ELEATION

ELEMENT SIMULATION ACADEMY

BY APOORV BAPAT

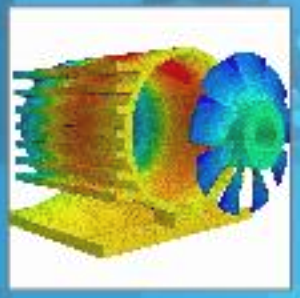
Element Analysis Frequency Responses Analysis

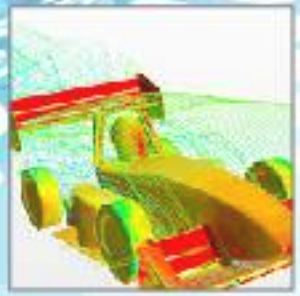
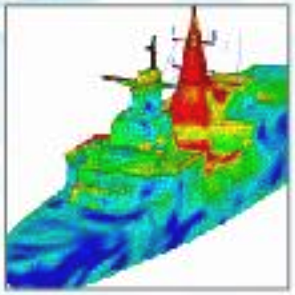
Front Impact Analysis

Bump Analysis Roll Cage Analysis Element 3D Meshing

Side Impact Analysis Thermal Analysis

Meshing Rear Impact Analysis Rool Over Anal




What are the Different Ways to Validate any design?

Design Validation means whether the component which we are going to manufacture will sustain for various loading conditions or not.
In other words , we can say that **Design Validation** is a process to find the results like Stresses, Displacement, Strain, Fatigue, Eigen Values, Heat Flux, etc. may or may not come on my model which leads failure.

There are 3 Methods to Validate any Design:

1. **Analytical Method**
2. **Numerical Method**
3. **Experimental Method.**

Analytical Method :

Analytical Method is classical approach to find results on any design / component which can me manufactured.
Analytical Method is a generic process which involves solution techniques based on formulas / Methods / theorems.

Advantages of Analytical Method :

1. It gives 100% accurate results - but only for Simple Problems (having uniform dimension)

Disadvantages of Analytical Method :

1. You can only find the results of which you have formulas.
2. Giving inputs related to certain Manufacturing processes is not possible. – Manufacturing Defects, Pre-Stresses in Material, corrosion effects, rate of corrosion.
3. Giving 100% accurate material properties is difficult.
4. Can't find Area of complex geometries.
5. Time Consuming.
6. Accuracy can't be believed blindly.
7. Calculations can be done only in single points

Numerical Method :

Numerical Method is a mathematical tool designed to solve numerical problems. The implementation of a numerical method with an appropriate convergence check in a programming language is called a numerical algorithm.
FEM / FEA is a branch of Numerical Method.

Advantages of Numerical Method :

1. We can solve real life complicated problems.
2. Results can be achieved faster.
3. Results can be achieved on Entire Geometry.
4. Applicable even if a physical prototype is not available.

Disadvantages of Numerical Method:

1. Approximate Results.

Experimental Method

Experimental Method is a technique to find results (stresses, deformation, strain, etc.) on Design / object via performing the various experimental process on Prototype.

Advantages of Experimental Method :

1. Physical / Manufacturing errors can be considered in Results.
2. Actual Measurements

Disadvantages of Experimental Method :

1. Time Consuming and need expensive set up.
2. Applicable only if Physical Prototype is available.
3. Results cannot be believed blindly and a minimum of 3 to 5 prototypes must be test.

Numerical Methods

There are total 4 Numerical Methods:

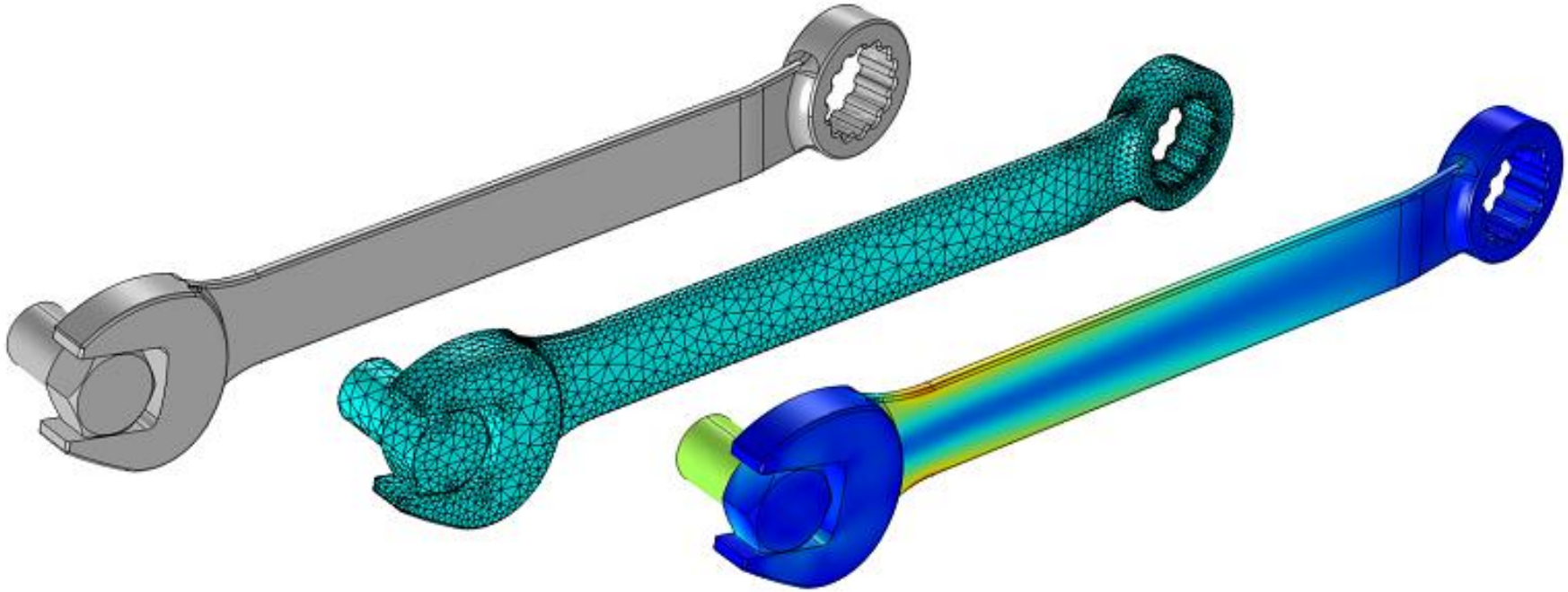
1. Finite Element Method (**FEM**) / Finite Element Analysis (**FEA**)
2. Boundary Element Method (**BEM**)
3. Finite Volume Method (**FVM**)
4. Finite Difference Method (**FDM**)

1. Finite Element Method (FEM) :

The **finite element method** (**FEM**) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as **finite element** analysis (FEA).

Are FEA and FEM different?

Finite Element Method (FEM) and Finite Element Analysis (FEA) both are one and the same. The term “FEA” is more popular in industries while “FEM” is more popular at universities.



What is FEM?

Finite – Any continuous object has infinite degrees of freedom and it is not possible to solve the problem in this format. The Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).

Element – All of the calculations are made at a limited number of points known as nodes. The entity joining nodes and forming a specific shape such as quadrilateral or triangular is known as an Element. To get the value of a variable (say displacement) anywhere in between the calculation points, an interpolation function (as per the shape of the element) is used.

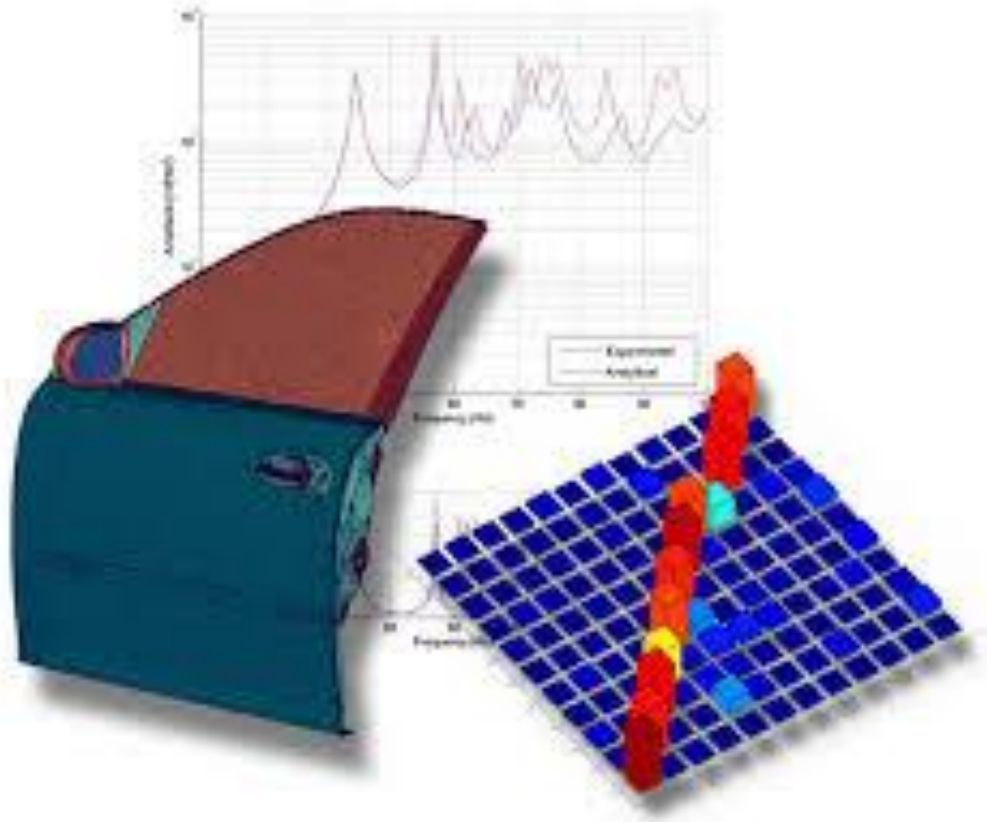
Method - There are 3 methods to solve any engineering problem. Finite element analysis belongs to numerical method category.

Advantages of FEA / FEM

1. Better **Visualization** of Failure Location
2. Lower down the **Design cycle time**
3. Decrease the **Number of prototypes**
4. Cut the **Testing** cost
5. **Optimum design** can be achieved faster

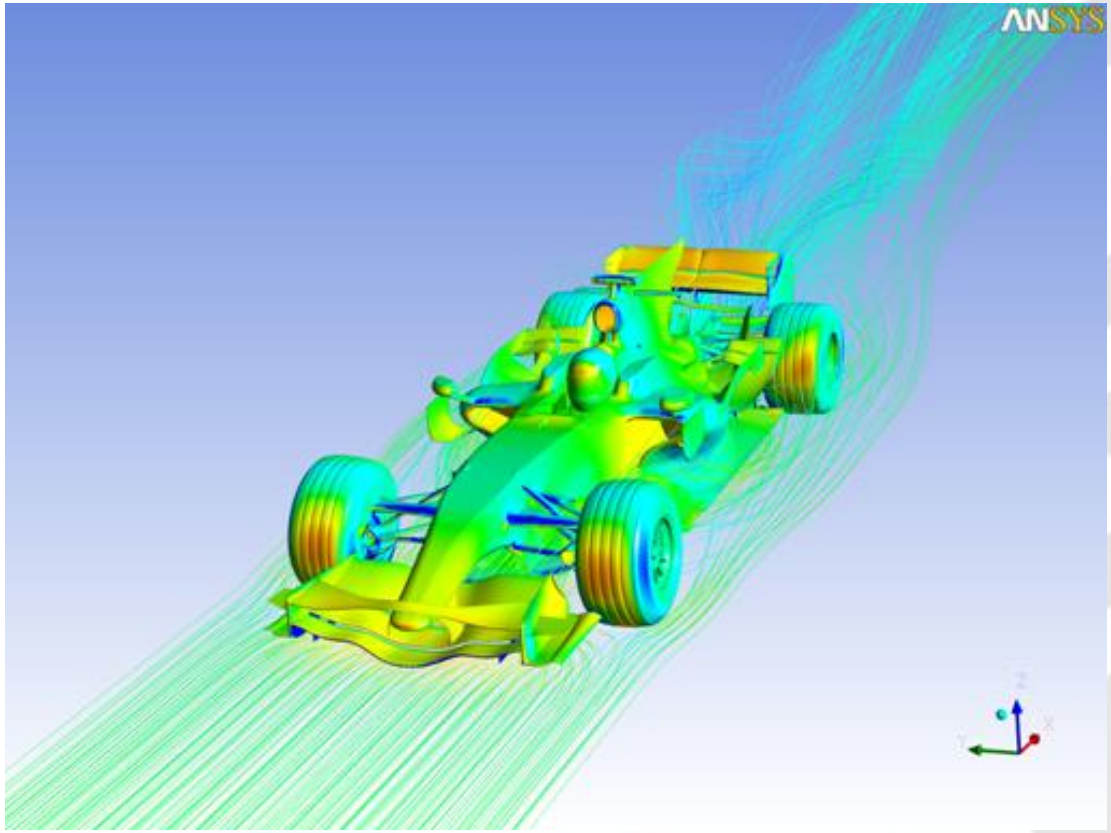
Boundary Element Method (BEM) :

The **boundary element method (BEM)** is a technique for solving a range of engineering/physical problems. It is most often used as an engineering design aid - similar to the more common finite element method - but the **BEM** has the distinction and advantage that only the surfaces of the domain need to be meshed.



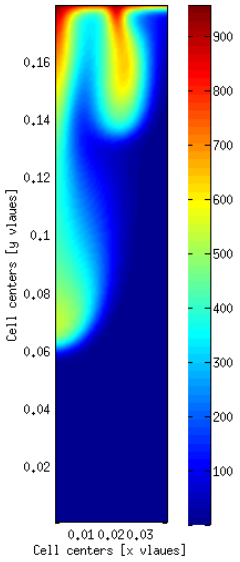
Finite Volume Method (FVM) :

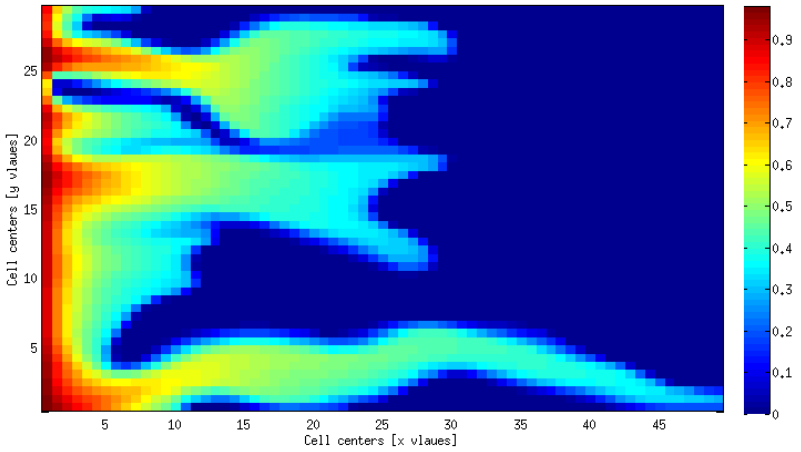
Most **Computational Fluid Dynamics (CFD)** software is based on FVM. The unit volume is considered in Finite Volume Method (similar to element in finite element analysis). Variable properties at the nodes include pressure, velocity, area, mass, etc. It is based on the Navier - Stokes equations (Mass, Momentum, and Energy conservation equilibrium equations).

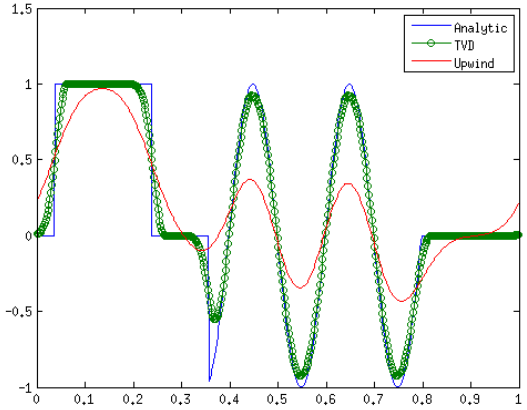


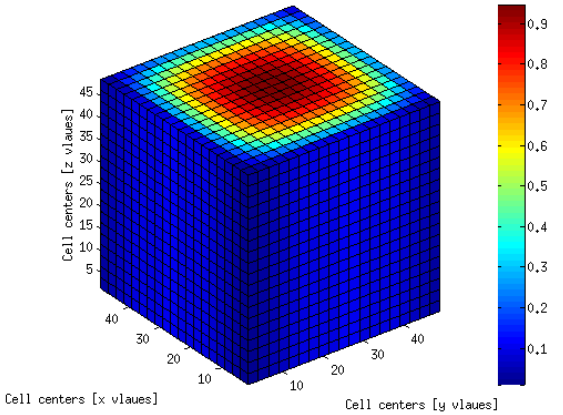
Finite Difference Method (FDM) :

Finite Element and **Finite Difference Methods** share many common things. In general the Finite Difference Method is described as a way to solve differential equation. 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.





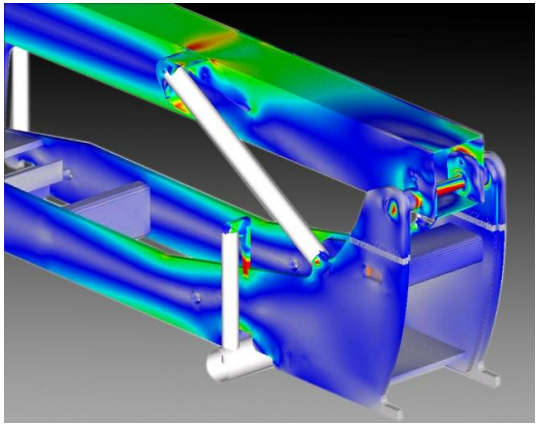


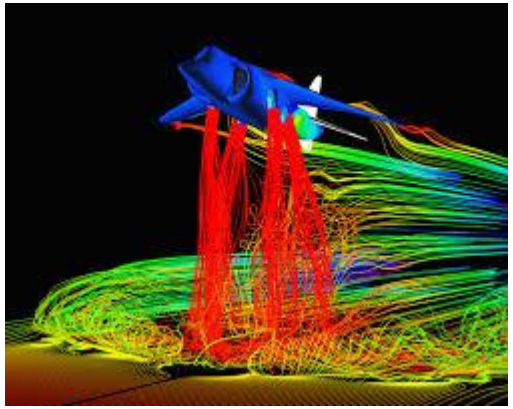


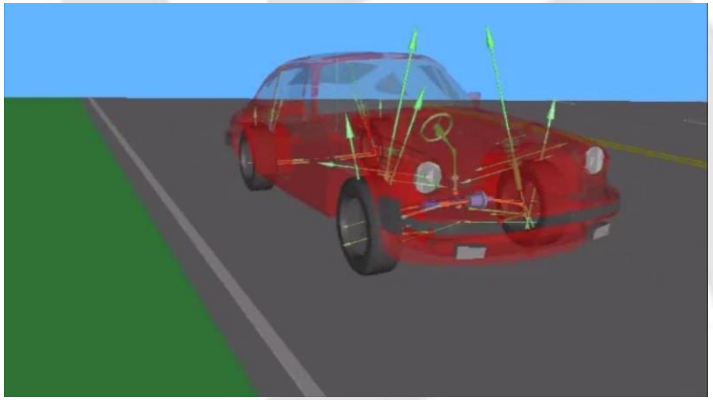
What is CAE ?

Computer-aided engineering (**CAE**) is the broad usage of computer software to aid in engineering analysis tasks. It includes Finite Element Analysis (FEA), Computational Fluid Dynamics (CFD), Multibody dynamics (MBD), Optimization, etc.

- CAE retrieves description and geometry from a CAD database
- It is used in almost every industry such as aerospace, automobile manufacturing
- CAE depends on CAD



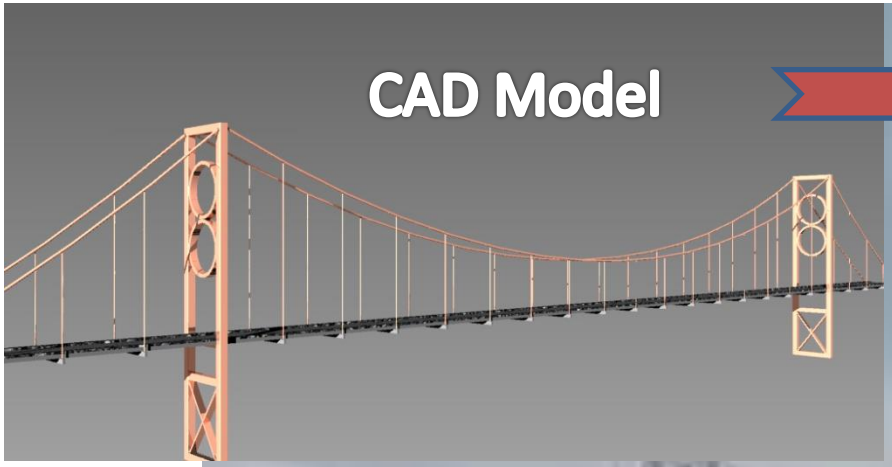




Difference Between CAD and CAE

CAD is about creation/generation of the 3D model (Design).

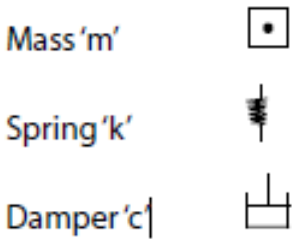
CAE is about testing its characteristics (Material, Strength, Fatigue…) using engineering methods.

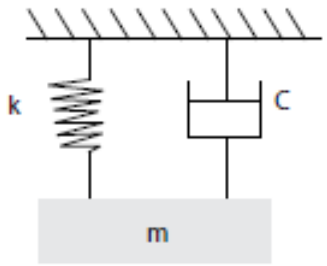


Discretization of problem:

All real life objects are continuous. This means there is no physical gap between any two consecutive particles. As per material science, any object is made up of small particles, particles of molecules, molecules of atoms, and so on and they are bonded together by the force of attraction. Solving a real life problem with the continuous material approach is difficult. **The basis of all numerical methods is to simplify the problem by discretizing (discontinuation) it**. **In other words, nodes work like atoms and with gap in between is filled by an entity called an element**. **Calculations are made at the nodes and results are interpolated for the elements.**

From a mechanical engineering point of view, any component or system can be represented by three basic elements:





All the numerical methods including the Finite Element Method follow the discrete approach. Meshing (nodes and elements) is nothing but the discretization of a continuous system with infinite degree of freedoms to a finite degree of freedoms.

Why do We Carry Out Meshing?

The basic idea of FEA is to make calculations at only limited (Finite) number of points and then interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it's just not possible to solve the problem in this format. Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).

The Finite Element Method only makes calculations at a limited (Finite) number of points and then interpolates the results for the entire domain (surface or volume).

What is DOF (Degree Of Freedom)

The minimum number of parameters (motion, coordinates, temp. etc.) required to define the position of any entity completely in the space is known as a degree of freedom (dof).

All of the elements do not always have 6 dofs per node:

- The number of dofs depends on the type of element (1-D, 2-D, 3-D).
- The family of element (thin shell, plane stress, plane strain, membrane, etc.).
- The type of analysis. For example, for a structural analysis, a thin shell element has 6 dof/ node (displacement unknown, 3 translations and 3 rotations) while the same element when used for thermal analysis has single dof /node (temperature unknown)

For a new user it is a bit confusing but there is lot of logical, engineering, and mathematical thinking behind assigning the specific number of DOFs to different element types and families.

Element Shape Characteristics

In general, four shapes are possible: point, line, area, or volume. :

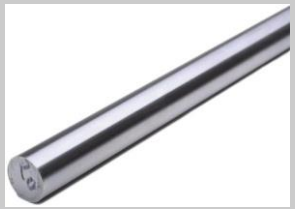
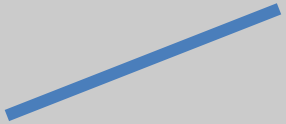
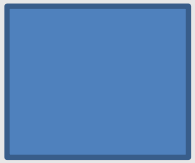
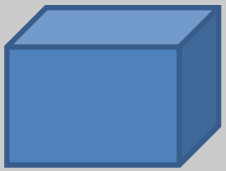
1. A point element is typically defined by one node, such as a mass element.

2. A line element is typically represented by a line connecting two or three nodes. Examples are beams, spars, pipes, etc.
   - **We Use 1D Elements when One of the Dimensions of the object / component is very very large as compared to other two dimensions.**

3. An area element has a triangular or quadrilateral shape. Example Quard and Tria
   - **We Use 2D Elements when Two of the Dimensions of the object / component is very very large as compared to the Third Dimension (Generally Thickness)**

4. A volume element has a tetrahedral or brick shape and is usually a 3-D solid element.
   - **We Use 3D Element when all the three dimensions are comparable.**

ELEATION
ELEMENT SIMULATION ACADEMY
BY APOORV BAPAT

Type	**Actual Model**	**Element Shape** (Geometric properties defined by Nodes)	**Additional Requirement** (Actual Volume Calculation)
1D Element	Steel Rod 	Beam element with 2 nodes (Length Only) 	Cross – Sectional Parameters (Area) V = L x A
2D Element	Sheet Metal Component 	Quard Element with 4 nodes (Area Only) 	Thickness (T) V = A x T
3D Element	Metal Cube 	Hexa Element with 8 Nodes (Volume) 	None Element itself has volume.

The 10 questions you should be able to answer…

The following is a list of the 10 questions you should be able to answer about any given FE simulation. It is also a good checklist to bring along to presentations by others as a guide to assure all your questions are answered. If you have not or cannot answer these questions you need to go back and understand why.

1. **What is the problem you are trying to solve?**
2. **What geometry are you using?**
3. **What solution type are you using?**
4. **What is your material model?**
5. **What are your element selections?**
6. **What are your constraints?**
7. **What are your loads?**
8. **How are you combining the loads and the constraints?**
9. **What are the results?**
10. **What are your recommendations?**