

# Introduction to FEA

Author: Arpit Agarwal, Structures Lead

## **Finite Element Method (FEM)**

FEM or Finite Element Method is a mathematical method for solving physical problems. FEM evaluates solutions at various points called nodes.

## **Finite Element Analysis (FEA)**

FEA or Finite Element Analysis on the other hand is the implementations of FEM. There are numerous different software programs available to implement FEM, known as FEA software.

## **Basics of FEA**

The whole process of any FEA could be divided into three broad sections irrespective of which software you use:

1. **Preprocessing:** When you do an analysis you have to do modeling and meshing. This is essentially the process to set up the what type of FEA you will be conducting and
  - a. **Meshing:** The process of breaking the geometric model into small pieces in order to create nodes and elements is called meshing. In other words meshing is the process of converting the geometric model to a FEA model.
  - b. **Boundary Conditions:** In order to solve a FEA problem you have to apply force (these force could be structural, thermal, magnetic or of any form) and you have to resist at some points of the model from the effects generated by these forces. For example to see the stress in a cube you may think of applying force at the top surface and can constrain it at the parallel bottom surface. These set of loads and constraints are called boundary conditions.
2. **Solving:** After preprocessing we have to use a solver to find solution out of the FEA problem. In solidworks we could either use the native solidworks solver or can export or save the pre-processed mesh for solving with other external solver like ANSYS, SimScale, etc.
3. **Post-Processing:** After solving you view the results of your FEA problem. This section is called post-processing. Here we can see different types of images, plots and graphs of the result.