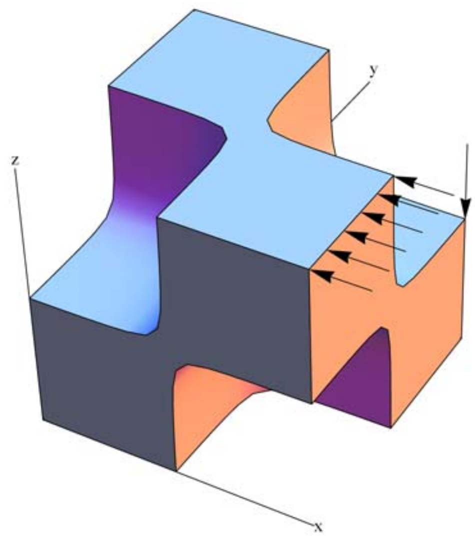
Axisymmetric analysis employs triangular and quadrilateral elements, similar to the plane stress/strain elements. The two nodal degrees of freedom have the interpretation of radial displacement in the r direction and axial displacement in the z direction. However Ansys, like most other general purpose finite element programs, still designates them as simply x and y displacements (u, v).



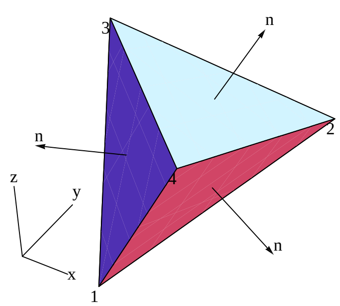
In Ansys use Structural solid PLANE182 element with the Axisymmetric option.

Three Dimensional Solid Model

Use for irregular solids



Four node tetrahedral element



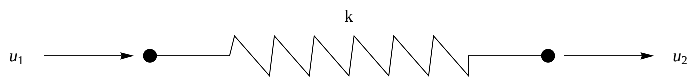
Three degrees of freedom per node.

In Ansys use SOLID185 element.

Spring, Gap, and Contact Elements

Useful for modeling supports and joints.

Axial Spring Element



$$k_s \begin{pmatrix} 1 & -1 \\ -1 & 1 \end{pmatrix} \begin{pmatrix} u_1 \\ u_2 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \end{pmatrix}$$

k_s = Spring constant

Spring force = $k_s(u_2 - u_1)$

Rotational/Torsional Spring Element

$$k_T \begin{pmatrix} 1 & -1 \\ -1 & 1 \end{pmatrix} \begin{pmatrix} \theta_1 \\ \theta_2 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \end{pmatrix}$$

k_T = Rotational/torsional spring constant

The torque (twisting moment) in the spring

$$T = k_T(\theta_2 - \theta_1)$$

Ansys elements

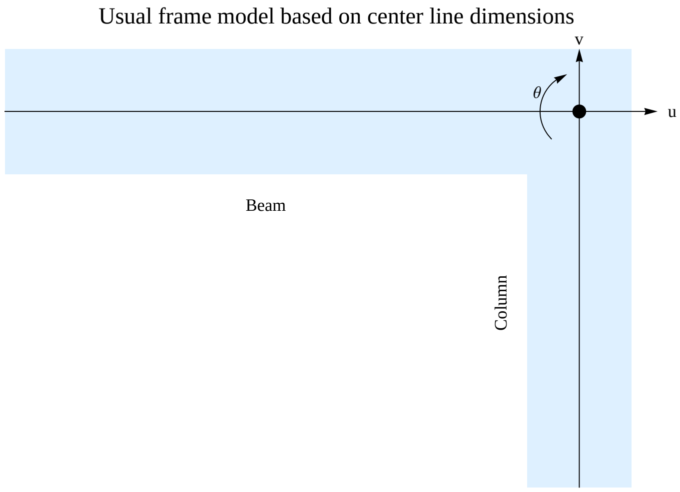
COMBIN14 - Combination Spring-Damper used for both 2D and 3D models. Behavies essentially as a truss (LINK1) element except that it needs spring consatnt and not EA values. Length of the element has no influence on the results.

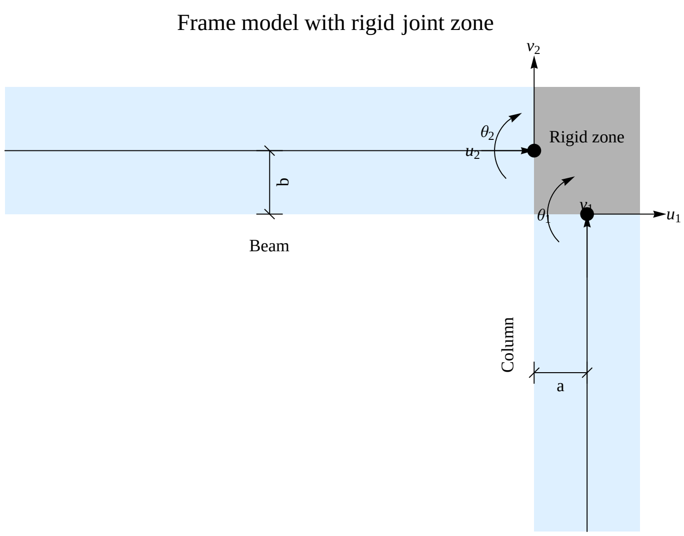
CONTAC elements for node-to-surface and surface-to-surface contact

Constraints Between Degrees of Freedom

Many practical situations require constraints between degrees of freedom.

Rigid-Zone at Beam-Column Connections





The following constraints are defined between the degrees of freedom at these nodes to create the effect of the rigid joint zone.

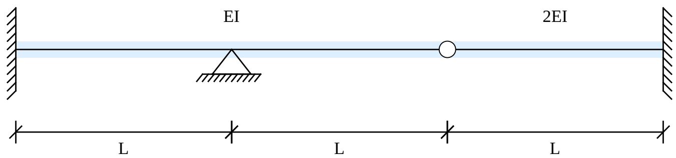
$$\theta_1 = \theta_2$$

$$u_1 = u_2 + b\,\theta_2$$

$$v_1 = v_2 + a\,\theta_2$$

Internal hinges

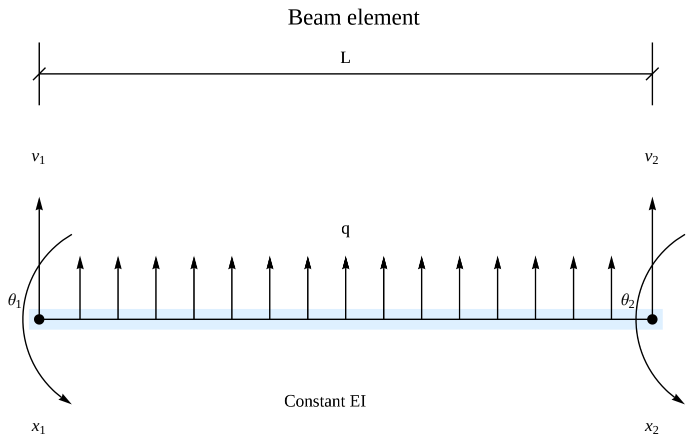
Create the simplest possible model using beam elements.



Solution

Constraints Using Lagrange Multipliers

Beam Element - Simplified version of frame element. No axial deformations. Thus only two degrees of freedom per node. All elements are aligned with the horizontal axis. Thus no need for transformation from local to global.



$$\frac{\mathrm{EI}}{L^3}\begin{pmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{pmatrix}\begin{pmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{pmatrix} = \frac{qL}{2}\begin{pmatrix} 1 \\ \frac{L}{6} \\ 1 \\ -\frac{L}{6} \end{pmatrix}$$

Class Activity

Analyze the beam if the right support settles downwards by 10 mm. Use $L = 3000$ mm

For the left portion up to the hinge: Rectangular section with $b = 100$ mm and $h = 476.22$ mm

For the right portion after the hinge: Rectangular section with $b = 100$ mm and $h = 600$ mm

$E = 200\,000\, N/\mathrm{mm}^2$

$\mathrm{EI} = 200\,000 \times (1/12)\,(107.583)\,(464.758)^3 = 180 \times 10^{12}\, N.\,\mathrm{mm}^2 = 180 \times 10^6\, \mathrm{MN}\cdot\mathrm{mm}^2.$

Solution

Element 1: $$\begin{pmatrix} \frac{2}{25} & 120 & -\frac{2}{25} & 120 \\ 120 & 240000 & -120 & 120000 \\ -\frac{2}{25} & -120 & \frac{2}{25} & -120 \\ 120 & 120000 & -120 & 240000 \end{pmatrix} \begin{pmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

Element 2: $$\begin{pmatrix} \frac{2}{25} & 120 & -\frac{2}{25} & 120 \\ 120 & 240000 & -120 & 120000 \\ -\frac{2}{25} & -120 & \frac{2}{25} & -120 \\ 120 & 120000 & -120 & 240000 \end{pmatrix} \begin{pmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

Element 3: $$\begin{pmatrix} \frac{4}{25} & 240 & -\frac{4}{25} & 240 \\ 240 & 480000 & -240 & 240000 \\ -\frac{4}{25} & -240 & \frac{4}{25} & -240 \\ 240 & 240000 & -240 & 480000 \end{pmatrix} \begin{pmatrix} v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

Assembling the elements equations in the usual way the global equations are as follows.

$$\begin{pmatrix} \frac{2}{25} & 120 & -\frac{2}{25} & 120 & 0 & 0 & 0 & 0 & 0 & 0 \\ 120 & 240000 & -120 & 120000 & 0 & 0 & 0 & 0 & 0 & 0 \\ -\frac{2}{25} & -120 & \frac{4}{25} & 0 & -\frac{2}{25} & 120 & 0 & 0 & 0 & 0 \\ 120 & 120000 & 0 & 480000 & -120 & 120000 & 0 & 0 & 0 & 0 \\ 0 & 0 & -\frac{2}{25} & -120 & \frac{2}{25} & -120 & 0 & 0 & 0 & 0 \\ 0 & 0 & 120 & 120000 & -120 & 240000 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & \frac{4}{25} & 240 & -\frac{4}{25} & 240 \\ 0 & 0 & 0 & 0 & 0 & 0 & 240 & 480000 & -240 & 240000 \\ 0 & 0 & 0 & 0 & 0 & 0 & -\frac{4}{25} & -240 & \frac{4}{25} & -240 \\ 0 & 0 & 0 & 0 & 0 & 0 & 240 & 240000 & -240 & 480000 \end{pmatrix} \begin{pmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

The known boundary conditions are $v_1 = \theta_1 = v_2 = 0$, $v_5 = -10$ mm, and $\theta_5 = 0$. The zero boundary conditions are incorporated by simply removing corresponding rows and columns to get

$$\begin{pmatrix} 480000 & -120 & 120000 & 0 & 0 & 0 \\ -120 & \frac{2}{25} & -120 & 0 & 0 & 0 \\ 120000 & -120 & 240000 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{4}{25} & 240 & -\frac{4}{25} \\ 0 & 0 & 0 & 240 & 480000 & -240 \\ 0 & 0 & 0 & -\frac{4}{25} & -240 & \frac{4}{25} \end{pmatrix} \begin{pmatrix} \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \\ v_5 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

Setting $v_5 = -10$ means we remove last row, and move -10 times the last column to the right-hand side. Thus the final system of equations is as follows.

$$\begin{pmatrix} 480000 & -120 & 120000 & 0 & 0 \\ -120 & \frac{2}{25} & -120 & 0 & 0 \\ 120000 & -120 & 240000 & 0 & 0 \\ 0 & 0 & 0 & \frac{4}{25} & 240 \\ 0 & 0 & 0 & 240 & 480000 \end{pmatrix} \begin{pmatrix} \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ -\frac{8}{5} \\ -2400 \end{pmatrix}$$

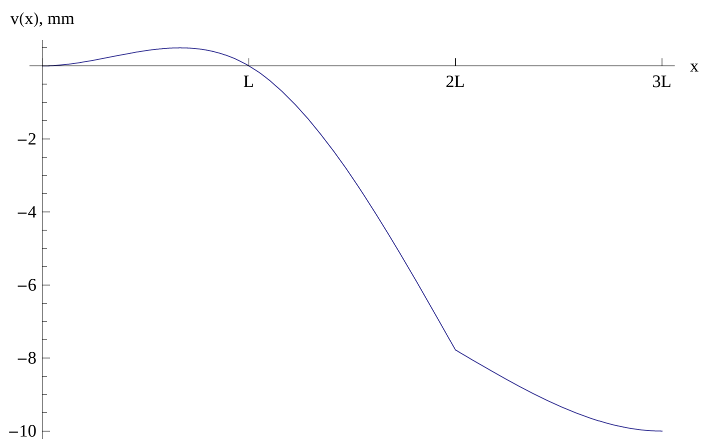
The solution for nodal unknowns can now be obtained, by using the Lagrange multiplier method to impose the multipoint constraint, as follows.

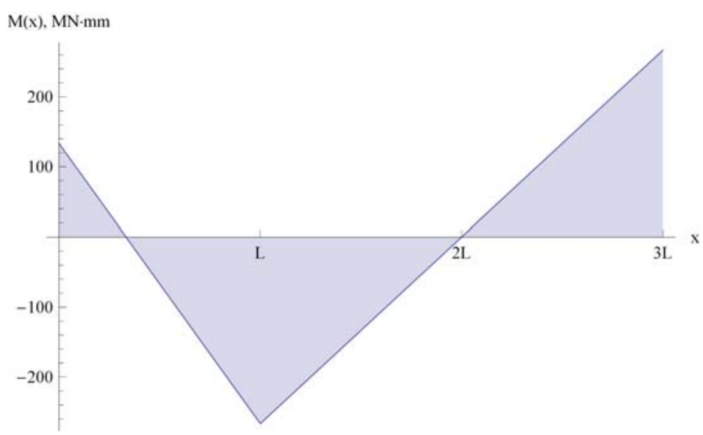
Augmented system of equations

$$\begin{pmatrix} 480000 & -120 & 120000 & 0 & 0 & 0 \\ -120 & \frac{2}{25} & -120 & 0 & 0 & 1 \\ 120000 & -120 & 240000 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{4}{25} & 240 & -1 \\ 0 & 0 & 0 & 240 & 480000 & 0 \\ 0 & 1 & 0 & -1 & 0 & 0 \end{pmatrix} \begin{pmatrix} \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \\ \lambda \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ -\frac{8}{5} \\ -2400 \\ 0 \end{pmatrix}$$

Solution

$$\left\{\theta_2 = -\frac{1}{900},\ v_3 = -\frac{70}{9},\ \theta_3 = -\frac{1}{300},\ v_4 = -\frac{70}{9},\ \theta_4 = -\frac{1}{900},\ \lambda = \frac{4}{45}\right\}$$





Solution

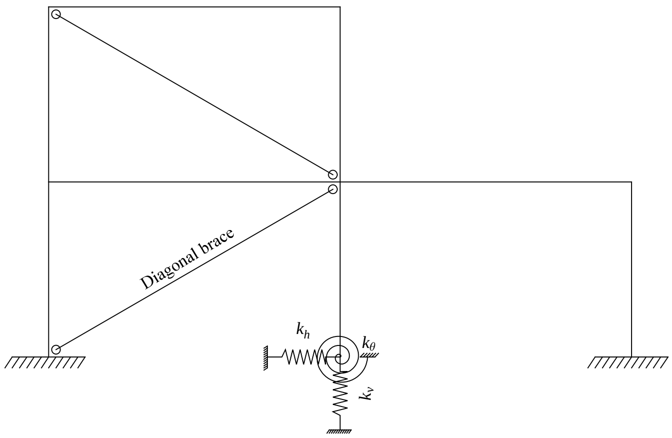
Solution

Class Activity: Ansys solution

Models with more than one element type

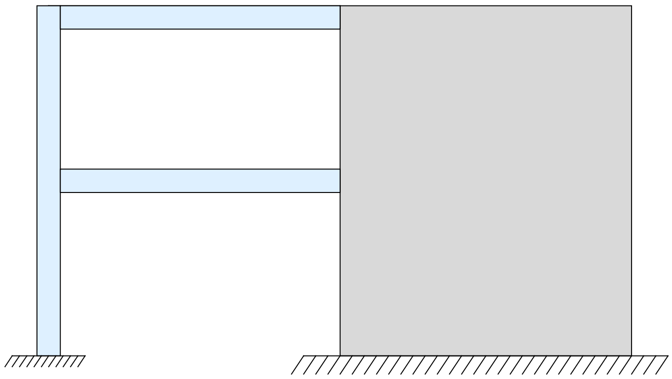
It is possible to create finite element models using different types of elements for different parts. It is common to represent flexible supports by using springs instread of rigid supports. A braced frame can be created by using plane frame elements for the frame members and plane truss elements, or springs, for the bracing members.

Braced frame with flexible middle support

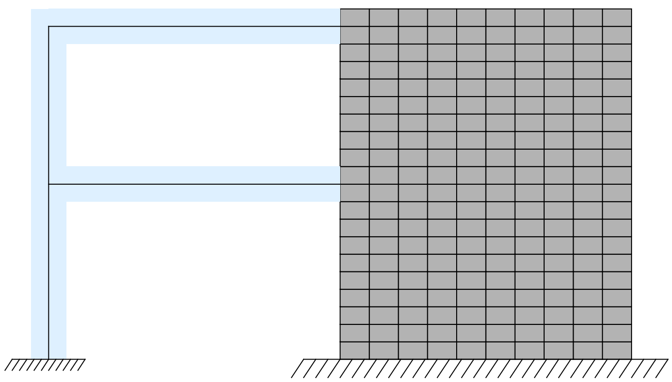


At the nodes where different element types meet it is important to be carefully consider the effect of different degrees of freedom of different elements. This is particularly important when combining elements that have rotational degrees of freedom wth those that only have the translational degrees of freedom. For example when combining a plane stress element with a plane frame element, one must pay attention as to how the rotation of the frame element will be translated to the displacements of the plane stress element. For these situations it is generally necessary to define appropriate relationships (constraints) between dgerees of freedom at the interface nodes.

Frame–shear wall system

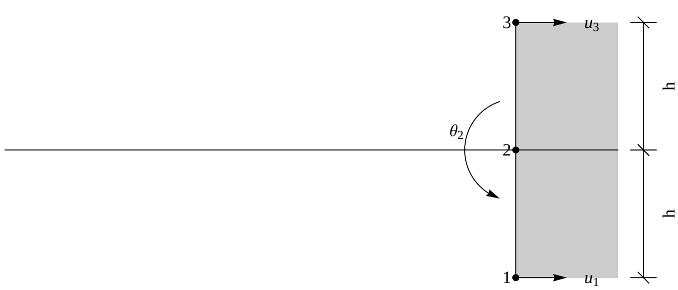


Finite element model with frame and plane stress elements



One possibility of defining an appropriate constraint between the rotational degree of freedom of the frame element and the displacements of the plane stress element is to use the plane sections assumption from the strength of materials and define the constraints based on the following illustration.

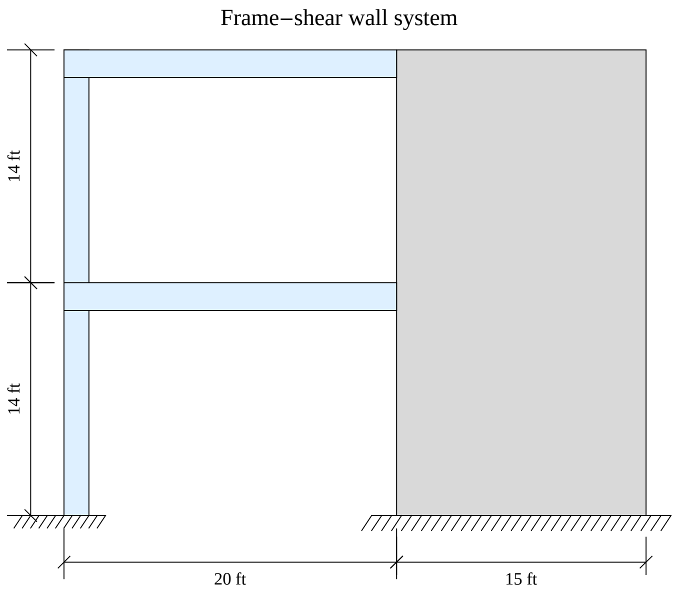
Interface constraints

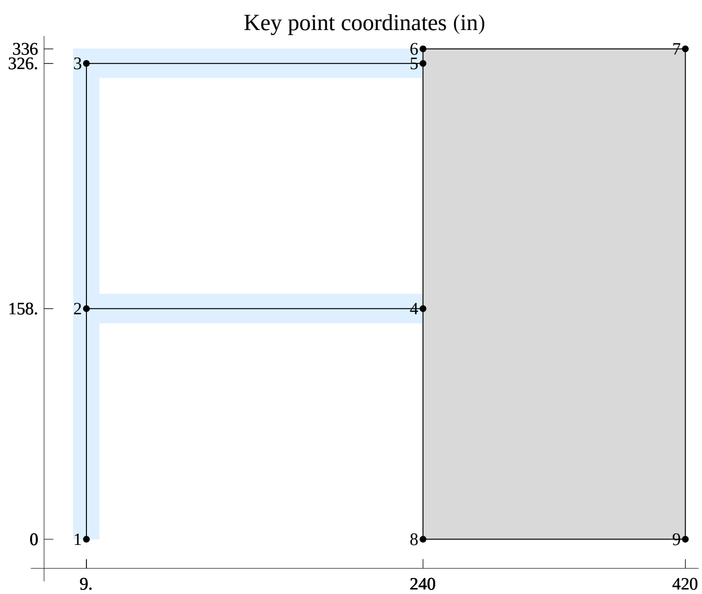


Constraints: $u_3 = -h\,\theta_2$ and $u_1 = h\,\theta_2$

Class Activity

Analyze the following concrete frame-shear wall system. All members are 10 inch thick (out of the plane). The column is 18 in wide and the beams are 20 in deep. $E = 4000$ ksi. Poisson's ratio = 0.20. Unit weight of concrete = 150 lb$/$ft^3. In addition to the self weight of the members, the right side of the wall is subjected to a uniform wind pressure of 600 lb/ft. The beams carry a distributed gravity load of 2000 lb/ft. Use frame element for the beams and column and plane stress element for shear wall. Employ appropriate constraints at the junction between frame and plane stress elements.





Use lb and inch units.

Add the two elements. Type 1: Beam: 3D finite strain 2 node 188. Set option K2 = Rigid classic, K3 to Cubic form, and K6 to at element nodes. Type 2: Solid: Quad 4node 182. Set option K3 to plane stress with thickness.

For the plane stress element use real constants command to define a thickness of 10 in.

Define a linear elastic material with E = 4,000,000 psi, ν = 0.20, and density = 150/(12×12×12) $\mathrm{lb}/\mathrm{in}^3$

Using section command create two rectangular sections. The column section ID 1: 10×18 in and the beam section ID 2: 10×20 in.

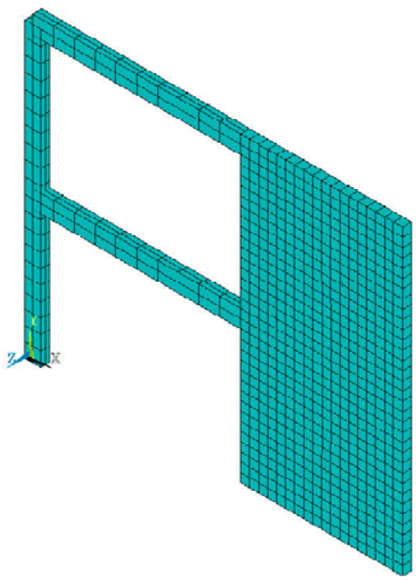
Using modeling create keypoints, lines, and the area for the shear wall.

Using mesh attributes pick column lines and define material as 1, element type as 1 (beam188), section as 1, and check the orientation key point box. Enter the keypoint 8 as the orientation key point. The real constant set number does not apply to this element. Pick beam lines and define material as 1, element type as 1, section as 2, and check the orientation key point box. Enter the keypoint 8 as the orientation key point. Pick shear wall area and set real constant to 1, material as 1, element type to 2 (plane 182). The section option is not relevant to this element.

Using size control pick column and beam lines and define number of element divisions to 10 to divide each line into 10 equal length elements. Pick shear wall area and set element edge length to 10 in.

Create mesh over beam and column lines and over the shear wall area.

Using PlotCtrls> Style> Size & Shape turn on display of elements based on section and real constants. Create a three dimensional view of the model to make sure that your model looks correct.



Set all degrees of freedom to 0 at the bottom keypoint of the column and the bottom line of the shear wall.

On the right side line of the shear wall apply a pressure (acting towards left) of 600/12/10 psi. Ansys wants pressure in psi for plane stress elements. Hence the division by 10 in (wall thickness).

Use Loads> Define Loads> Apply> Structural> Inertia> Gravity> Global. Enter 1 for acceleration in the global y direction to simulate loading due to self weight of members.

Use Loads> Define Loads> Apply> Structural> Pressure> On Beams, select all beam elements and apply a load of 2000/12 lb/in. Keep the load key as 1 (this specifies normal load).

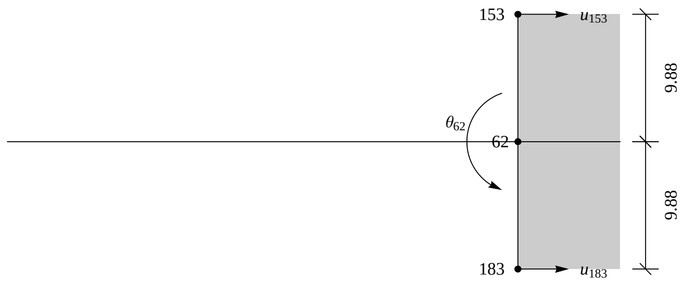
Zoom-in into the regions where the beams are connected to the shear wall. Determine the three node numbers at each interface and their coordinates (Use List> Nodes to get a table of nodal coordinates). In my model the numbers and the corresponding constraint equatons are as follows.

```
NODE X Y Z THXY THYZ THZX
62 240.00 158.00 0.0000 0.00 0.00 0.00
153 240.00 167.88 0.0000 0.00 0.00 0.00
183 240.00 148.12 0.0000 0.00 0.00 0.00
```

Difference in y coordinates

Between nodes 153 and 62 $= 167.88 - 158 = 9.88$

Between nodes 183 and 62 $= 158 - 148.12 = 9.88$



Constraints: $u_{153} = -9.88\,\theta_{62}$ and $u_{183} = 9.88\,\theta_{62}$

```
NODE X Y Z THXY THYZ THZX
42 240.00 326.00 0.0000 0.00 0.00 0.00
135 240.00 336.00 0.0000 0.00 0.00 0.00
168 240.00 316.12 0.0000 0.00 0.00 0.00
```

Difference in y coordinates

Between nodes 135 and 42 $= 336 - 326 = 10$

Between nodes 168 and 42 $= 326 - 316.12 = 9.88$

Constraints: $u_{135} = -10\,\theta_{42}$ and $u_{168} = 9.88\,\theta_{22}$

Use Coupling/Ceqn> Constraint Eqn to define constraint equations

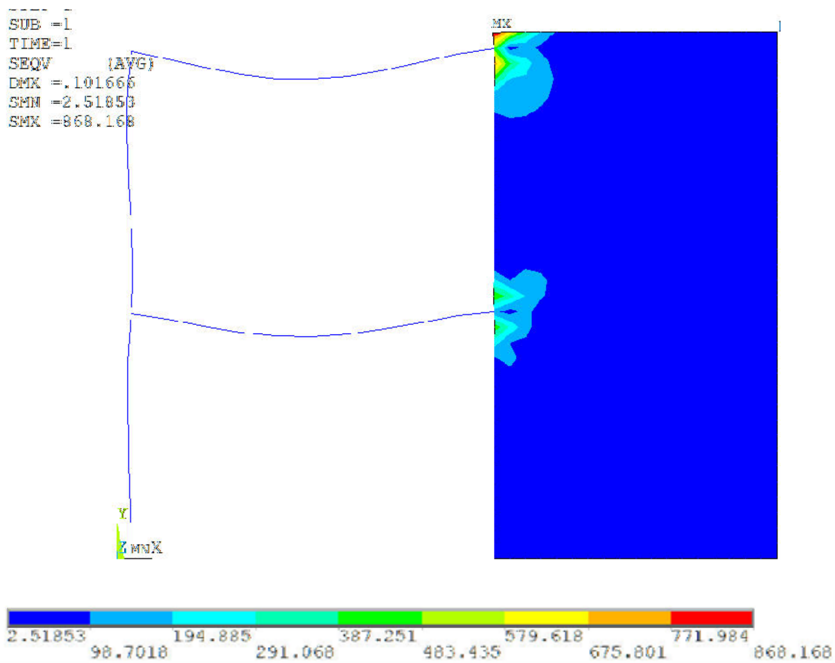
Constraint equation 1: $u_{153} + 9.88\,\theta_{62} = 0$. In the constant term enter 0 (right hand side of the equation). For the first term in the equation enter node number 153, label as UX, and coefficient as 1. For the second term in the equation enter node number 62, label as ROTZ, and coefficient as 9.88. Leave the third term blank.

Constraint equation 2: $u_{183} - 9.88\,\theta_{62} = 0$. In the constant term enter 0 (right hand side of the equation). For the first term in the equation enter node number 183, label as UX, and coefficient as 1. For the second term in the equation enter node number 62, label as ROTZ, and coefficient as -9.88. Leave the third term blank.

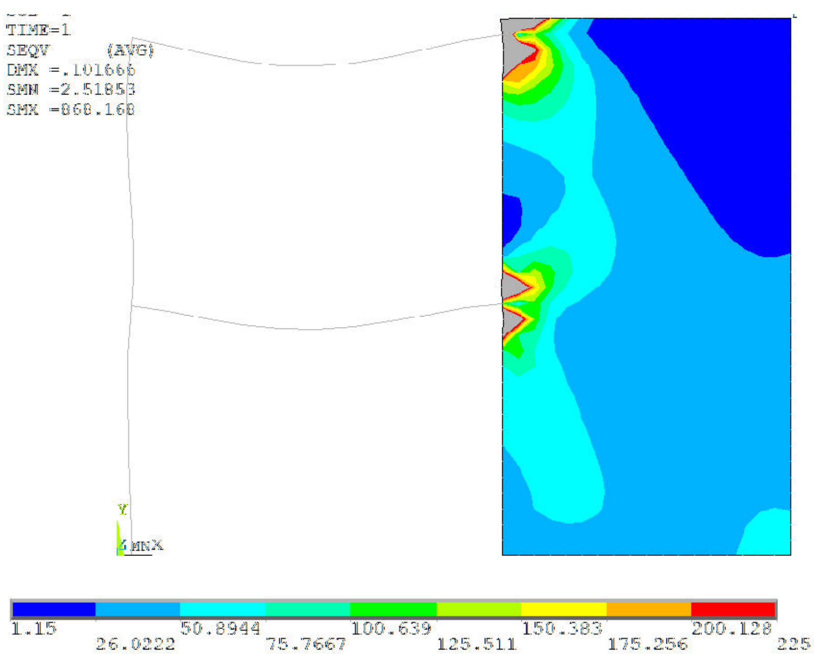
Constraint equation 3: $u_{135} + 10\,\theta_{42} = 0$

Constraint equation 4: $u_{168} - 9.88\,\theta_{42} = 0$

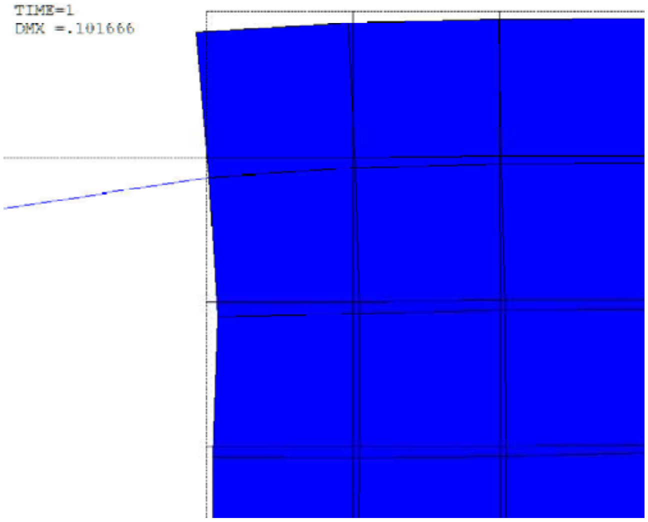
von Mises stress plot



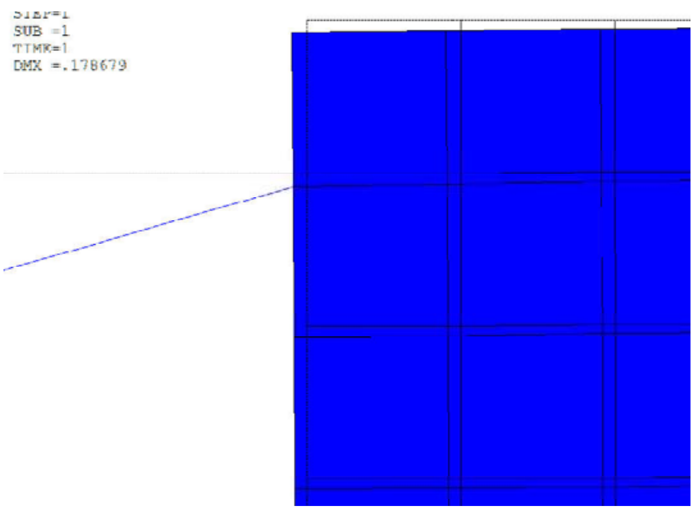
von Mises stress plot showing contours in the range from1.15 psi - 225 psi.



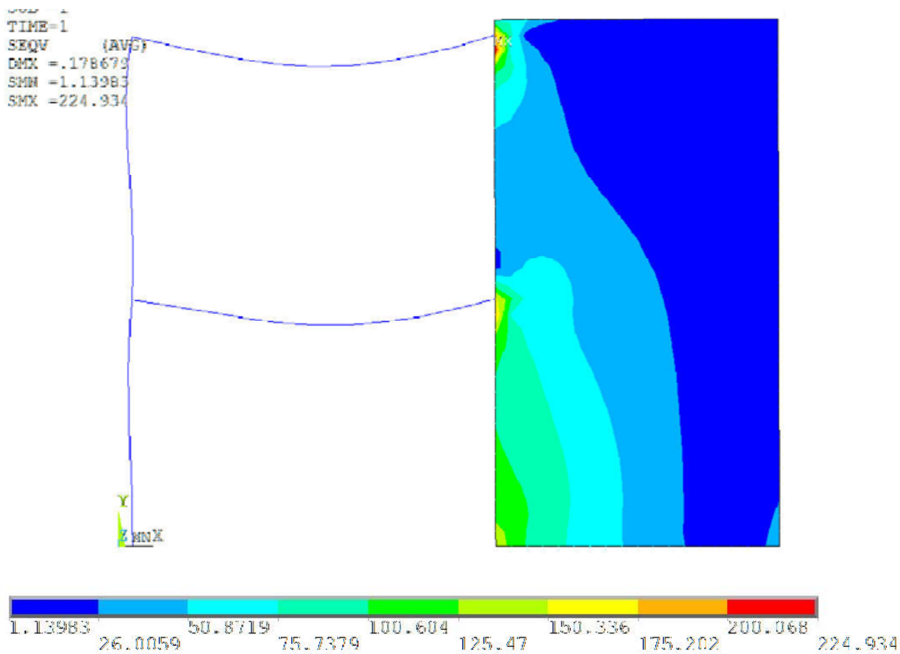
Plot of deformed shape. The maximum vertical displacement of 0.10165 in occurs near the middle of the upper story beam.



Plot of deformed shape without the constraint equations. The maximum vertical displacement of 0.17855 in occurs near the middle of the upper story beam.



Plot of von Mises stress without the constraint equations


TIME=1
SEQV (AVG)
DMX =.178679
SMN =1.13983
SMX =224.934
Y
Z MN X
1.13983
26.0059
50.8719
75.7379
100.604
125.47
150.336
175.202
200.068
224.934

Solution