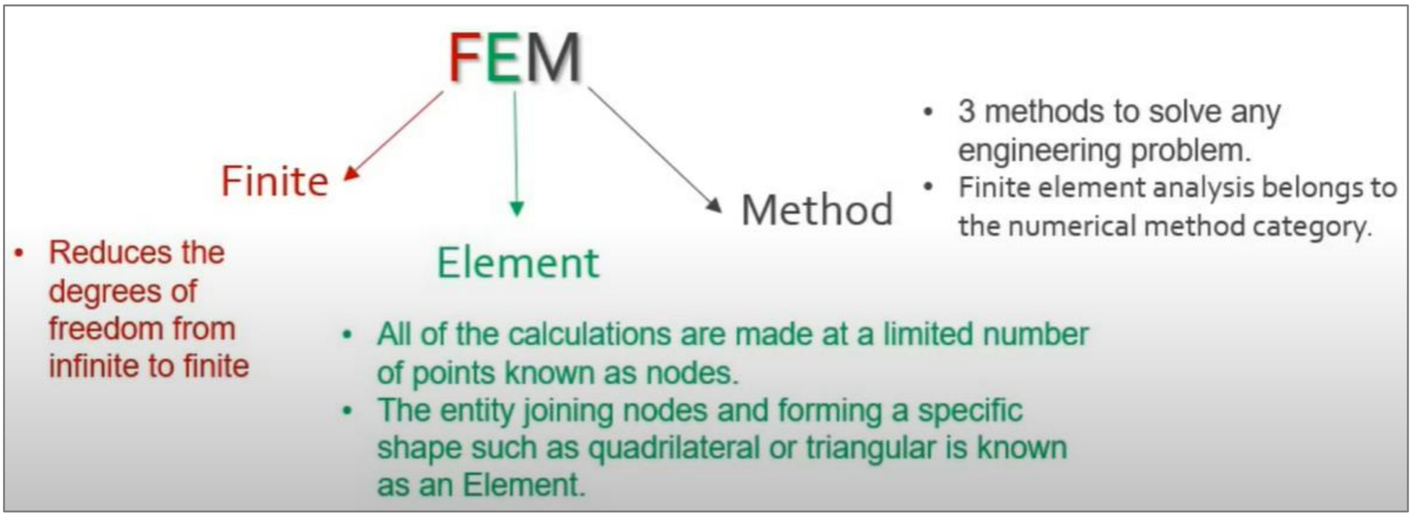
FEA Learning Notes [Sketch]

by André Duarte B. L. Ferreira
First version: 2020-05-05
Last edit: 2023-10-13

Since this isn't for sale and only for learning purposes, (and unless people find this online by chance) for personal purposes, I didn't pay much attention to making references of sources. For that reason several images here are not mine, and I don't take credit for them. And also take note: there probably are errors, as these notes are made as I learn, so please do DYOR. This is a very basic, ELI5-type, introduction to FEA, mostly focused on Static Structural FEA.

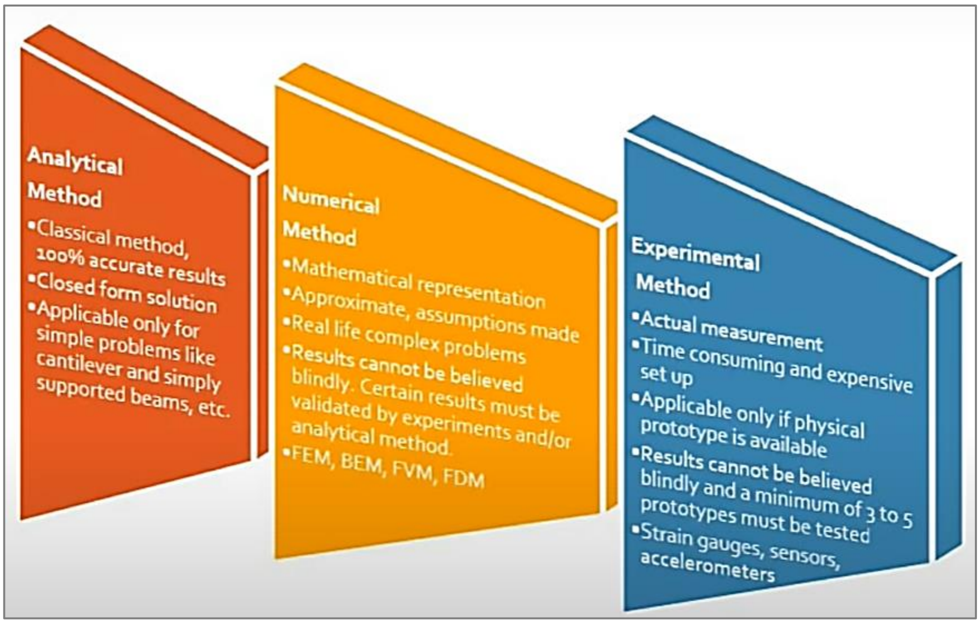
Intro

Finite Element Analysis (FEA) is an analysis that uses the Finite Element Method (FEM) to study a physical (statics, dynamics, thermal, fluids, …) problem. What is the FEM? Basically, it's a method based on the principle that it's harder to calculate 1 complex thing than 5 simple things that resemble that hard thing. For example, it's easy to calculate the deflection of a cantilever beam. But what about a beam that makes an S-shape attached to a C-shaped beam? We don't have a simple equation for that. So what we do is we slice up that complex beam into, say, 100 straight beams (the ones we can easily calculate the deflection for). If we divide it into enough small simple parts, it will look approximately the same as the big complex part. The simple equations for a straight beam will still work and in the end we still manage to get the solution for the complex part. The trouble is that now we need to use that simple equation 100 times for each beam. If there's anything computers excel at is doing simple calculations very fast.



Now, we can apply this logic not only to structural problems but also fluid, magnetic, thermal and electric problems. It's very versatile. But FEM is just one of the methods where we split a complex problem into smaller easier problems. Others are Finite Difference Method (used to solve coupled thermal and Computational Fluid Dynamics, CFD, problems), Finite Volume Method (used in CFD and computation electromagnetics), and the Boundary Element Method (often used in acoustics).

Methods to solve an engineering problem:



Different numerical methods:

1. **Finite Element Method (FEA):**
   - The Finite Element Method (FEM) is a popular numerical technique used to determine the approximated solution for a partial differential equations (PDE).
   - Applications - Linear, nonlinear, buckling, thermal, dynamic and fatigue analysis, etc.
2. **Boundary Element Method (BEM):**
   - Powerful and efficient technique to solve acoustics or NVH problems.
   - Just like FEA, it also requires nodes and elements, but it only considers the outer boundary of the domain.
   - So when the problem is of a volume, only the outer surfaces are considered. If the domain is of an area, then only the outer periphery is considered.
3. **Finite Volume Method (FVM):**
   - FVM method representing and evaluating partial differential equations as algebraic equations method is used in many computational fluid dynamics packages.
   - CFD (Computational Fluid Dynamics) and Computational Electromagnetics
4. **Finite Difference Method (FDM):**
   - It uses Taylor's series to convert a differential equation to an algebraic equation. In the conversion process, higher order terms are neglected.
   - It is used in combination with BEM or FVM to solve thermal and CFD coupled problems.

Types of analysis:

Static Analysis	Dynamic Vibration Analysis	Explicit Dynamic Analysis	Fatigue / Durability	Thermal Analysis	Optimization	High End Analysis	Computational Fluid Dynamics
•Linear Analysis •Non Linear Analysis •Quasi-Static Analysis •Contact Bolt-joint Simulations •Leakage, Interference Analysis	•Modal Analysis •Transient Dynamic Analysis •Frequency Response Analysis •Spectrum Analysis •Multi Body Dynamics	•Impact Analysis •Drop Test •Can Crush •Projectile Impact •Car Crash Analysis	•Stress-Life Approach •Strain-Life Approach •ASME standard Fatigue Analysis •DNV standard Fatigue Analysis	•Steady State Thermal Analysis •Transient Thermal Analysis •Coupled / Thermal Stress Analysis •Transient Weld Analysis	•Topology Optimization •Size and Shape Optimization •Topography Optimization	•Mold Flow Analysis •Multi-physics analysis •Manufacturing Simulations (Weld analysis) •NVH	•Pressure drop ration & Flow characteristic •Cavitation study •Thermal, transient flow analysis •Laminar turbulent flow •Flow coefficient & pressure recovery

CAD Software's

Commercials:

- AutoCAD/Inventor (Autodesk)
- Creo/Pro-Engineer(PTC)
- Unigraphics/Solid Edge (Siemens)
- CATIA/Solidworks/Draftsight (Dassault systems)
- Spaceclaim/Design Modeler (ANSYS)
- MicroStation(Bentley)
- TurboCAD Pro
- Onshape

Freeware and open source:

- 123D
- LibreCAD
- FreeCAD
- BRL-CAD
- OpenSCAD
- QCad
- SolveSpace

CAE Software's

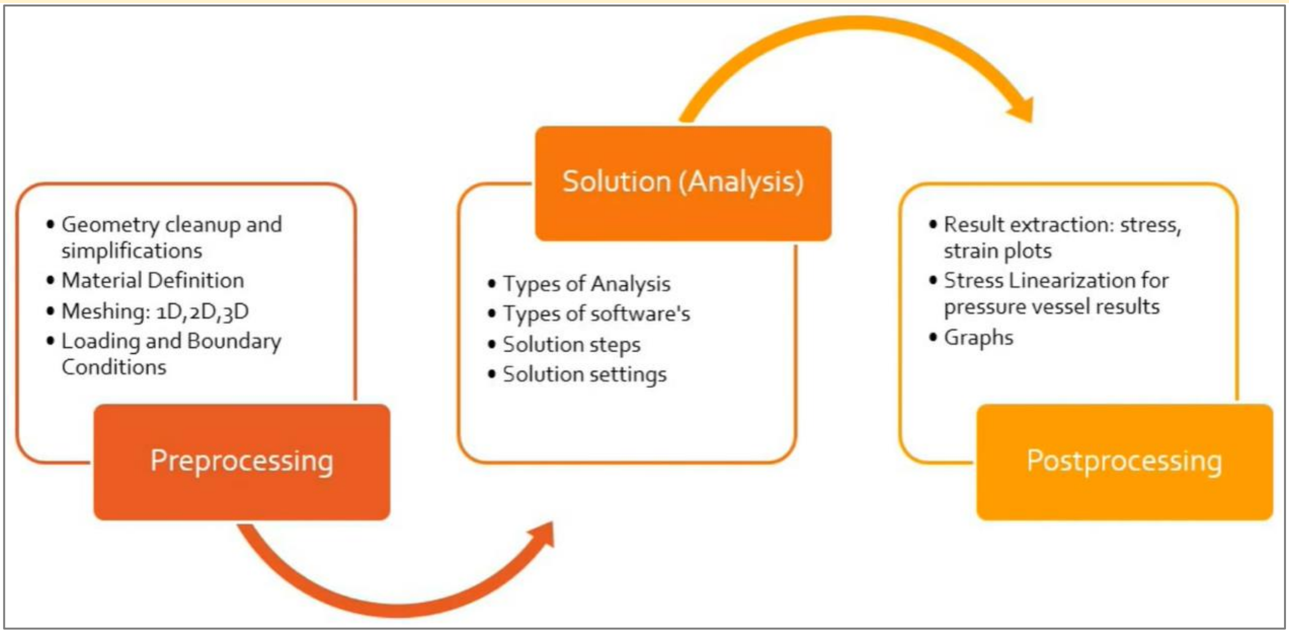
Commercials

- ANSYS/Fluent/CFX
- Abaqus
- Nastran/Patran
- Hypermesh
- Radioss
- LS-Dyna
- Simufact Welding
- Autodesk CFD/Moldflow
- Modex3D
- ANSA
- COMSOL Multiphysics

Freeware and open source:

- CalculiX
- JCMsuite
- ANSYS (student version, 32000 nodes)
- FEBio
- FeatFlow
- SimScale
- Z88/Z88Aurora
- Fidesys (Academic version)
- VisualFEA (Educational version)
- CAEplex (Free Plan Available)

FEA Process Flow



Pre-Processing

Before the software initiates calculation of our values of interest, the software and us have to prepare everything so that it goes smoothly and correctly. This includes:

- Define analysis type
    - static / dynamic
    - linear / nonlinear
- Geometry cleanup and simplifications – many times it is even better to make a new CAD model just for FEA purposes;
- Applying materials and their properties (elastic, elasto-plastic, creep, viscoelastic, anisotropic).
- Define boundary conditions
- Define loads

- Define other constraints such as contacts and movements;
- Defining the mesh and geometric properties of the problem.
    - Types of elements – 2D (tri or quad) 3D (tet or hex);
    - Order of elements – most come in linear (aka first-order) (tri3, tet4, quad4, hex8) or quadratic (aka second-order) form (tri6, quad8, tet10, hex20);
    - Type of integration – full or reduced.
- Definition of symmetries and other simplifications;
- Definition of the “solver” and other solution techniques to use.

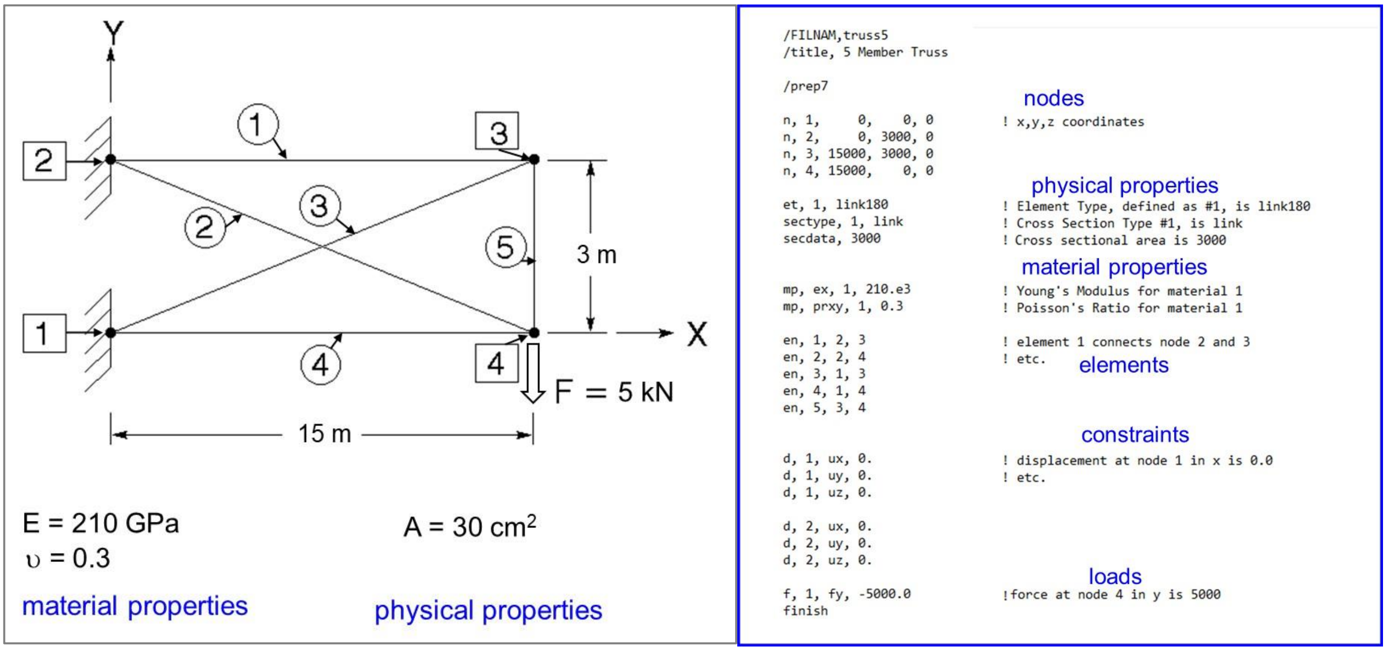


Figure 1 Example of a truss with 4 nodes modeled with truss elements and the text input file that the solver of the FEM software gets to work with. Note how all the units had to be converted to be consistent. That means – everything in mm, mm^2 and N.

Solution Analysis

This is where the CPU gets heated. Where what we want to discover is calculated. That’s why all the processing before is ‘pre’ and after is ‘post’. The thousands of equations are solved, special solution techniques may be used depending on what the user chose for example to save time at the cost of accuracy.

FEM Calculation sequence:

(Very simplified, only to get a “feeling” and broad brush of what’s going on)

Each node contributes a certain part of the stiffness to the element which when we combine (via the element shape functions) to get a stiffness matrix for each element that has those nodes (for structural analysis) for each of their DOF in their local coordinates. The software combines all of them into a big stiffness matrix in the global coordinate system with all elements, their DOF and their stiffnesses. There are intermediate steps for example the numerical integration to obtain stiffness in the volume of the element calculated which also use the integration points.

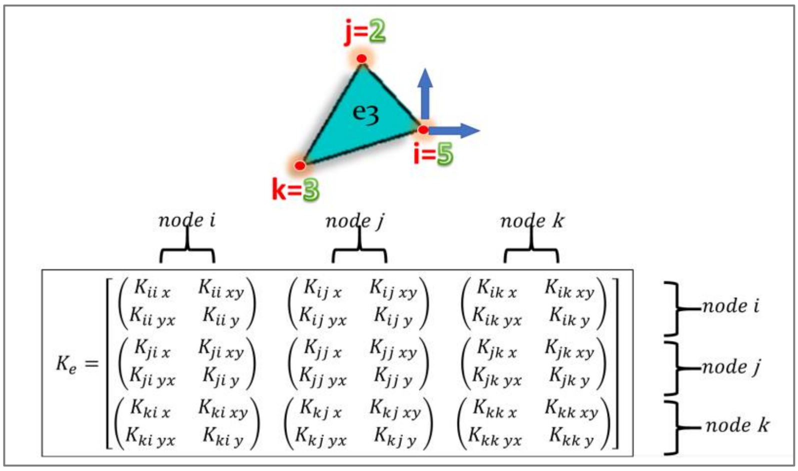


Figure 2 *Stiffness matrix of the element e3 composed by the combination of the contributions of the stiffnesses from its 3 nodes.*

Having all boundary conditions applied (reduction in DOF due to constraints + forces for structural analysis) the displacements for each node (and reaction forces where applicable) are calculated. So $\mathbf{F} = \mathbf{K} \times \mathbf{d}$. Therefore $\mathbf{d} = \frac{\mathbf{F}}{\mathbf{K}}$ or $\mathbf{d} = \mathbf{K}^{-1} \times \mathbf{F}$. This means the system inverts the global system matrix to multiply by the force vector and then get the displacement vector, the vector with the displacements of each node. Nowadays due to the size of the global matrixes the system doesn't do inversion, but numerical integration to get the stiffness matrix. These forces, placed in a force vector, are for each node and each of the nodes' DoF.

Then to get the displacements inside the element we use again the shape functions to interpolate the values from the nodes to the inside.

To have a stress we need a delta displacement (or alternatively calculate the internal forces and with $\sigma = \frac{F}{A}$ get the stresses). So firstly, the strains are calculated. The stress between two nodes is related to how much the delta displacement or between them is or the strain. Therefore, we don't calculate the stress at nodes but inside the elements (at the integration points). From here these stresses are extrapolated to nodes. Because extrapolations incur error, the stresses and strains are the most accurate at integration points.

This means each node will have as many different stress values as many elements share that node. This is where the function **stress averaging** comes from. This is done just to have a smooth stress plot, but often you can turn this off just to see the unaveraged stress values.

Post-Processing

After the "juice" is gotten all the processing that follows is called "post". This includes:

- Plotting of graphs
- Calculation of factors of safety
- Display of graphics
- Execution of animations
- Plot deformed structure
- …

General

Pre-simulation

Pre-simulations are quick, draft simulations you do to get some results asap. The mesh is coarse and not much attention is paid to its quality. You do things approximately. The point is to get some quick results so we know the general direction of where the simulation may be going. This can also help acknowledge important considerations for further pre-simulations and simulations.

FEA quality

Draft, minimum viable FEA

- Quick FE analysis. Often using the CAD software used for the designs.
- Quick iteration between FEA and CAD. Use feedback from FEA into CAD and vice-versa.
- Getting approximate solutions based on having the right fundamentals of meshing, loading, and boundary conditions rightly setup, without going into too much detail.
- Using approximate material properties.
- Implementing conservative performance criteria margins.
- Creating an informal analysis report as an option.

Certifying FEA

- FE analysis will probably take life as a project on its own with realistic longer timeframe.
- CAD team and CAE team probably different. Close collaboration between two will be needed. CAE team to obtain CAD data, understand design intent, and clarify engineering assumptions.
- Applying a comprehensive range of advanced FEA techniques, high quality meshes, super-elements, among others.
- Ensuring verification and validation at each phase of the analysis.
- Delivering accurate results backed up with convergence analysis, accurate material data, clearly specified loading conditions, and well-defined boundary conditions.
- Incorporating regulatory safety margins into performance criteria.
- Generating a formal report.

Simplification of the model to reduce computational cost

- Use symmetry - reduce the number of elements and number of DoF in the mesh --> lower computation cost. For example something that we do as axisymmetric can't have rigid body motion (RBM) in the radial direction.
- Types of symmetry: mirror, axisymmetry (= something that can be done with revolution), rotation, pattern.
- This also includes noticing repetitiveness and doing the simulation only for it
- Defeaturing – See whether you can simplify the model. Whether certain details (which would require smaller or adapted mesh) can be simplified/removed with minimal impact on the results.
    - Remember the principle of St. Venant. Things that are far away enough may not impact our points of interest and therefore be removed.

  - See where the load path is. Things outside of the load path may affect the results very little and therefore be simplified.
  - See what are the points of interest.
- Exclude areas of low stress from the simulation.

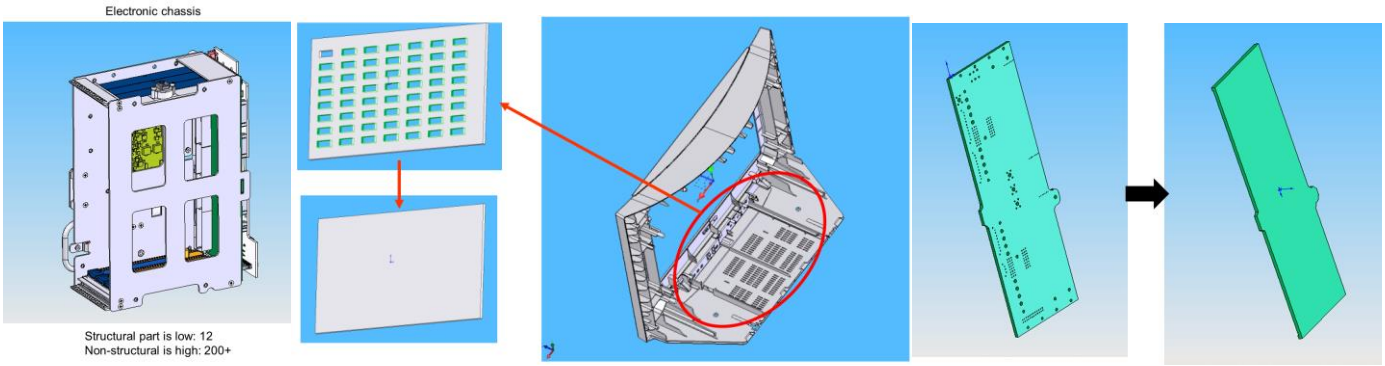


Figure 3 *Defeature examples*

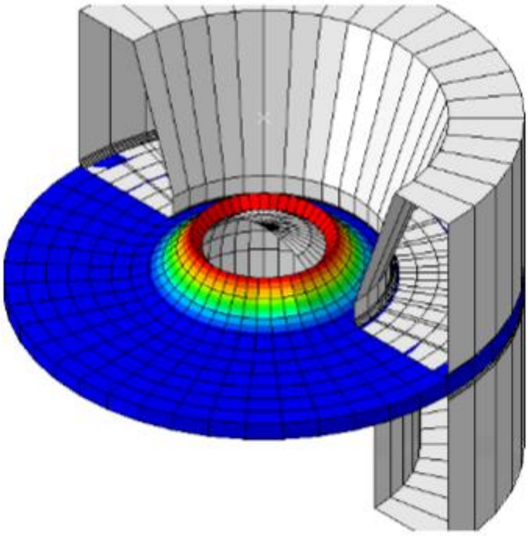


Figure 4 *Nonlinear analysis of the plastic expansion of a hole of a sheet metal circular plate using symmetry condition for the die and punch, and 2D elements [Source].*

Flying structure and rigid body motion

- Mechanism = freedom with zero stiffness - movements that do not produce strain.
- The FEM only looks at the stiffness matrix to see if a system is stable. If there are unconstrained DoF then they will be part of the solver of equations. The discretized geometry is not exact, there is numerical roundoff, etc. So the FEM sees these small imbalances as an ill conditioned matrix (i.e. RBM).
- Imagine FE analysis of scuba tank. How do we constrain the model? We cannot run a static FE analysis with just a load balance, we need a constraint set that will remove all rigid body motion and ground the structure properly. We need something that will ground the model, but not introduce unwanted load paths and incorrect stress distributions.
- Basically you don't have $\Sigma F = 0$.
- Minimum fixings
  - 1D and 2D axysymmetric - 1

    - 2D plane - 3
    - 3D - 6
- Solutions
    - 3-2-1 method
    - Weak springs - need to add enough to prevent DoF that would induce the rigid body movement. Also need to check the reaction forces on the springs after simulation to make sure they are negligible and therefore not affecting the rest of the results.
    - Inertia relief - assumes that the out of balance forces will be absorbed by inertia (mass of the body). Use this only when body has enough mass compared to forces, that these out of balance forces won't cause much RBM.

Consistent units

When software is not smart enough that we can specify units for everything, we need to be consistent. If we place stress as MPa (=N/mm^2), then we must use mm (not eg. m) and N (not eg. kN).

Also remember F = ML/T^2. Force units we usually set with young modulus. So then if we define E as N/mm^2, and then either mass or time, we are defining the other. If we put second as unit of time, then mass has to be tonne because 1 N = 1 ton * 1 mm / s^2 (1 newton applied on a ton makes it accelerate at 1mm/s^2)

Sources of error inherent in FEA

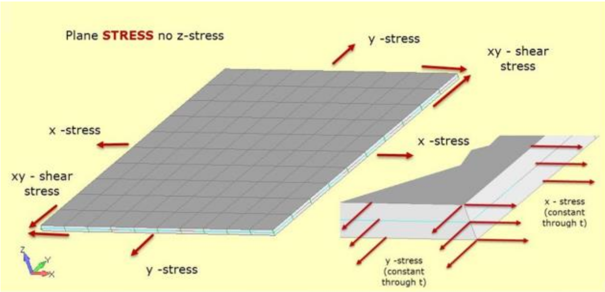
- Geometry + Mesh (geometry not well represented by mesh, eg linear element used to represent curve)
- Mesh (element distribution - not refined where it needs to eg. in stress concentration, low element quality (very distorted shapes, mesh errors like wrong element normals, free faces and edges
- Material properties not good enough (especially important for non linear material problems)
- Boundary conditions not acc to real life
- RBM allowed
- Loading not acc to real life
- Numerical integration (eg reduced integration may cause errors. Reduced integration uses a lesser number of Gaussian co-ordinates (the ones inside the element) when solving the integral. The more Gaussian co-ordinates you have for each element, the more accurate your answer will be, but this has to be weighed up against the cost of computation time.)
- Rounding errors. Due to memory limitations, numbers can't go to infinitesimal. So this can pile up. Occurs more when there's big and small numbers (1E10; 1E-10) eg which occurs in joints of materials with very different Young modulus, and also in harsh element size transitions and with many elements (more algebraic operations)
- Errors due to singularities (point loads, point boundaries, sharp corners)
- Due to how stresses are calculated from displacements there can be jumps between elements.

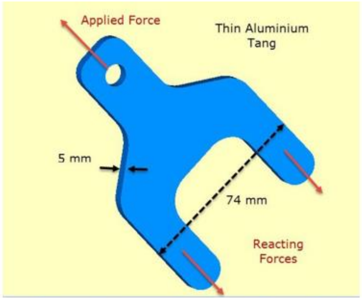
Plane stress and plane strain

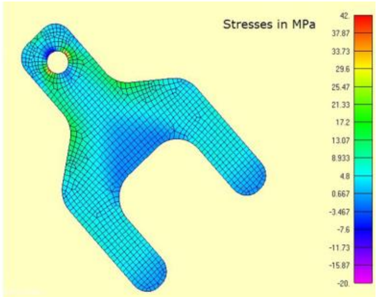
Plane stress

- Used mostly for very thin objects.
- Stress is assumed to exist only in the plane, out of plane are rounded off to zero.

- Plastic parts with thin walls often suffer from this stress state, where σ3 <<< σ1, σ2. Only a tiny fraction of the stresses acting parallel to the surface are developed in the thickness direction.

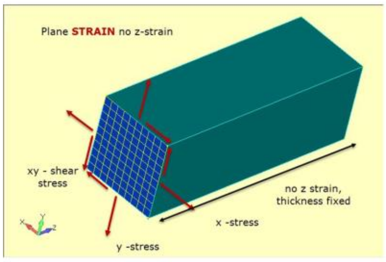


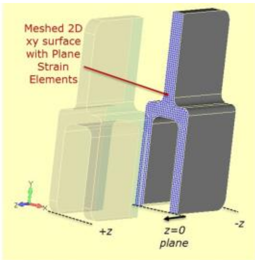


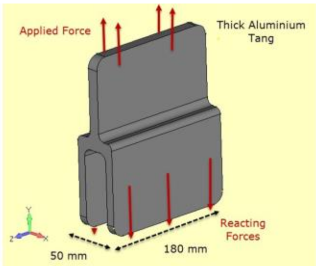


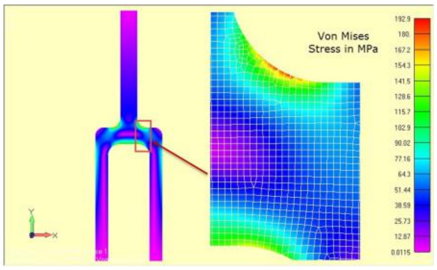
Plane strain

- It's an exact situation that occurs. Used for thick objects where through thickness (aka out of plane) strain is close to or zero. Infinite thickness, strain tends to zero.









Nonlinear Analysis

In real life not many things are perfectly linear. Technically we should always do nonlinear analysis. But we don't because it takes way more computing power, and makes it more difficult to calculate analytically. So we simplify an assume linearity for many situations, where the difference to reality is not big enough to warrant such cost. For example, we make calculations based on undeformed shapes. However, at some point the nonlinearities become too much to ignore. The error too big to discard. Then we are recommended to do nonlinear analysis.

In a static non linear analysis, we correctly take into account the stiffness matrix (**K**) changes with the solution. (In a linear analysis we calculate **K** at the start, and calculate one solution, done). That is, **K** is a function of the displacements. What this means is that when we're doing a nonlinear analysis, when the FEA software achieves a solution, it will recalculate **K** based on it, and re-run for a solution based on this new **K**. This process stops either based on convergence or based on what we decided (for example limit deformation to a certain value). For this reason alone, nonlinear FEA is more computationally costly than linear. Non-linear analysis is therefore an iterative process.

There are 3 types of non-linearities in Statics FEA:

Geometric nonlinearity: occurs when the magnitude of changes to the geometry affect its response. When so called large deformations occur which affect for example the load itself. See example **Figure 5**. Another example is buckling.

In a geometrically linear setting, the equations of equilibrium are formulated in the undeformed state and are not updated with the deformation. In most engineering problems, the deformations are so small that the deviation from the original geometry is not perceptible. The small error introduced by ignoring the deformations does not warrant the added mathematical complexity generated by a more sophisticated theory.

When we do analytical napkin calculations we are assuming linearity. This is kind of quasi-static (that we are moving things very little (less than 5%) and very slowly). We assume things aren't deforming that much (less than 5%). The linear FE models require lesser computational resources and are quicker.

With large deformations, we need to consider whether the load changes.

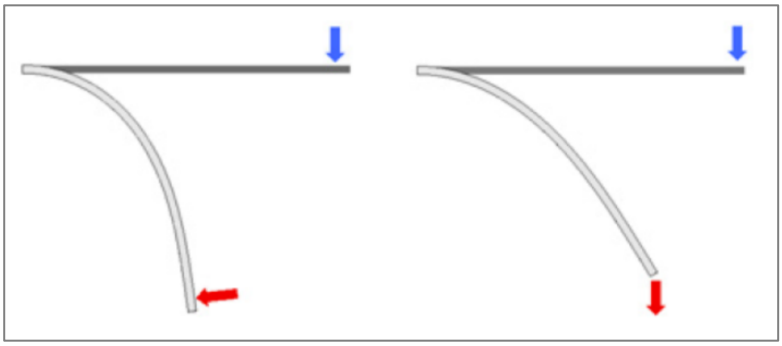
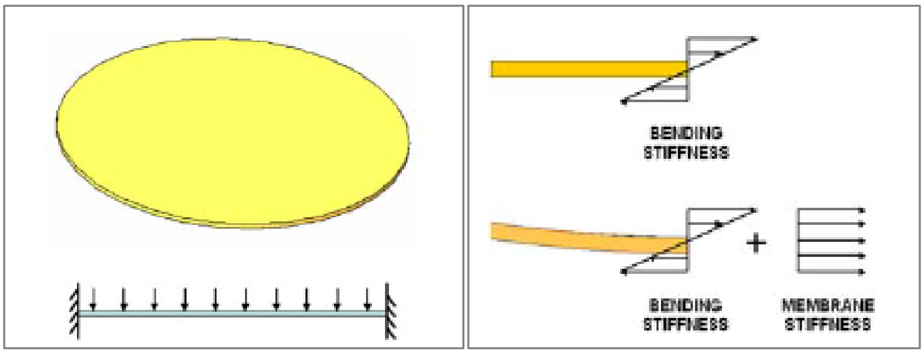


Figure 5 *Non-linear loading on the left. As the deformation increases, so does the load direction. This being the case, the solver needs to calculate deflection, update the loading, calculate deflection again, new loading etc (until we tell it to stop). Right side is linear loading. Does not change with displacement.*

Another typical example of a nonlinear situation is when we have a membrane where we put enough pressure that it deforms to the point where not only is there bending but membrane tension: the in-plane stiffness comes into play. Initially, the membrane resists the pressure load only with bending stiffness. After the pressure load has caused some curvature, the deformed membrane exhibits stiffness additional to the original bending stiffness. The deformation changes the membrane stiffness so that the deformed membrane is much stiffer than the flat membrane. This therefore is non-linear because at a certain point the solver needs to update the stiffness matrix to take into account membrane stiffness besides bending stiffness.



Material nonlinearity: plastic region of the stress-strain curve. The Young's modulus is not a neat constant value, but starts changing depending on deformation. This means the material properties (this one at least) is a function of displacement.

Contact nonlinearity: occurs when due to one of the parts in contact pushing or pulling leads to a change in geometry that warrants change in **K** or the forces between the parts in contact, forcing another iteration on approaching the solution.

In a simulation where we have large deformations/deflection it may become non-linear. Some software have an option that you can turn that on in the analysis settings for large deflections so that the software updates the model during the simulation. This will give more accurate solutions (and in fact even when the deformations are small), but takes more computing time. To evaluate the need to include geometric nonlinearity try to run the simulation with it on and off. A substantial difference indicates the need for the account of geometric nonlinearity.

Mesh

A finite element mesh is defined by a set of nodes together with a set of finite elements.

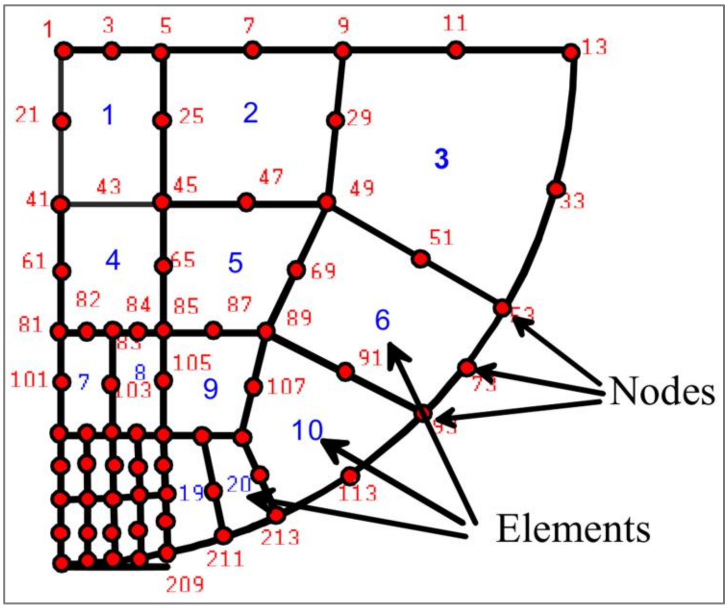
Nodes: The nodes are a set of discrete points within the solid body. Nodes have the following properties:

1. A node number. Every node is assigned an integer number, which is used to identify the node. Any convenient numbering scheme may be selected.

2. Nodal coordinates. Each node is assigned a set of coordinates, which specifies the position of the node in the undeformed solid.

3. Nodal displacements. When the solid deforms, each node moves to a new position. For a three dimensional finite element analysis, the nodal displacements specify the three components of the displacement field u(x) at each node.

For a two dimensional analysis, each node has two displacement components. The nodal displacements are unknown at the start of the analysis, and are computed by the finite element program.

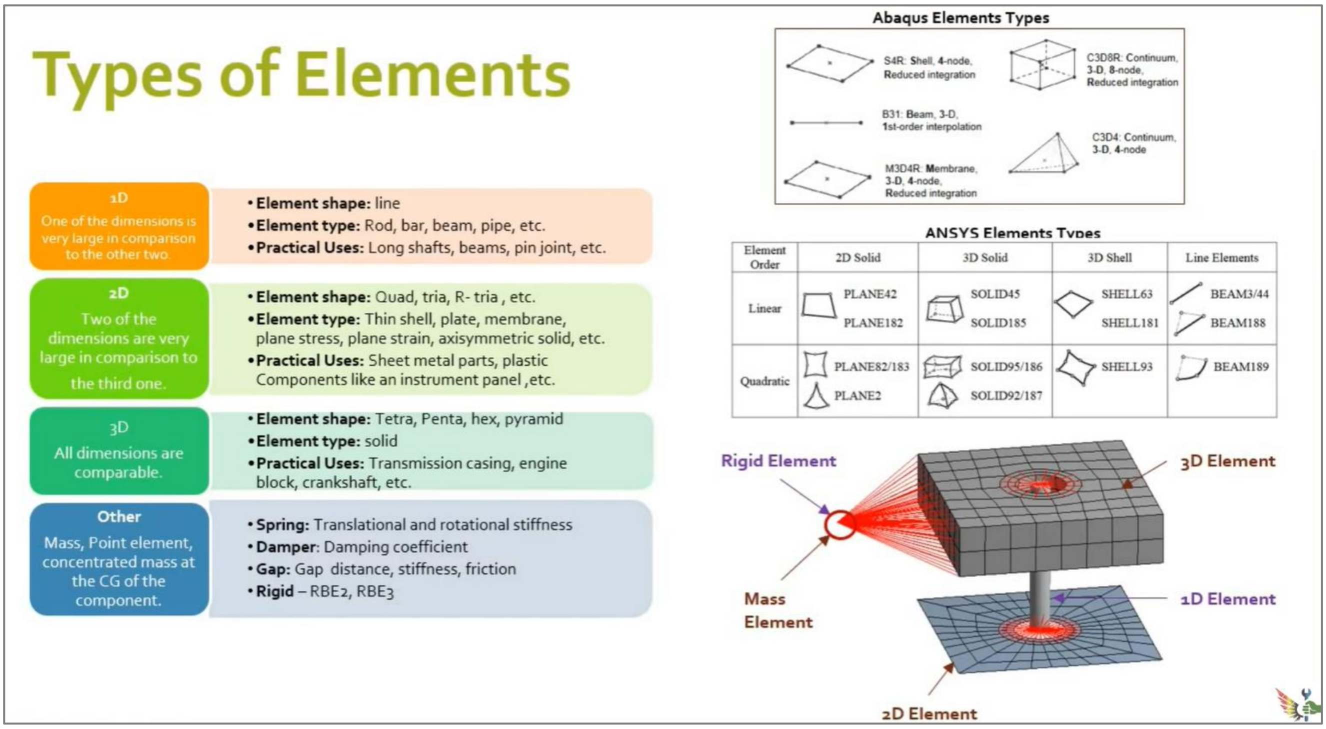


Element types

Meshing a model is simply taking a complex part or assembly and filling it or dividing it into simple lines, surfaces or volumes called elements. A continuous system with infinite degrees of freedom becomes a system with finite DOF.

This is the general process of decisions you will have to go through:

1. Decide which dimension of element to use (1D, 2D, 3D);
2. Decide which type of element to use (triangular/tetrahedral or quadrangular/hexahedral);
3. Decide the order of the element to use;
4. Decide the type of integration to use.



Degrees of freedom

In a 1 dimensional space only 1 dimension exists. Therefore only 1D elements can be used. Only 1 DoF exists per node: translation in the only dimension.

In 2D space, we can use both elements with 1 dimension and elements with 2 dimensions. Only 3 DoF exist per node: translations in the 2 dimensions and 1 in plane rotation.

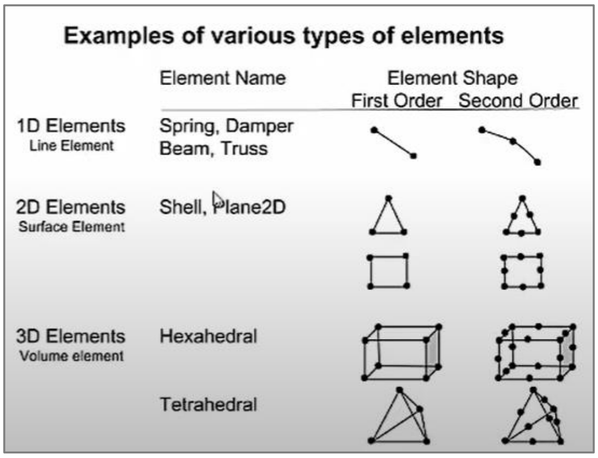
In 3D space we can use all elements. Each node can have 6 DoF, 3 translations and 3 rotations.

Structural DOF - Each node has as many DoF as many movements are possible. This means:

- 1 is a 1D space (1 translation)
- 3 in a 2D space (2 translation, 1 rotation)
- 6 in a 3D space (3 translations, 3 rotations)
- Sometimes, depending on element type it's unnecessary to have all DoF in the nodes. The best example comes from 3D elements, that usually have only 3DoF (3 translations) in each node. They have more nodes per element, and this allows them to "capture" more complex behaviors with less DoF per node.
- We can have 1D and 2D elements in 2D analysis, and 1-3D elements in 3D analysis. In 2D analysis therefore only 3DOF are possible.

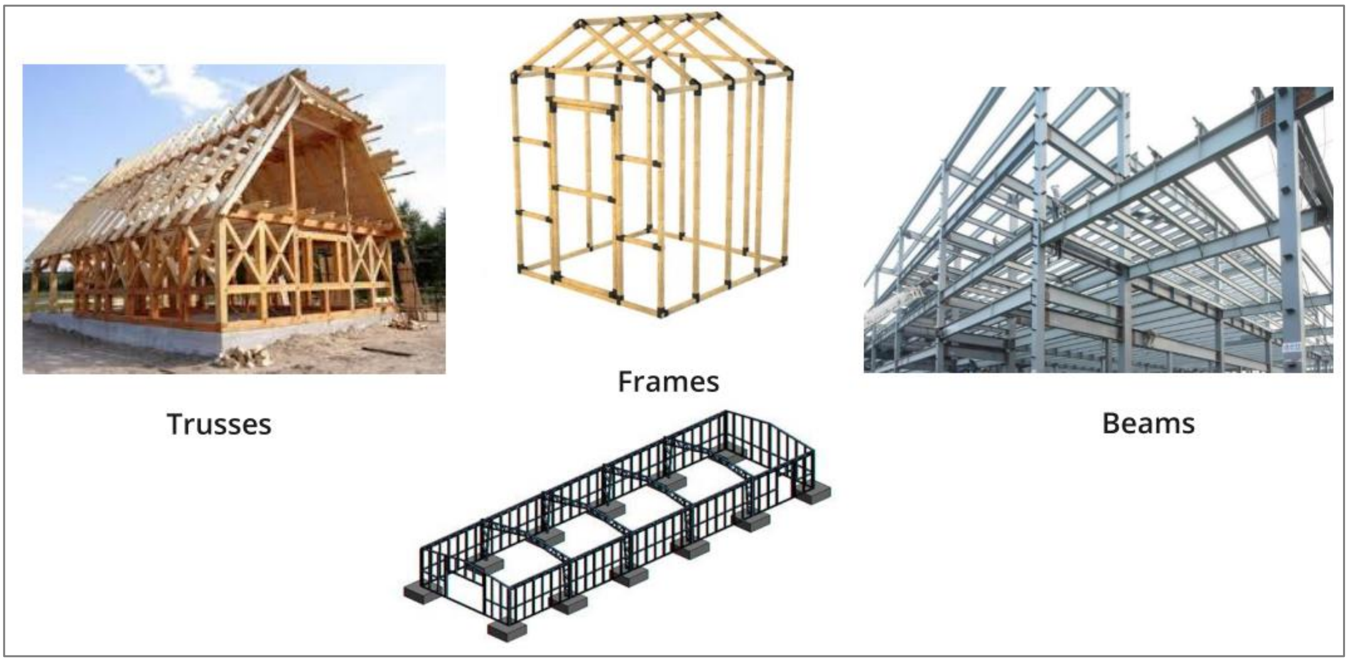
- 1D Elements in 3D space (1 dimension much greater than the other 2)
    - Beam
        - bending, shear, axial
        - 6 DOF
    - Truss
        - only transmit axial forces [they rotate freely on the nodes, they take no moments]
        - 3 DOF, only translations
- 2D Elements in 3D space (2 dimensions much greater than the 3rd)
    - Pure 2D element
        - Only 2D stress, thickness is not modelled
    - Membrane (thin membranes like fabric, thin metal shells)
        - Only has in-plane stiffness (no flexural stiffness, can't resist moments)
    - Plate (pressure vessels, aircraft fuselages, automobile bodies, ...)
        - Resists bending, has flexural stiffness. Plates are designed only to take transverse loading (loading perpendicular to the plane of the element). Think of a bookshelf: this would be a plate structure. A plate structure is one in which curvilinear geometry cannot be accurately captured i.e. use these elements for "flat" structures only. Curved geometry would look massively faceted using plates. If you're using something like ABAQUS, you wont get the choice of using plates, so just go for something like the S4R (say) which doesn't really care (up to a point) whether your structure is "thin" or "thick", as it will adapt as required.
    - Shell = Plate + Membrane
        - Shell triangle (3 nodes (1st order), 6 nodes (2nd order), ...)
        - Shell rectangle (4 nodes (1st order), 8 nodes (2nd order), ...)
        - Rule of thumb to use plate/shell when smallest side dimension is 15x larger than thickness.
        - Thin plates/shell when transverse shear is of no importance
        - Thick plates/shells when it is.
- 3D Elements
    - Tetrahedral (triangular pyramid - 4 nodes)
    - Pentahedral (quadrangular pyramid - 5 nodes, and wedge - 6 nodes)
    - Hexahedral

There's over 200 different types of elements.



Line elements

There are many real world objects that can be modelled as line bodies. As long a structure, or part of it, has small lateral dimensions and uniform cross section, it is a good candidate to being modelled as a line body.



The advantages of using line bodies are similar to those of using surface bodies, but even greater. Engineers should consider using them whenever possible over solid models, and even over surface models.

A truss is defined as a structure consisting of two-force members. By two-force member I mean that the members are pin-jointed at the ends and the loads apply on the joints so that the members are either stretched or compressed but not bent. Two members connected by a pin-joint can rotate about the axis independently. In reality, structural members are rarely connected by a pin-joint. Modern structures are constructed using either welds or multiple bolt-and-nuts. The members are rigid jointed, not pin-jointed. Main reason of pin-joint assumption is computational difficulty.

How much error is caused by the pin-joint assumption? The amount of error depends on the slenderness of the structural member. If the members are slender enough there is no substantial difference between the two.

If in real life members of truss do not rotate about supports, then why do we idealize the joints as pin joints?

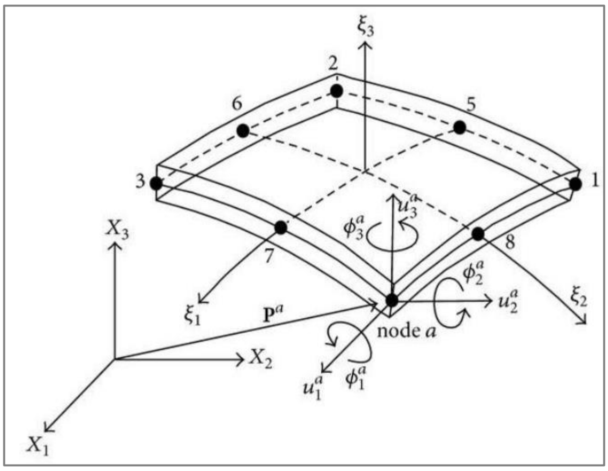
In a frame structure of building, we assume one end of columns as hinged. Actually, it is made of cement and steel monolithically. Then how it will rotate? Actually, our idea is- there should be no moment at that point or moment is zero.

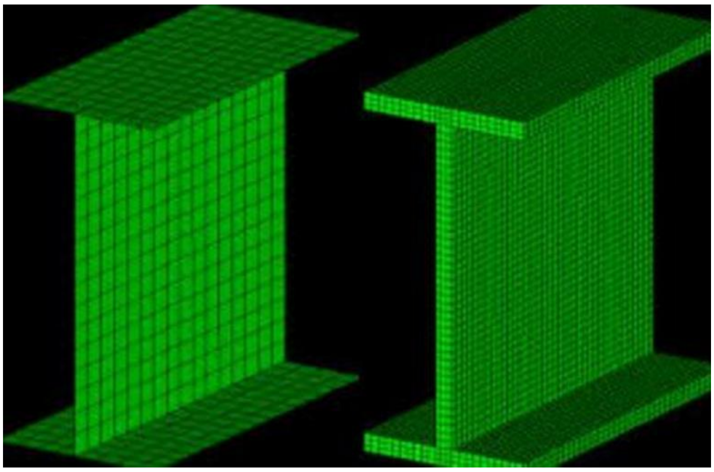
When using Line Bodies, mesh with a very large element size so that you get only an element per beam allowing you to get a solution equal to theoretical values

Shell Elements

A shell element is a planar (2D) element that can be arranged in the 3D space. It is used to mesh a body when one of its dimensions is much smaller than the other two. Each node has 6 degrees of freedom: 3

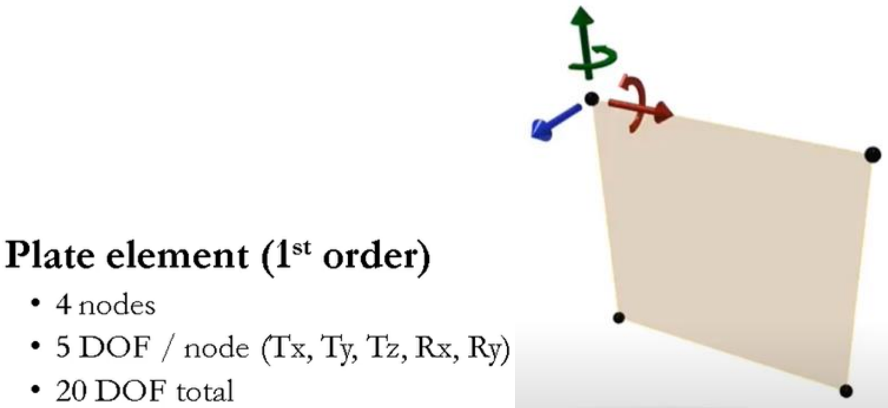
translational and 3 rotational. Due to the presence of the rotational DOF it is very efficient to model the problems dominated by out-of-plane bending modes, contrasting to a solid element, which does not have rotational degrees of freedom.



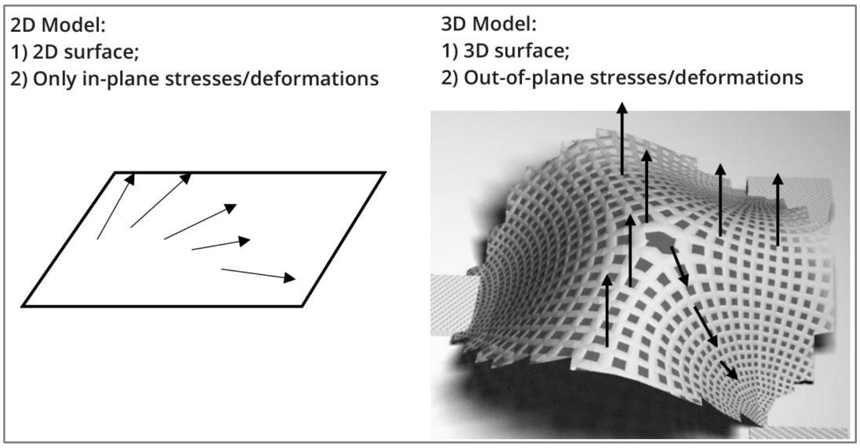


The essential difference between shell elements and 2D solid elements is that shell elements can have out-of-plane deformation (warpage) while solid 2D elements cannot.

Plate elements are similar to shell elements, but plate elements don't have curvature while shell elements do.



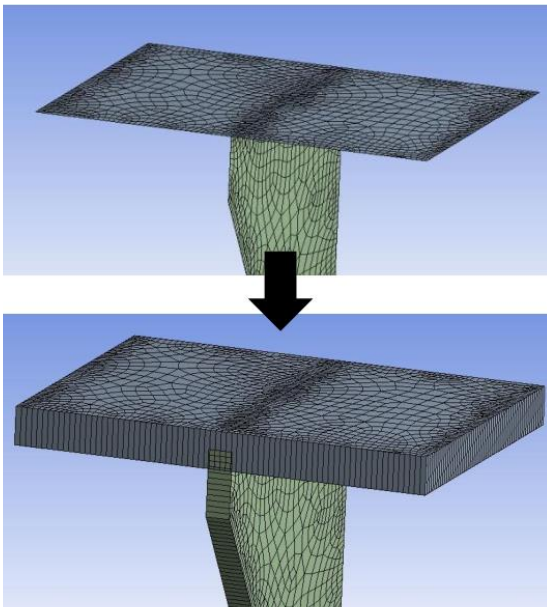
You should use a 2D model whenever possible (as well as symmetry) as it saves on processing power and improves accuracy. For the same processing time we can have more nodes/elements, and the more elements, the better the results. Remember that the difference between a 2D model and a surface, is that a 2D model doesn't have out-of-plane bending/stresses/...

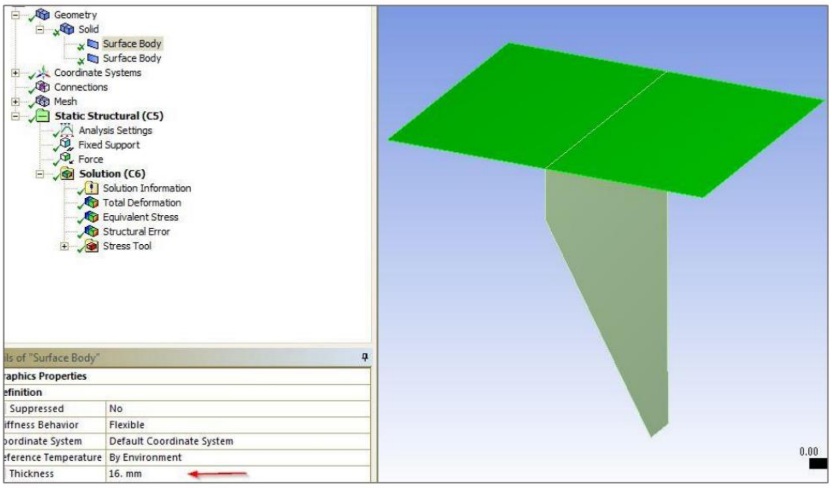


When a real world body is thin and subject to bending, it is generally a good candidate for a 3D surface body. ANSYS will mesh surface bodies with shell elements. There are many advantages of using surface models.

1. Creating surface models is usually easier than creating solid models.
2. The problem becomes simpler to solve for ANSYS compared to using solid models, due to the efficiency of shell elements. This results in
    a. a much faster computation of the result
    b. more accurate results

Consider using surface models over solid models whenever possible. In the old days surface models were visually weird because they had 0 thickness. Now ANSYS can render thickness in order to the engineer to visually better imagine what they're working with.





By using the surface model instead of the solid model we were able to achieve very similar results. However the surface model only consisted of 1000 nodes while the solid model consisted of 6500 nodes. Before using a solid model, consider the possibility of using a surface/line model first.

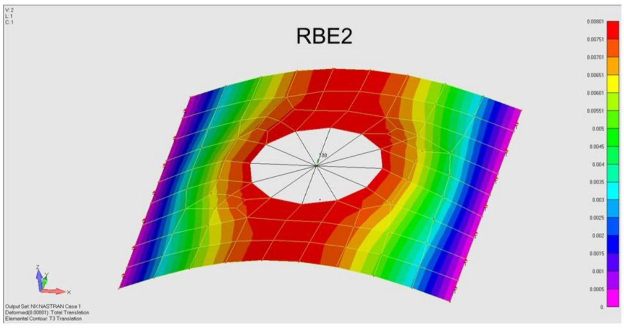
Solid elements

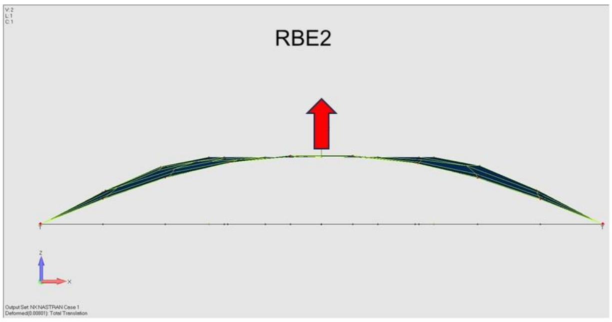
Others

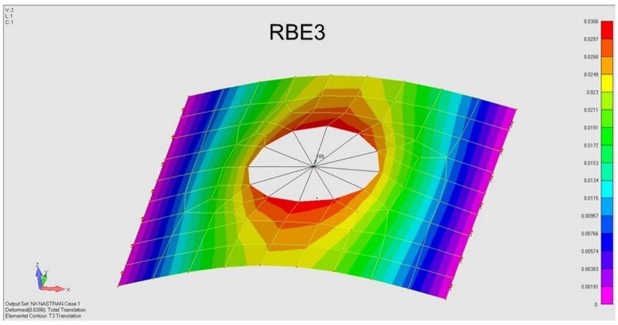
Spider elements (RBE2 rigid and RBE3 flexible)

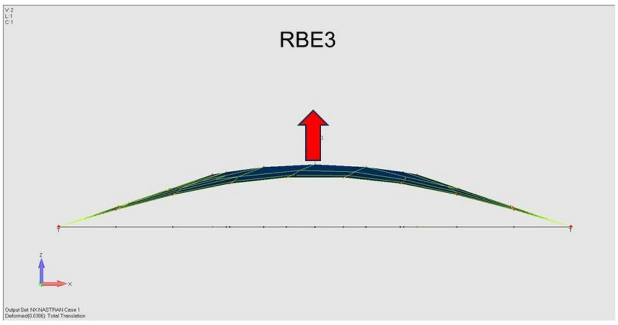
RBE2 – besides transmitting loads or displacements, it's stiff.

RBE3 – transmits loads, no contribution to stiffness









These elements are commonly used

Non structural mass and lump sum mass elements

Non-structural mass elements: think of things that don’t contribute to stiffness, but yet have enough mass that we should consider them (depending on the case): paint, electric components in a circuit board, deposits and fluid content on pipes, cargo on vehicles, decks or bridges, people, (…).

Lump sum mass elements: Used in dynamic analysis or where inertia effects are to be modeled in statics using body forces. Usually an element at a node, with both mass and mass moment of inertia terms.

Crack tip elements

Elements specifically formulated for fracture mechanics for the prediction of propagation of cracks.

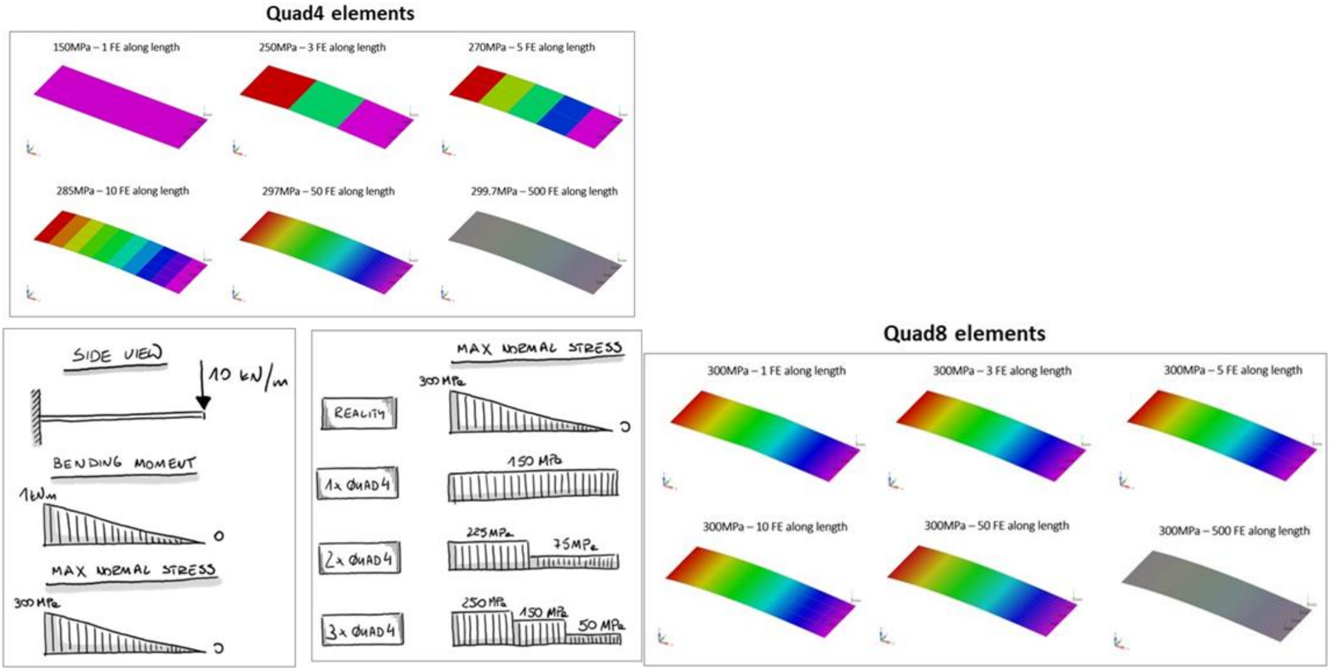
Element order

Shape functions

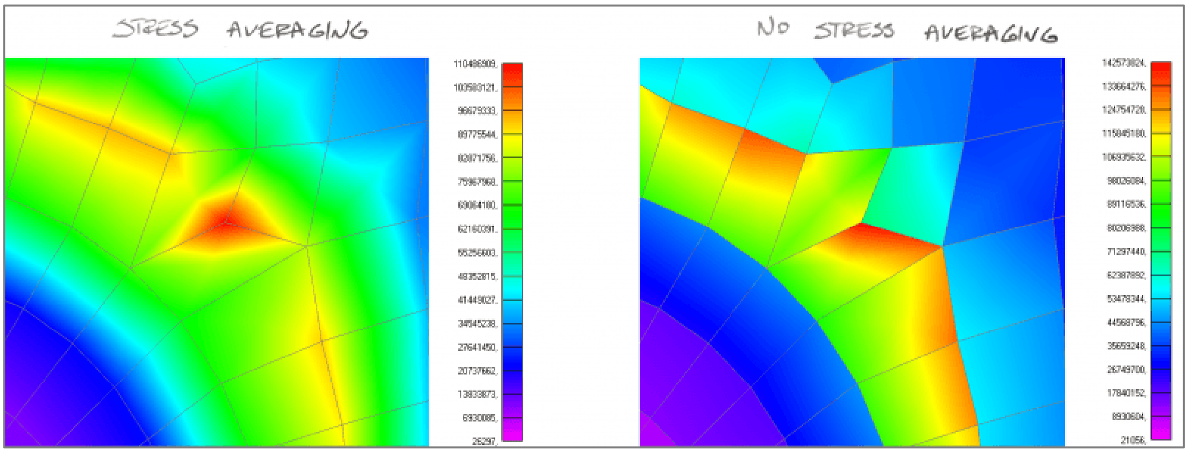
Post discretization in FEA, all the elements are assigned a function (a polynomial) which would be used to represent the behavior of the element. Polynomial equations are preferred for this as they can be easily differentiated and integrated. Order of an element is the same as the order of the polynomial equation(s) used to represent the element.

The FEM only makes calculations at a limited (finite) number of points and then inter/extrapolates the results to the entire domain (line, surface or volume). More specifically, the displacements are calculated at the nodes. To know the displacement at any other part of the element we need the shape functions which interpolate those values to the inside. These shape functions not only serve to help describe the actual shape/volume of the element, but also the field of displacements/temperature/stress (…) inside of it.

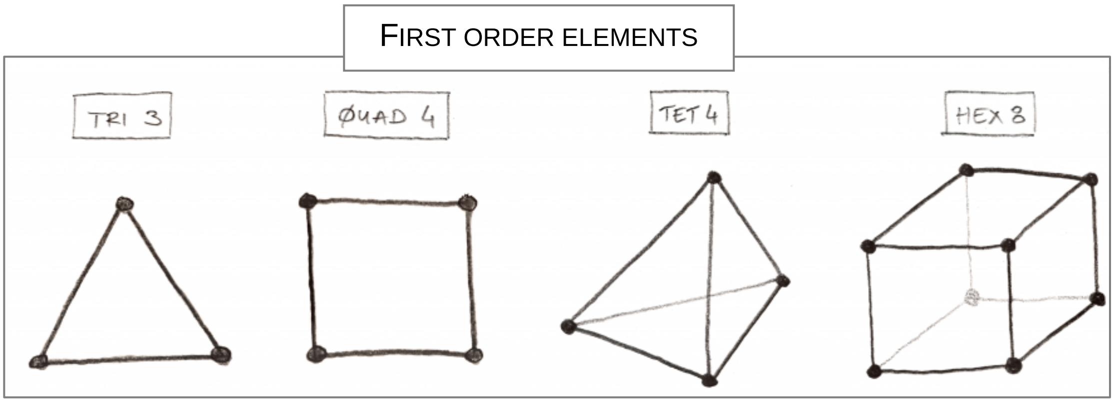
(Polynomial) functions of x. In 3D we would have f(x,y,z).	
$f(x) = 0$	Zero function
$f(x) = a$	Constant function
$f(x) = ax + b$	Linear function (order 1)
$f(x) = ax^2 + bx + c$	Quadratic function (order 2)
$f(x) = ax^3 + bx^2 + cx + d$	Cubic function (order 3)

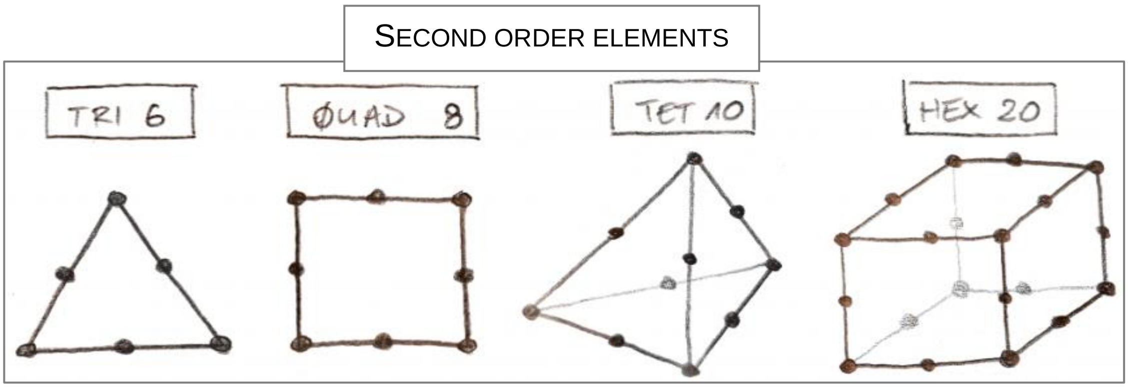


Note that for a distributed force, then Quad8 elements dont converge as quickly because they also only allow linear variation of stress but not quadratic.

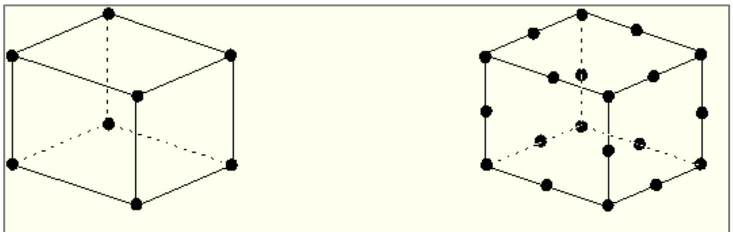


Displacements are calculated at nodes then interpolated to the inside of the element. There we calculate the strains. From the strains we can get the stresses. Then these are extrapolated back to nodes. From the nodes to interpolated to the whole edge of the element. This is why we see stress jumps from element to element. From this image we can see these rectangular elements are not 1st order because the unaveraged stress varies within the elements.





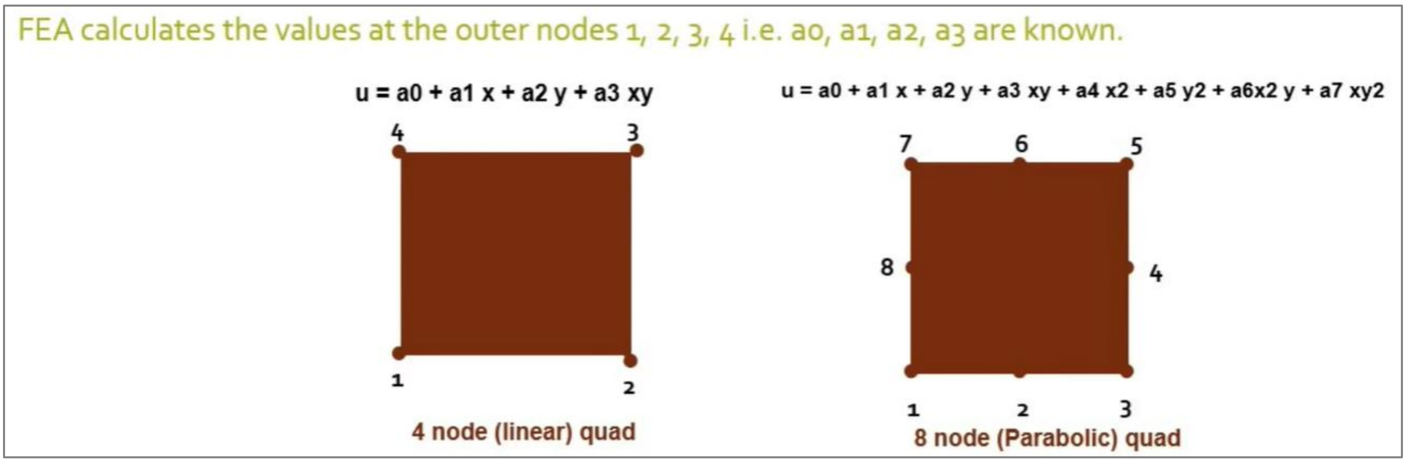
A Second order element or Quadratic element will have mid-side nodes in addition to nodes at the corners. It’s possible to know the values anywhere inside the element with the interpolation functions. These two points can be joined by a straight line (a linear interpolation) or a curved line (higher-order interpolation).



(a) Linear element (8-node brick, C3D8)

(b) Quadratic element (20-node brick, C3D20)

However, a Quadratic element needs a quadratic equation to describe its behavior as it has three nodes.



In FEM, it is wrong to directly assume that second- (or higher-order) elements would perform better or are more accurate. For example: In problems involving contact, first order elements perform much better than second-order elements. Elements of much higher order are never used due to significant oscillations. On the contrary, in problems involving bending / incompressibility, second-order elements perform much better than the first-order elements.

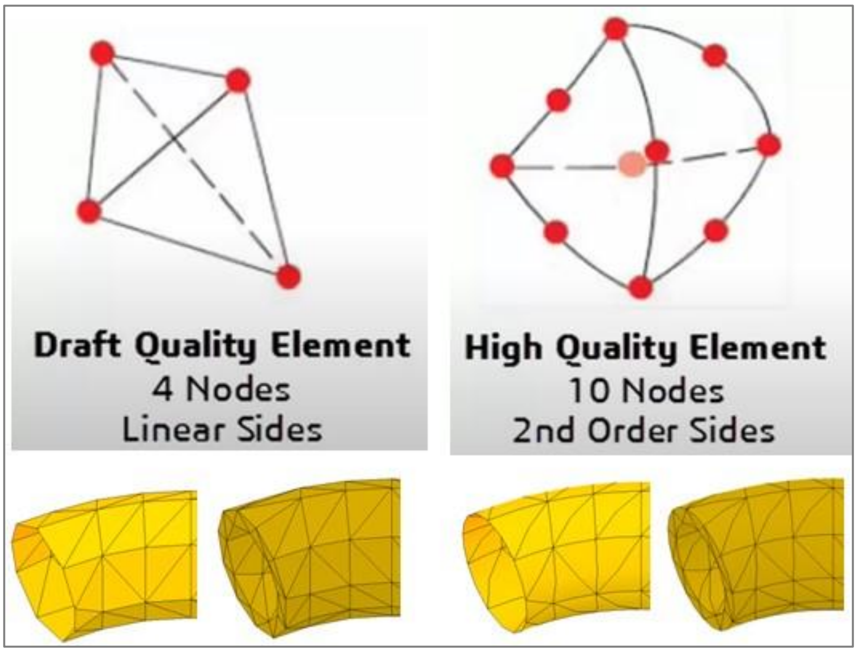
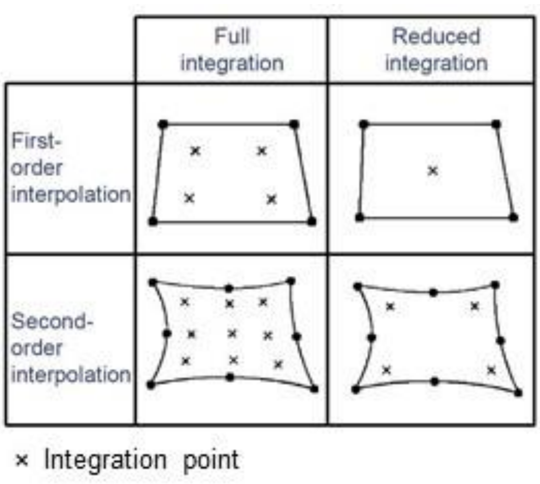


Figure 6 *1st order tetrahedron, and 2nd order tetrahedron. Notice that because of the increased number of nodes the 2nd order mesh better approximates the geometry. Note that on top there are tetrahedrons represented, but the first image on each side are triangles. There are also triangles of 1st and 2nd order.*

1st order elements vs 2nd order elements

- 1st order have corner nodes only whereas 2nd order have also midside nodes.
- Because 1st order only have two nodes we can only make a straight line, so they use linear function (P(x)= ax + b) whereas 2nd order use polynomial function (P(x)= ax^2 + bx + c). Because of this not only do 2nd order can describe geometry better (curved geometry at least) as they can have curved lines, but also the interpolation of displacements is better since it can allow for variation other than linear one.
- 1st order elements want to have constant stress 2nd order allow for linear variation

Integration points



Because the real values are calculated for nodes (no interpolations, extrapolations or integrations which come with loss in accuracy) therefore place them when possible where results are important. For example: loads, boundaries coinciding with nodes is good.

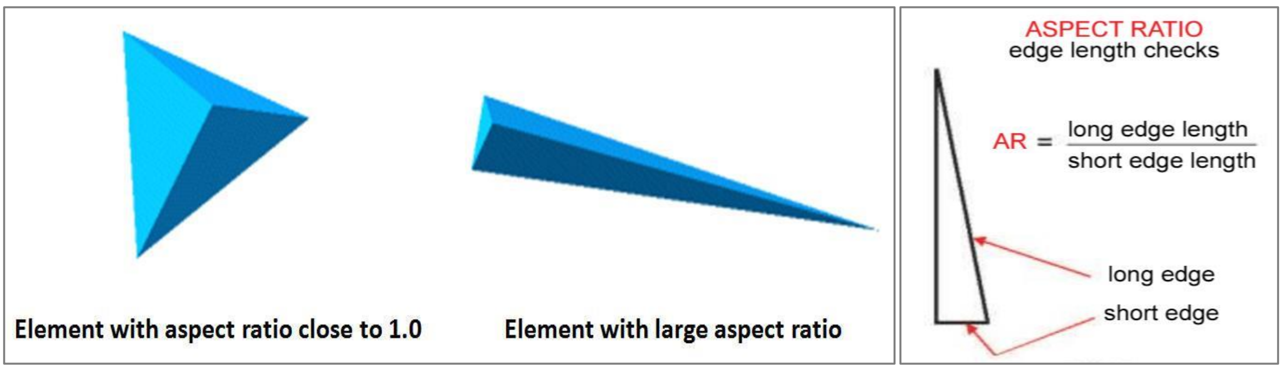
Mesh Quality

Skewness & Aspect ratio

Skewness

Skewness is the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. So it has to do with the internal angles of the shape. A value ranging from 0 to 1, the smaller the better, and, as a guideline, elements of skewness of more than 0.95 are considered unacceptable.

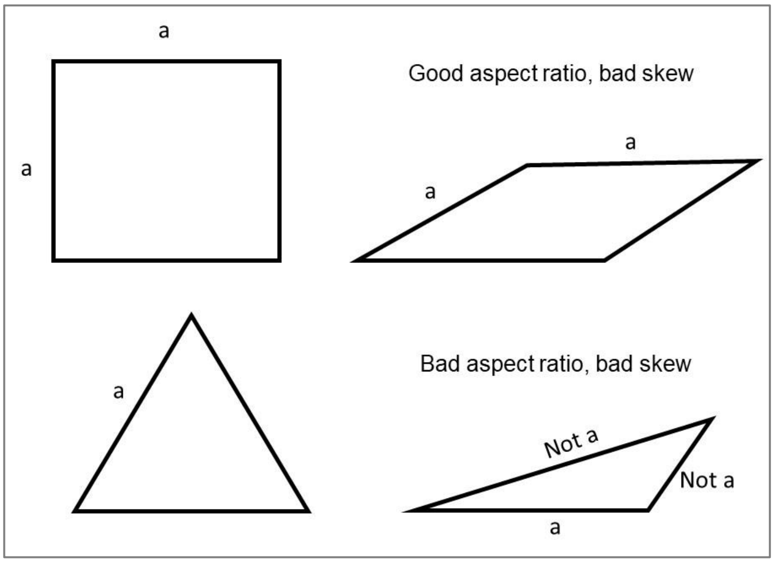
Aspect ratio is the ratio between size of the longest and shortest side of the shape. We prefer equilateral things here. (The reason we do is because at some point in the calculations, the equations are made for equilateral perfect shapes, and the more we deviate from them, the more error is introduced).



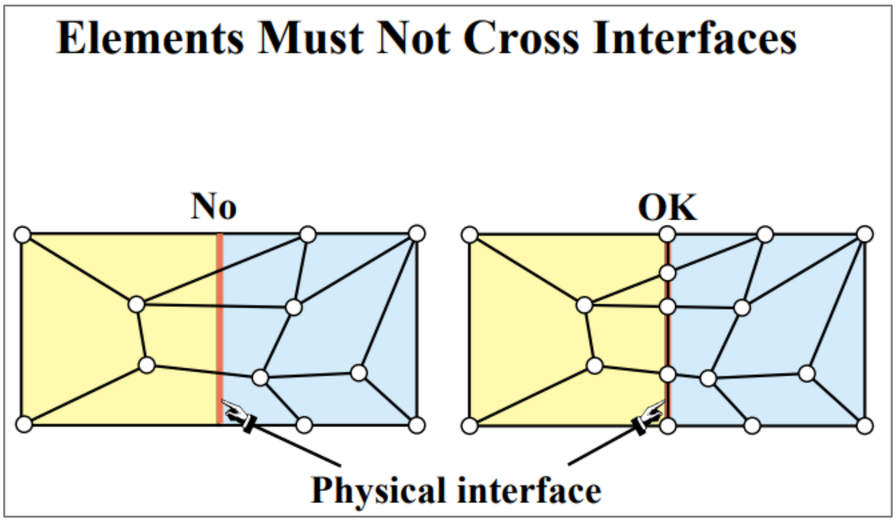
If the aspect ratio of an element is 10 it means its largest edge is 10x bigger than the shortest. Rules of thumb:

- Keep the aspect ratio below 5 in areas where stress is of crucial importance (high stresses) because you need accurate results there.
- Don't have more than 10% of the elements with an aspect ratio higher than 10. A way to fix this can be to introduce mesh controls in the places where the AR exceeds the value you want. Extremely large values >> 40 should be closely examined to determine where they exist and whether the stress results in those areas are of interest or not,

For tri/tet elements a high aspect ratio means high skew. For a quad/hex element high aspect ratio can still have zero skew.



Interfaces



The first unknown in linear static FEA that is solved is the displacement of each node. Based on this, the stresses and strains are calculated. The stress values at nodes in an FEA study are calculated at Gauss, or Quadrature points in the element, and then averaged with the stress values from the surrounding elements. While the displacements are solved explicitly at the nodes, the stresses are an averaged value and if there are insufficient stress values present in an area, the stress value averaged at the node can be inaccurate.

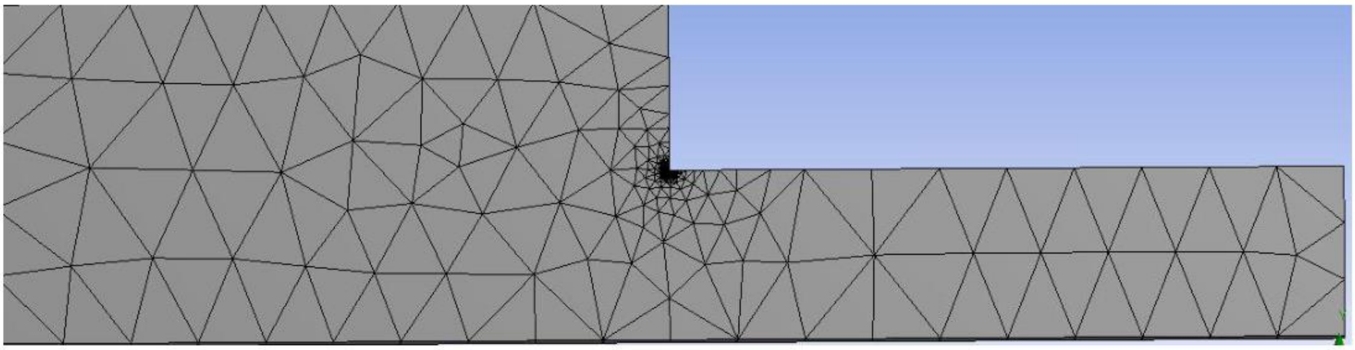
Calculations are first made for the nodes and then for stresses, which are averaged between themselves. That makes sense within a part or material, but not between materials or parts.

A good mesh will put the elements in the regions of geometry change as they tend to be stress concentrators, and thus have higher stresses, and so that's where the part will tend to break first.

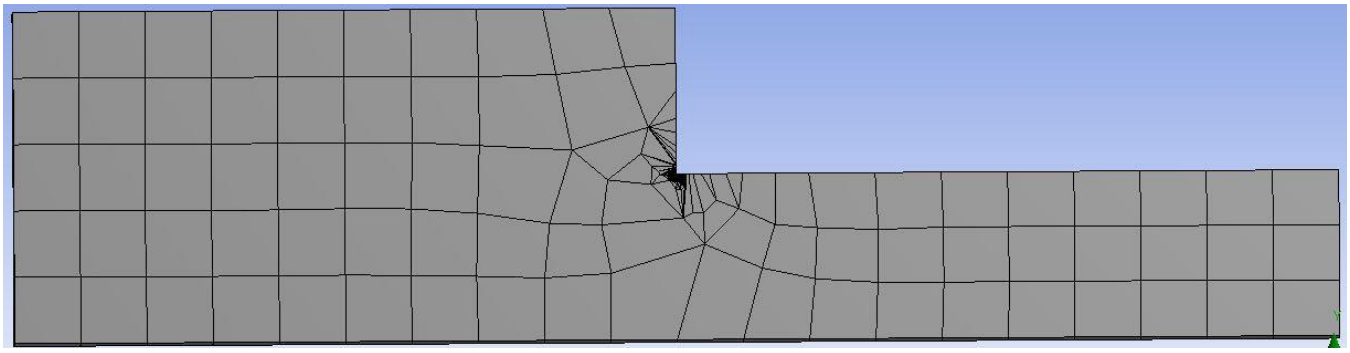
Element size transition

Transition: Fast means that there may be large jumps in element size. Slow means opposite, that is, the transition from big to small elements is smooth (with medium sized elements in between).

Slow transition

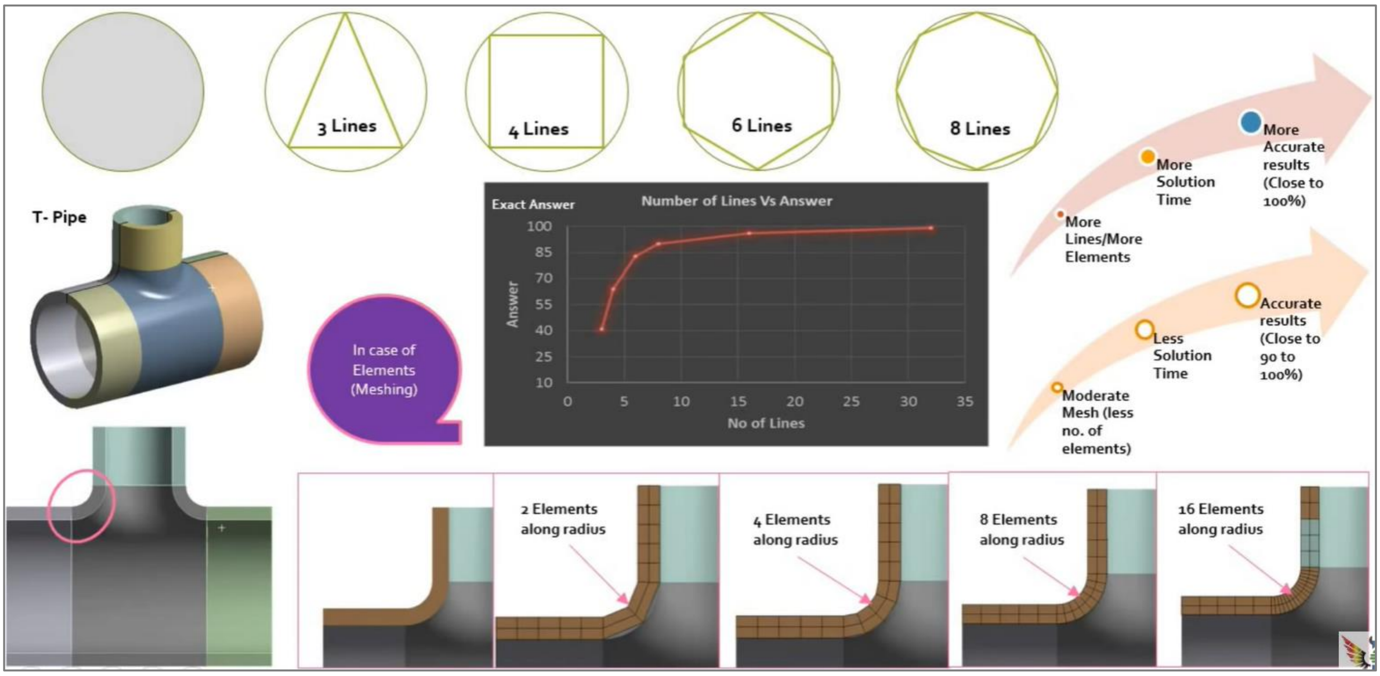


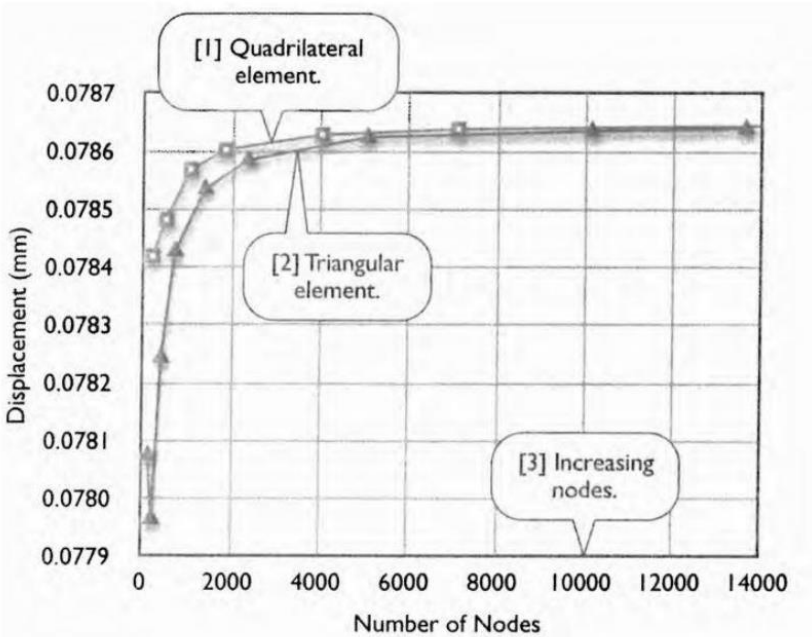
Fast transition

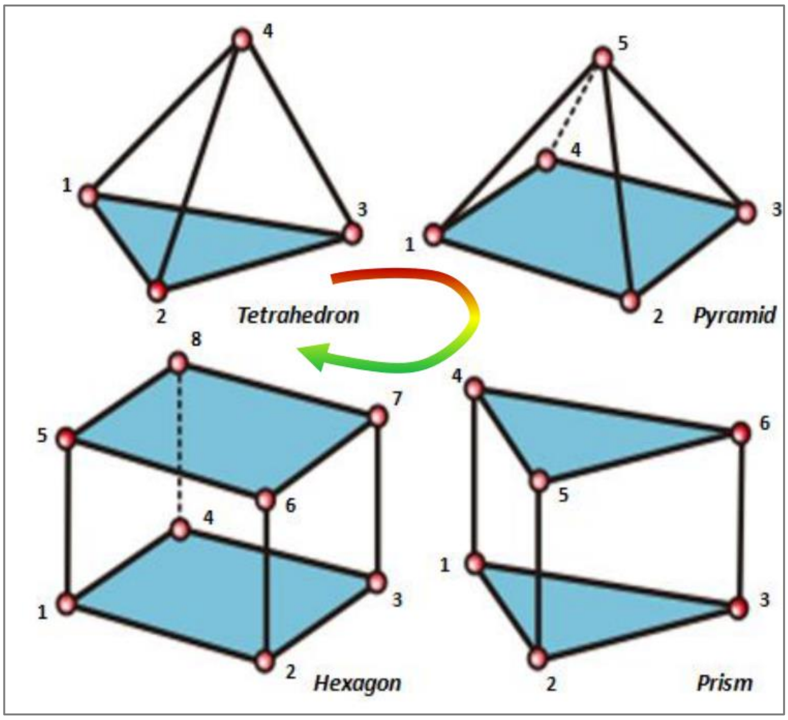


Element shape and convergence speed

In 2D, quadrilateral elements converge faster than triangular. In 3D, from faster to slower convergence are the hexahedral, prism, pyramid, and tetrahedral.

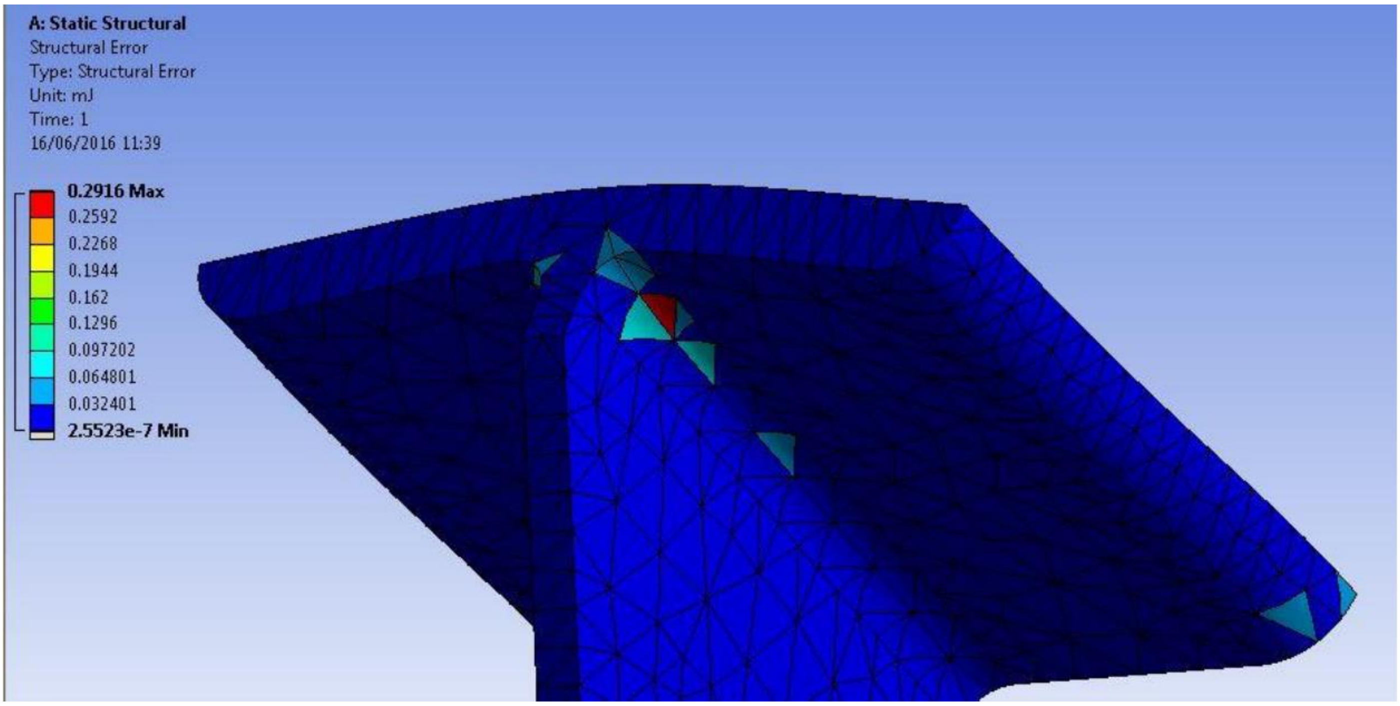

3 Lines
4 Lines
6 Lines
8 Lines
T- Pipe
Exact Answer
Number of Lines Vs Answer
Answer
No of Lines
In case of Elements (Meshing)
More Lines/More Elements
More Solution Time
More Accurate results (Close to 100%)
Moderate Mesh (less no. of elements)
Less Solution Time
Accurate results (Close to 90 to 100%)
2 Elements along radius
4 Elements along radius
8 Elements along radius
16 Elements along radius


[1] Quadrilateral element.
[2] Triangular element.
[3] Increasing nodes.
Displacement (mm)
Number of Nodes

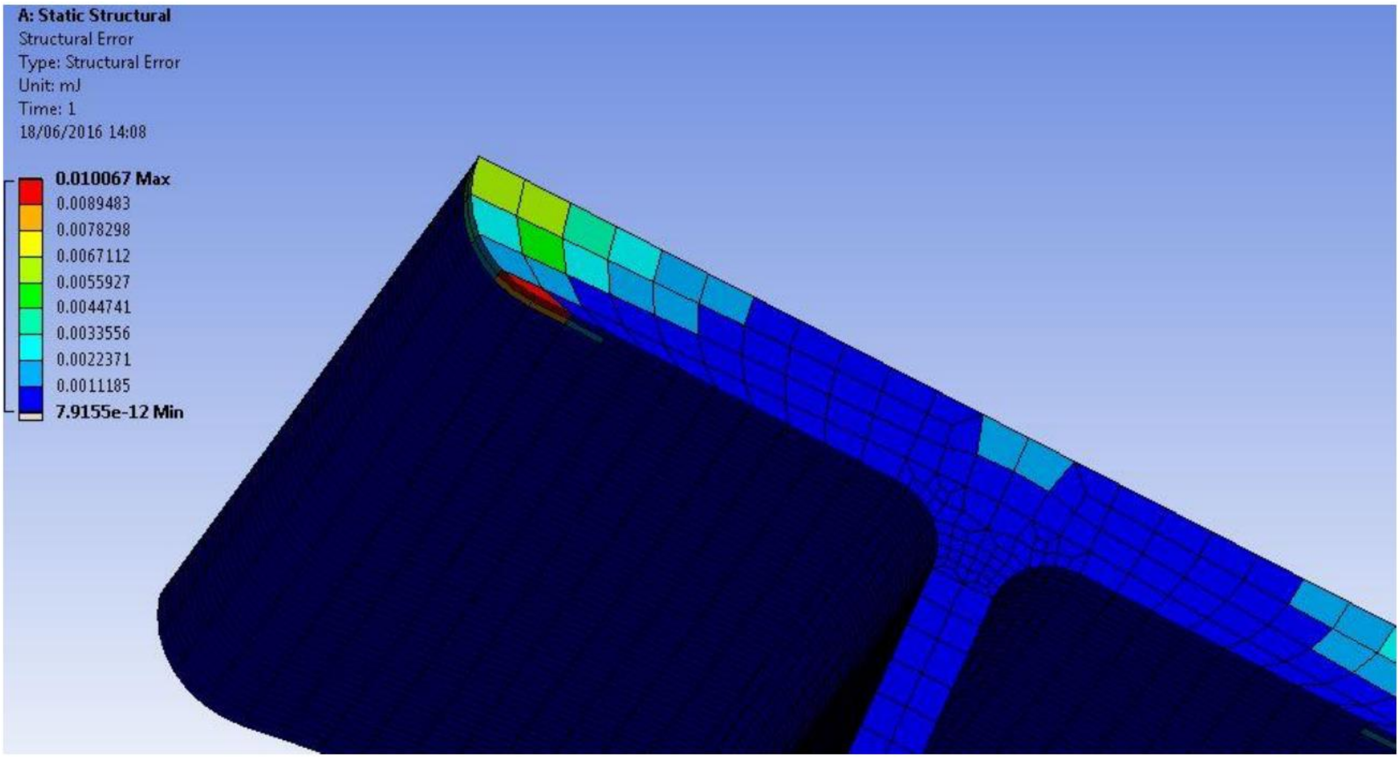


Besides converging faster, by changing the element shape you can also reduce structural error.

Let’s change the element shape from tetrahedral to hexahedral.

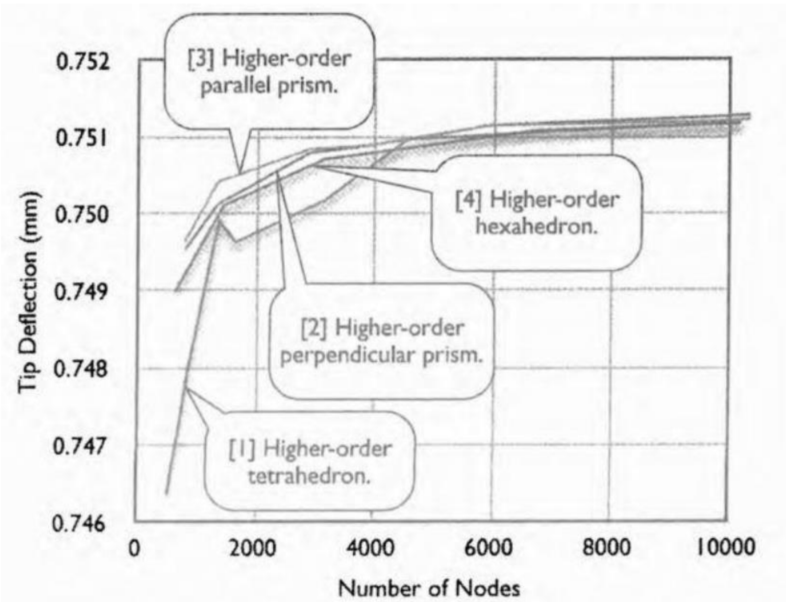


Now see the structural error significantly decreasing:

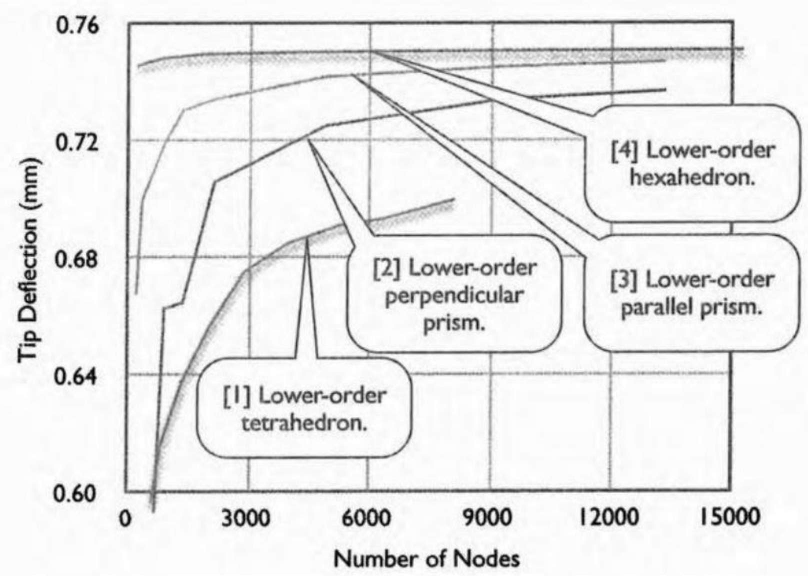


Besides, the structural error now occurs on a singularity point, so it’s not as important. The singularity point is simulating a right angle that in real life doesn’t exist (it’s rounded, even if only very slightly, so the stress is not infinity).

The chart below is made by a collection of the convergence curves in Sections 9.3-9, 9.3-10, 9.3-11, and 9.3-12 to compare the convergence behaviors of the higher-order elements. Except the tetrahedron, the order of the convergence speed among other elements is not obvious. The tetrahedron performs poorly only when the mesh is coarse (below 5000 nodes, for this case), otherwise it may be as good as other elements. Contrasting to the lower-order tetrahedron, the higher-order tetrahedron is still practically useful as long as the mesh is fine enough.

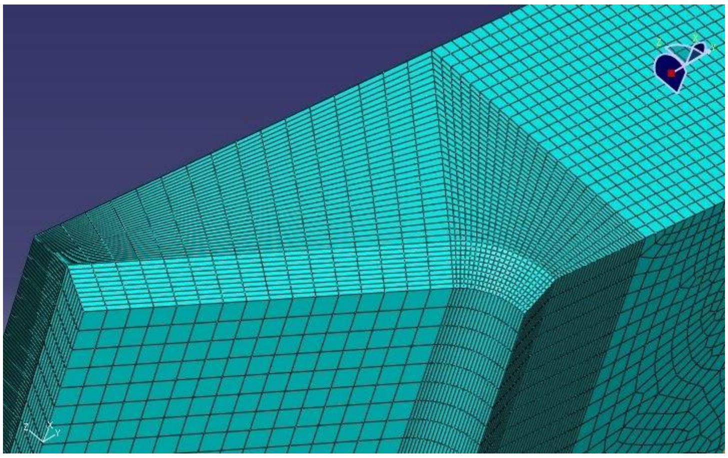


The chart below is made by a collection of the convergence curves in Sections 9.3-5, 9.3-6, 9.3-7, and 9.3-8 to compare the convergence behaviors of the lower-order elements. The order of the convergence speed is, from fast to slow, hexahedron, parallel prism, perpendicular prism, tetrahedron. The differences between them are obvious and quite evenly spaced. The lower-order tetrahedron converges so poorly that it is not practically useful. As a guideline, NEVER use lower-order tetrahedral elements.



Combining the observations in Section 3.5 and this section, we may summarize the conclusions as follows: (a) Never use lower-order tetrahedra or triangles. (b) Higher-order tetrahedra or triangles can be as good as other elements as long as the mesh is fine enough. In cases of coarse mesh, however, they perform poorly and are not recommended. (c) Lower-order prisms are not recommended. (d) Lower-order hexahedra and quadrilaterals can be used, but they are not as efficient as their higher-order counterparts. (e) Higher-order hexahedra, prisms, and quadrilaterals are among the most efficient elements so far we have discussed. Mesh your models with these elements whenever possible. If that is not possible, then at least try to achieve a higher-order hexahedra-dominant or quadrilateral-dominant mesh.

Besides the above guidelines, mesh quality requirements, in terms of mesh metrics such as skewness, should also be met.

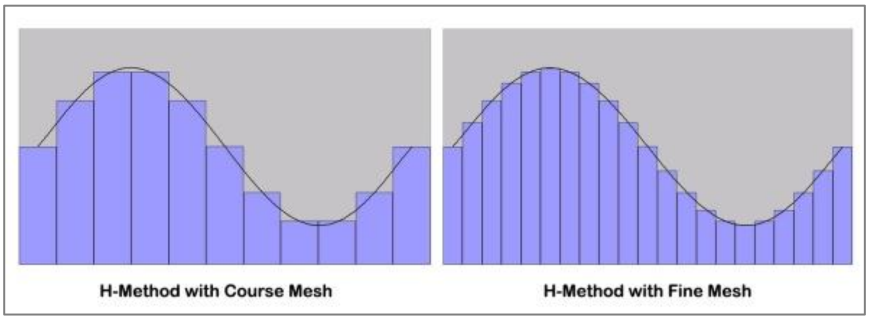


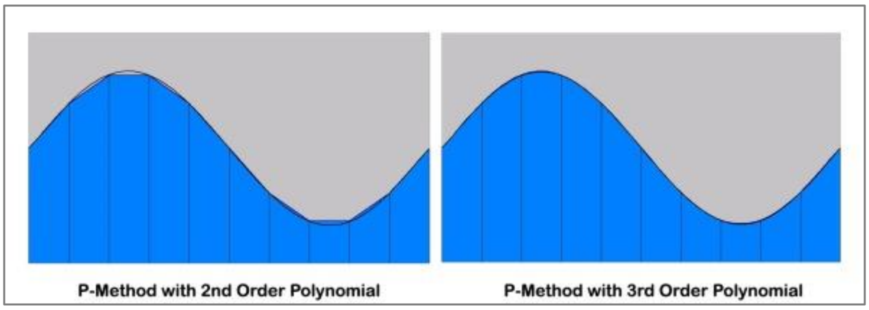
Mesh Independence & Convergence

- The results of a FEA should be independent of mesh size.
- We should strive to find the point at which a change in the mesh does not significantly impact the results. When this happens we say we have **convergence**. This means, as we improved the mesh, the results are converging to the same values (hopefully the real ones, if the simulation is well done).
- **Mesh independence study** is trying to figure out if the results do not depend on the mesh. For this we have two main ways to verify.

- **H-method:** “h” usually denotes the characteristic length of elements. This method is about reducing the “h” value which results in increased number of elements. In this, we increase the number of elements until we see the results are not changing anymore (they have converged).

- **P-method:** “p” denotes the polynomial order. Here, the order of the shape functions is increased instead of increasing the number of elements.



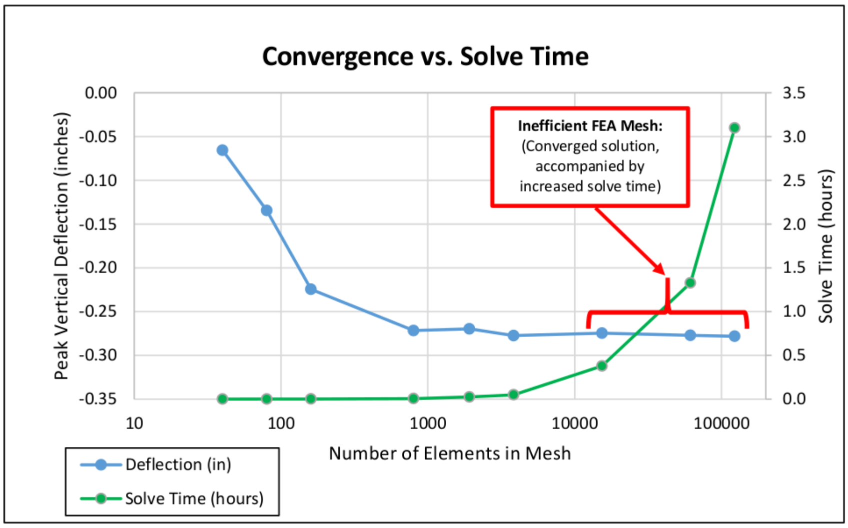


Linear elements	Quadratic elements
• Can support only a linear variation of displacement and therefore (mostly) only a constant state of stress within a single element. • Highly sensitive to element distortion. • Acceptable if you are only interested in nominal stress results. • Need to use a large number of elements to resolve high stress gradients.	• Can support a quadratic variation of displacement and therefore a linear variation of stress within a single element. • Can represent curved edges and surfaces more accurately than linear elements. Not as sensitive to element distortion. • Recommended if you are interested in highly accurate stresses. • Give better results than linear elements, in many cases with fewer number of elements and total DOF.

- You might not want to change the overall mesh. It saves time if you only change the areas that you predict influence the results the most, and only perform a convergence analysis for those areas.

- With further changes to the model, you then use the bigger mesh size that gave you mesh-independent results.

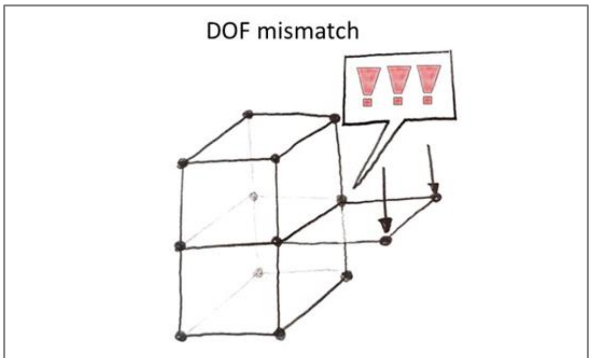
In this image, at ~1000 elements we achieve mesh size independence. A further decrease in element size (increase in number of elements) will not improve the results. So for further analysis if we change the design, provided the changes are not so impactful that we need to run another convergence analysis, we can stay at 1000 elements.



- When should we do a mesh independence study?
    - Whenever we want to be certain that the results are accurate and are not being influenced negatively by the mesh.

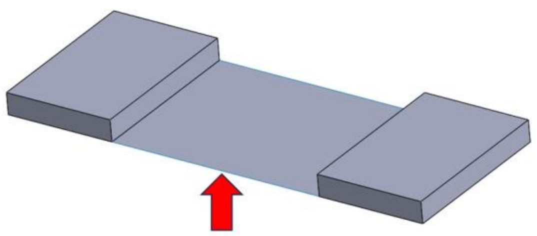
Mismatched DoF

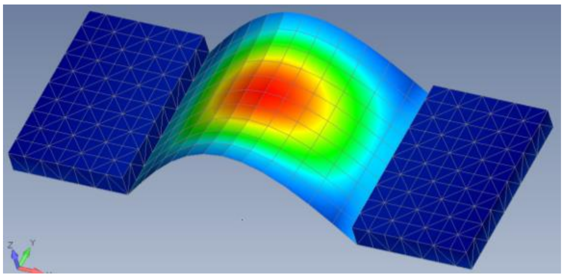
In 3D, each beam of a node element has 6 DoF, each node of a shell element has also 6 DoF, but each node of a solid element only has 3 DoF. This means that when we connect Shell or beam elements to solid elements we need to take this mismatch into account.



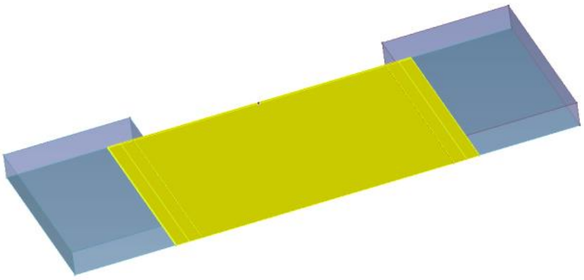
Connecting 3D elements to 2D elements. The area above will rotate like an hinge as the nodes of 3D elements don't take rotations as DOF.

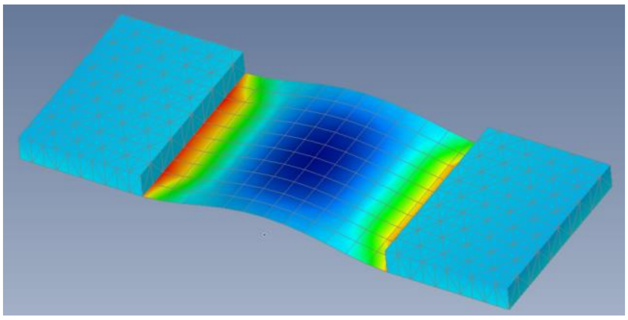
In this situation we meshed the 2 bars on the sides as solids and the middle thin part as shell.





What happened was that the shell was free to rotate on the connection as if it was a hinge. It couldn't translate, but it could rotate. The solution to this is to create an overlap and bond it.

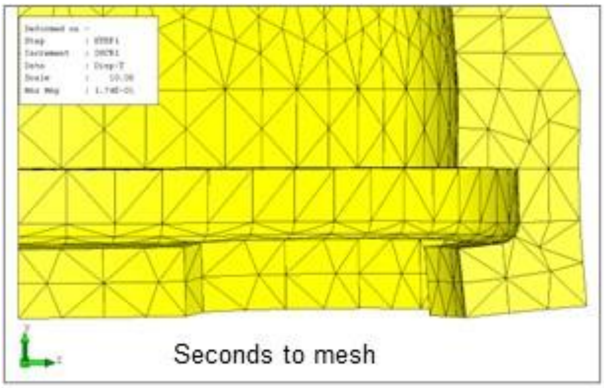


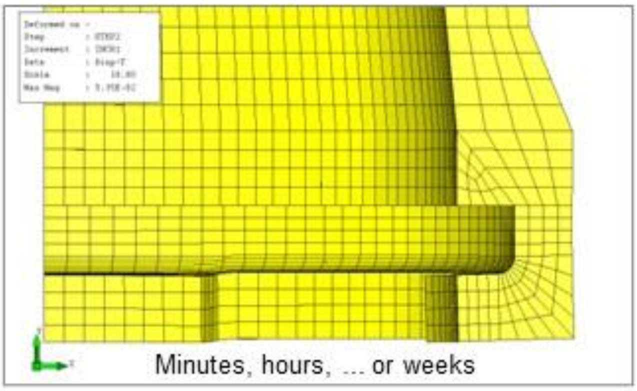


Hexahedral vs tetrahedral elements

Hexahedral elements are…

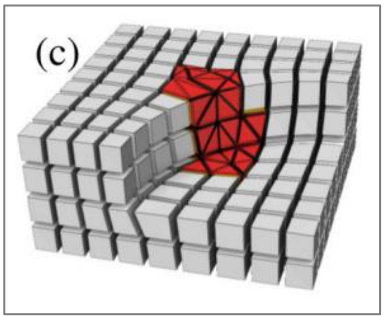
- … more time consuming to mesh often requiring manual intervention. Most likely needs defeaturing.
- … especially difficult for curved surfaces
- … quicker to solve. We need less hexahedral elements than tetrahedral for the same accuracy. Less elements = less DoF = less equations for the solver to calculate = faster.
- … able to provide more accurate results (for similar numbers of elements).
- Is time saved on solving more than time spent on manually meshing hexs? Often not. Therefore Tets are usually the best choice. But if you know for a fact you can mesh quickly a part with hexs, then go for it.





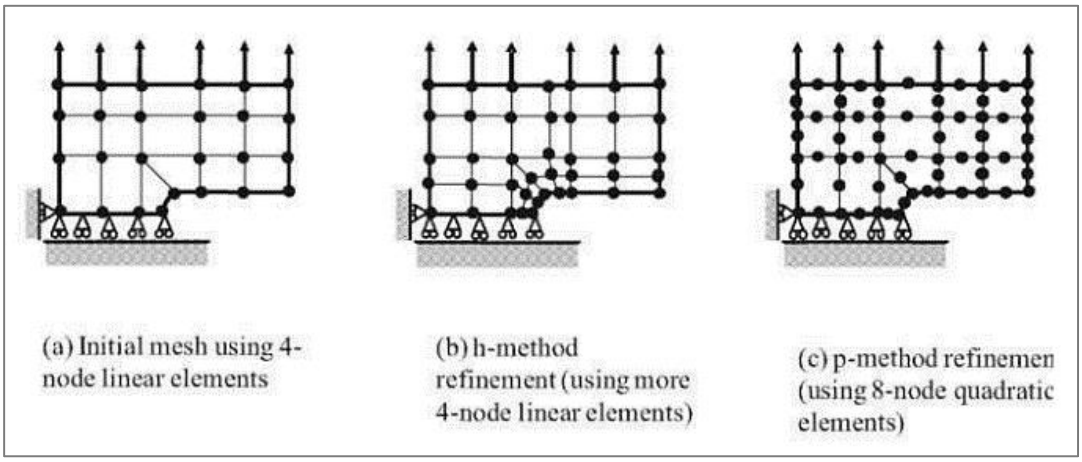
Tets are an element which can mesh any geometry* which is great, except they are usually more expensive to solve. On a per-element basis they are faster: 1000 tets will solve faster than 1000 hexes, but meshing a part at a similar density results in about 5 times more tets than hexes.

Hexahedral-dominant meshing is a relaxation of full hexahedral meshing, that allows to greatly improve robustness at the expense of introducing a small number of tetrahedra.



Mesh refining

- Mesh refining options via adaptive meshing (automated iterative mesh refinement):
  - h-method (smaller elements, interpolation order remains same)
  - smaller elements are placed in critical regions like sharp variations of stress and vice versa
  - "h" usually denotes the characteristic length of elements. This method is about reducing the "h" value which results in increased number of elements.
  - p-method (same element size, increase interpolation order)
  - eg. from linear to quadratic, quadratic to cubic, and so on.
  - same no. elements, but more nodes = more accuracy because each element gets better degree of approximation.
  - "p" denotes the polynomial order.



Checks to do on your FEA

"Garbage in, garbage out."

Pre-processing

- Inspect mesh - especially on curved surfaces and transitions between small and large elements:
  - Mesh is smooth (no very big near very small elements, should be gradual change) and continuous (otherwise results are suspect).
  - Identify elements that are not fully connected with adjacent elements.
  - Identify very distorted elements
  - Check if elements have material properties assigned

    - Quadratic elements for curved places (edges in 2D, surfaces in 3D). Linear elements can also, but need to be small enough to represent curve.
    - Same order of interpolation is used for all elements, such as all linear elements or all quadratic elements. Which one to choose depends on how much stress variation we expect.
    - Adjust element size to speed of stress/strain change (small elements for areas with rapid stress changes, so areas where stress varies a lot)
    - Start with relatively coarse mesh (it is difficult for the user to work out, prior to the FE analysis, the appropriate level of mesh refinement that is needed to obtain a good FE solution)
    - Check for free edges.
- Check mesh quality graph
    - Element aspect ratio, which reflects element distortion (over 1:5 is large, unacceptable on places with rapid stress change). Elements should be approximately their theoretical shape (squares, cubes, equilateral triangles, ...)
    - For 2nd order elements, check Jacobian ratio.
    - Also check skewness
    - If mesh quality is only bad on areas that are unimportant, continue as is. Otherwise remesh area.
- Assign colors to different element types used, to make sure they were assigned correctly
- If possible same for materials
- Check applied loads (direction, value and points of application are correct) and boundary conditions (check also if it's not over constrained)
- Check whether meshed model mass is similar to design mass (especially important for dynamic sims)
- Usually possible to add mass elements. Increase only density if its uniform and true in all volumes.
- Run modal analysis (just first 10 modes) to quickly check for lack of boundaries, connections between bodies that are missing

Post-processing

- Check whether results converged (in singularities they don't) and to a good enough degree. Start with coarse mesh (big elements) as first analysis, then successively refine it (make them smaller) until change results in negligible differences in the output we're more interested in (usually stress).
- In some complex problems where it may be not be obvious to the user where the highest stress gradients will occur, particularly when large 3D geometries are involved, it may be advisable to deliberately start with a coarse mesh with the sole purpose of identifying the hot spots, and then run a new FE analysis with a more refined mesh in these regions.)
- Mesh refining options via adaptive meshing (automated iterative mesh refinement).
- Also check peak values of stresses to see whether mesh was sufficiently refined there
- Use napkin calculation to see if both stresses and reactions make sense
- Check if deformed shape is plausible
- Check deformation values to see if they make sense
- Check if boundaries value make sense (deformations and stresses)
- Check stress values smoothness (no strange jumps in stress)

- Check if difference between averaged and unaveraged stress is not too big (say more than 5% or that everything is still below yield even in unaveraged situation)
- Imagine how stuff will deform and where the critical parts will be.
- Is the mesh of good quality?
    - Skewness;
    - Aspect ratio;
    - Interfaces;
- Are the applied boundary conditions correct?
- Are the contacts correct?
- Are the material properties correct?
- Do a “napkin sketch” and some hand calculations to check if the overall results make sense.
- Is the stress continuous?
- Have you achieved mesh independence?
- If any, can you identify what is and what isn’t a stress singularity?

Sensitivity studies

How does the output of your FEA model changes with respect to variations in the input parameters? (Material properties, boundary conditions, geometry, mesh size, and loads).

Most common example is mesh sensitivity study, which we need to do to know whether the mesh refinement we have is enough, i.e., whether we have achieved convergence.

Stress singularities

In mathematics and physics a singularity is a point at which a function takes an infinite value, such as in space–time when matter is infinitely dense, such as at the centre of a black hole, or in FEA where a stress seems to keep increasing no matter how much we refine the mesh around it.

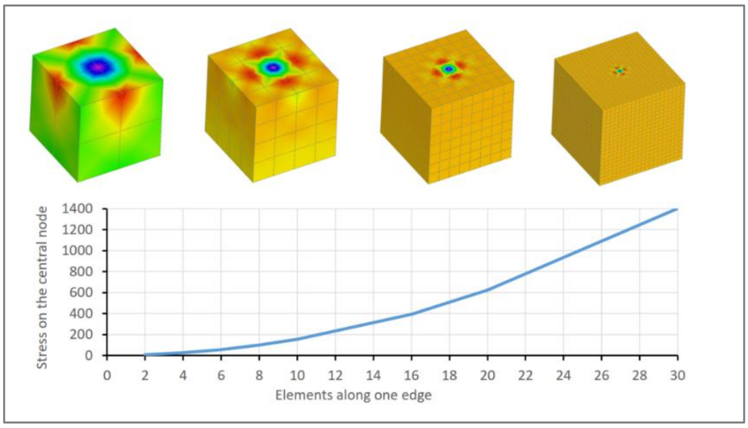
Most common types on singularities in FEA:

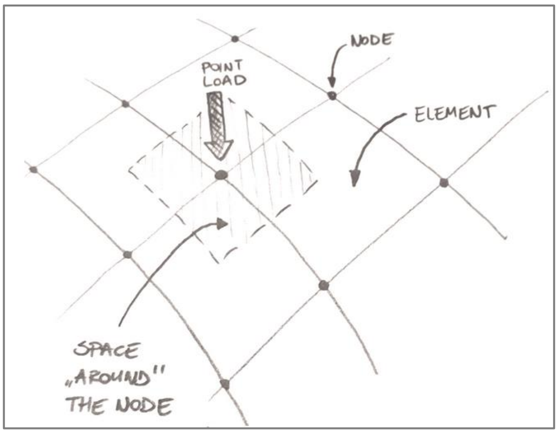
- Sharp corners
- Point loads ($\sigma = \frac{F}{A}$, if point load then A=0, therefore $\sigma = \infty$) and point boundaries

To solve:

- Make fillet on the sharp corner, or increase its radius.
- Convert point load to distributed on a small area
- Ignore singularity (see Saint Venant principle to see if applicable).
- For cracks, where the nature of stress singularity is known, use crack-tip quadratic elements.

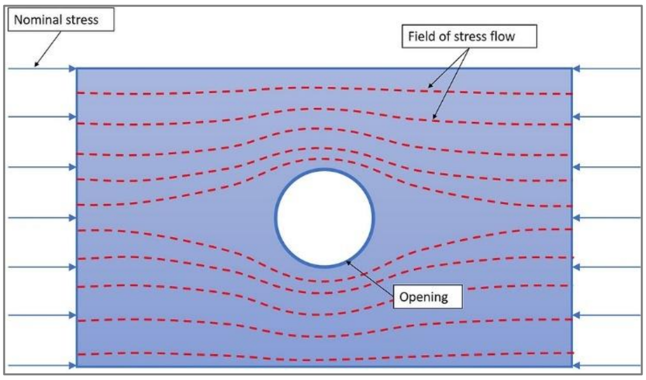
Point load singularities

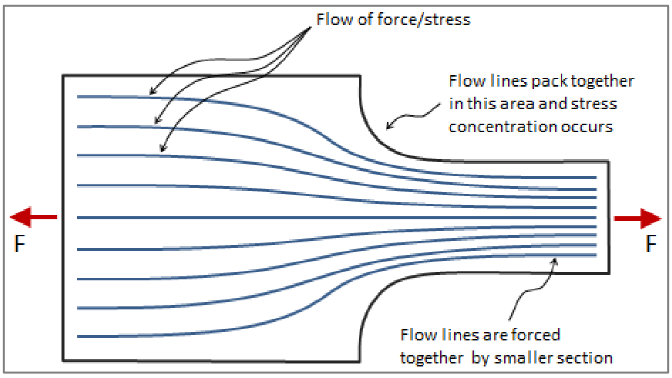


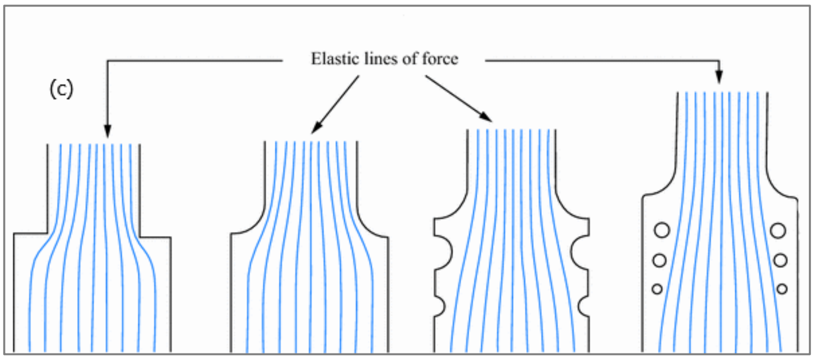


Sharp corner singularities

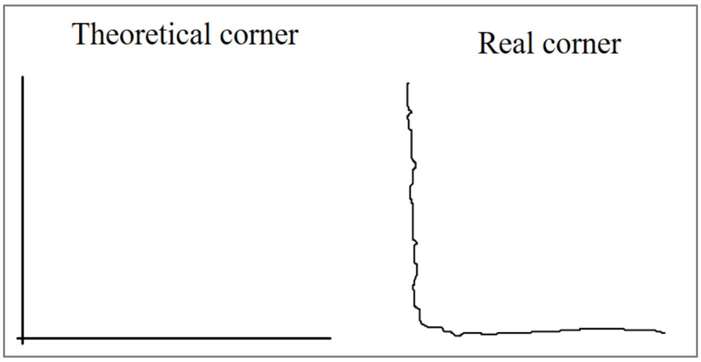
Stress propagates kinda like sound (except more statically without vibrating as much). As you pull on one end, the atoms pull on each other successively. As if stress flowed. And it flows through paths, the so-called “load path”. Just like with a fluid, if we have the same flow of water and we reduce the tube, it wants to go “faster”. Since load flow can’t go faster, instead it gets “concentrated”. In a way, it’s as if the water didn’t go faster, but increased its pressure.







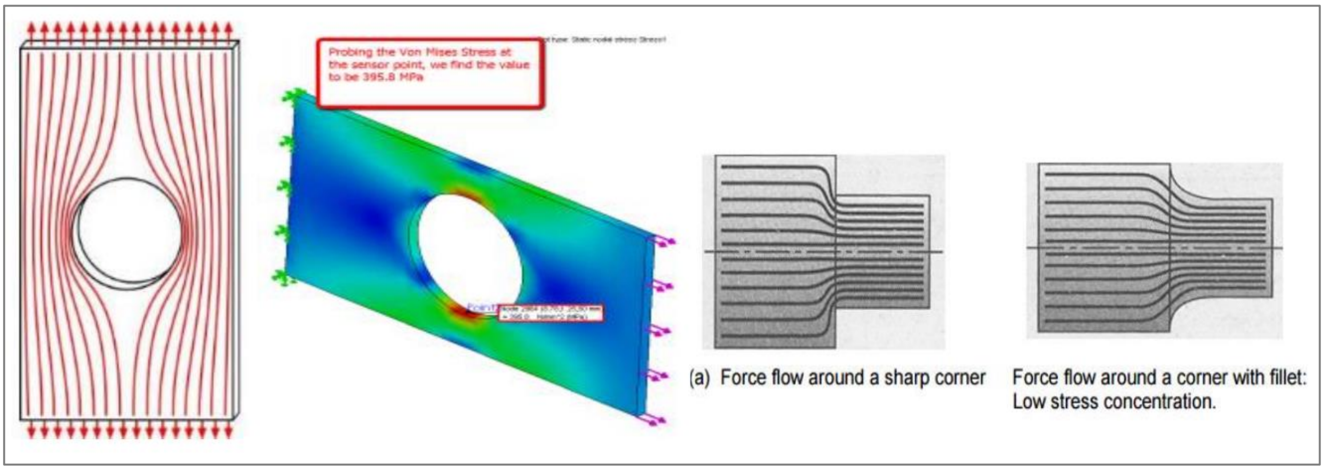
If a corner is in the path of the flow of a force, then the stress will concentrate. The sharper the corner, the more it will concentrate. In a sharp corner, the stress has trouble flowing. There’s no curve. In real life we can’t really have exactly sharp corner, it’s always a sort of a curve (or maybe a crack). But in a CAD model we can have such theoretical sharp angles.



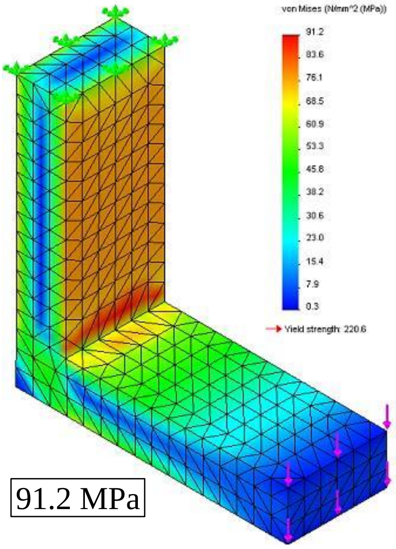
The less round the corner is, the higher the stress concentration. When the corner has no radius, then in that sharp corner the stress is infinite, and the corner is called a point of stress singularity.

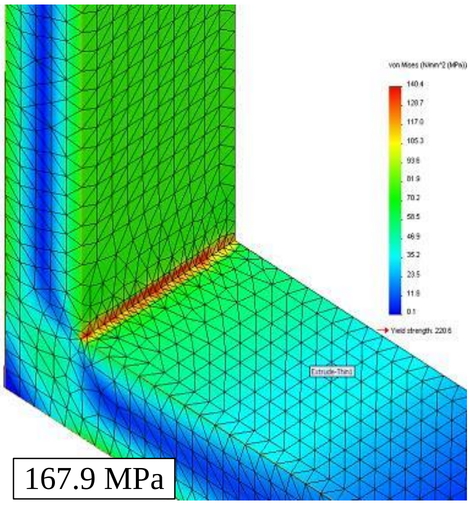
In the real world, though, corners like that don't happen as it's near impossible to manufacture such a sharp corner as the precisions of current machining don't allow for that (maybe with 3D printing ;)).

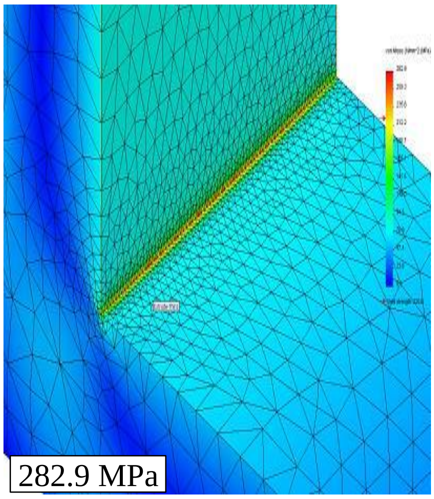
In the computer though, since we're dealing with the theoretical, zero-radius fillets are frequent. More, because certain small features such as small fillets don't contribute much to the global behavior of the structure, they are often not included in the model for simulation. It becomes, then, important that engineers are aware of the existence of these points. They may mistakenly use the maximum stress as the design stress, but glazing over the fact that it came from a singularity – it wouldn't exist in the real world. So always check if the maximum stress comes from a singular point.

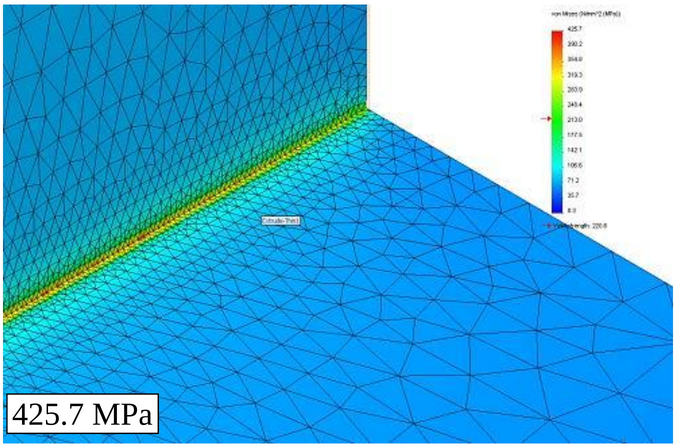


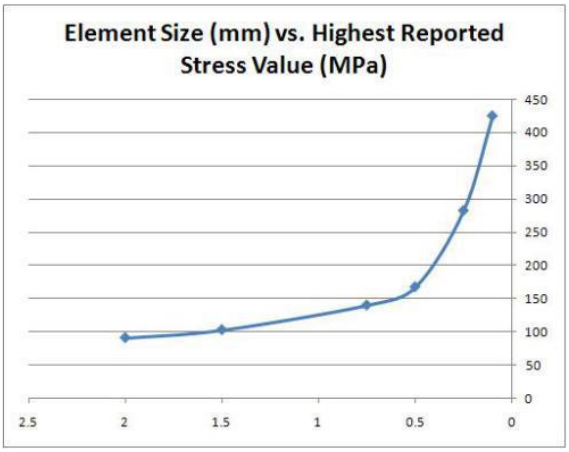
A singularity is characterized by non-converging stress. It goes to infinity. Applying mesh control to the edge in question, and locally reducing the element size, we see the stress progressively increases.











To fix this, add a small realistic fillet/chamfer that the part would have in reality.

Verification & Validation

- 3 Models: physical, mathematical, computational.
- Verification: checking that computational matches mathematical:
    - Code verification
        - For: software engineers
        - Goal: check that software code is according to math
        - Example: compare code outputs with analytical solutions
    - Numerical Calculation verification:
        - For: analyst
        - Goal: estimate errors due to discretization (mesh).
        - Example: Check convergence, mesh grading (variable refinement in different areas), adaptive meshing. If not possible to have theoretical / manual calculations of whole thing, possible to calculate manually just the stress/displacement at some point of interest and compare?
- Validation: checking that solution of physical matches solution of computational
    - Doing experiments and observations

Other types of analysis

Modal

Modal analysis is the study of the natural frequency and resonance frequencies and shapes of the different modes of vibration in those frequencies. This depends only on the geometry of the components and how they are constrained.

Topics:

- Mode Shape/Eigenvector
- Modal Mass - a mass that is activated in a certain vibration mode.
- Modal Damping - energy dissipation in order to reduce vibratory motion. Each mode has a particular mode has a particular modal shape which dissipates energy differently.
- Modal Stiffness
- What modes drive the peak response?

Dynamic

For Static Problems the finite element method solves the equilibrium equations $\sum \mathbf{F} = 0$.

For Dynamic Problems the finite element method solves the equations of motion for a continuum – essentially a more complicated version of $\sum \mathbf{F} = \mathrm{m}\mathbf{a}$.

The difference between static and dynamic analysis is that in dynamic we have to take into account inertia. Why do I say inertia and not mass? Because in a static analysis we do take into account the mass. If something is not moving at all, we may still have to take into account gravity. However, if that thing moves very fast, then we also have to introduce the "force of the mass that doesn't want to change its moving state". So, if something moves, do we necessarily need to account for inertia? Not necessarily. If an event happens so slowly that it's as if we could divide it into time steps where it's not moving, we can approximate it to a static situation. We call these situations **quasi-static**, because they're almost static.

- {F} = force vector
- [K] = stiffness matrix[1]
- {x} = displacement vector

For static linear analysis:

$$\{\mathrm{F}\} = [\mathrm{K}]\{\mathrm{x}\}$$

For static non-linear analysis (the stiffness matrix depends on x):

$$\{\mathrm{F}\} = [\mathrm{K(x)}]\{\mathrm{x}\}$$

For dynamic analysis:

[1] (Note: this stiffness matrix is not to be confused with the other stiffness matrix that is used with stress and strain, whose units are Pa. This one has units of N/mm.)

$$\{F\} = [M]\{x''\} + [C]\{x'\} + [K]\{x\}$$

Note about notation:

[…] For matrices

{…} For vectors

Or sometimes simply in bold. [M], {v}, or **M**, **v**.

Implicit vs Explicit Analysis Approaches

Both are only of concern for dynamic analysis. They do the calculation with 'time' in mind but in different ways:

- Implicit
  - for when the time is large (say 1s - 10s)
  - method of choice for problems where inertial loading is important, but rapid transients are not the focus
- Explicit
  - for when it's shorter (say <1s). For larger times implicit is faster and for shorter, explicit is the faster.
  - for when time plays an important role, e.g., sudden acceleration or deceleration in crashes or impacts. So the first and second derivative of movement equation (speed and acceleration) will also be considered.

Thermal analysis

- Material properties (material conductivity, specific heat, density, (optional - emissivity)
- Finer mesh for high gradients of change
- Boundary conditions: fixed temperature and heat flow
- Loads: heat generation
- Transient (specify initial conditions) and steady-state analysis
- Temperature is like displacement, so calculated at nodes. Heat flow is like forces and stress.
- Thermal time constant - Imagine we have a body at stable temperature. Instantly get a spike of deltaT. The time constant is the time it takes to reach 63.2% (about 2/3) of initial deltaT.
- The time constant is a measure that characterizes that thing's ability to retain heat. With no influence from a heating or cooling system, theoretically, the thermal time constant provides an indication of how fast the it will take to achieve a new thermal equilibrium in response to changes in its internal and external thermal conditions.

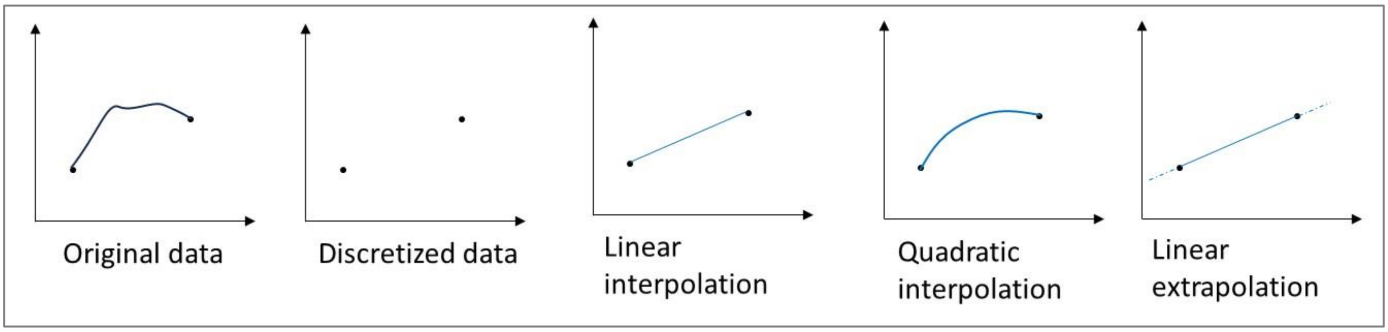
Thermo-mechanical analysis

- The effect of temperature change (which comes from the coefficient of thermal expansion) is incorporated into thermal strain
- Uncoupled - deformations of the body don't affect temperature. 1st thermal stresses are calculated, then used as loads for structural analysis.

- Coupled - thermal and structural analysis depend on and affect each other. Example: braking disc. Temperature depends on sliding/force. Or mould which cools down and reduces heat flow significantly.

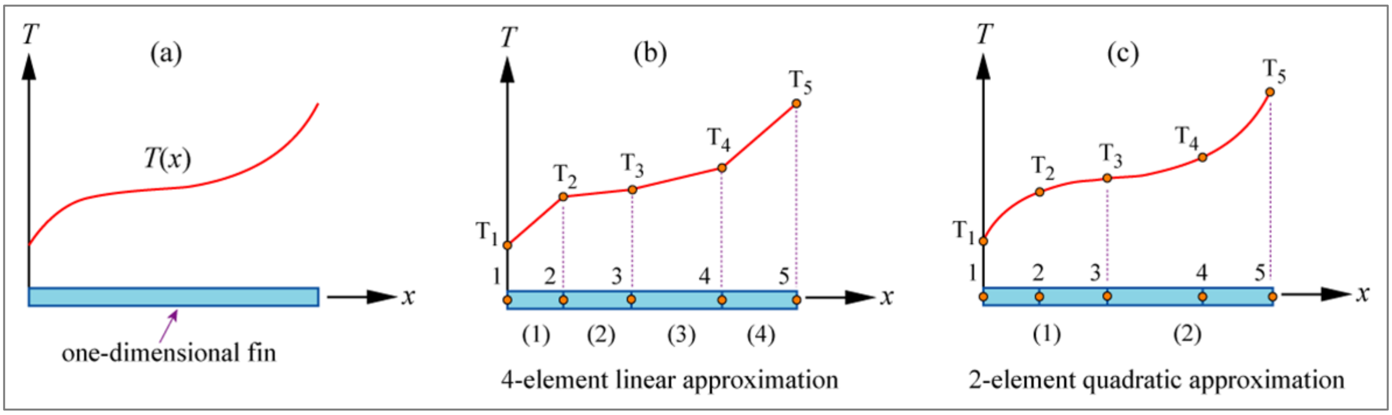
Background Knowledge

Interpolation and extrapolation



Original data could be geometry (surface of a shape), could be stress or displacements values, temperatures, …

Discretized data: We cannot store the full complex shape of the data, so we just store some points (discretization). The more points, the closer to the original data. Then having points that correspond to the real data, we can try to get something to look again like the real data but that is representable by equations (lines, curves). Here comes the interpolation. The extrapolation comes for example when we calculate stresses or strains in the center of the element, and we want to know them on the nodes (which lay on the outside). Then we need to extrapolate those values.



Isotropy and anisotropy

An **isotropic material** is one whose properties don't change that much depending on the direction. Metals are very commonly isotropic materials, as they are mostly a matrix of atoms sharing electrons. Materials whose properties change depending on the direction are called **anisotropic**. Several composites are an example of this, such as carbon fiber reinforced polymers.

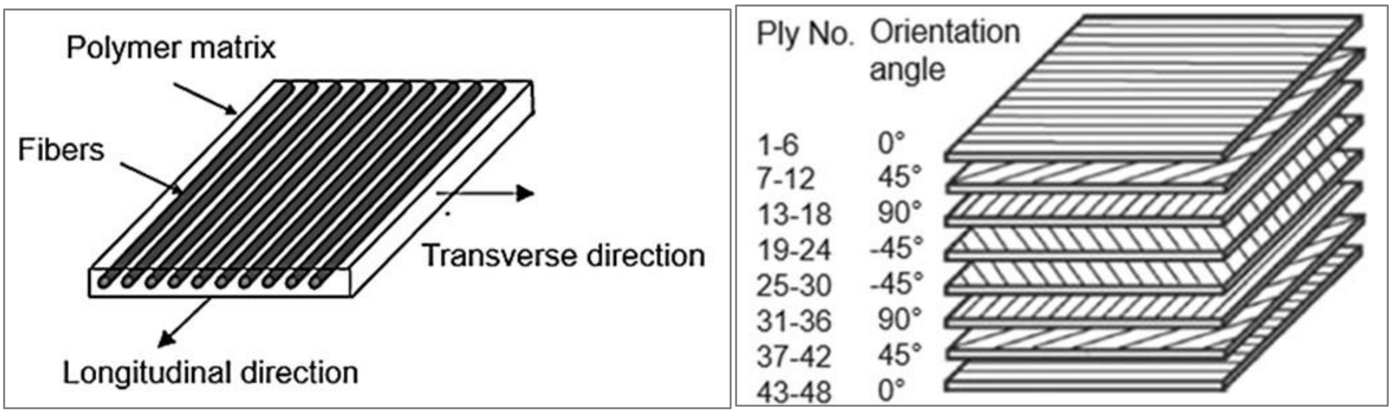


Figure 7 *Fiber reinforced polymer stronger in longitudinal direction (fibers are great in tension) and weaker in transverse direction (only the polymer matrix is resisting stress). To overcome weak non-longitudinal stiffness, layers with fibers in other orientations can be added.*

Within anisotropic materials, there is a special group called **orthotropic materials** which are those whose properties do depend on the direction, like all anisotropic materials, but only in some 3 perpendicular (orthogonal) axis. Wood is a good example of an orthotropic material. The directions would be radial (least stiffness), circumferential and axial (highest stiffness). Another example is sheet metal that has been rolled.

Figure 8 *Wood is a specific kind of anisotropic material – orthotropic.*

For an isotropic material, regarding mechanical elastic behavior, we mostly only need to know two independent values to know all we need for engineering purposes. These can be the Young's modulus and Poisson's ratio. Or the bulk modulus and shear modulus. These are related in pairs so we can get ones from the others. For the anisotropic, we might have to know up to 21 values.

Numerical integration

This basically means to perform integration of definite integrals using numerical methods (lots of simple calculations a lot of times, basically calculating area under function by eg. dividing into small rectangles). We learned how to do integrations analytically. For example:

To solve

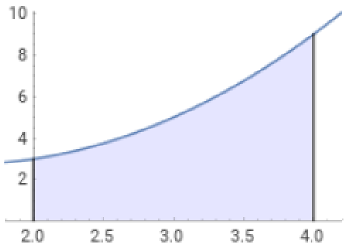
$$\int_2^4 (y^2 - 3y + 5)\,dy$$

We would first integrate the term

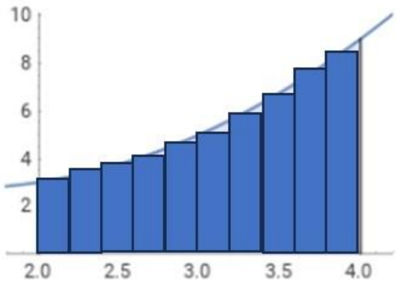
$$\frac{y^3}{3} - \frac{3\,y^2}{2} + 5\,y + \text{constant}$$

And then insert the interval from 2 to 4.

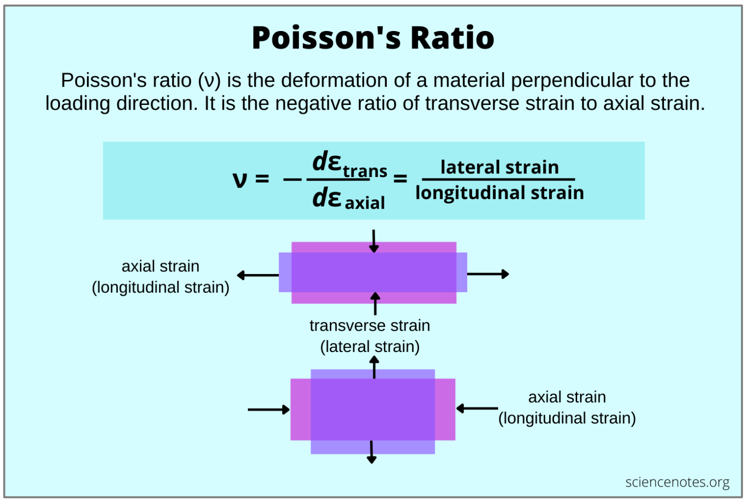
$$= \frac{32}{3} \approx 10.667$$



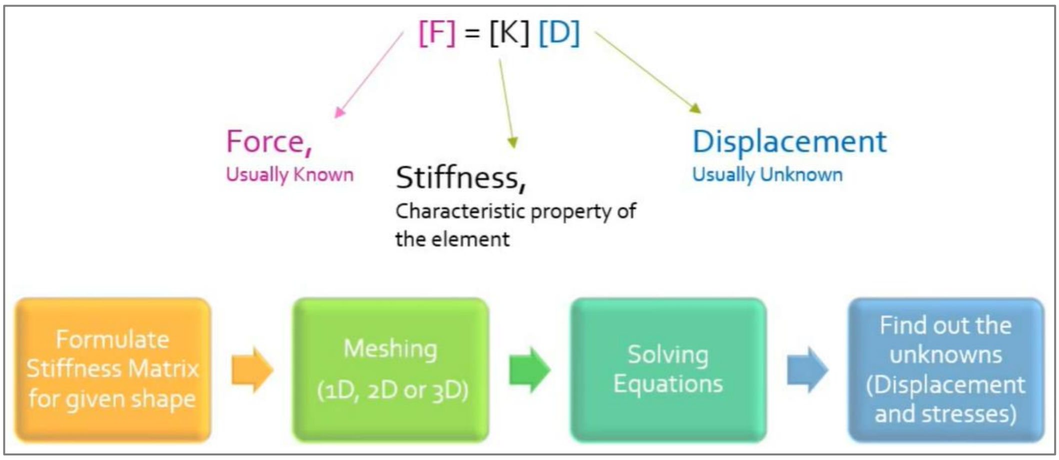
But that requires a certain degree of intelligence and adaptability that is not available in FEM software. Therefore we use numerical integration. Which in a way is kind of like FEM for integration. We find ways to calculate by dividing the area in smaller simple areas that the computer can easily calculate.



Poisson’s Ratio



Stiffness / Elasticity matrix



- It is a property of the material, and depends on its characteristics. Crystalline structure, or laminates (in case of composites), … . This is related to its isotropy, anisotropy or orthopy.
- The elasticity matrix is 3x3 for 2D and 6x6 for 3D.
- Let's talk about the 3D matrix. A 6x6 matrix contains 36 elements. However, for FEA even in the worst case we only need to know 21. The worst case would be when the material is completely anisotropic.

For one direction, we have

$$\sigma = \mathrm{E} \cdot \varepsilon$$

For 2D or 3D this becomes

$$[\sigma] \times \frac{[k]}{E} = [\varepsilon]$$

$$[\sigma] = [\mathrm{C}] \times [\varepsilon]$$

Where [C] is the elasticity or stiffness matrix. Some people also use D to represent it.

For an isotropic material that becomes. And some people seem to call this "compliance matrix" which they say is the inverse of the stiffness matrix.

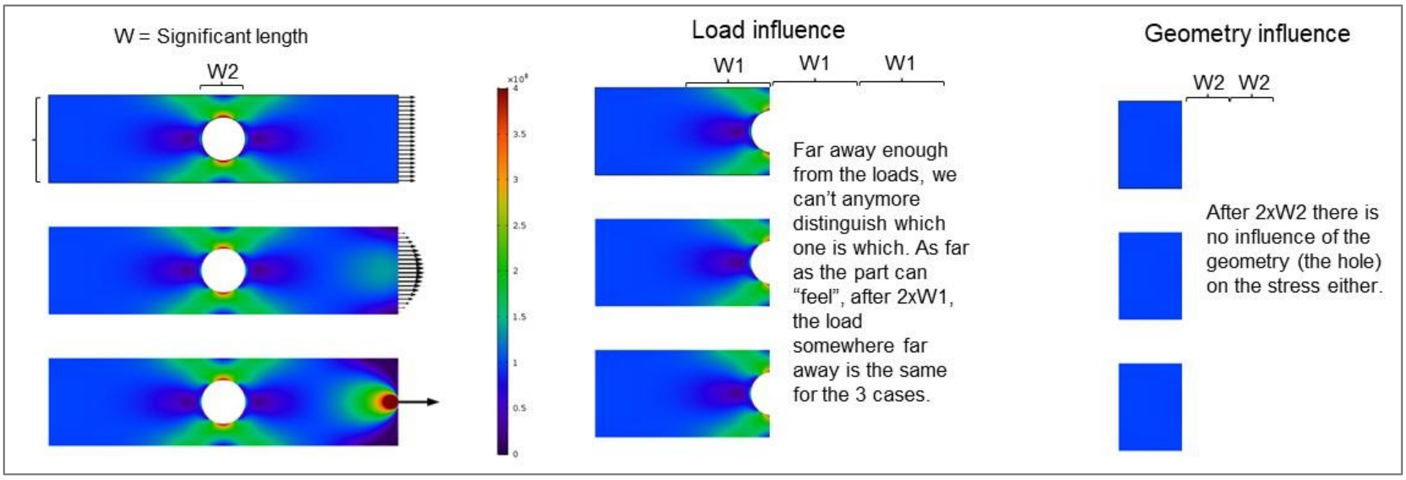
$$\begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \varepsilon_{yz} \\ \varepsilon_{zx} \\ \varepsilon_{xy} \end{bmatrix} = \frac{1}{\mathrm{E}} \begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1+\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 1+\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 1+\nu \end{bmatrix} \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{yz} \\ \sigma_{zx} \\ \sigma_{xy} \end{bmatrix}$$

Source: https://en.wikipedia.org/wiki/Hooke%27s_law

For more information: https://www.comsol.com/blogs/modeling-linear-elastic-materials-how-difficult-can-it-be/

St Venant's principle

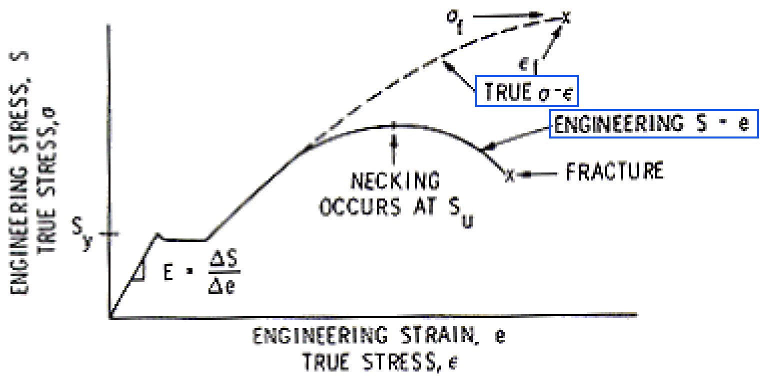
"The difference between 2 different but equivalent loads (eg. force applied spread over on an edge or applied on a point) becomes very small at sufficiently large distance."



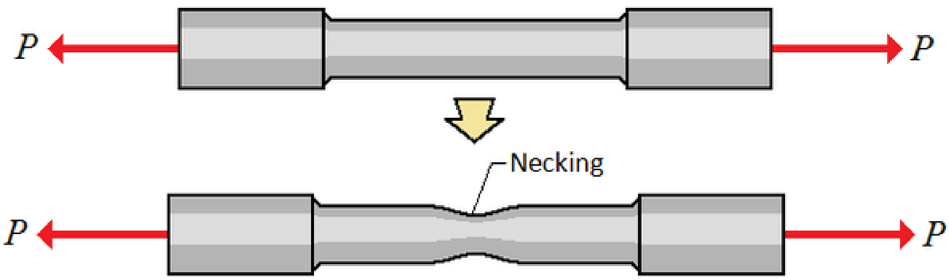
Rule of thumb include 2-3 significant lengths away from local influence. This can be geometric features such as widths, diameters.

True vs Engineering stress

The Stress-strain curves for materials have typically been obtained by testing on a displacement controlled tensile testing machine. The machine applies a steadily increasing displacement and measures the force. We then get a Force vs displacement graph. Knowing the (initial) length of the sample we convert displacement to strain, and the (initial) section area the force to stress. In other words, the machine measures force and displacement, and these are converted into stress and strain by dividing by the original sample size. Quantities calculated in this way are called engineering stresses and strains.



Above the Ultimate Tensile Strength (UTS, aka σ_u), the sample begins to exhibit necking. This occurs when plastic deformation starts to be concentrated at a particular weak spot in the sample:



Necking reduces the area of the sample. While the stress on the neck may continue to go up, the reduction in area means there are less atoms there holding it all together, and therefore the force the sample is able to endure decreases.

If we keep track of the reduction in sample area, we can divide the force by the actual area (rather than the original area), and the displacement by the actual length. This leads to quantities called true stresses and strains. Indeed, the true stress-strain graph doesn't show any negative gradient:

True stress is the actual material's stress, the stress the material, the atoms are really experiencing. Engineering stress is more the "stress" as calculated from the undeformed shape. So it's more useful for engineering purposes.

Now, to actually answer the question: UTS is an engineering property, not a materials scientist's property. For many applications, engineers don't really care about the shape of a component when it fails - only the maximum load it can carry. Defining the UTS in terms of the engineering stress (and hence the original dimensions of the component) makes it easy to work out how much load can be carried. If the UTS is exceeded, the sample will fail (unless the load is removed) because it's all downhill from there!

Tensor

A tensor is a container of values. One way to think about it is

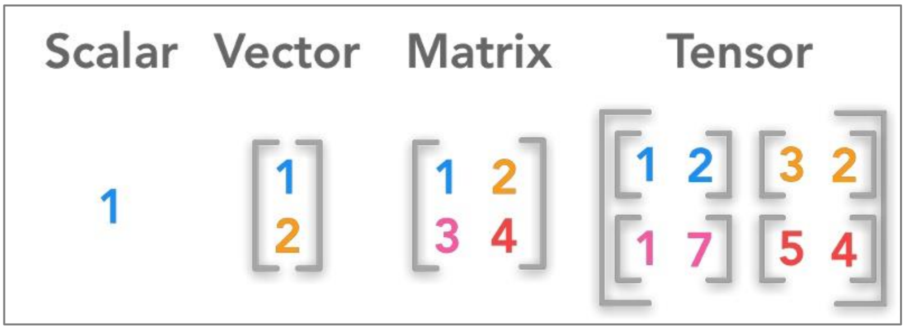
- A scalar is a 0^{th} dimension, or 0^{th}-order tensor.
- A vector (also called an 'array') a 1D or 1^{st} order tensor.
- A matrix a 2D or 2nd order tensor.
- From then on we stopped having fancy names and we just call them n-th order tensors. A 3D tensor would be like a cube of numbers. A 4D tensor is harder to visualize because we are 3D beings. But that's the idea.
- Like a matrix can be the result of vector multiplications (cross and dot product), we can hold inside a tensor also operations between other tensors.

We call out specific elements in a matrix as

$$c_{ij}$$

For a 3x3 matrix we get 9 elements. But what if we want something like a 3x3x3x3 thing? That we call a 4^{th} order tensor and it would have 81 elements.

$$c_{ijkl}$$

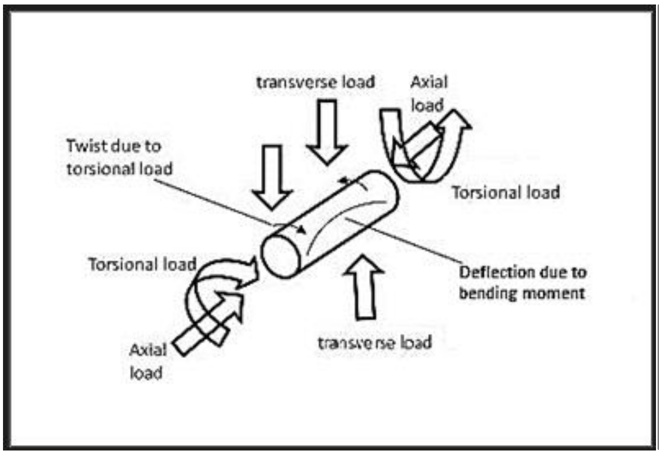


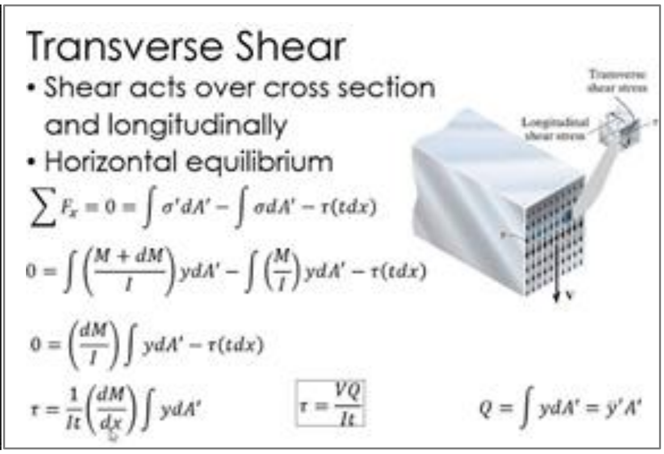
$$\underline{\mathbf{f}} = \underline{\underline{\boldsymbol{K}}}\,\underline{\mathbf{d}} \implies \begin{bmatrix} f_1 \\ f_2 \\ f_3 \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix} \begin{bmatrix} d_1 \\ d_2 \\ d_3 \end{bmatrix}$$

Examples of physical problems that fit the above template are listed in the table below.[2]

Problem	$\mathbf{f}$	$\mathbf{d}$	$\boldsymbol{K}$
Electrical conduction	Electrical current $\mathbf{J}$	Electric field $\mathbf{E}$	Electrical conductivity $\boldsymbol{\sigma}$
Dielectrics	Electrical displacement $\mathbf{D}$	Electric field $\mathbf{E}$	Electric permittivity $\boldsymbol{\varepsilon}$
Magnetism	Magnetic induction $\mathbf{B}$	Magnetic field $\mathbf{H}$	Magnetic permeability $\boldsymbol{\mu}$
Thermal conduction	Heat flux $\mathbf{q}$	Temperature gradient $-\boldsymbol{\nabla} T$	Thermal conductivity $\boldsymbol{\kappa}$
Diffusion	Particle flux $\mathbf{J}$	Concentration gradient $-\boldsymbol{\nabla} c$	Diffusivity $\boldsymbol{D}$
Flow in porous media	Weighted fluid velocity $\eta_{\mu}\mathbf{v}$	Pressure gradient $\boldsymbol{\nabla} P$	Fluid permeability $\boldsymbol{\kappa}$

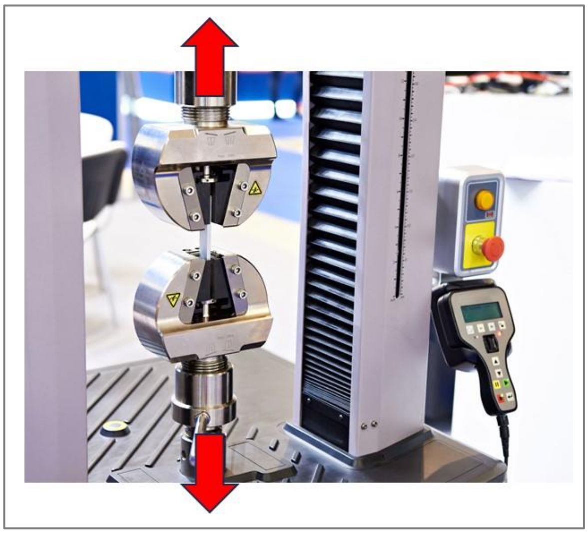
Transverse things





Von Mises (equivalent) Stress

When we go to a material database and get that the yield strength of a certain steel is 600 MPa, this value was obtained from a uniaxial test, meaning, the force was applied on a single direction. This is how most test data is established for materials.



However, what the material often experiences in combined types of loadings is more complex. Imagine for example, instead of just tension: tension + torsion + bending. So we have material data for uniaxial stress, but we need to know how it behaves in a 3D stress state. What do we do? We try to find an equivalency.

Several yield criteria have been developed to deal with this, by relating analytical definitions with real life tests.

In general a yield criteria will define whether an isotropic material has reached a state of yield due to a triaxial load state. The big advantage is that the stress state is described as a scalar - there is now a single value we can compare against yield.

However, caution is needed when using Von Mises stress as a general stress indicator as sign (compression/tension) and sense (direct/shear) are lost. Therefore, component stresses should be investigated, not just Von Mises.

Questions

- What is a linear elastic material?
    - It's a material that obeys to Hooke's law. The question is already a bit tricky because it's assuming there are such materials that are 100% linear elastic. Ceramics are probably very close to that though. But metals are only linear elastic until app. the yield limit. After that they don't follow Hooke's law anymore and so the linear elastic model cannot be used anymore. They become non-linear.
- What is a degree of freedom?
    - The minimum number of parameters (motion, coordinates, temperatures, etc.) required to define the position and state of any entity in that space and analysis. The total DOF of a mesh = no. nodes * DoF / node. In static structural analysis you can have maximum 6 DoF per node (3 translations and 3 rotations).
    - The number of DOF depends on the element type (1D, 2D, 3D), the family of the element (thin shell, plane stress, plane strain, membrane, etc.) and the type of analysis.
- Why do we say "non-X". Non-linear, non-static analysis. Instead of …, dynamic.
    - Because there are several types of analysis and we cannot just give them one name or it doesn't matter for that purpose. For example, for the non-linear, because if a study is not linear, it could have some random form (polynomial, whatever). But it doesn't matter which. Linear means the

response is proportional to the cause. As for dynamic, because some people consider analysis like fatigue or modal not to be dynamic but also not static. So things can be moving quickly but are not considered dynamic.

Sources

- Youtube channel where I took some of the pictures. Great about Ansys-FEA learning:

https://www.youtube.com/channel/UCAMyTkJwYHtkDxNG_ROCg_g/videos

- A blog with a good way of explaining:

https://enterfea.com/blog/

- Good comprehensive training:

http://www.fetraining.net/

J. Dean, Introduction to the Finite Element Method (FEM) - Lecture 2 First and Second Order One Dimensional Shape Functions

---