

SimStreamlinedforDesign:

TUTORIAL: *Understanding Simulation Terms*

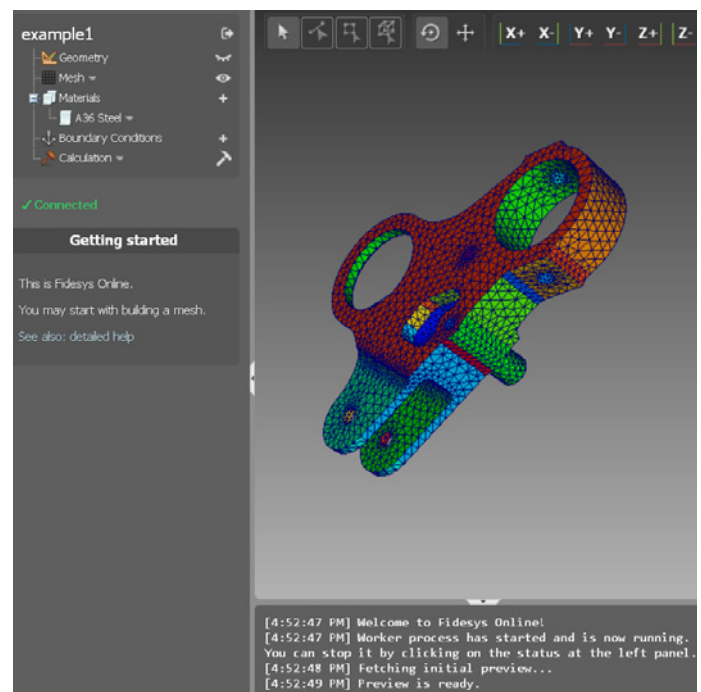
This tutorial provides a brief and simple explanation of the terms that you need be familiar with in order to use and understand basic Structural Analysis. The terms are explained in clear English with no math involved. Understanding basic terms like Boundary Conditions, Forces, Pressures, Displacements, and Stresses will allow you to effectively use simulation to improve your understanding of how structural performance is affected by design changes.

Setting up a structural analysis Case may seem daunting at first. SimForDesign.com has streamlined the process of Simulation For Design to dramatically reduce the required level of expertise and to demystify the process. However, a basic understanding of some of the principles and terms used is quite helpful.

Let's start with a few fundamental concepts. SimForDesign.com leverages the industry proven Finite Element Method. You can go to the [Wikipedia definition](#) for a full definition in precise mathematical terms, however, the fundamental principal is that any object can be subdivided into a subset of simple parts that each span a finite space and whose behavior can be understood. You can think of this as mathematical lego blocks.

Then it is just a matter of managing the information for each simple part and its neighbors. The parts are referred to as elements and the collections of elements is referred to as a mesh. Meshing is the process of creating the mesh from your design geometry. There are several approaches to meshing. SimForDesign.com

[.com](http://SimForDesign.com) provides a fully automatic creation of the mesh with default characteristics (that you can change) to make the process of going from your design to a mesh as simple as possible. Unlike other tools for structural analysis, with SimForDesign.com you can proceed with effective analysis without needing to understand the details about the mesh that was generated and you only need to be aware that one was generated based on your settings. The accuracy of your results is a function of the mesh - but we will discuss that in a separate tutorial.



SimStreamlinedforDesign:

TUTORIAL: *Understanding Simulation Terms*

The benefit of this approach is that is widely used and is a proven technology. Indeed the Finite Element Method is the workhorse technology for Structural Analysis and other simulations.

Now that you have an understanding of what the Finite Element Method is we will explain how this is used to calculate structural response. The Finite Element Method organizes the mesh elements by their connections to neighbors to define a total response model. For those who like to explore this is usually called the [Stiffness Matrix](#) and can be thought of as a collection of 3D springs with various stiffnesses. The Finite Element Method then uses this to create a set of relationships between forces applied to the model and the resultant movement of the model at the element connection points (nodes).

The first thing that is solved for is how much each node moves in each direction when the force is applied. This movement is called the Displacement. For 3D solid meshes used by [SimForDesign.com](#) there are three (3) components of displacement, one in each direction

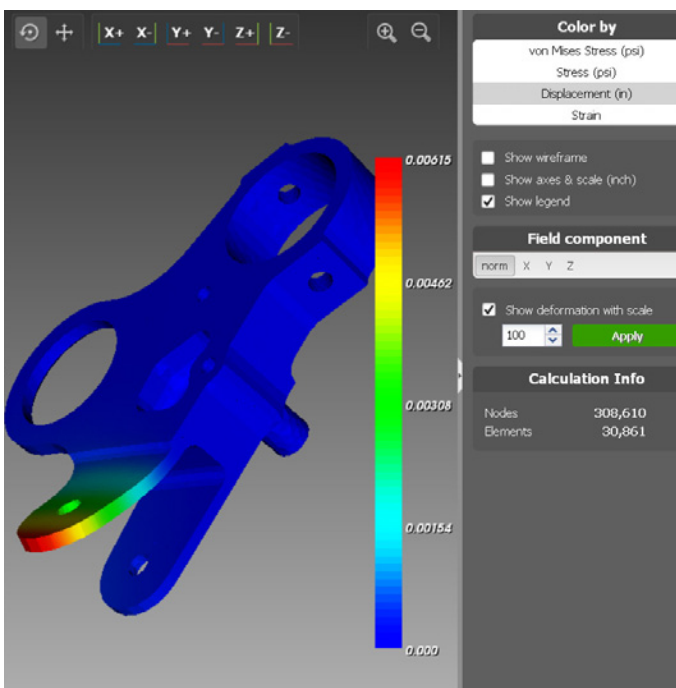
(x,y,z). [SimForDesign.com](#) also calculates the magnitude of the displacements which is the amount of combined displacement from each direction to form a single movement along a single path. The Displacements are calculated for each node in the model.

In order to calculate these displacements you need to specify sections of the model that are not allowed to move in particular directions or have a specified displacement. These specified displacements are referred to as displacement boundary conditions. For 3D solid meshes used by [SimForDesign.com](#) displacements can be applied to parts of the model (curves, surfaces, and volumes) in each of the three major directions (x,y,z) or combinations of these directions.

The other missing part to calculate the forces applied to model. These forces are referred to as loading boundary conditions. Loads can be defined as a gravity force, a force on each node in a particular direction, or as a pressure applied to part of the model. A pressure is a force distributed over surfaces on the model and is applied normal to the model.

Once the displacements have been calculated for each node the strains for each element are then calculated for each direction and each combination of directions (shear). Strain is the definition of how much a material has been stretched (or compressed) when compared to its original size. Here's the [Wikipedia definition](#) for those who want to know.

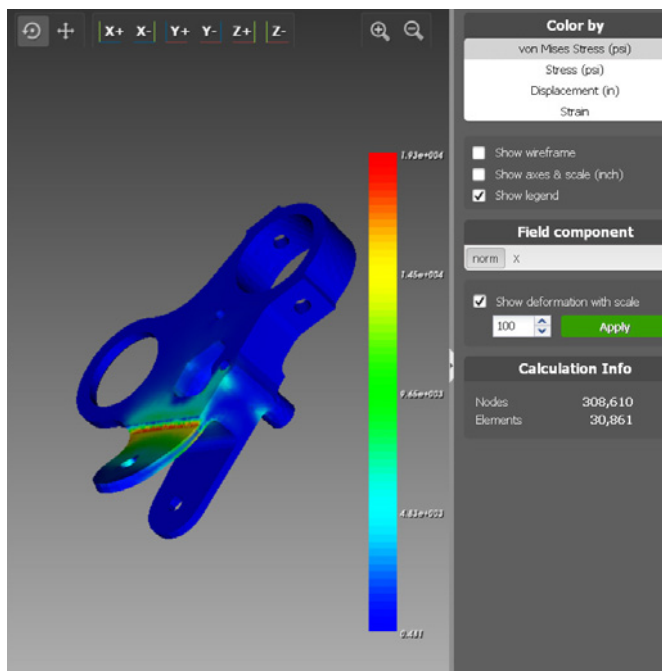
The determination of strains allows the Finite Element Method to then calculate stresses. The stress in a particular direction for an element can be thought of as the force applied on an element from its neighbors divided by the effective area of the element in that direction. Here's the [Wikipedia definition](#) for those who want to know. Stresses for each element are then calculated for each direction and each combination of directions (shear). There are various techniques to convert the combination of stresses into a single value that can be used to compare against allowable stresses. SimForDesign.com allows you to review all of the



SimStreamlinedforDesign:

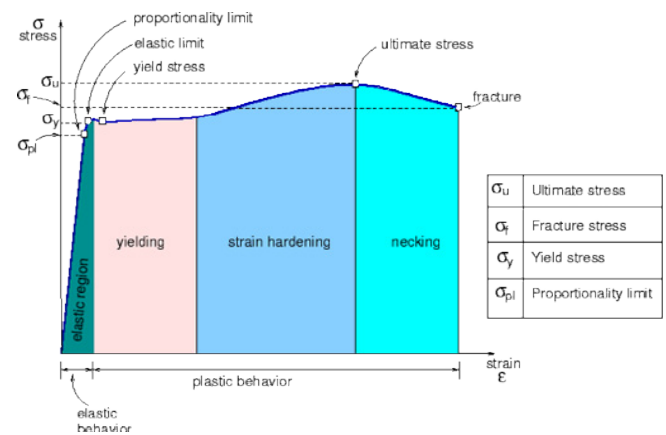
TUTORIAL: *Understanding Simulation Terms*

components and a commonly used single value based on combination of the stresses in different directions called the Von-Mises Equivalent Stress.



In order to calculate the displacements, strains and displacements the Finite Element Method has to understand the material properties. This is one of the areas where the Finite Element Method can be very complex to setup. Wow, that sounds complicated - not really though. Linear response area just means the part of the response that is linear to the load applied, for example twice the load means twice the displacement and twice the stress. For most metallic materials and many others the design space is usually defined as the stress that is the yield stress of the material divided by a safety factor. If you are using non-metallic materials you may want to review the stress-strain curve of the material to determine if you are in the linear range (elastic limit).

Materials like rubber and some plastics need special consideration. Below is a typical stress-strain curve for metallic materials.



Since the behavior is limited to the linear range we only need two parameters as follows that can easily be looked up for your materials:

- Young's Modulus = the ratio of the stress along an axis over the strain along that axis in the linear range of material behavior ([Wikipedia definition](#))
- Poisson's Ratio = the negative ratio of strain in normal directions to strain in a particular direction. When a material is compressed in one direction, it usually tends to expand in the other two directions perpendicular to the direction of compression. ([Wikipedia definition](#))
- Density = mass or weight per unit volume. ([Wikipedia definition](#))

SimForDesign.com has simplified this dramatically by working only within the linear response of materials and by providing default values for A36 Steel based upon values from Wikipedia. [Young's modulus](#) for A36 steel is 200 GPa (29,000,000 psi). A36 steel has a [Poisson's ratio](#) of 0.260