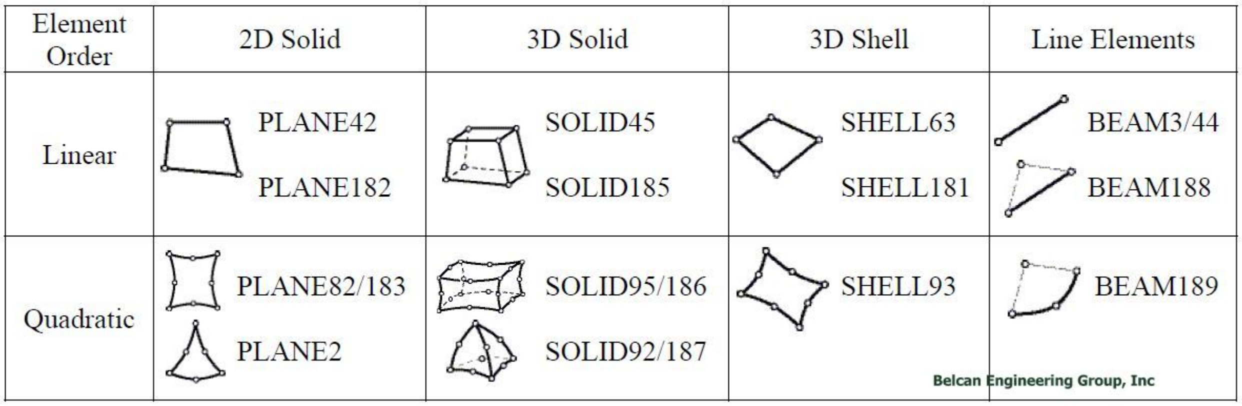
ME 304 Finite Element Analysis
Basic types of FEA Elements

See Section 1.3.3 in the FEA textbook for description of various elements:



Element Order	2D Solid	3D Solid	3D Shell	Line Elements
Linear	PLANE42 PLANE182	SOLID45 SOLID185	SHELL63 SHELL181	BEAM3/44 BEAM188
Quadratic	PLANE82/183 PLANE2	SOLID95/186 SOLID92/187	SHELL93	BEAM189

Belcan Engineering Group, Inc

Elements with nodes only on the ends or corners are linear elements (for example PLAN182) and the shape functions are linear (for shape function, see "page 155 in: https://faculty.up.edu/lulay/ME304/BeamElements.pdf). Elements with mid-side nodes (for example PLANE183) are called quadratic elements. The shape functions of quadratic elements are non-linear. The results produced by quadratic elements are generally more precise than equivalent linear elements if the strain is non-linear.

In the vernacular, 3D elements such as SOLID185 and SOLID186 are referred to as "brick" elements and 3D elements with 6 edges (SOLID92/187) are referred to as "tets" (tetrahedron).

It might seem confusing as to why Line Elements and 2D Solid elements exist since everything in the real world is 3D. However, each element has its purpose for existing – there are good reasons for selecting each type. Selecting appropriate elements is an important decision for FEA modelers – poor choices can lead to erroneous conclusions and can result in excessive computation time.

For example, one of the important feature of 2D and 3D elements is their aspect ratio (length of the longest side to the shortest side). In many cases, inaccuracy of the numerical result increases with increased aspect ratio (D. Logan – A First Course in FEM, Thomson publishing). Let's say you are wanting to model a long slender plate that is 2 inches wide, ¼ inch thick and 100 inches long. An aspect ratio of 1 would require 800 brick elements (¼ by ¼ by ¼). With a linear brick element (SOLID185) this would result in about 7200 nodes (2x9x400). Assuming the loading is axial or bending, a single simple line element (BEAM188) would suffice; that would be only 2 nodes and result in an equally good answer. Remember, that the stiffness matrix is $(DOF)^2$. If each node has 6DOF, then essentially the 3D model is $(6*7200)^2/(6*2)^2 = 13X10^6$ larger than the 2-node problem. …and remember how much more difficult it is to find the inverse of a 4X4 than a 2X2 matrix!? Even to the computer, (6*7200)X(6*7200) is a big deal!

It should also be noted that it is possible for a single FEA model to use a combination of elements.

Let's take a look at a few common ANSYS elements.

BEAM188 Element Description (from ANSYS Help)
BEAM188 is suitable for analyzing slender to moderately stubby/thick beam structures. The element is based on Timoshenko beam theory which includes shear-deformation effects. The element provides options for unrestrained warping and restrained warping of cross-sections.

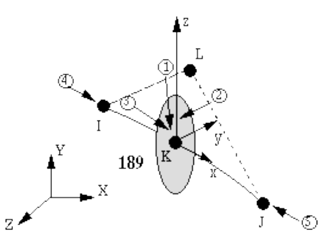
The element is a linear, quadratic, or cubic two-node beam element in 3-D. BEAM188 has six or seven degrees of freedom at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. A seventh degree of freedom (warping magnitude) is optional. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

The element includes stress stiffness terms, by default, in any analysis with large deflection. The provided stress-stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling, or collapse studies with arc length methods or nonlinear stabilization).

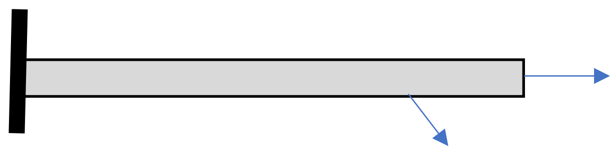
Elasticity, plasticity, creep and other nonlinear material models are supported. A cross-section associated with this element type can be a built-up section referencing more than one material. Added mass, hydrodynamic added mass and loading, and buoyant loading are available.

BEAM189 (used in HW6), Element Description (from ANSYS Help):
The BEAM189 element is suitable for analyzing slender to moderately stubby/thick beam structures. The element is based on Timoshenko beam theory which includes shear-deformation effects. The element provides options for unrestrained warping and restrained warping of cross-sections.

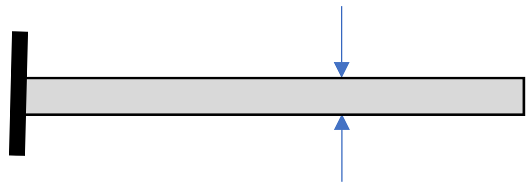
The element is a quadratic three-node beam element in 3-D. With default settings, six degrees of freedom occur at each node; these include translations in the x, y, and z directions and rotations about the x, y, and z directions. An optional seventh degree of freedom (warping magnitude) is available. The element is well-suited for linear, large rotation, and/or large-strain nonlinear applications.



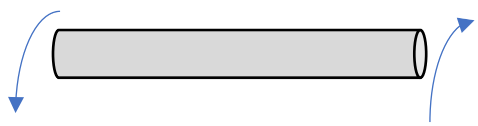
Lulay's notes on line body elements:
BEAM189 is a line body element, but can determine stresses in 3D due to bending and axial loads, such as shown below – but BEAM189 is NOT a 3D element! It will NOT determine the loads caused at the point of contact – it will only determine the bending and axial loads that the forces produce (similar to the analysis conducted in Strength of Materials using My/I and F/A):



BEAM189 cannot determine stresses through the thickness other than those based on bending (My/I) and axial (F/A) loads. For example, BEAM189 will NOT determine stresses based on loading such as:



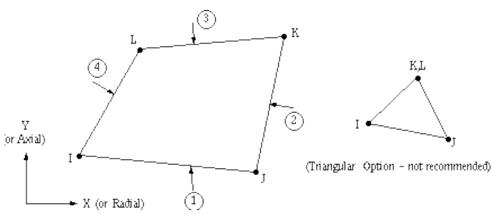
Nor can BEAM189 analyze torsion or "twist":



Line elements are useful when 1 dimension is much larger than the other 2 dimensions.

PLANE182 Element Description (from ANSYS Help):

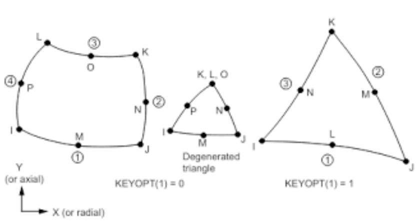
PLANE182 is used to model 2-D solid structures. It can be used as either a plane element (plane stress, plane strain or generalized plane strain) or an axisymmetric element with or without torsion. In most cases, the element is defined by four nodes with two degrees of freedom at each node: translations in the nodal x and y directions. For the axisymmetric option with torsion, it is still defined by four nodes, but with three degrees of freedom at each node: translations in the nodal x and y directions, and rotation in the nodal y direction. The element has plasticity, hyperelasticity, stress stiffening, large deflection, and large strain capabilities. It has a mixed-formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. (Lulay's note: "hyperelasticity" refers to non-linear elastic behavior. Rubber is a hyperelastic material – stress and strain are NOT linearly related, but the material returns to its original length when unloaded).



PLANE183 Element Description (from ANSYS Help):

PLANE183 is a higher order 2-D, 8-node or 6-node element. PLANE183 has quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems).

This element is defined by eight nodes or six nodes. It can be used as a plane element (plane stress, plane strain and generalized plane strain) or as an axisymmetric element (with or without torsion). In most cases, the element has two degrees of freedom at each node: translations in the nodal x and y directions. For the axisymmetric with torsion option, however, the element has three degrees of freedom at each node: translations in the nodal x and y directions and rotation in the nodal y direction.



Lulay's notes on 2D elements: 2D elements are useful for modeling thin plates (for example propane tanks, sheet metal, etc.) where 1 dimension is much less than the other 2 dimensions, or for loading conditions where it is known that the through-thickness stresses are zero. There are different types for different purposes (eg. "membrane" or "plate" elements) having different DOF.

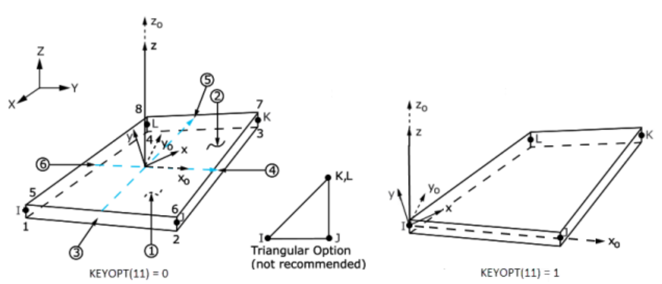
“3D” SHELL Elements, SHELL181 Element Description

SHELL181 is suitable for analyzing thin to moderately-thick shell structures. It is a four-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.

SHELL181 is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.

SHELL181 can be used for layered applications for modeling composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first-order shear-deformation theory (usually referred to as Mindlin-Reissner shell theory).

The element formulation is based on logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time



Lulay’s comments: ANSYS lists shell elements as being 3-dimensional, but they are actually 2-dimensional. They provide 3-dimensional results much in the same way that beam elements (which are line elements…1-dimensional in a sense) provide 3-dimensional results. Solid elements (discussed next) are truly 3-dimensional.

SOLID185 Element Description (from ANSYS Help):

SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

SOLID185 is available in two forms:

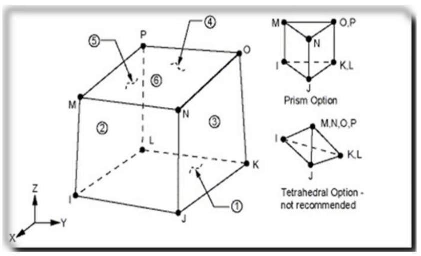
- Homogeneous Structural Solid (KEYOPT(3) = 0, the default) -- See *"SOLID185 Homogeneous Structural Solid Element Description"*.
- Layered Structural Solid (KEYOPT(3) = 1) -- See *"SOLID185 Layered Structural Solid Element Description"*.

See SOLID185 for more details about this element.

A higher-order version of the SOLID185 element is SOLID186.

SOLID185 Homogeneous Structural Solid Element Description

SOLID185 Structural Solid is suitable for modeling general 3-D solid structures. It allows for prism, tetrahedral, and pyramid degenerations when used in irregular regions. Various element technologies such as B-bar, uniformly reduced integration, and enhanced strains are supported.



SOLID186 Element Description (from ANSYS Help):

SOLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

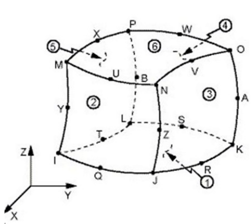
SOLID186 is available in two forms:

- Homogeneous Structural Solid (KEYOPT(3) = 0, the default) -- See *"SOLID186 Homogeneous Structural Solid Element Description"*.
- Layered Structural Solid (KEYOPT(3) = 1) -- See *"SOLID186 Layered Structural Solid Element Description"*.

SOLID186 Homogeneous Structural Solid Element Description

SOLID186 Homogeneous Structural Solid is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems). The element may have any spatial orientation.

Various printout options are available. See SOLID186 for more details.



(Lulay’s note, meshing of 3D elements can result in a mixture of “tets” and “bricks”).