

FEA - A Practical Guide

FEA – Understanding the Physical Problem.

This is arguably the most important step in the FE procedure as it will determine all steps that follow. The first thing you need to establish is;

WHAT ARE YOU TRYING TO DO?

Answers to the above question may include “*I want to determine the maximum stress in a plate due to an applied pressure load*” or “*I want to optimise the shape of a steel beam so that it can carry a load of 250kN*”.

Once you have established what you are trying to do, you then need to identify all the distinctive features of the structure that must be analysed. You should consider the following;

1. What are the dimensions of the structure?
2. Which dimensions are particularly important?
3. Can the structure be simplified into a number of simpler structures for analysis?
4. Can the structure be simplified by removing any geometric entities that will not effect the analysis results? (i.e. defeaturing)
5. Does the structure exhibit any symmetry?
6. What material is the structure made from?
7. What type of loading is present?
8. Are there any locations where stress concentrations are to be expected (e.g. holes, fillets etc?)
9. Are the loads expected to be large enough to induce yielding, permanent deformation or failure?
10. Are the loads constant or do they vary with time?
11. Are the loads symmetrical or applied all in one plane?
12. Is the structure expected to come into contact with other structures as a result of the loads or as a result of an impact event?
13. Are there any significant environmental factors which will affect the structures response?

Types of mathematical models used to model geometry

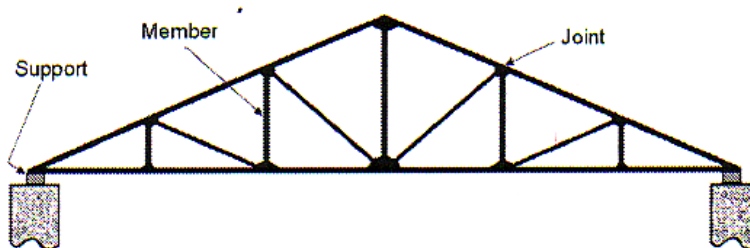
Before creating your FE model you need to decide what type of structure you are dealing with, and if simplification or approximations can be made to obtain an acceptable answer more efficiently. Your structure will generally fall into one of the below categories;

1. 1D Spar or Truss

This type of geometry does not change its cross sectional area along the length of the structure and can only change in length along its greatest dimension, and can only carry axial loads. A typical example might be a cable in tension.

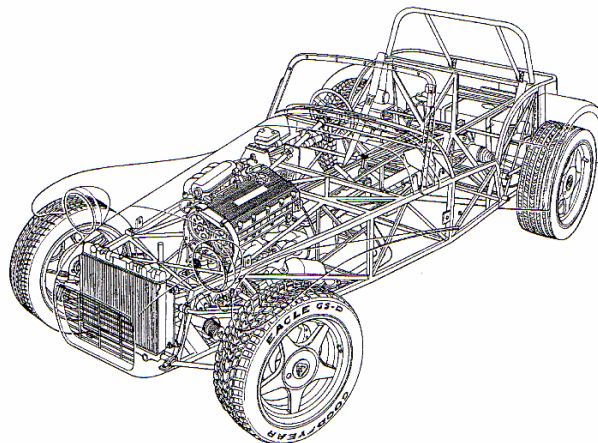
2. 2D Spar or Truss

This is a 2D version of the above. Every member within the structure is assumed to carry only axial loads and has no resistance to bending. The members are assumed to be connected via frictional pin joints. A typical example may be a 2D truss used in a bridge;



3. 3D Spar or Truss

This is a 3D version of the above. Examples may be bridges, towers or space frame structures;

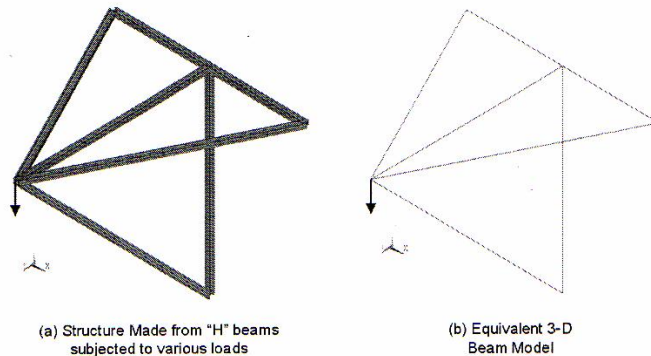


4. 2D Beam

A beam is a structural element whose length is large compared to its width and height. Beams can carry axial AND BENDING loads. Properties of the beam cross section are considered, i.e. a 2D beam analysis will differentiate between the use of a “C” section and a “I” section, a spar or truss would not.

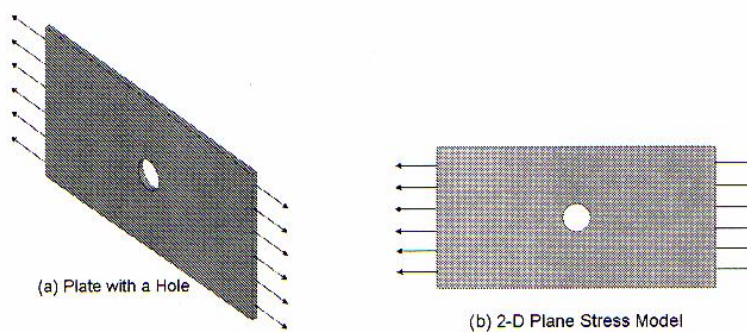
5. 3D Beam

This is a 3D version of the above. This is used to model structures such as space frames when BENDING and TORSION are considered to be important.



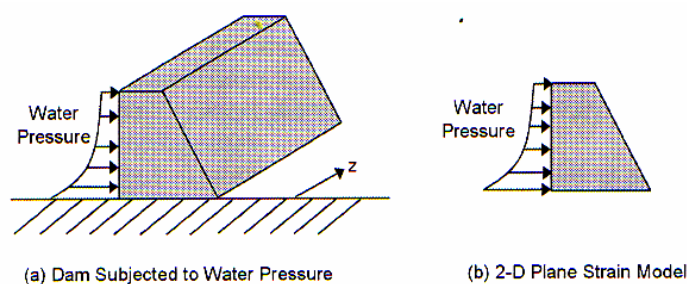
6. 2D Plane Stress

A plane stress structure assumes that there are ZERO STRESSES (and non zero strains) through the thickness of the structure. This type of model is only valid for plate like structures where the thickness dimension is much smaller than the other two dimensions.



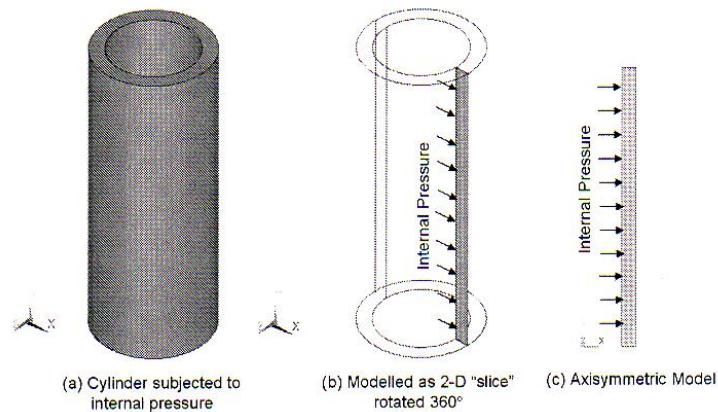
7. 2D Plane Strain

A plane strain structure assumes that there are ZERO STRAINS (and non-zero stresses) through the thickness of the structure. This type of model is used for long, constant section structures, where the thickness dimension is much larger than the other two dimensions for example rolling, drawing and forging, or if you were to model a dam;



8. 2D Axisymmetric

This type of structure has geometry, loading and boundary conditions that are symmetric around a central axis, and can thus be described by rotating a 2-D section 360° around the central axis. Typical examples may be pressure vessels, straight pipes and shafts



9. 2D Axisymmetric with torsion

This is based on the above, with the addition that the structure can carry a torsion load. This is useful for modelling a axisymmetric structure that is carrying a non-axisymmetric load, for example a shaft carrying a torque.

10. 3D Thin shell

This is used for modelling thin panels. A thin shell model includes both bending and membrane or stretching effects and assumes that stresses through the thickness of the material are not significant. Typical applications include automotive body panels or any structure made up of sheet metal.



11. 3D Solid

This is used to model structures which can not be modelled with any of the simpler mathematical models above.

Summary

1. **1D Spar or Truss:** Cannot carry bending
 2. **2D Spar or Truss:** Cannot carry bending
 3. **3D Spar or Truss:** Cannot carry bending
 4. **2D Beam:** Can carry bending
 5. **3D Beam:** Can carry bending and torsion
 6. **2D Plane Stress:** Zero stresses through the thickness
 7. **2D Plane Strain:** Zero strains through the thickness
 8. **2D Axisymmetric:** Rotational symmetry
 9. **2D Axisymmetric with Torsion:** Rotational symmetry with torques
 10. **3D Thin Shell:** Assumes stresses through the thickness are not significant
- 3D Solid:** None

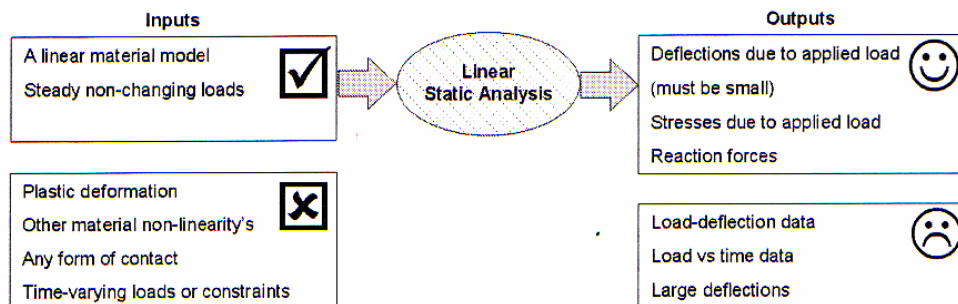
FEA Solve Types

Once you have decided what type of structure you have, you need to decide what type of analysis you are going to do on that structure. The basic structural computational methods offered by most FE software are;

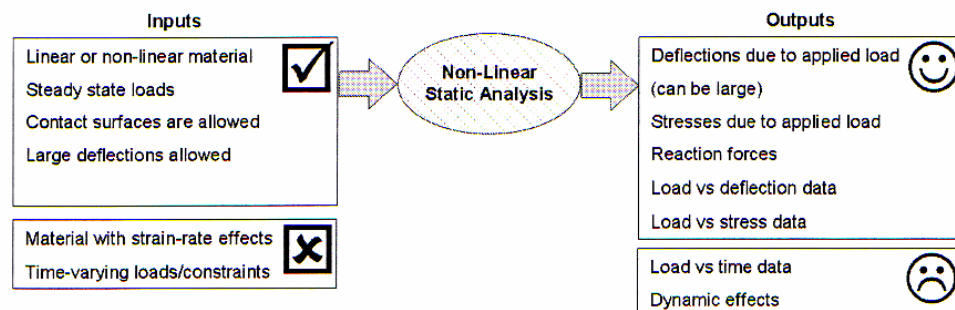
1. Static Analysis

This assumes that there are no dynamic effects in the system. A static analysis can consider static loads or steady inertia loads such as gravity or rotational velocity.

- a. **Linear Static:** This assumes that material properties are linear, deflections are small and that loads and boundary conditions do not change with time.

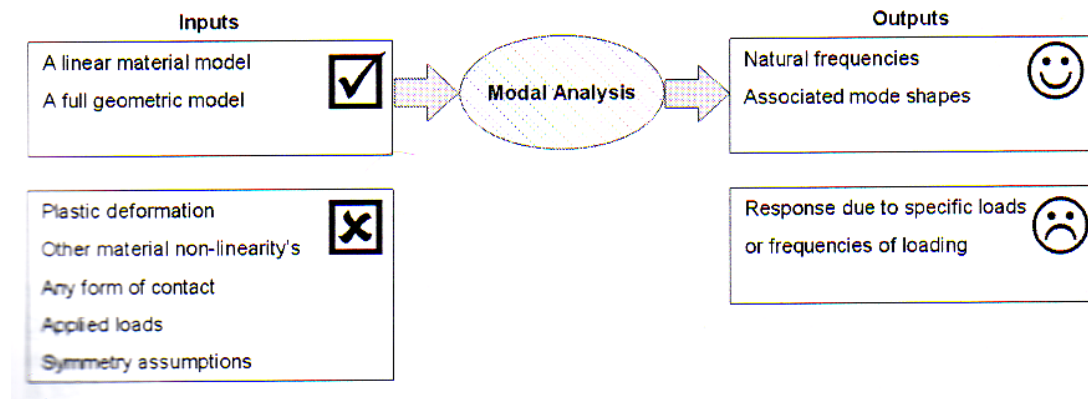


- b. **Non-linear Static:** This can include large deformations, contacts, plasticity, creep, and time varying loads. However, it is still assumed that these time varying loads do not introduce any dynamic effects. (if a load changes quickly such that it would introduce dynamics effects the solver will use solution sub-steps i.e. split the problem into smaller sub-steps to ensure the solution runs smoothly)



2. Modal Analysis

A modal Analysis is used to determine the natural frequencies and associated mode shapes of a structure. This is a dynamic analysis, so dynamic effects are included and the density of the material must be included in the material model. Also note that symmetry assumptions can not be used for a modal analysis. If you use reflective symmetry (for example) you will eliminate a percentage of the mass of the structure and thus the results from your modal analysis will be meaningless. In general modal analysis is considered a linear analysis hence non-linearities such as non-linear materials and contacts are not allowed.

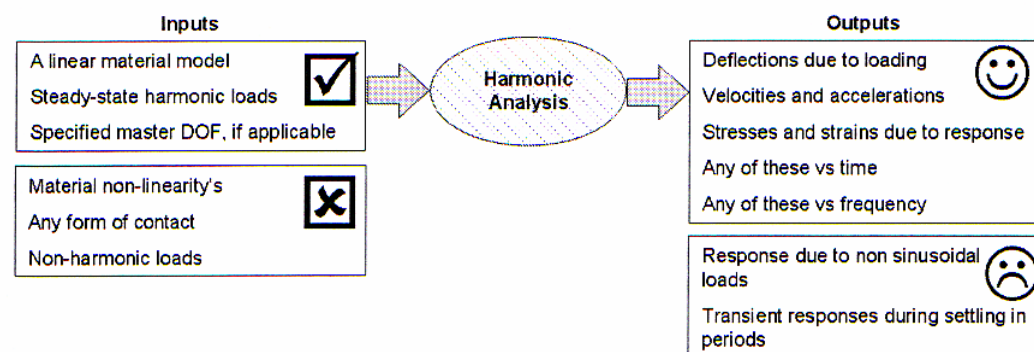


3. Harmonic Analysis

A structure that is exposed to a sustained cyclic load will respond with a sustained cyclic response (i.e. a harmonic response). A harmonic response allows for the prediction of the response of the structure, thus providing information as to whether a design will successfully overcome resonance, fatigue and other harmful effects of forced vibrations.

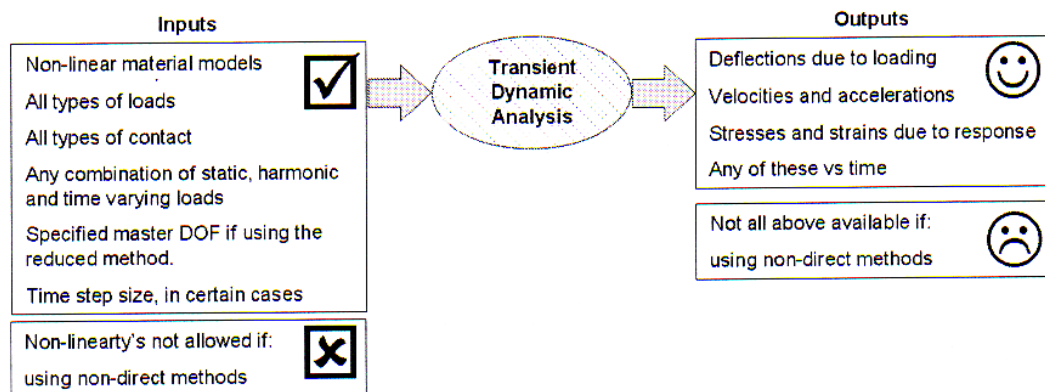
A structure is subjected to loads that vary sinusoidally with time and the response of the structure (generally displacement) is generally plotted against frequency. Peak responses on the graph are then identified and stresses reviewed at these frequencies. Harmonic loads are normally imparted on structures due to rotating or reciprocating machinery. In general harmonic analysis is also considered a linear analysis hence non-linearities such as non-linear materials and contacts are not allowed.

All the loads applied in a harmonic analysis must be at the same frequency, however, the load amplitude and its phase may be specified as being different at different locations in the model. It is important to note that when a steady state harmonic load is applied to a structure there will initially be an unsteady response from the structure as it adapts to the newly applied load and moved from rest, after which the structure will have a steady state response. This “settling in” period is not predicted by a harmonic analysis, so if you are interested in this part of the analysis you should perform a transient analysis.



4. Transient Dynamic Analysis

A transient dynamic analysis is a method used to determine the dynamic response of a structure due to the action of any type of loading that varies with time. This type of analysis is used to determine time-varying displacements, velocities and accelerations at each time step in the analysis. Unlike in static analysis, inertia and damping effects are considered to be important. Transient dynamic analysis can be used to model any structure subjected to any combination of static, harmonic and transient loading.

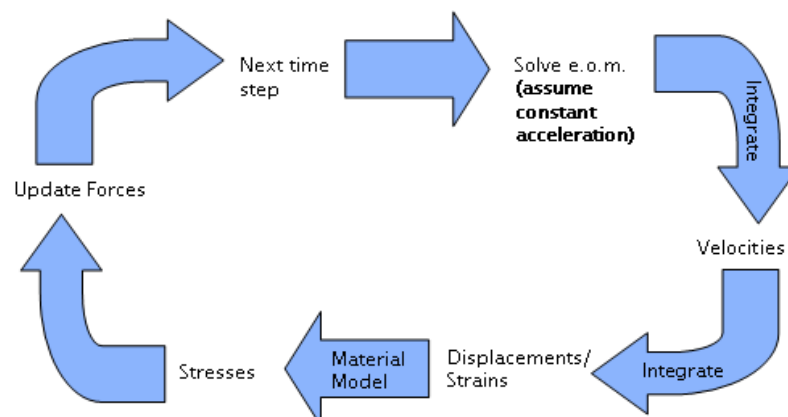


5. Explicit Dynamic Analysis

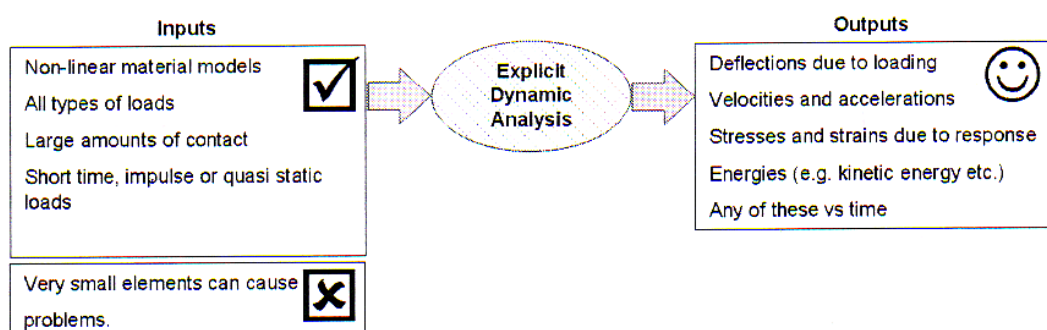
This is used to obtain fast solutions, for short-time, large deformation dynamic, quasi-static problems with large deformations and nonlinearities, and complex contact/impact problems. In general, if a problem is highly non-linear, with non-linear materials, contacts, large deflections etc. and subjected to dynamic or short time or impulse loads then an explicit dynamic solution should be used. The explicit solution method is different to the more common implicit solution method used in most other types of analysis.

The implicit time integration method requires the formulation and inversion of the global system equations at each time step. This is very costly in terms of computational resources, particularly when the model is very large. When the number of DOF becomes large and the CPU per increment becomes very long. For problems with few time increments this is not so much of a problem, but when a large number of time increments are required then it is likely that computational resources will be exhausted.

In contrast explicit codes assume that acceleration is constant over each time step, this leads to linearization of the set of non-linear equations. Thus to obtain realistic results a small time step must be used. However, this is computationally cheaper as iterations are not used. Thus explicit dynamic analysis carries out the procedure seen below;



The main disadvantage of this method is that the time step size must be small for a stable solution. For stable computations, the time step is normally selected by the solver code such that the time step is less than the time required for a stress wave to travel through the smallest element. This means that the solution time is essentially based on the size of the smallest element in the mesh and this must be taken into account when designating the mesh.

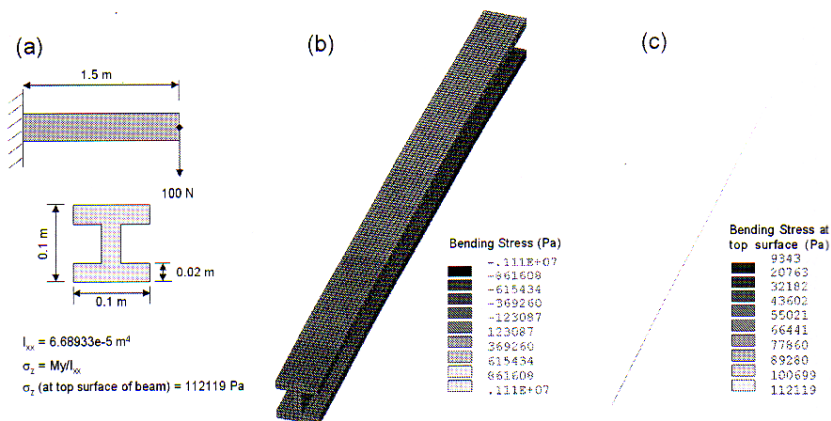


Modelling and Meshing

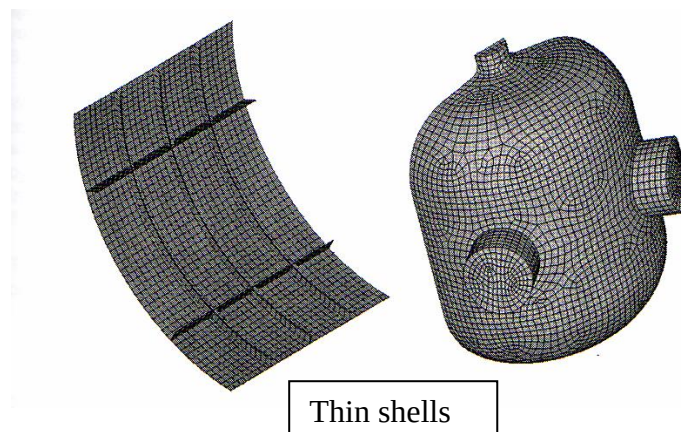
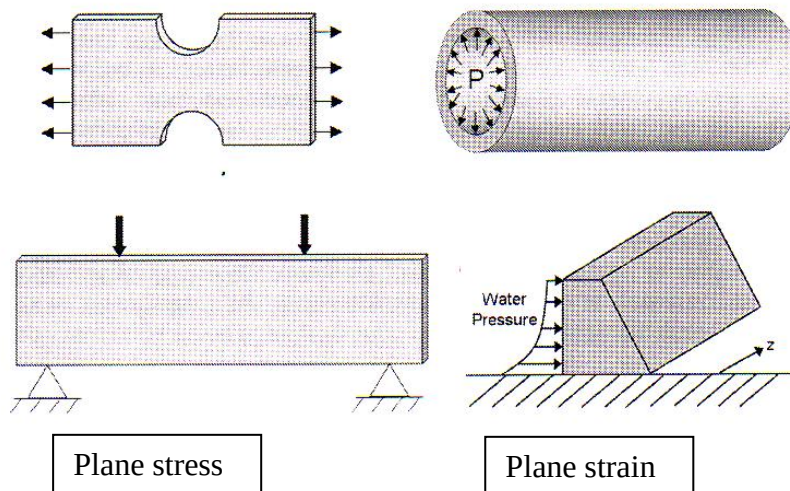
In order to make calculations as simple as possible then it is good practice to make as many assumptions as possible to simplify the calculations PROVIDED that these simplifications do not alter the results that we are interested in, rendering them useless. Once you have done this you can mesh the model.

Simplification through element choice

You should always use the simplest type of geometry and element that will get you the answer that you require. For example, could the problem be reduced to a simple beam model, a planar stress model, a planar strain model, an axisymmetric model or a thin shell model?

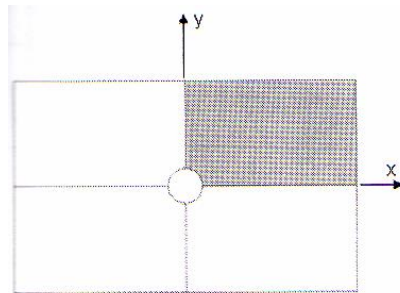


Modelling a structure with beam elements (c)

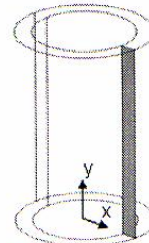


Simplification through Symmetry

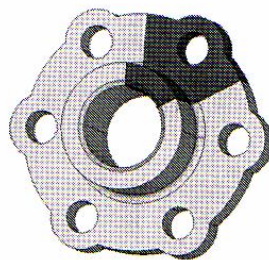
In some cases it will be possible to reduce the model complexity by using any symmetry in terms of geometry and loading. Types of symmetry that might exist are reflective, axial, cyclic and repetitive;



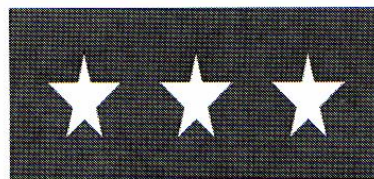
(a) Reflective Symmetry



(b) Axial Symmetry

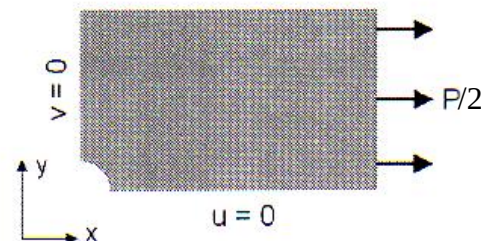
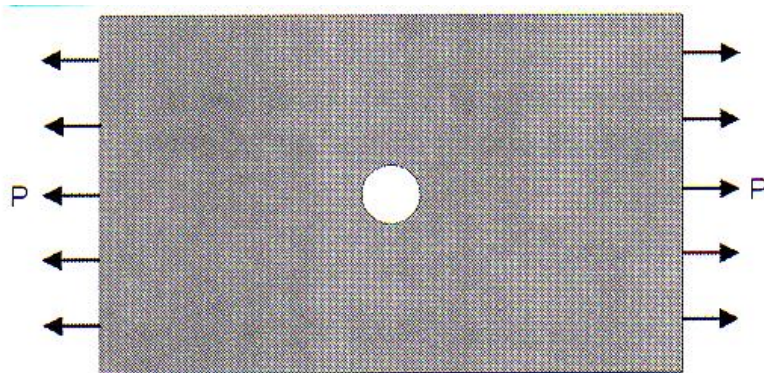


(c) Cyclic Symmetry



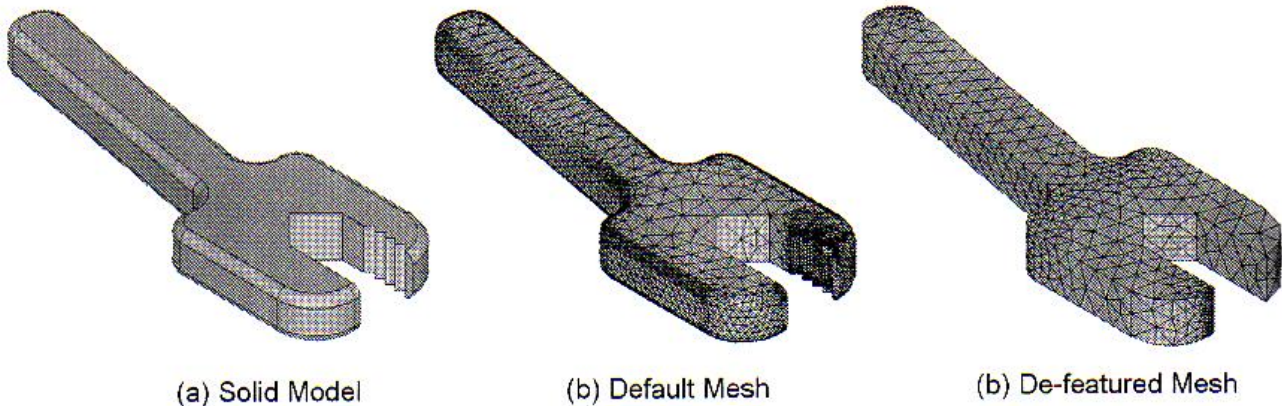
(c) Repetitive Symmetry

If the geometry and the external conditions are symmetrical then only part of the structure needs to be simulated. However, it is important to apply the correct constraints to the 'joining' edges and that the loads are changed as appropriate. For example, consider a large plate with a hole in it subjected to a tensile load;



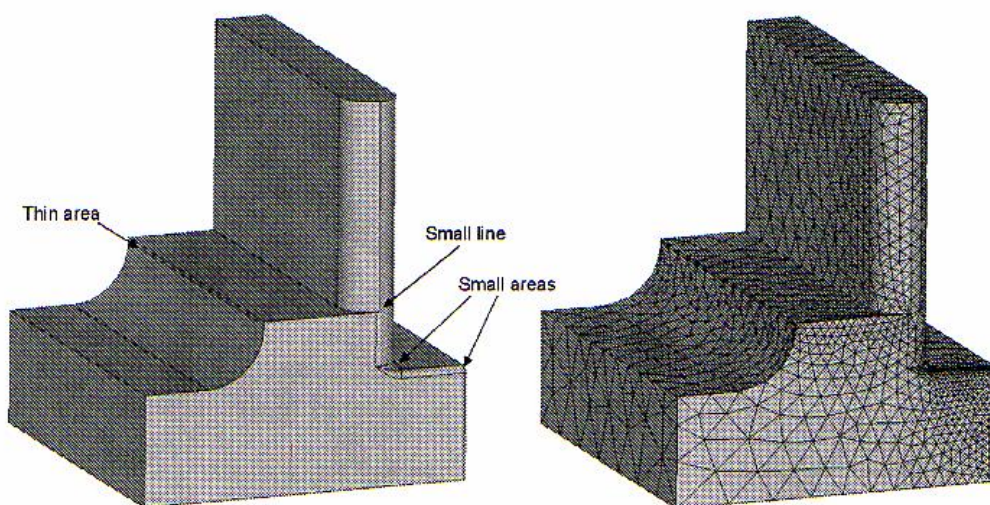
Simplification through Defeaturing

Many parts will have small features that are required for manufacturing or for some other function (e.g. to clip a harness or piece to trim to them), which will have no impact on the structural performance of that part. To model these features would require small elements which would increase the run-time. In order to simplify the calculations and reduce the run-time, these features should be removed wherever possible. In general small fillets, screw threads, splines, small holes, small protrusions, small bosses, decorative features etc should all be considered for removal;



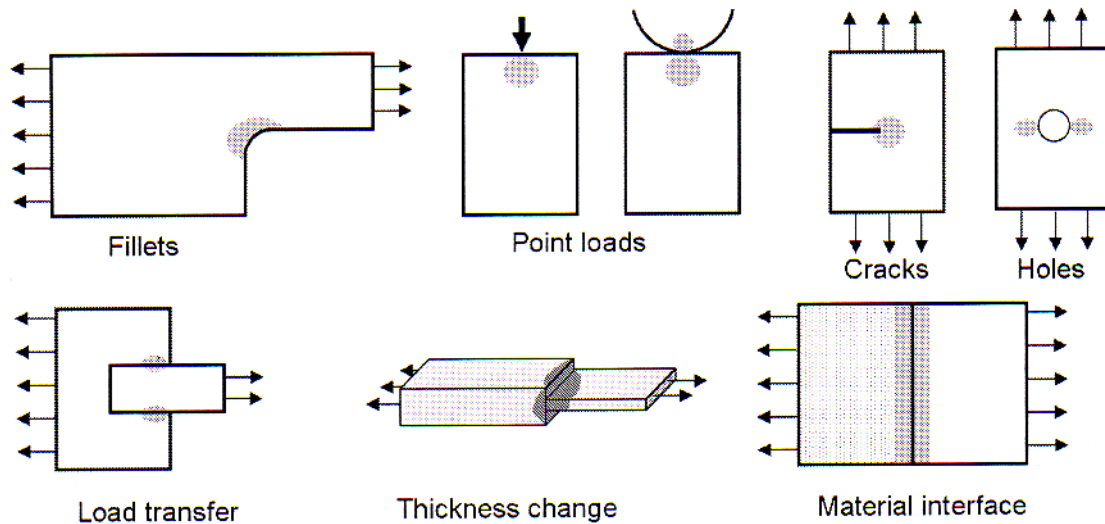
Simplification through Model Clean-up

In many cases when a solid model is imported into a FE software package the transfer process may have resulted in some unwanted geometric effects being introduced to the model. In such cases model clean-up is imperative as it will reduce the probability of meshing problems later in the analysis procedure. While solid modelling is a powerful tool, if models are generated using poor procedures then undesirable features such as short edges, short lines, small voids or gaps, sliver surfaces, small areas etc. may be present in the model. A solid CAD geometry will almost always require significant modifications before it is suitable for meshing with finite elements.



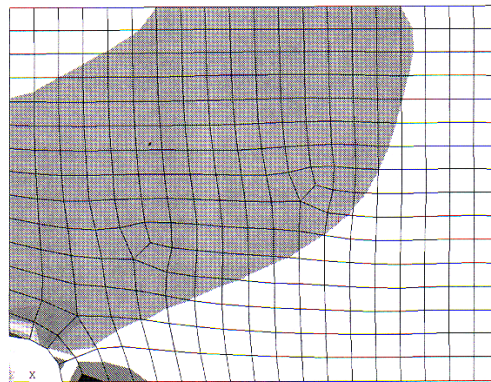
General Meshing Guidelines

In general you should use a coarse mesh for areas where stress/strain is not changing rapidly and a fine mesh in areas where stress/strain is rapidly changing;

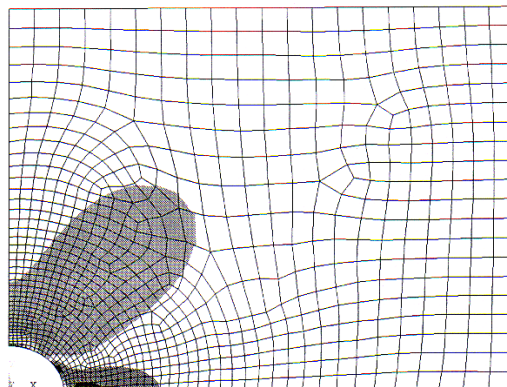


Examples of situations where a more refined mesh is needed in the shaded area

If you are not sure where the areas of high stress are going to occur, you may need to do an initial analysis to determine these areas and then a second analysis to determine the magnitude of these high stresses. Note that there should be a gradual change in stress from one element to the next. Any large changes from one element to the next or discontinuities suggest a poor mesh.



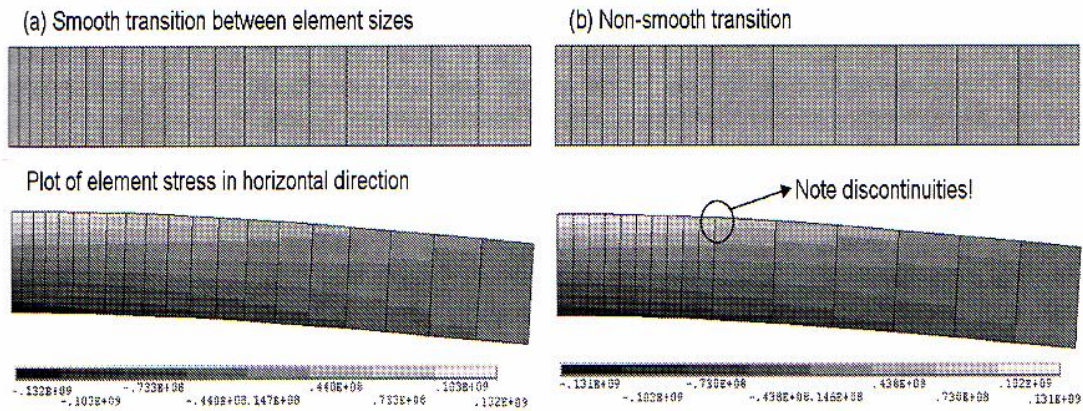
Example of an Unsuitable Mesh to Analyse the "Plate with Hole" Problem; Plot of Element Stresses Shows Discontinuities



Example of a Suitable Mesh to Analyse the "Plate with Hole" Problem; Plot of Element Stresses Shows Smooth Transitions

Changes in Element Size

As discussed earlier, smaller elements are required in areas where stress/strain changes rapidly and larger elements are required elsewhere to avoid unnecessary calculations. It is important that there is a gradual change of element size from small to large, otherwise discontinuities will be introduced into the model.



Smooth transitions in element size (left) produces better results.

Element Quality

The finite element method essentially works by approximating the distribution of an unknown variable in a piecewise manner across the problem to be analysed. The element formulations are designed with the optimal shape of the element in mind (i.e. squares, equilateral triangles, cubes or equilateral pyramids), hence, the more the elements diverge from this optimal shape the more errors in the element formulations start to become important. Therefore all elements should be a “regular” as possible. In practice, it will not be possible to build completely regular meshes as complex boundaries, holes, fillets etc will require elements to deviate from the ideal shapes. The allowable limits of distortion vary according to the problem but generally speaking if stress is changing rapidly throughout the problem then the results will be very sensitive to element shape.

Most FE software packages have facilities for checking poorly shaped elements and attempting to remedy them. Typical qualities that are checked are;

- **Warpage:** A measure of how far from a 2D planar surface a quad shell is.
- **Aspect ratio:** The ratio of the longest side to the shortest side.
- **Internal angles:** measurements of internal angles of the elements.

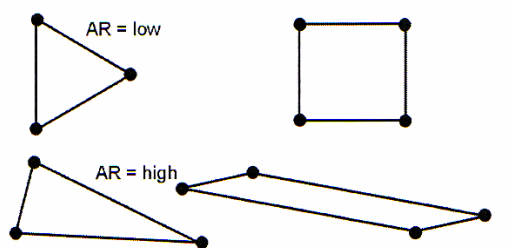
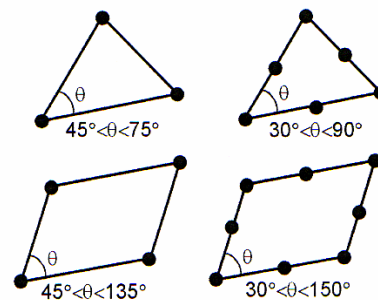


Illustration of low and high aspect ratio 2D elements



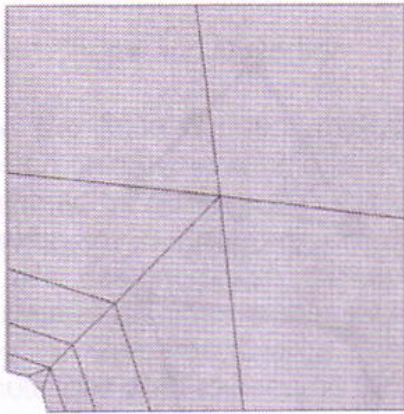
Typical Allowable internal angles in linear and quadratic 2D elements

In general in 2D modelling given a choice between triangles and quadrilaterals for a similar nodal distribution you should pick quadrilaterals. Triangles are great for easy mesh generation and for meshing corners and fillets etc. but quadrilaterals will give much more accurate results.

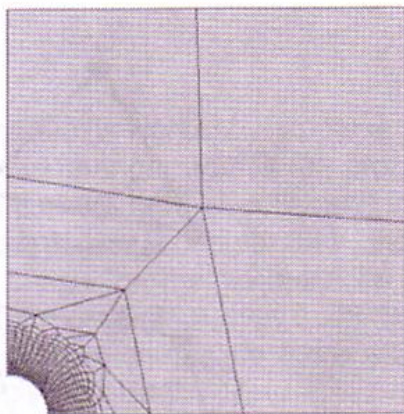
In 3D modelling, bricks should be used over wedges and wedges used over tetrahedrals. Most modern FE software will automatically mesh in tetrahedrals as it is easier to generate a mesh in this manner. Tetrahedrals and wedges can produce acceptable displacement results but generally do not give accurate results for stress. This is basically to do with the fact that these element shapes are very stiff. In recent years there have been great improvements in element formulations for tetrahedral elements, but they should still be used with caution.

What makes a good quality mesh?

Consider the below $\frac{1}{4}$ symmetry plane stress model of a plate with a hole subjected to uniform tension. We will now consider several meshes that have been generated to solve this problem. The first is the default mesh generated by the FE software;

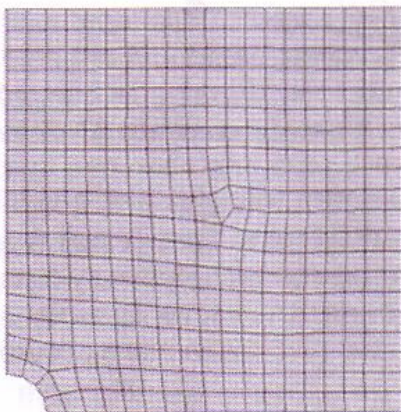


- Only 3 elements have been used around the curve of the hole. This is not enough to correctly model its highly curved geometry.
- We know there will be a stress concentration around the hole, but the mesh around the hole is not fine enough to correctly model this.
- The size ratio between the largest and smallest element in the mesh is too large, generally this should be no greater than 10:1
- The mesh mixes quadrilaterals and triangles (near the hole) which is generally not good practice. A full mesh of quadrilaterals would be preferable.



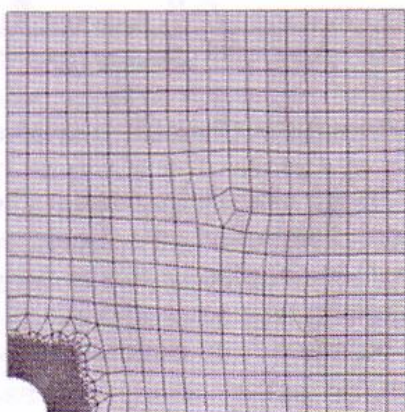
The mesh has now been updated specifying refined elements around the hole.

- The automatic mesh refinement has introduced many poorly shaped and distorted elements into the mesh.
- The size ratio between the largest and the smallest element is even worse.
- The transition from tiny elements to very large elements happens too quickly.



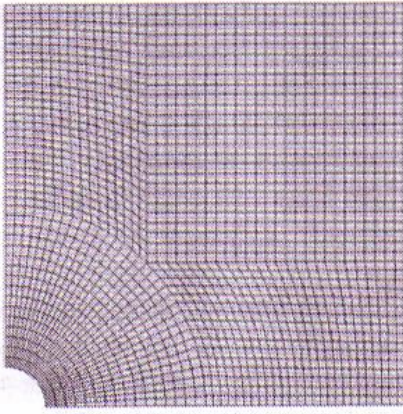
The mesh has now been updated to make it more regular by specifying a constant element edge length throughout the model

- The mesh density in the region away from the hole is appropriate, but there is no refinement around the hole.
- There are only 4 elements around the hole which is not enough to model the curvature or the expected high stress.
- There is only one poorly shaped element in the centre of the mesh. While it is within acceptable limits it would be preferable to not have it there.



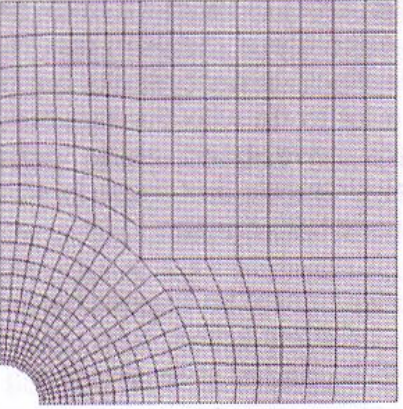
The mesh has now been updated specifying refined elements around the hole.

- The automatic mesh refinement has introduced many poorly shaped and distorted elements into the mesh.
- The size ratio between the largest and the smallest element is once again a problem
- The transition from tiny elements to very large elements happens rather quickly and is localised.



A mapped mesh has now been generated by splitting the plate up into separate surfaces then meshing each surface individually, specifying a constant element edge length throughout the model.

- This mesh is far more regular and superior to any of the above.
- However, there is no mesh refinement around the hole.
- The aspect ratio of the elements next to the hole is a little high.
- The mesh density is probably a little high in regions away from the hole where the stress will be relatively constant.

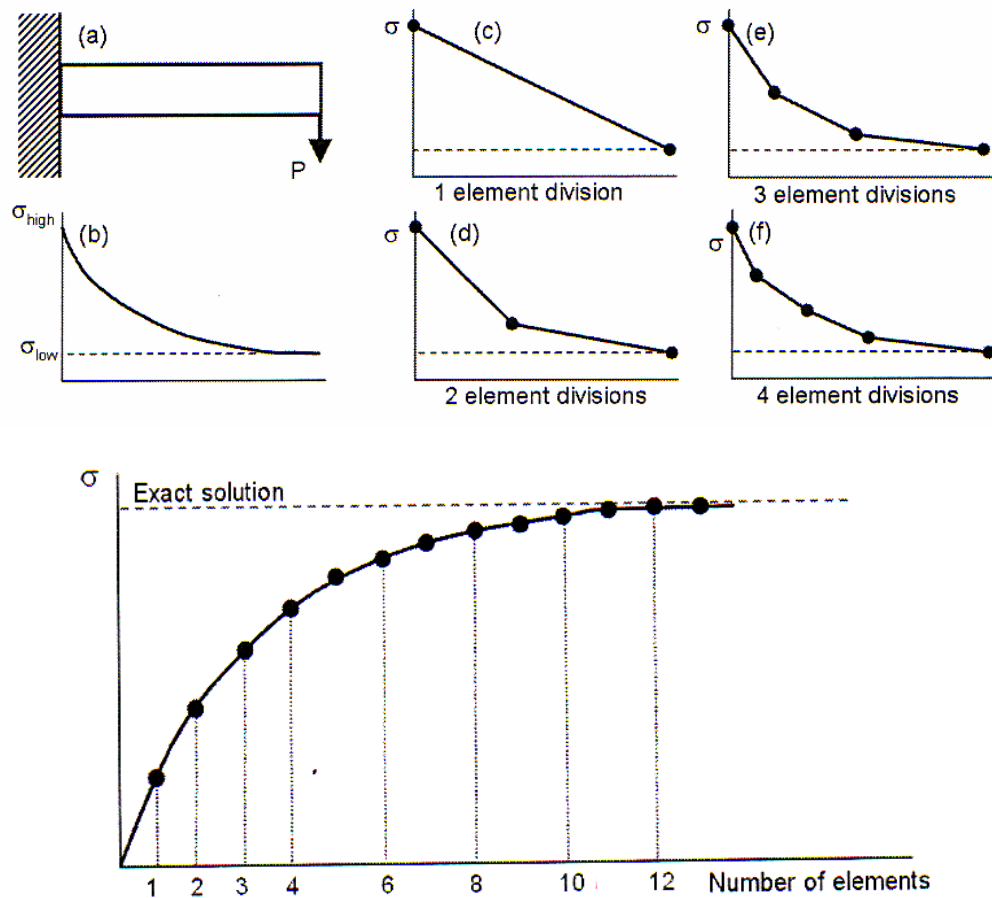


The mesh has now been updated using biasing and by specifying different element edge lengths for each surface, resulting in a good mesh with the following characteristics.

- A refined mesh near the hole.
- Smooth transitions between small and large elements.
- A regular, organised and smooth mesh made up from near square shaped elements.

How many elements to use

Unfortunately there is no simple answer to this. The method used to determine the optimum number of elements in a FE mesh is called **mesh convergence**. Having an acceptable mesh density is very important. If the mesh is too coarse then the results may contain serious errors and vital information like regions of high stress can be missed due to poor mesh resolution. If, however, you take a brute force approach and use an overly fine mesh throughout the model, your model will take a lot of time and computational resources to solve. Consider modelling a cantilever beam with beam elements. The following compares the theoretical results (a quadratic distribution) with the results from using more and more elements.



Error Estimates

Most FE post-processors offer a facility to calculate element error estimates and plot these values on the FE mesh. These error estimates measure the relative difference between results (e.g. displacement or stress) across an element edge or at a node. These will give an indication to the quality of the mesh.

Element Types

Before meshing your problem you need to decide what element type you are going to use. Elements for structural analysis can generally be classed as follows;

Basic Shape	Subtypes	Representative Geometry
Point	Mass Element	
Line	Spar/Truss Element Spring Element Beam Element	
Area	2D Solid Element - Plane Stress - Axisymmetric Plate Element	
Curved Area	Shell Element 3D Contact Element 3D Surface Effect Element	
Volume	3D Solid Element	

Point Elements

- Mass Elements

These are used to define a concentrated point mass at a point in a FEM

Line Elements – 1D

- Spar/Truss Elements

These can extend/compress axially, but cannot bend or change cross section.

- Spring Elements

Essentially similar to spar/truss elements and are typically used with point masses to build mass-spring models of engineering problems.

- Beam Elements

These are used to model beams where tension, compression and BENDING occur. These elements allow the definition of sectional properties to model the beam cross sectional shape.

Area Elements – 2D

- 2D Solid Element – Plane Stress

These elements are rectangles or squares that are used to model thin sheet structures assuming plane stress (i.e. there is zero stress through the thickness of the structure)

- 2D Solid Element – Plane Strain

These elements are rectangles or squares that are used to model thick structures where the cross-sectional properties do not vary significantly along the thickness assuming plane strain (i.e. there is zero strain through the thickness of the structure)

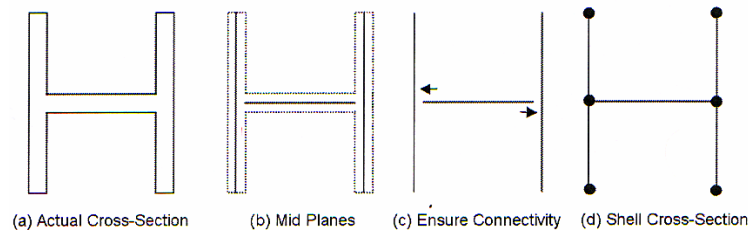
- 2D Solid Element – Axisymmetric

These elements are rectangles or squares that are used to model axisymmetric problems. These elements can deform in the radial and axial directions and may also be able to model torsional effects.

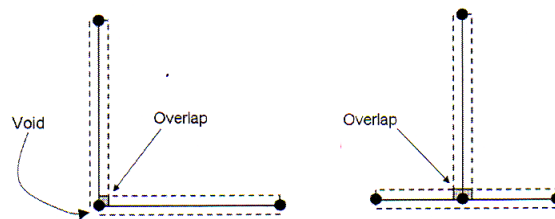
Curved Area Elements

- **Shell Elements**

These elements are plate elements that include both bending and membrane or stretching effects and are suitable for modelling 3D thin shell type structures. These elements can deform in the x,y, and z direction and element thickness is generally inputted during the element definition, hence these elements should be placed on the midsurface of the structure they are representing so that when half the thickness is applied to either of the element, the final geometry closely represents the original;

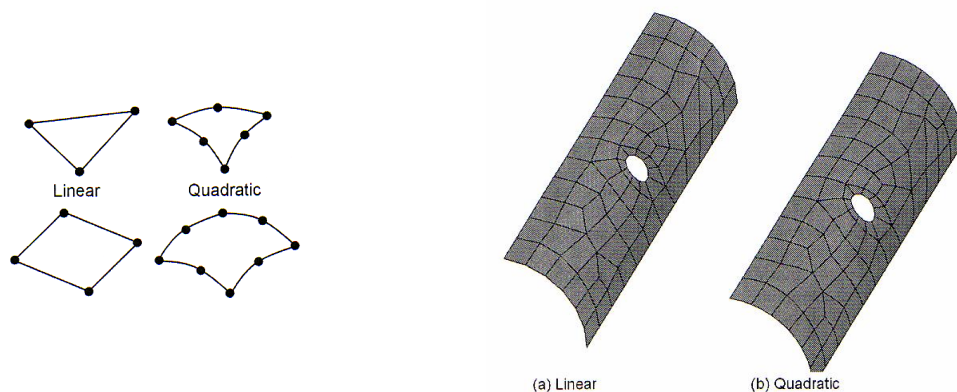


Another consequence of projecting half the thickness on either side of the elements is where elements join at a corner and a void or an area of overlap may occur which will affect the results in this region;



When determining if a structure is suitable for analysis with shell elements, the main criteria to be satisfied is that the wall thickness of the structure must be significantly less than the other dimension. A structure may generally be described as a thin-walled structure when its surface area to wall thickness ratio is 10:1 or greater.

Shell elements are available in both rectangular and triangular shapes and in both first order (linear) and second order (quadratic), or higher, forms. The type and order of the element used will be determined by the geometry of the shape being modelled and the accuracy of the results that are required;



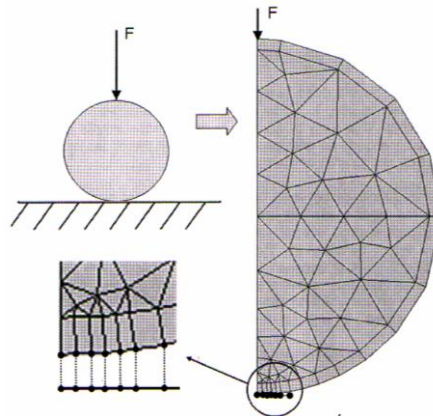
When post processing shell elements, you should be clear which surface (top, middle or bottom) of the element you are looking at as the stress etc. may be different on each surface.

- **3D Contact Elements**

These are used to define contact between two structures. Types include the following;

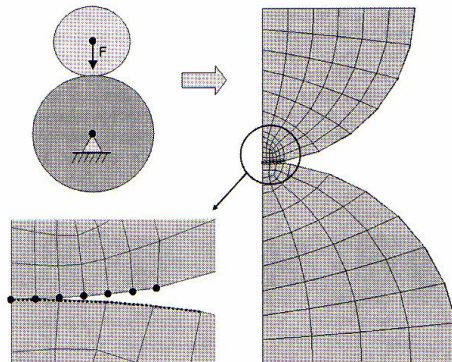
- **Node to node contact elements**

These are typically used to model point-to-point contact applications. This means that the location of contact is known prior to running the model. This type of problem usually involves little relative sliding between the contacting surfaces.



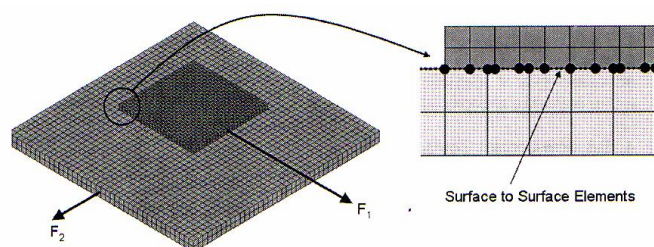
- **Node to surface contact elements**

These elements can be used to model point-to-point, point-to-surface or edge-to-surface type contacts. This form of contact is much more simple to implement as the nodes on one body do not have to line up with the nodes on the other body. This allows large deformation and large relative part sliding, without requiring knowledge of the exact location of the contact areas beforehand.

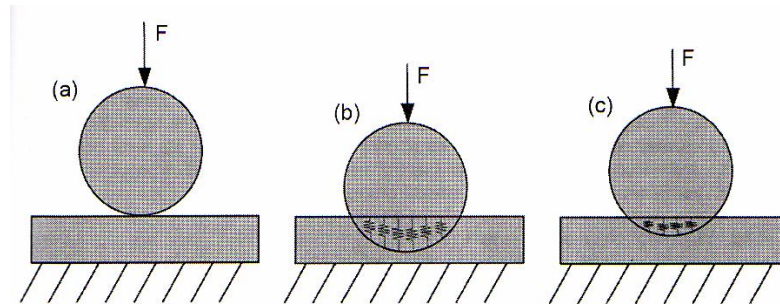


- **Surface to surface contact elements**

These elements represent the most useful method for the solution of real world problems. These elements use a target surface and a contact surface to form a contact pair. These elements are well suited to problems where large areas of contact exist or there is large relative sliding, however, they are not well suited to point-to-point contacts or point-to-surface contacts when one of the other contacts discussed may be more suitable.

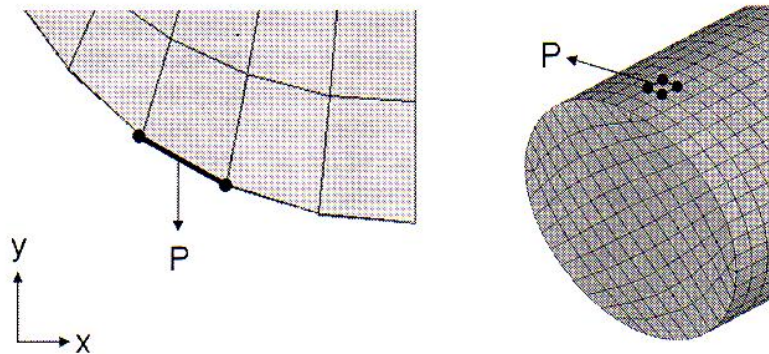


There are many different methods for formulating contact elements in FEA, however the most common methods are based on the **penalty approach**. In this method when one surface comes into contact with another a spring is placed between the two contacting entities (e.g. nodes), and this applies a restoring force on the contact entity which has penetrated the target entity. The stiffness of this spring is determined by the stiffness of the two contacting surfaces or may be defined by the user;



- **3D Surface Effect Elements**

These surface effect elements are overlaid onto a face of a 3D solid or shell element and used to apply various loads and surface effects. For example, the below shows a surface element which is used to apply a pressure load in a particular direction;



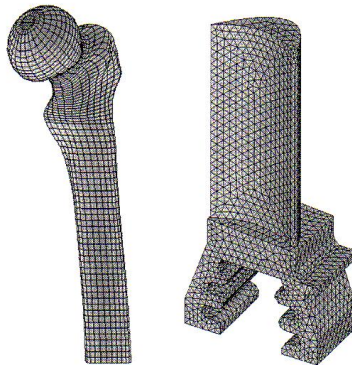
Volume

- 3D Solid Elements

These come in two fundamental shapes, bricks (hexahedrons) and pyramids (tetrahedrons) and are used for 3D modelling of solid structures.

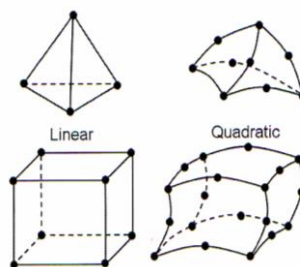
In reality all problems are three dimensional, however, to simplify calculations, wherever possible, assumptions are made so that simpler element types may be used. When this is not possible and the full 3D stress regime must be considered, then 3D solid elements are used. However, in most cases it will still be possible to simplify the problem by taking advantage of symmetry and thus modelling only part of the problem.

Examples of models using 3D solid elements are shown below;



An advantage of using solid elements is that the FE model will look like the actual problem and assumptions regarding loads will generally not be required, so a 3D FE model is created upon which pressures, forces or displacements can be applied without any need to take account of plane stress or axisymmetry. However, the disadvantages are difficulty in producing the solid model, difficulty in meshing in 3D, more computing power to solve the problem and more post processing effort required to interrogate the results.

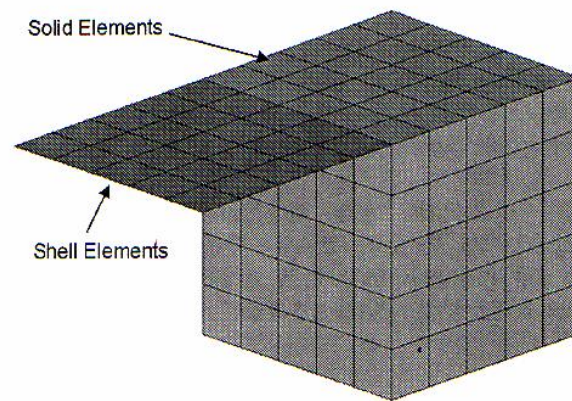
Just as with shell element, solid elements can come in two different shapes and be linear or quadratic etc



In recent years much effort has been put into improving the formulation of tetrahedral elements. Theoretically, tetrahedral elements are too stiff during bending and hexahedral elements were preferred, however, tetrahedral elements allow for easier meshing of complex curved structures. If this is an issue in your analysis then check the help system of your FE package for more information. Note also that solid elements can not model bending behaviour so when using them you should ensure that you have at least three elements through the thickness of the structure.

Changes in element type

Often when you are creating an FE model you will need to use more than one element type. Great care must be taken when joining different element as there will be inconsistencies at the interface that could result in the loads being transferred. In order to be consistent the two elements must have the same DOF. For example, consider where shell elements are joined to solid elements along the edge of the solid part as shown below. The shell elements have rotational DOF whereas the solids do not. The unintended result here is that the interface will act as a hinge as the solid element cannot deal with the local rotations on the attached shells. This can easily be fixed by having some overlap between the solids and the shells, or by applying constraints between the nodes on the shells and the nodes on the solids.

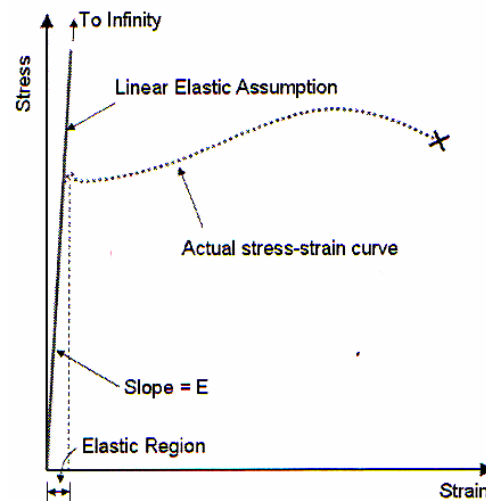


Materials

In addition to modelling the geometry of your FE model you must also define what the material properties of your model are. Materials used in structural codes generally fall into one of the following categories;

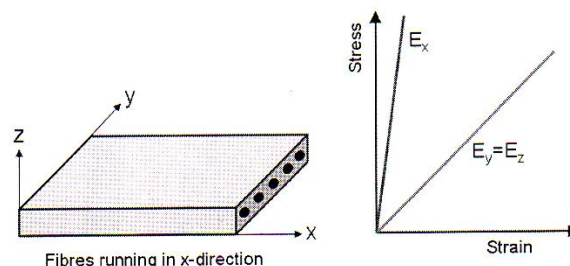
1. Linear Elastic Isotropic

Assumes that Hooke's law holds in all directions. Typically used for modelling metals that are not expected to yield. This assumes that the material behaves linearly regardless of the stress, i.e. for metals this is approximately correct as long as the metal remains BELOW YIELD. When defining a linear elastic material model the parameters needed are the Young's modulus of the material and usually, the Poisson's Ratio and the Density. When using this model, results should be constantly checked to ensure that the yield stress has not been exceeded and that the displacements are small. If this is not the case then a more complex model must be used.



2. Linear Elastic Orthotropic

Assumes that Hooke's law holds in all directions, but allows different values for Young's modulus in different mutually perpendicular directions. Used to model simple reinforced composite materials such as wood where yielding is not expected.



3. Linear Elastic Anisotropic

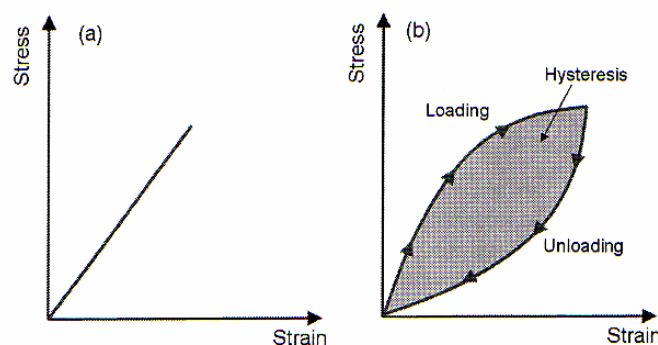
As above but different values for Young's modulus can be defined in each direction (i.e. they do not need to be mutually perpendicular). Used for materials that are highly anisotropic such as bone and certain more complex composite materials.

4. Non-Linear Elastic

Used for representing the behaviour of materials such as rubber, foams and certain polymers that behave in this manner.

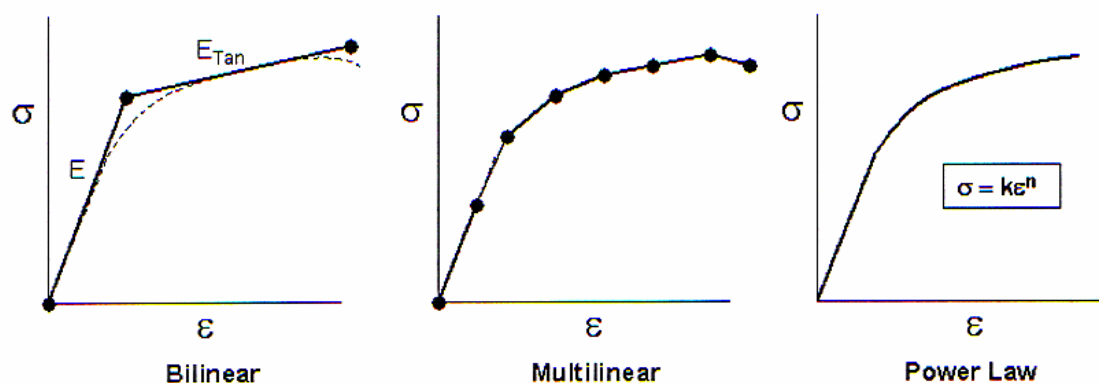
5. Visco-Elastic Models

These materials exhibit Creep, Hysteresis, and stress relaxation. The strain rate is dependent on time, i.e. it will exhibit some creep. Energy will be lost when a load is applied then removed, i.e. hysteresis is observed in the stress-strain curve, with the area of the loop being equal to the energy lost during the loading cycle, and finally if a visco-elastic material is given a prescribed strain, then the stress will decrease with time, i.e. stress relaxation. Generally the creep, stress relaxation and hysteresis properties will all change with temperature.



6. Elasto-Plastic Rate Independent

These materials allow for yielding and subsequent plastic deformation of the material due to the applied loading. The plastic deformation is assumed to be strain rate independent. The most simple material model is the bilinear elasto-plastic material model which uses two lines to approximate the behaviour of the material, one being the Young's modulus, and the other being the tangent modulus. Other models include the multilinear model where several points are plotted to more accurately represent the non-linear part of the curve (typically derived from experimental data), or a power law model may be used;



7. Elasto-Plastic Rate Dependent

As above but now the speed at which the deformation takes place will affect the response of the structure.

8. Specialised Plasticity Models

These are used to model materials that although exhibit plastic behaviour do not conform to the standard models above. Examples include cast iron (which behaves very differently in compression and tension), concrete (which exhibits a non-linear stress-strain curve with strain SOFTENING), soils and rock.

9. Damage Models

These models are used to model damage to composite or reinforced concrete materials, which typically occur due to impact loading and subsequently significantly alter their material properties.

10. Specialised Material Models

Used to model any material which can not be modelled in one of the above categories, e.g. shape memory alloys.

Loading

Once you have modelled your structure you must then apply loading and boundary conditions to your model. These will generally fall into one of the following;

1. Static Point Force

A simple load applied to a point and only acts through this point. This load is applied so slowly that it does not introduce any dynamic effects into the system.

2. Static Distributed Load

This is used to model any static force that is applied over an area. For example a u.d.l. in building structures or aerodynamic loads over an aircraft. This load can vary linearly or non-linearly. Again the load is assumed to be static which significantly simplifies the calculations.

3. Static Pressure Load

This assumes that an applied force is distributed over a certain area of the loaded geometry. This load can vary linearly or non-linearly. This type of loading would be used in examples such as the analysis of pressure vessels, metal forming or underwater structures.

4. Static Inertia Load

This loading refers to applied inertia loads that will not result in motion of the structure under investigation, but will effect the predicted stresses and strains. The main example of this is self weight due to gravity.

5. Time Varying Point Force

As one, but applied at any speed and can vary with time, thus introducing dynamic effects into the system. Since time is now a factor, velocity and acceleration in the system must be considered.

6. Time Varying Distributed Load

As two, but applied at any speed and can vary with time, thus introducing dynamic effects into the system. Since time is now a factor, velocity and acceleration in the system must be considered.

7. Time Varying Pressure Load

As three, but applied at any speed and can vary with time, thus introducing dynamic effects into the system. Since time is now a factor, velocity and acceleration in the system must be considered.

8. Velocity

A velocity can be applied to a whole model, e.g. an initial velocity applied to a vehicle, or part of a model, e.g. a headform.

9. Acceleration

An acceleration may also be applied to a whole model or part of a model.

Boundary Conditions

1. Regions Fixed in all Degrees of Freedom

A part of the model is fixed in all 6 dofs, e.g. the built-in end of a cantilever beam.

2. Regions Fixed in Specific Degrees of Freedom

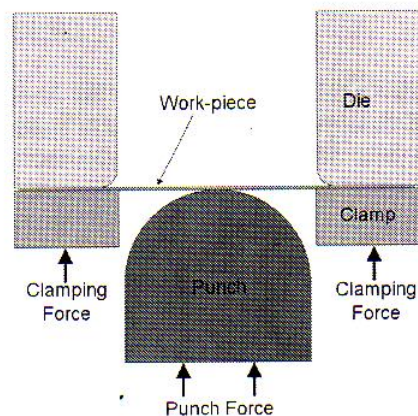
A part which is allowed to move in certain direction, but not others. For example, a pin joint which allows rotation about one axis, but not about any other axis or any translational movement.

3. Prescribed Displacement of Specific Degrees of Freedom

Used when the motion of a particular part of the model is known and will not change. For example, the movement of a die in metal forming.

4. 2D Contact Surfaces

A good example of this is in the analysis of any metal forming process, such as the one shown below;



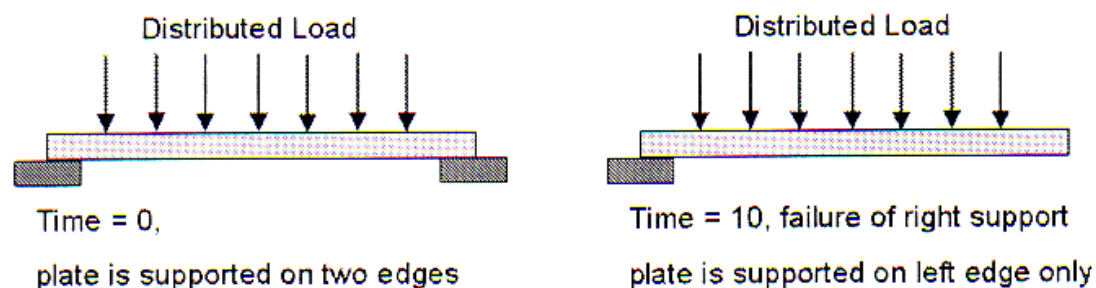
In the above example, contact surfaces would exist between the die and the work-piece, the clamp and the work-piece and the punch and the work-piece. Friction will obviously significantly effect the behaviour of the model so will always be defined.

5. 3D Contact Surfaces

A 3D version of the above where the contact surfaces will be areas rather than lines. For example a vehicle crash model will have many 3D contact surfaces.

6. Time Dependent Boundary Conditions

These boundary conditions are active for only part of the analysis, for example it may be necessary to model an engine mount breaking at a certain point in a vehicle crash, or part of a structure failing after a certain time as shown below;



Note that some FE packages now also have CAD capabilities (or CAD packages have now been developed to have FE capabilities). One aim of doing this is to have a CAD part with a FE model associated with that part. If the CAD part is then updated, then the mesh will automatically update also. If this is the case, then loads and boundary conditions should be associated to the CAD not to the mesh, so that if a node or element moves when the part is remeshed, then the load or boundary condition stays in the correct place.

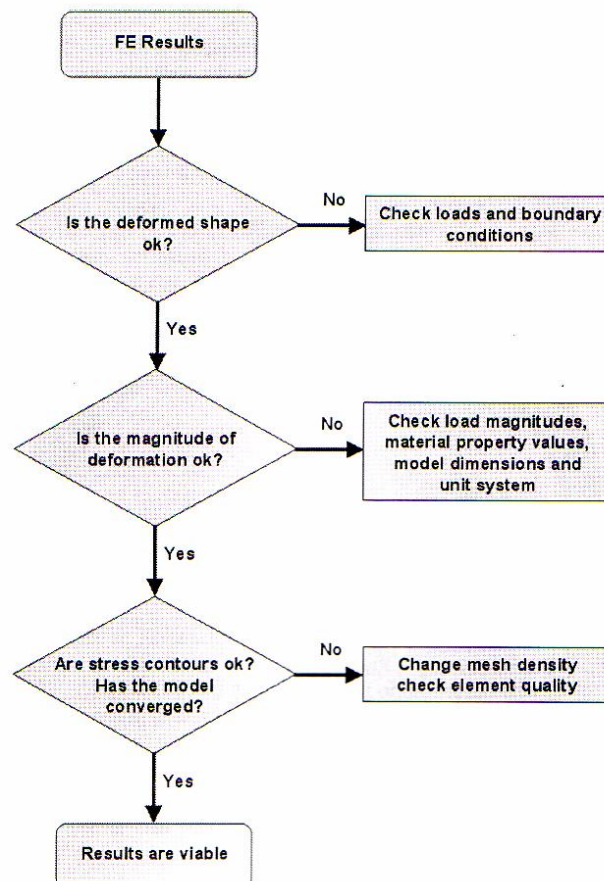
If codes are used that automatically generate a mesh, then great care should be taken as to the quality of that mesh.

Post Processing

Once you have created your model (Pre-Processing) and you have solved (or Processed) your model, you then need to Post-Process your results.

The first step is always to check the deformed shape of the structure. This tells us straight away if the loads have been applied correctly and if the boundary conditions are behaving as intended.

Once the general deformation behaviour has been confirmed, the next step is to check the magnitude of the displacement. If the magnitudes are too high or too low then this tells us that there is a problem with our applied load magnitudes, our model dimensions or our material model definition. The next step is to examine the stress results. By checking contour plots of stress across the elements we can quickly determine how good the mesh we used was. If unacceptable discontinuities in stress contours are found then the model requires re-meshing with a more suitably refined mesh in order to capture the problem behaviour. If the model is re-run, then the displacement and stress results should be compared to the previous run to investigate if the model has converged. If the answer to all of the below is “yes”, then the results are viable – although not necessarily CORRECT as any of the assumptions that you have made when creating your FE model may have altered the results.



When checking the deformation of your model it is important to note that some FE software will automatically scale the deformation of the model in order to aid the user's appreciation of the deformed shape. Consequently the magnitude of maximum deformation should always be checked and, if necessary, the scaling of deformed shape should be reduced/increased or even turned off.

Most modern post-processors offer the facility of animating the model deformation. In a linear static analysis, the post-processor takes advantage of the linear assumption to interpolate the intermediate deformations. However, the solver has simply calculated the final deformed shape.

These notes are adapted from

“Practical Stress Analysis with Finite Elements” – Bryan J Mac Donald