Mathematical Elements of CAD

Finite Element Method/Analysis (FEM/FEA)

- It is necessary to use mathematics to comprehensively understand and quantify any physical phenomena, such as structural behavior. Most of these processes are described using partial differential equations (PDEs). Differential equations can not only describe processes of nature but also physical phenomena encountered in engineering mechanics.
- Engineering analysis of structural systems have been addressed by deriving differential equations relating the variables of basic physical principles such as equilibrium. However, once formulated, solving the resulting mathematical models is often impossible, especially when the resulting models are non-linear partial differential equations. Only very simple problems of regular geometry such as a rectangular or a circle with the simplest boundary conditions were manageable. These partial differential equations (PDEs) are complicated equations that need to be solved in order to compute relevant quantities of a structure (like stresses (ϵ), strains (ϵ), etc.) in order to estimate a certain behavior of the investigated component under a given load.
- For a computer to solve these PDEs, numerical techniques have been developed over the last few decades and one of the most prominent today is the finite element method.

Mathematical Elements of CAD

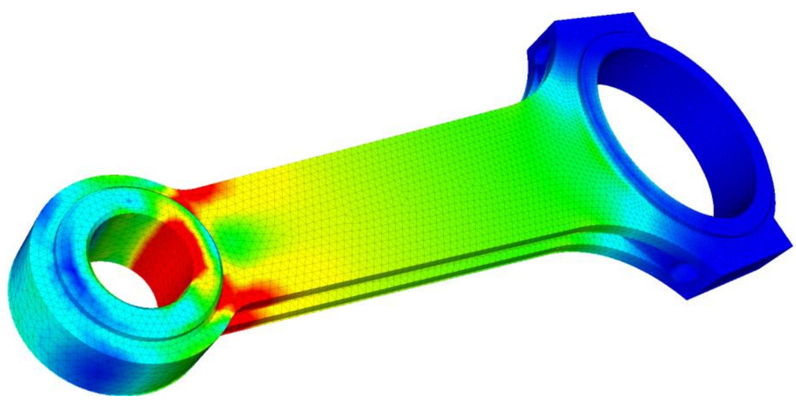
Finite Element Analysis/Method

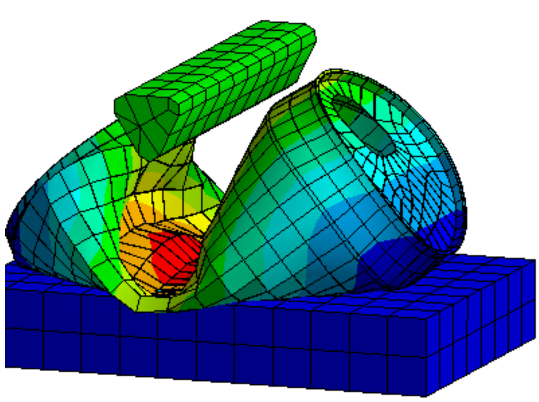
- The Finite Element Analysis (FEA) is the simulation of any given physical phenomenon using the numerical technique called Finite Element Method (FEM). Engineers use it to reduce the number of physical prototypes and experiments and optimize components in their design phase to develop better products, faster.
- The finite element method (FEM) is the dominant discretization technique in structural mechanics. The basic concept in the physical interpretation of the FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at a set of nodal points.
- The response of the mathematical model is then considered to be approximated by that of the discrete model obtained by connecting or assembling the collection of all elements. The disconnection-assembly concept occurs naturally when examining many artificial and natural systems. For example, it is easy to visualize an engine, bridge, building, airplane, or skeleton as fabricated from simpler components.

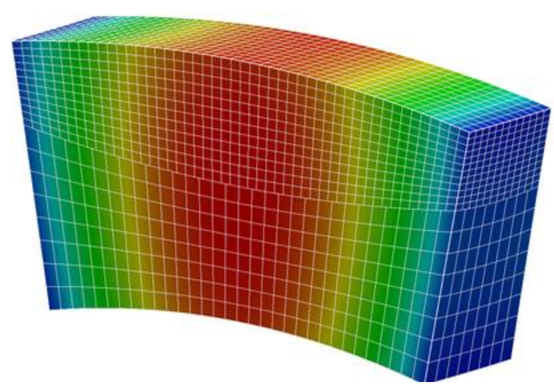
Mathematical Elements of CAD

Finite Element Analysis/Method

It is important to know that FEA only gives an approximate solution of the problem and is a numerical approach to get the real result of these partial differential equations. Simplified, FEA is a numerical method used for the prediction of how a part or assembly behaves under given conditions. It is used as the basis for modern simulation software and helps engineers to find weak spots, areas of tension, etc. in their designs. The results of a simulation based on the FEA method are usually depicted via a color scale that shows for example the pressure distribution over the object.







Mathematical Elements of CAD

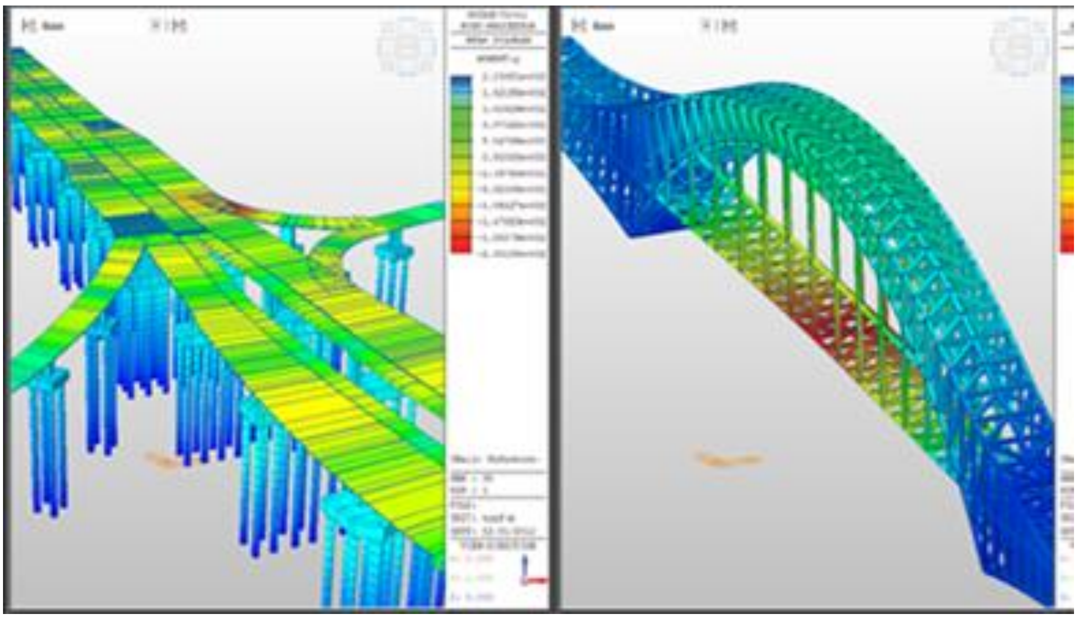
Finite Element Analysis/Method

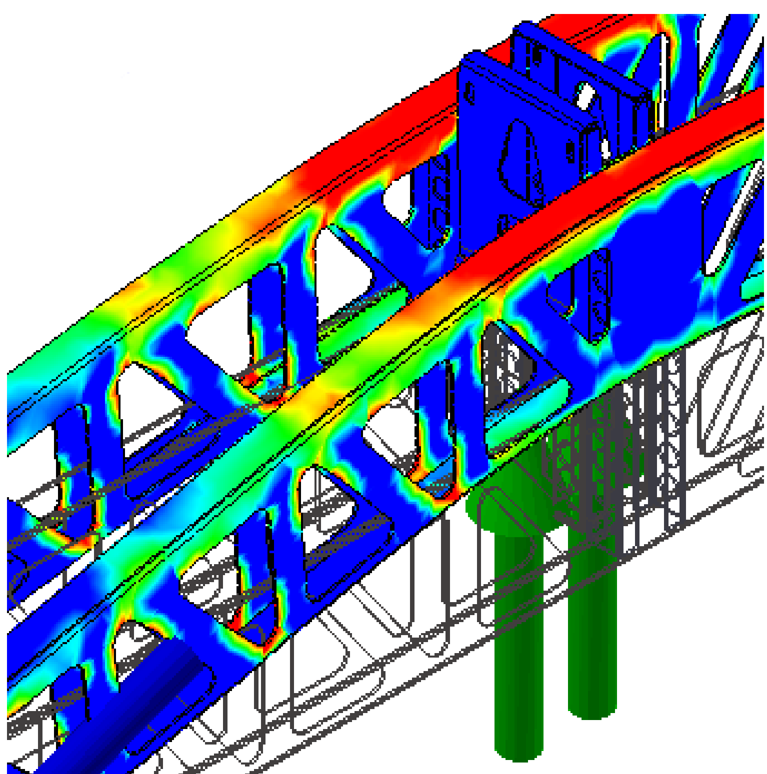
Depending on one's perspective, FEA can be said to have its origin in the work of Euler, as early as the 16^{th} century. However, the earliest mathematical papers on Finite Element Analysis can be found in the works of Schellbach [1851] and Courant [1943].

FEA was independently developed by engineers in different industries to address structural mechanics problems related to aerospace and civil engineering. The development for real-life applications started around the mid-1950s as papers by Turner, Clough, Martin & Topp [1956], Argyris [1957], and Babuska & Aziz [1972] show. The books by Zienkiewicz [1971] and Strang & Fix [1973] also laid the foundations for future developments in FEA software.

Mathematical Elements of CAD

Finite Element Analysis/Method

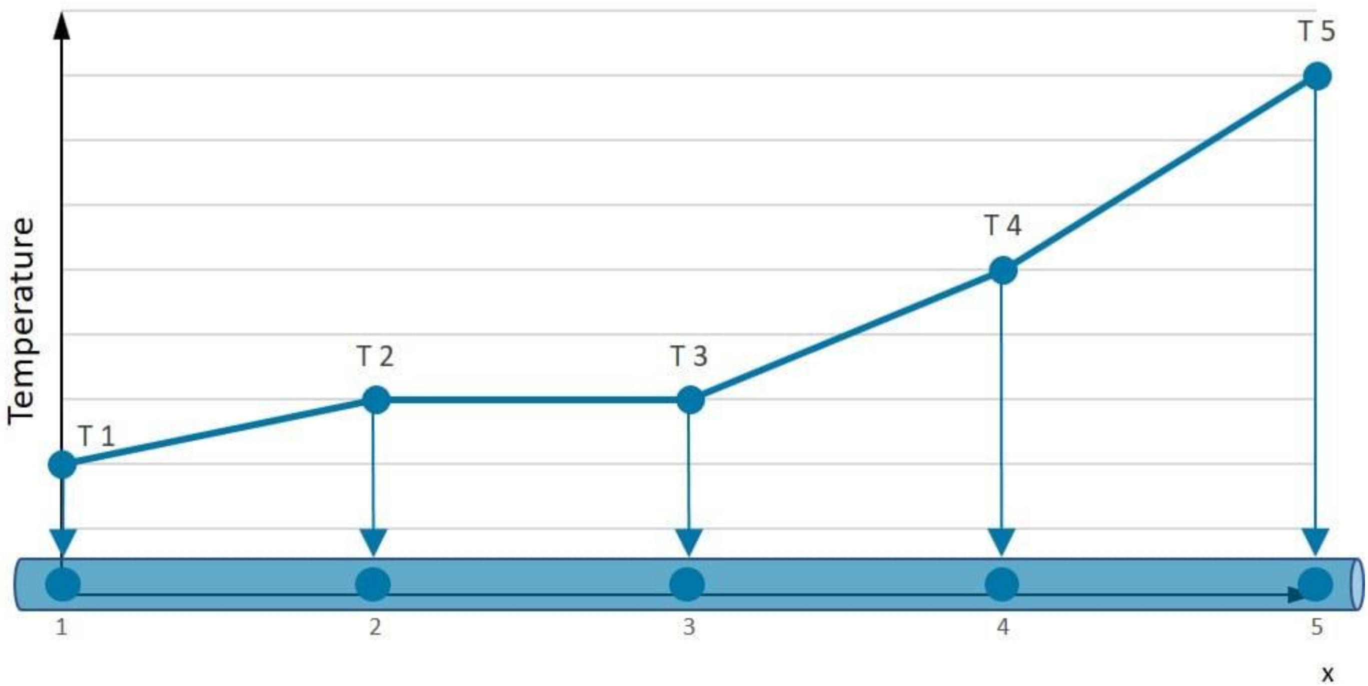




Mathematical Elements of CAD

Finite Element Analysis/Method

To be able to make simulations, a mesh, consisting of up to millions of small elements that together form the shape of the structure, needs to be created. Calculations are made for every single element. Combining the individual results gives us the final result of the structure. The approximations we just mentioned are usually polynomial and in fact, interpolations over the element(s). This means we know values at certain points within the element but not at every point. These ‘certain points’ are called nodal points and are often located at the boundary of the element. The accuracy with which the variable changes is expressed by some approximation for eg. linear, quadratic, cubic, etc. In order to get a better understanding of approximation techniques, we will look at a one-dimensional bar. Consider the true temperature distribution T(x) along the bar in the picture below:


Temperature
T 1
T 2
T 3
T 4
T 5
1
2
3
4
5
x

Mathematical Elements of CAD

Finite Element Analysis/Method

Let's assume we know the temperature of this bar at 5 specific positions (Numbers 1-5 in the illustration). Now the question is: How can we predict the temperature in between these points? A linear approximation is quite good but there are better possibilities to represent the real temperature distribution. If we choose a square approximation, the temperature distribution along the bar is much more smooth. Nevertheless, we see that irrespective of the polynomial degree, the distribution over the rod is known once we know the values at the nodal points. If we would have an infinite bar, we would have an infinite amount of unknowns (DEGREES OF FREEDOM (DOF)). But in this case, we have a problem with a "finite" number of unknowns.

A system with a finite number of unknowns is called a discrete system. A system with an infinite number of unknowns is called a continuous system.

The line illustrated at the top shows this principle for a 1D problem. These can be two or three dimensional problems as well.

Mathematical Elements of CAD

Finite Element Analysis/Method

Let's assume we know the temperature of this bar at 5 specific positions (Numbers 1-5 in the illustration). Now the question is: How can we predict the temperature in between these points? A linear approximation is quite good but there are better possibilities to represent the real temperature distribution. If we choose a square approximation, the temperature distribution along the bar is much more smooth. Nevertheless, we see that irrespective of the polynomial degree, the distribution over the rod is known once we know the values at the nodal points. If we would have an infinite bar, we would have an infinite amount of unknowns (DEGREES OF FREEDOM (DOF)). But in this case, we have a problem with a "finite" number of unknowns.

A system with a finite number of unknowns is called a discrete system. A system with an infinite number of unknowns is called a continuous system.

The line illustrated at the top shows this principle for a 1D problem. These can be two or three dimensional problems as well.

Mathematical Elements of CAD

<u>**Finite Element Analysis/Method**</u>

<u>**FEM**</u> Solution Process / Procedures

- Divide structure into pieces (elements with nodes) (discretization/meshing)
- Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure (forming element matrices)
- Solve the system of equations involving unknown quantities at the nodes (e.g., displacements)
- Calculate desired quantities (e.g., strains and stresses) at selected elements

Mathematical Elements of CAD

Finite Element Analysis/Method

Element Types

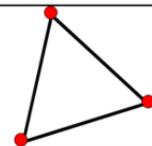
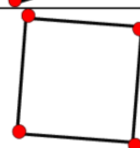
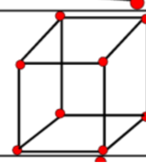
- Dictated by features of FEA software
- 1-D to 3-D
  - ✓ Pure stress, stress + bending, thermal, fluid
- Triangular Element
  - ✓ Model transitions between find and coarse grids
  - ✓ Irregular structures
  - ✓ Warped surfaces
- Quadrilateral element
  - ✓ Should lie on an exact plane; else moment on the membrane is produced.
- 3-D two-node truss
  - ✓ 3 DOF (DEGREE OF FREEDOM), no bending loads
  - ✓ Stress constant over entire element
  - ✓ Typically used in two-force members ("RBAR")
- 3-D two-node beam
  - ✓ Specify Moments of inertia in both local X and local Y
  - ✓ 6 DOF Mx My Mz
  - ✓ A typical beam

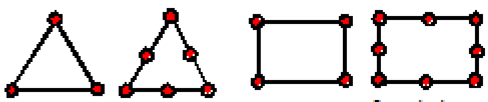
Mathematical Elements of CAD

Finite Element Analysis/Method

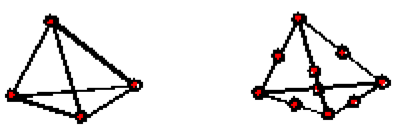
Element Types

- 3-D 4-node plate
  - ✓ Modeling of structures where bending (out of plane) and/or membrane (in-plane)
  - ✓ Stress Play equally important roles in the behavior of that particular structure
  - ✓ Each node has 6 DOF
  - ✓ Must specify plate thickness
- 2-D 4-node solid
  - ✓ “Isoparametric four-node solid"
  - ✓ Common in 2-D stress problems and natural frequency analysis for solid structure.
  - ✓ "Thin" in that stress magnitude in third direction is considered constant over the element thickness.
  - ✓ 2 translational DOF and no rotational DOF
- 2-D 2-node truss
  - ✓ No bending loads
  - ✓ Uniaxial tension-compression
  - ✓ Straight bar with uniform properties from end to end
  - ✓ 2 DOF, no rotation
- 2-D 2-node beam
  - ✓ 3 DOF - translation x, y and rotation about z
- Any cross section for which moment of inertia can be computed

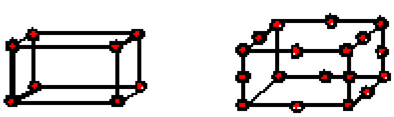
Finite element name	Type	Geometric Representation
One-dimensional finite element	Straight nodal lines	
	Curved nodal lines	
Two-dimensional finite element	triangular	
	quadrilateral	
Tridimensional finite element	hexahedron	



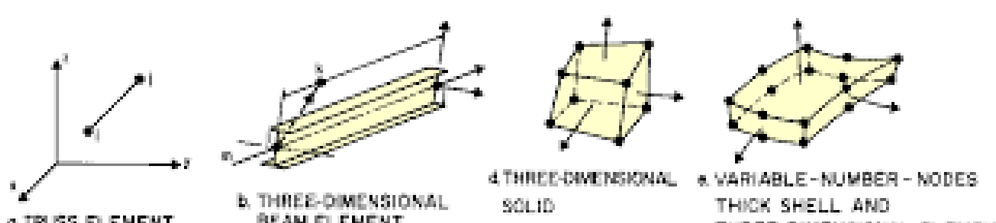
3 noded triangle | 6 noded triangle | 4 noded quadrilateral | 8 noded quadrilateral

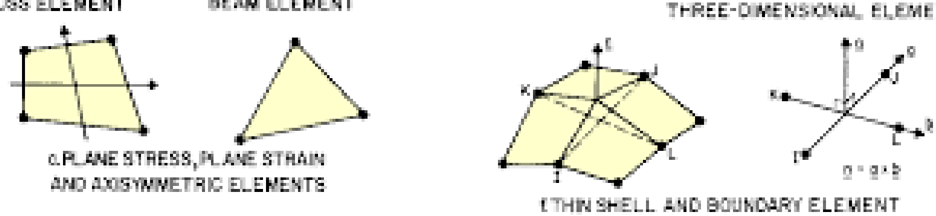


4 noded tetrahedron | 10 noded tetrahedron



8 noded brick | 20 noded brick





Finite Element Analysis/Method

Shape Functions

The shape function is the function which interpolates the solution between the discrete values obtained at the mesh nodes. Therefore, appropriate functions have to be used and, as already mentioned, low order polynomials are typically chosen as shape functions.

Mathematical Elements of CAD

Finite Element Analysis/Method

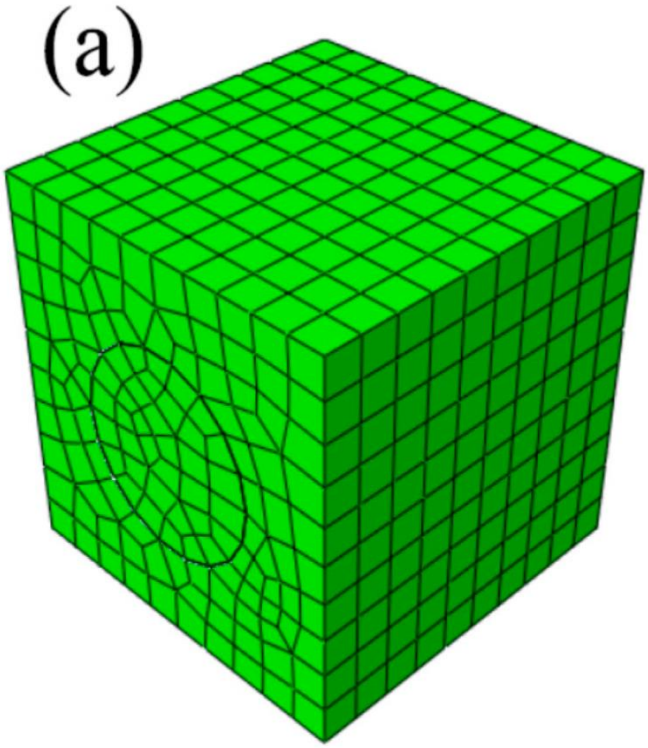
Meshing

- **Coarse:** Faster computation; not concerned about stress concentrations, singularities, or warping. Not near changes in geometry or displacement constraints or changes in material including thickness.
- **Fine:** Best approximation but at the cost of the computation time. Look for disproportionate stress level changes from node to node or plate to plate and large adjacent node displacement differences to determine if need to refine the mesh.
- Nodes should be defined at locations where changes of geometry or loading occur. Changes in geometry relate to thickness, material and/or curvature.
- A simple check, if you can, is to decrease the mesh size by 50%, re-run analysis, and compare the change of magnitude of stresses and strains. If there is no significant change, then ok.
- In most companies, all of this knowledge of mesh size will be known and might be set a FEA control file.

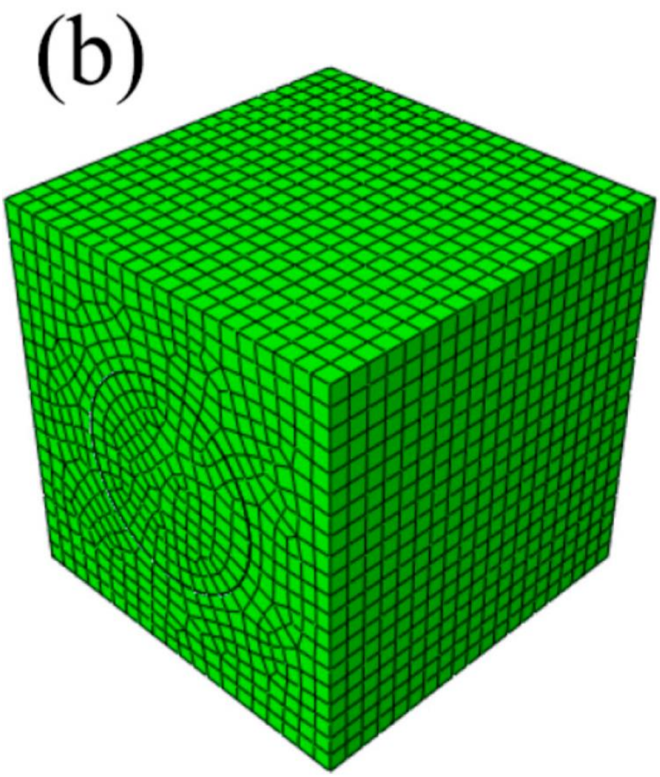
Mathematical Elements of CAD

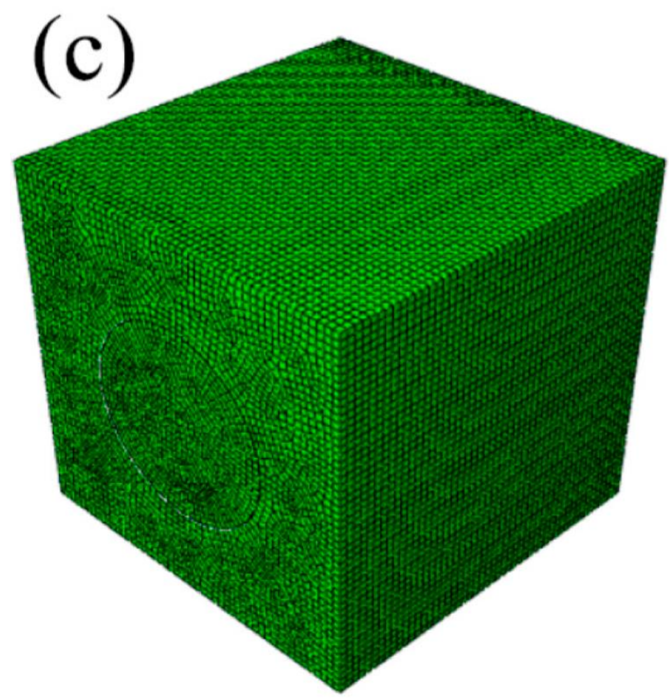
Finite Element Analysis/Method

Mesh & Elements



level	elements	max. error
0	256	15.8252%
1	565	9.9129%
2	1202	6.3601%
3	2279	4.8492%



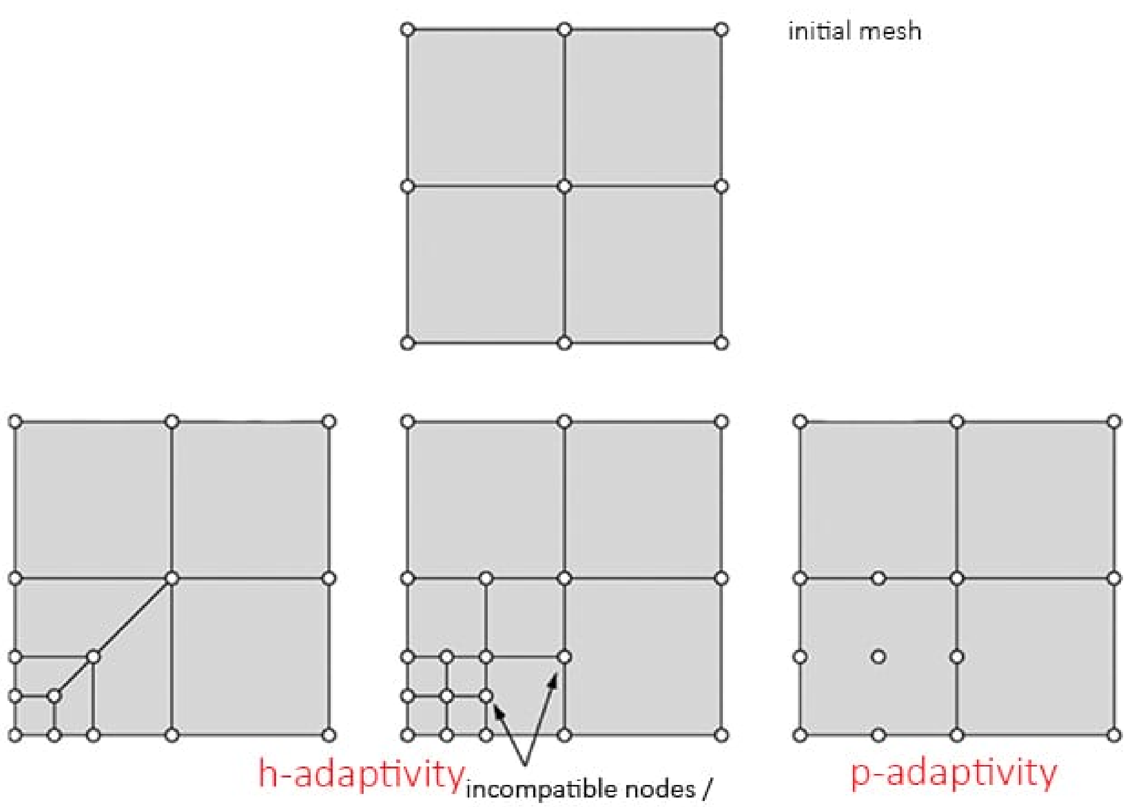


Finite Element Analysis/Method

Mesh Convergence

One of the most important issues in computational mechanics that affect accuracy is mesh convergence. This is related to how small the elements need to be to ensure that the results of an analysis are not affected by changing the size of the mesh.

It is important to first identify the quantity of interest. At least three points need to be considered and as the mesh density increases, the quantity of interest starts to converge to a particular value. If two subsequent mesh refinements do not change the result substantially, then one can assume the result to have converged.



Mathematical Elements of CAD

Finite Element Analysis/Method

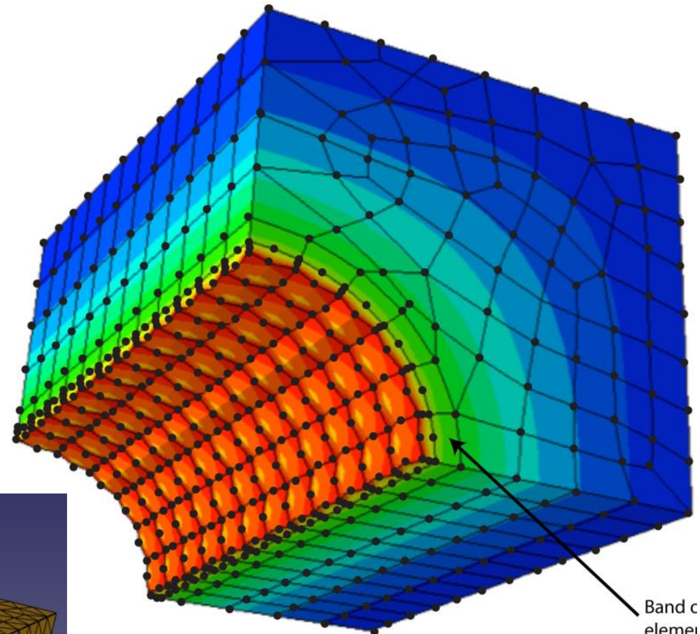
Meshing Refinement

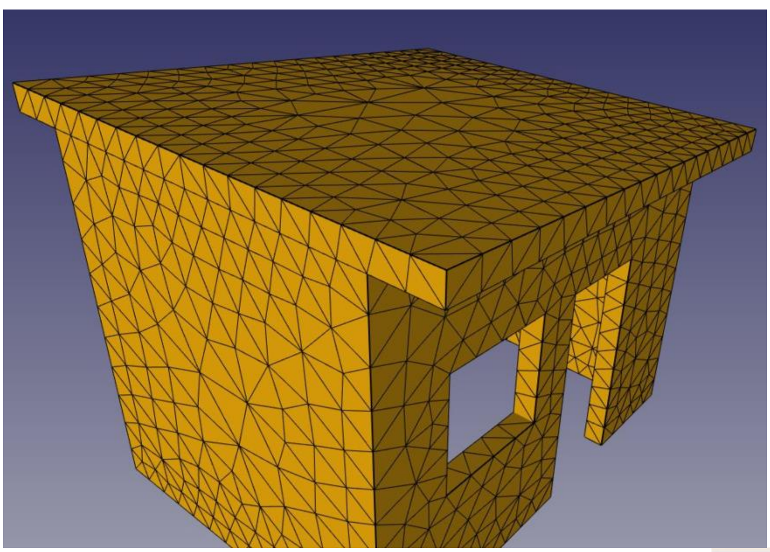
- Going into the question of mesh refinement, it is not always necessary that the mesh in the entire model is refined. St. Venant's Principle enforces that the local stresses in one region do not affect the stresses elsewhere. Hence, from a physical point of view, the model can be refined only in particular regions of interest and further have a transition zone from coarse to fine mesh. There are two types of refinements (h- and p-refinement) as shown in the figure, h-refinement relates to the reduction in the element sizes, while p-refinement relates to increasing the order of the element.

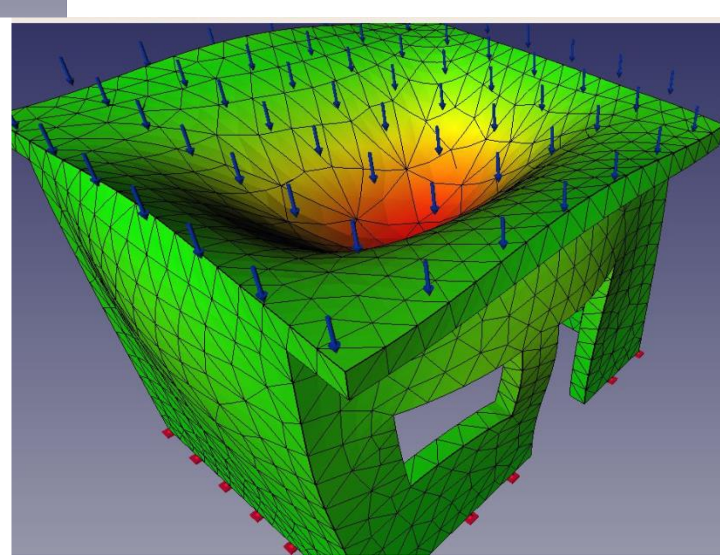
Mathematical Elements of CAD

Finite Element Analysis/Method

Mesh & Elements







Mathematical Elements of CAD

Finite Element Analysis/Method

Degrees of Freedom

- Constrain structure to prevent rigid body motion
- Restrict motion in non-desirable directions

Applied Forces

- Static
- Static distributed
- Transient
- Harmonic vibratory

Mathematical Elements of CAD

Finite Element Analysis/Method

A typical finite element analysis on a software system requires the following information:

- Nodal point spatial locations (geometry)
- Elements connecting the nodal points
- Mass properties
- Boundary conditions or restraints
- Loading or forcing function details
- Analysis option