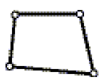
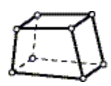
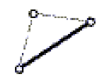
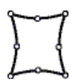
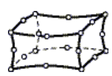
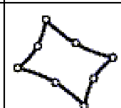
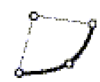
ANSYS TIPS

Picking an Element Type For Structural Analysis:

by Paul Dufour

Picking an element type from the large library of elements in ANSYS can be an intimidating thing for a beginner. Out of the almost 200 available which one should I select? Why are there so many? Are there 181 kinds of shell elements? This article will address this issue for a structural analysis. The table below shows the most commonly used element types for stress analysis.

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

Element Naming Conventions:

Elements are organized into groups of similar characteristics. These group names make up the first part of the element name (PLANE, SOLID, SHELL,etc). The second part of the element name is a number that is more or less (but not exactly) chronological. As elements have been created over the past 30 years the element numbers have simply been incremented. The earliest and simplest elements have the lowest numbers (LINK1, BEAM3, etc), the more recently developed ones have higher numbers. The “18x, 20x, and 22x” series of elements (SHELL181, SOLID187, etc) are the newest and most modern in the ANSYS element library.

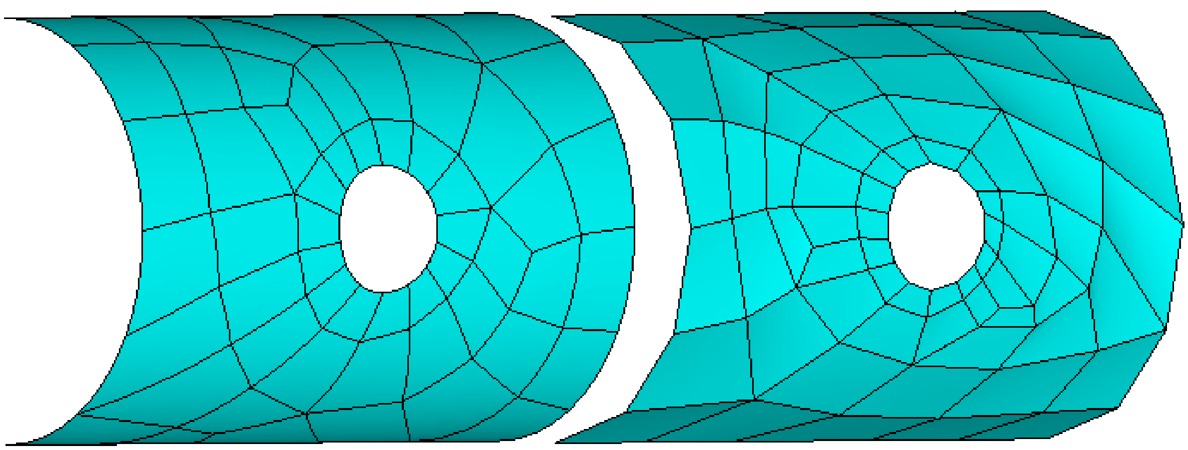
Some FEA codes (NASTRAN comes to mind) have just a handful of elements that do everything. There are several reasons why there are such a large number of elements in ANSYS. First, there was a conscious plan from the beginnings of the program that, for example, if you want to do a thermal analysis, there is no reason to use an element with structural degrees of freedom. That just requires an element to carry a bunch of baggage around that it is not using. While ANSYS does have coupled field elements that can do both, generally speaking if you want to do a thermal analysis you use an element with a temperature degree of freedom only, and structural elements do not drag around an unused temperature degree of freedom. This was done in the name of computational efficiency. Secondly, old elements are rarely deleted out of the program library for the sake of compatibility with older models. When a new and improved BEAM4 element was created, BEAM4 was not revised but left alone, and the BEAM44 was introduced. So the library keeps getting larger as new and more capable elements are formulated. The last reason is that ANSYS is a huge program with integrated capabilities in many physics fields. So there are elements for fluids, electromagnetics, electric field, acoustics, and explicit dynamics analyses as well.

<u>Element Order:</u>

Element order refers to the interpolation of an element's nodal results to the interior of the element. This determines how results can vary across an element, and is important if you have high gradients of strain. A beam or plate in bending is a good example of this phenomenon, where the strain is changing sign over a potentially very small distance. Element order can be linear or quadratic.

Linear elements do not have midside nodes. In general the strain can only vary linearly from one node to another. This is complicated in ANSYS with many "linear" elements having so called "extra shapes" or being "fully integrated". This makes the elements behave more like a quadratic element, but not completely so. Naturally a linear element is computationally faster than a quadratic element.

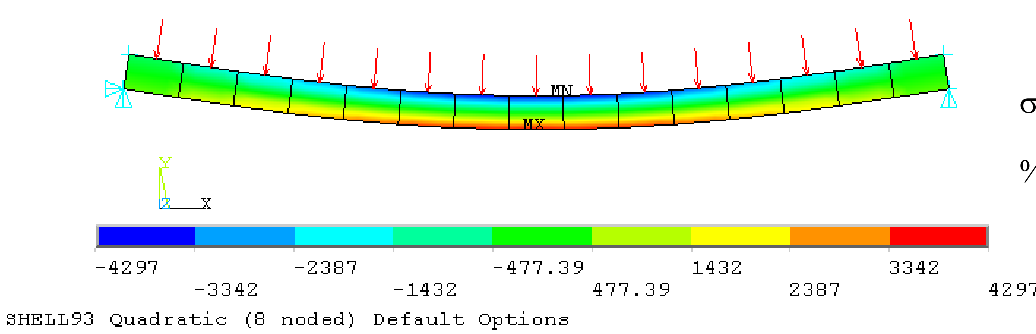
Quadratic elements have midside nodes. The shape function for strains varies in some nonlinear fashion between the corner nodes. Whereas linear elements are flat on both the sides and in-plane, a quadratic element can follow a curvature in both directions and is more accurate for a given number of nodes in the model. For example, in a tube with a hole, the curvature is maintained around the edges of the hole as well as on the curved edge of the panel; it is not a faceted representation as illustrated below.



SHELL93 Quadratic Elements (8 nodes) SHELL181 Linear Elements (4 nodes)

Some considerations in picking an element order:

- For in-plane bending type problems, quadratic elements are much better than linear elements. Even several linear elements with "extra shapes" or "full integration" turned on are not as good one quadratic element.



$\sigma_{theo} = 4318$ psi

$\%$ error $< \frac{1}{2}\%$

- Linear elements are more sensitive to non-perfect element shapes (i.e., have more error). A linear element is probably better in a mapped mesh than a free mesh.

- For out-of-plane, plate type bending, there isn't much difference between linear and quadratic elements.

- Linear bricks give good results due to the "extra shapes" (full integration) if the elements are regularly shaped. If the volume has a shape complex enough to make skewed elements, mesh it with 20 node bricks or tetrahedral elements. An important side note here…linear triangles and tets give poor results since they are actually constant-strain elements (one value of strain calculated for the whole element. No gradient at all. These elements are no longer even available in ANSYS except as degenerate elements.).

- As a general rule of thumb, one quadratic element containing three nodes should be more accurate than two linear elements having a total of three nodes across the same dimension.

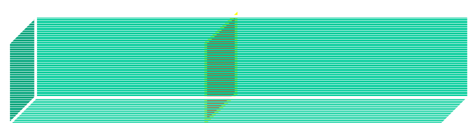
- In nonlinear analysis, a finer mesh of linear elements is computationally more efficient, robust, stable and more accurate than a coarser quadratic mesh with a comparable number of nodes.

Make *very* simple models and experiment! Lame "advice", I know…but it will *never* fail you!

2D Solid Elements:

How can a 2D element be a "solid"? That's a question most new ANSYS users ask. There are two reasons in ANSYS-speak. First, they are considered solid because they are not a shell or a beam element. Shells and beams are structural abstractions that will be discussed in the following sections. Secondly, they are solids because they fully represent a solid chunky part by modeling a *cross section* of that part. There are three main kinds of 2D solid elements:

Plane Strain: In a plane strain analysis we are modeling something that is so long that there is no strain in the thickness direction. A cross section of a long beam or a dam holding back water would be examples of this kind of behavior. Stresses in the thickness direction are significant.



Axisymmetric: In this type of analysis we are modeling a cross section of a structure that is fully defined by rotating the cross section around a central axis. A pressure vessel or bottle would be examples of this kind of structure. The thickness direction here represents the circumferential (hoop) direction and those out-of-plane stresses are very significant (picture a tank under pressure).

Plane Stress: This is not really a "solid" element per say but still falls into this category (since it doesn't really fit anywhere else). A plane stress analysis is a 2D model of a thin sheet with a small thickness and in-plane loading. You would get the same results with a shell model with in-plane loads only and the out-of-plane displacement constrained.

All 2D solid elements can be any one of these three types by changing the element key options.

PLANE42: Basic 2D solid element.

PLANE82: 8 noded version of PLANE42. Also has a triangular option.

PLANE182: Similar to PLANE42 except has capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

PLANE183: 8 noded version of PLANE182.

<u>3D Solid Elements:</u>

These types of elements have their geometry fully defined by the element nodes. They are elements used to mesh volumes in ANSYS. These volumes could be created in the ANSYS preprocessor or imported from a CAD system. Hexahedral elements (bricks) can be used to mesh regularly shaped rectangular type volumes, while tetrahedral elements (tets) can be used to mesh any volume. Even linear bricks (if they are not distorted much) can model a thin plate in out-of-plane bending with one element through the thickness since they all by default have the “extra shapes” turned on.

SOLID45: Basic linear brick.

SOLID95: 20 noded version of SOLID45. It can tolerate irregular shapes without as much loss of accuracy. Well suited to model curved boundaries. These elements can also be tetrahedral and can automatically transition between hexahedral and tetrahedral using pyramids.

SOLID185: Newer version of SOLID45. Also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

SOLID186: Newer version of SOLID95. Also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

SOLID92: Tetrahedral only element (10 noded quadratic).

SOLID187: Newer version of SOLID92. Similarly to other 18x elements also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

3D Shell Elements:

A shell element is a surface type element. It is really a 2D element that is called 3D because it is not restricted to the XY plane like a 2D solid element; it can be located anywhere in three-dimensional space and it can deform out-of-plane. Shell elements are engineering "abstractions" because a geometric surface has no physical thickness. An ANSYS *real constant* is used to assign a thickness to a shell element. Shell elements are also called plate elements, and are used to model panel type structures where the thickness is small compared to the other dimensions of the part. They can carry in-plane loads (also called membrane loads) and also out-of-plane bending moments and twisting.

SHELL63: This is an older but still very widely used shell element. It can analyze large deflections but not plasticity.

SHELL93: Quadratic version of SHELL63. In addition it can analyze plasticity.

SHELL181: Newest linear shell element. Has plasticity and can also be used to model laminated composites and sandwich panels. Note: There is no eight noded 18x series shell element.

Line Elements:

Line elements are a further engineering abstraction from shells; they are physically idealized as a simple line, which represents a long slender structure that can carry axial, bending, shear and twisting forces. Real constants define the cross sectional properties of the beam element.

BEAM3: Simple 2D beam element. Must be created in XY plane.

BEAM4: Beam element that can be modeled in 3D space. Has large deflection.

BEAM44: Tapered beam element. Can use the beam tool to define a section and have real constants automatically calculated.

BEAM188: Timoshenko beam that includes shear deformation. Can handle plasticity and can define a section with the beam tool. Can have a built-up cross section with more than one material. By default this is a true linear element. You need a lot of them to give the same results as *one* BEAM3/4/44/189. In release 8.0 turn on KEYOPT(3) = 2 to give an internal quadratic shape function to eliminate this problem. Can display stresses on the actual beam shape like it was solid.

BEAM189: Three noded quadratic version of BEAM188. This is pretty much the ultimate beam element that can do everything well.

Conclusion:

ANSYS has many elements in its library, but it's fast to narrow them down to just a few for your structural analysis once you select an analysis approach. Try to stick with the 18x/20x series of elements when possible because they incorporate the latest technology.