Finite Element Analysis Revealed:
uncovering engineering's latest design tools

So you've heard of Finite Element Analysis or FEA, but what's it really all about? Brett Longhurst from Bremar Automotion outlines the basics of a FEA stress analysis using a suspension upright as an example.

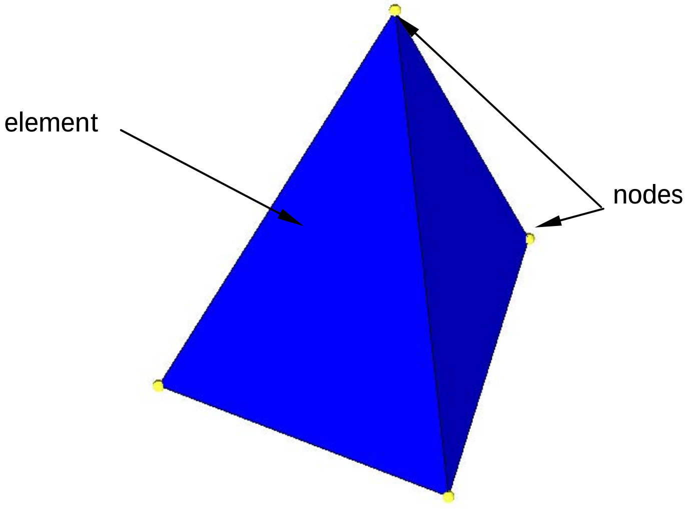
Finite Element Analysis or FEA as it's known today, has been around since the 60s when it was first put into practice by the knowledgeable folks at NASA. Thanks to the advances in computing power since then, FEA has become a common tool in engineering design work, allowing the designer or engineer to evaluate the strengths and weaknesses of their designs 'virtually'. This allows huge savings in both time and money, significantly reducing the amount of physical prototypes and testing required throughout the design process.

So how does it all work then? Well the general process for a FEA stress analysis involves three main steps:

1. Pre Processing
2. Solving
3. Post Processing

1. Pre Processing

Pre processing involves building a mathematical model of the part you want to analyse. There are a few ways to do this, but by far the most common method is to take a 3D CAD model of the part and break it down into thousands of tiny pieces that are a regular shape, such as a cube or a pyramid, through a process called meshing. Each tiny piece is called an element, (hence '**finite element** analysis') and the corners of the elements are called nodes.



Tetrahedral (pyramid) Element and Nodes

FEA Benefits:

:: less physical prototypes
:: less physical testing
:: more design iterations
:: better understanding of part
:: time & money savings
:: lighter, stronger part

FEA Process:

1. Pre Processing
(meshing)

2. Solving
(computer calculation)

3. Post Processing
(results interpretation)

So why split the model up like this? Well, not surprisingly, there's no mathematical formula to directly calculate the stress in a complex shape such as a suspension upright or a crankshaft. There are however, formulas to calculate the stresses and displacements in a cube or pyramid when a load is applied to it. So the whole premise of FEA is to take a complex shape and break it down into tiny, regular shaped elements for which stress and strain can be calculated, then add all those results together to figure out the overall stress and strain within the part and the way it deforms due to the applied loads.

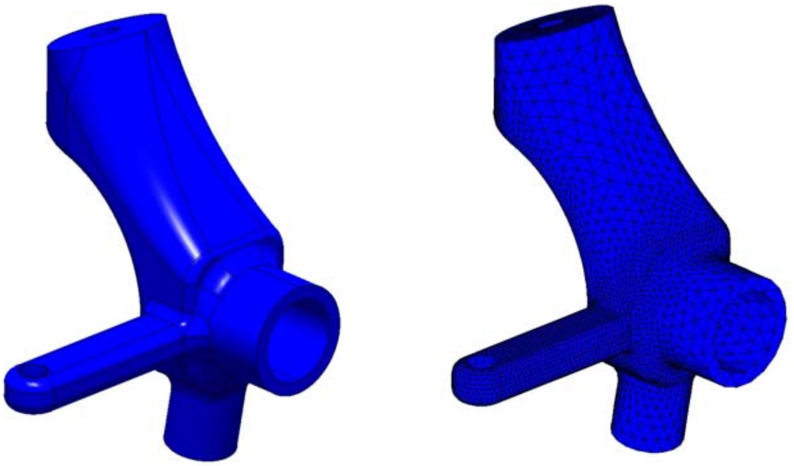


Figure 1.1: Original CAD Model → Meshed FEA Model

Pre processing software these days will take a CAD model and automatically mesh it with minimal input from the user. Element type, size, shape and quality does have a big effect on the accuracy of your results though, so it's not quite as simple as it sounds! The more elements you have, the more accurate your results will be, but the analysis will take longer to run so it's a matter of finding a balance between accuracy and run time. Often a mesh is refined in areas of high stress or around complex shapes to increase the accuracy without increasing processing times.

Once the model is meshed, material properties need to be defined and applied to the meshed part. These properties include the Young's modulus (a measure of material stiffness), its density, Poisson's ratio and more depending on the complexity of the analysis.

The next step in the pre processing stage is to apply loads and boundary conditions to the model. Loads are usually defined as forces acting on a certain point, but can also be torques, pressures, temperatures, or even a velocity or acceleration such as gravity. Boundary conditions are constraints that define how and where the part is held or bolted on, and are required to stop the part flying off into space when a force is applied. They basically tell the software which nodes aren't allowed to move during the analysis.

Pre Processing:

:: mesh CAD model
:: define material properties
:: apply loads & boundary conditions

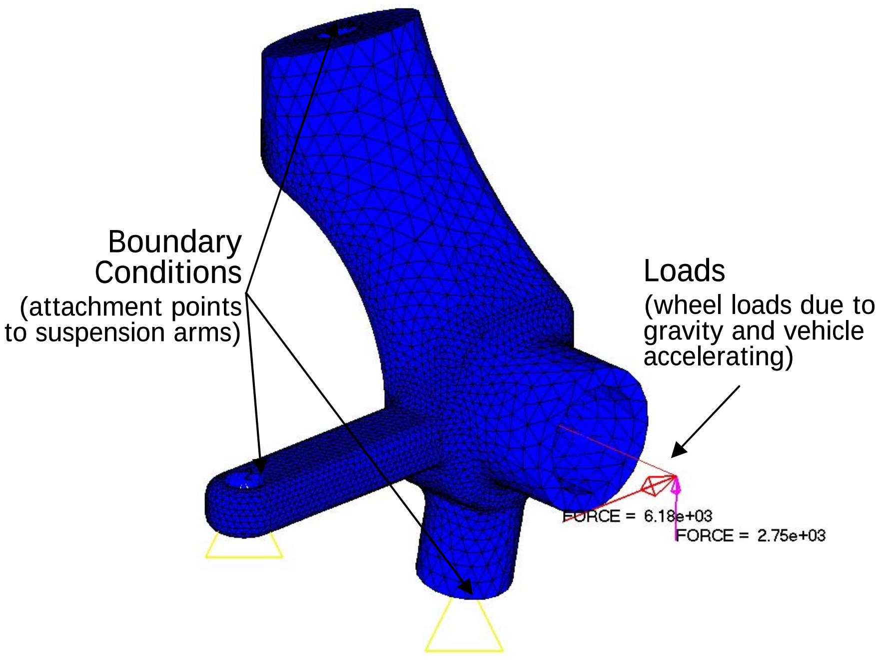


Figure 1.2: Loads & Boundary Conditions

Once the model's been meshed, materials defined and loads and boundary conditions applied, you've now got a pre processed FEA model ready for solving.

2. Solving

The next step in the FEA analysis is to send the model off and let the computer do all the calculation work. The software that does all the calculations is called the solver and it goes through the meshed model you've created and solves a bunch of mathematical equations for each of the nodes to figure out overall stress and deformation of the part.

These equations are based on the old F=kx equation for a spring, which you may remember from year 8 physics. F is the force, k is the spring stiffness and x is the displacement of the spring, or how much it stretches due to the applied force.

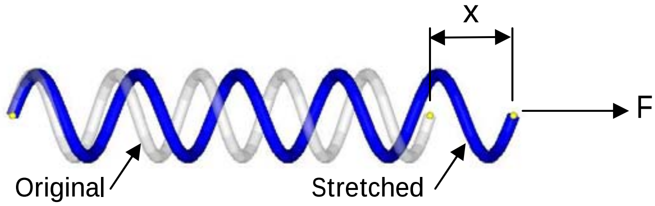


Figure 2.1: Spring Equation F=kx

In FEA most structures can be considered as a big, complex spring. However, instead of having to calculate x once, like in the simple spring example, the displacement needs to be calculated for each node in the model. Remember having to solve simultaneous equations in high school? Well the solver is doing exactly the same thing, solving literally thousands of these F=kx equations simultaneously using the forces and the material stiffness you've defined, to come up with the displacement of every node in the model.

Solving:

:: based on F=kx equation

:: computer solves lots of equations simultaneously

:: calculates displacements of nodes & stresses within elements

Once it knows the nodal displacements and how each element is deforming, it can also calculate stress within the element. This is particularly handy, as this is what determines whether your part is going to break or not.

3. Post Processing

Post processing is the part of the analysis process that involves reviewing and interpreting the results from the solver. Back in the 60s the poor guys at NASA would have just got reams of paper from their dot matrix printers, full of numbers that they'd have to manually review to figure understand the analysis results. But again, thanks to modern computers and software, the post processing software now offers the engineer nice coloured pictures on the screen to show the deformed shape of the part and where any stress 'hot spots' may be.

Whilst this may seem a little gimmicky, these coloured pictures, technically known as contours, are a very intuitive way of interpreting the results and quickly getting a practical picture of the overall state of the part, regardless of your technical knowledge… anyone can see if it's red, that's generally bad! The post processor will also show you the deformed shape which helps the analyst understand how the stresses are developing and what changes can be made to improve the design.

Although modern post processors makes viewing results quite straight forward, accurate interpretation of the analysis still requires a thorough knowledge of engineering principles, stresses and material properties. A good analyst will know what changes need to be made to the part to reduce areas of high stress and even determine how much material can be removed from areas of low stress, resulting in a stronger, lighter part.

Here are some of the results from the solver for our suspension upright example:

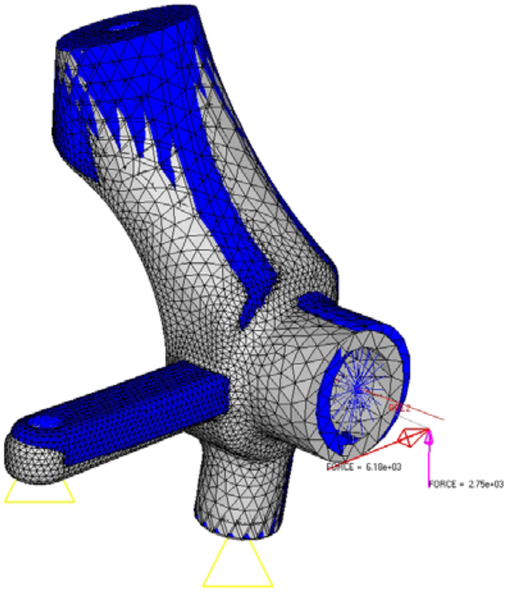


Figure 3.1: Original and Deformed Shapes (scaled up)

Post Processing:

:: interpreting results
:: examining deformed shape
:: viewing stress contours
:: determine changes to improve design

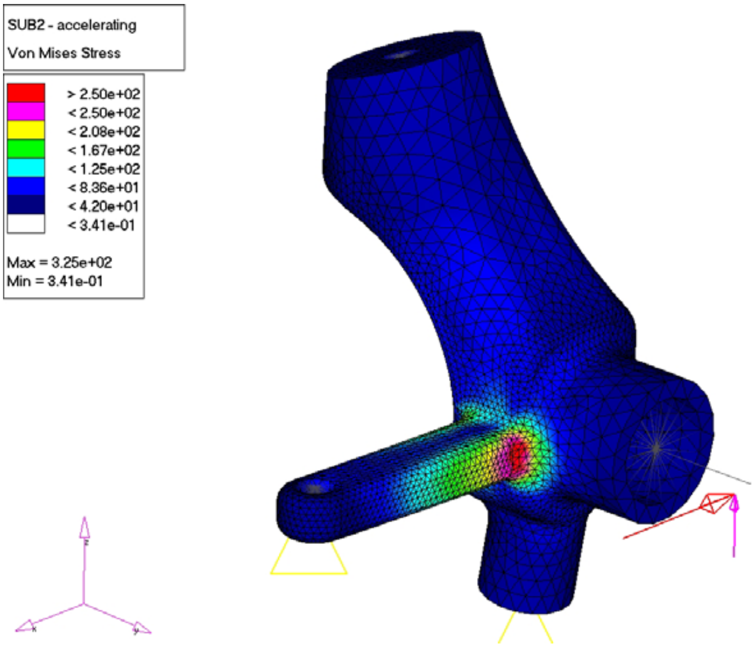


Figure 3.2: Stress Contour (units are MPa)

The next step is to determine whether a part will break by comparing the stress values from the analysis results to the strength of the material. Every metal and most plastics have what's called a yield strength and an ultimate strength. If the stress within the part exceeds the material yield strength, then the part will not return to it's original shape when the load is removed. Although the part is still in one piece, it's going to remain bent, which generally isn't good. If the stress exceeds the ultimate strength, then the part will fracture and break. Ideally, the whole aim of the analysis is to make sure the stresses within the part remain below the yield strength of the material.

In our suspension upright example, the stress contour shows a maximum stress of 325MPa, which is above the material's yield strength of 250MPa, but below the tensile strength of 345MPa. So this means the part will bend under these loads, but won't actually break in two. Still, a bent upright is of no use to anyone so one possible change to make to the part would be to increase the radius of the fillet in the high stress area to add some extra material.

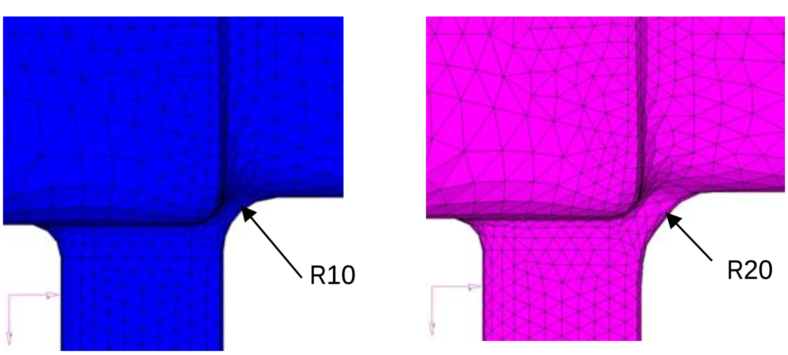


Figure 3.3: Increased Fillet Radius

Design Improvement:

:: make changes to ensure stress remains below material yield strength

:: add material to areas of high stress

:: remove material from areas of low stress where possible

And after rerunning the analysis, it's clear that the maximum stress in the part has dropped to a much more acceptable value of 120MPa. Much better to have figured this out now, rather than after having parts made and tested (not to mention cheaper and safer!).

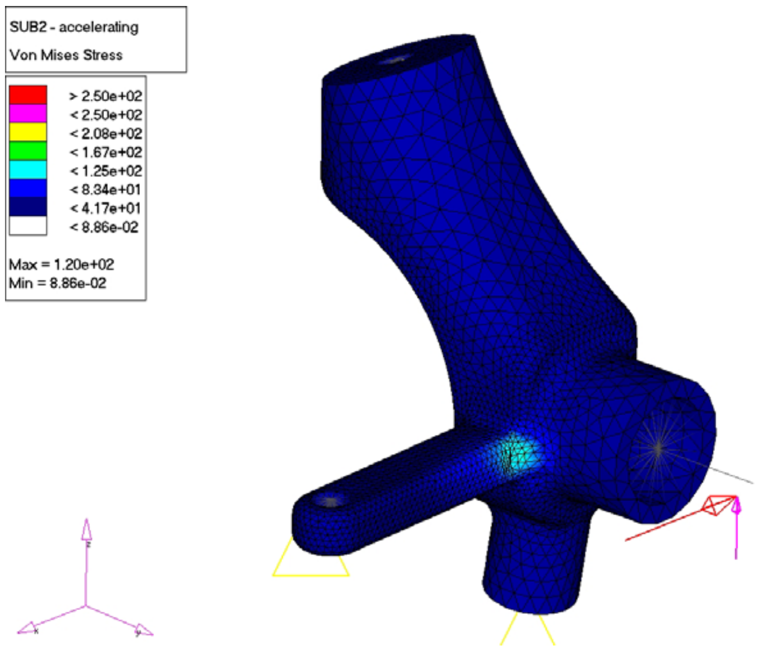


Figure 3.3: Results for Revised Design

So there you have it… the basics of a stress analysis using FEA. It's clear that there are many benefits to using this type of simulation tool in engineering: reduced costs; reduced design time; being able to assess a wide variety of designs; and ending up with a stronger, lighter part.

Keep an eye out for other practical tech articles from Bremar Automotion.

Brett Longhurst, BEng (Mech) Hons, is a director of Bremar Automotion Pty Ltd.

Bremar Automotion is an engineering design company based in Melbourne, Australia specialising in computer simulation of engineering problems at a practical, hands on level and is committed to making these simulation tools accessible to all levels of engineering and motorsport. For more information, see www.bremarauto.com or contact Brett Longhurst on 03 9029 3323.

Contact us now and let us work with you to find solutions to your engineering design problems.

Bremar Automotion Pty Ltd
ABN: 109 806 144
p: +61 3 9029 3323
e: enquiries@bremarauto.com
w: www.bremarauto.com