# FEA—Finite Element Analysis

Finite Element Analysis is sometimes called Stress Analysis (but don't use that term in exams) or FEA for short. It is a mathematical process that is used to calculate how an object will perform when it is subject to forces applied on it.
In basic terms, if you fit a shelf onto a wall, how will it react when you put a heavy object on top of it? By carrying out an FEA test on a computer, you can see if the shelf will stay up, fall off, or just bend due to the weight.
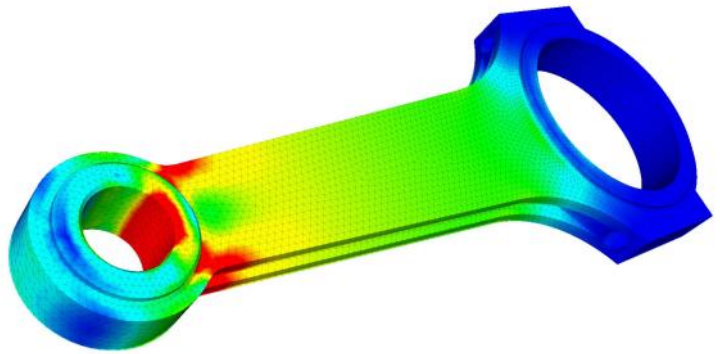
The good news, for Advanced Higher Graphics, is that we do not need to know the maths behind the process! What we need to do, however, is demonstrate that as a CAD operative, we can create 3D CAD items that are ready for testing, and which can be analysed by a **Stress Analyst**. It would then be their job to determine if the object performs to a suitable standard, or whether tweaks to it are required.

Preparing a 3D object

In a 3D CAD programme, draw the item that you wish to use in an FEA simulation. As this would reflect a real life object, you will need to specify all of the **Dimensions** correctly, you need to make sure that you indicate what **Material** the object (or all objects in an assembly) is made from.
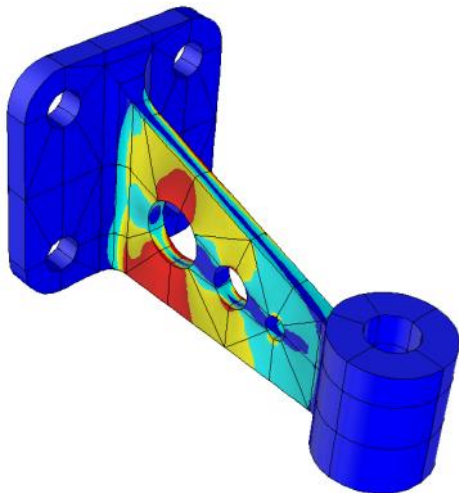
In **Inventor** the Environments>Stress Analysis tabs are used to run an FEA simulation. Once the object is imported, there are certain things that need to be specified:
Ground—a part of the object is normally grounded, i.e. doesn't move—this could be the wall the shelf is fixed to, or the ground under a building.
Force—a series of forces are applied to the object—these can either be **Point Forces**, where the force is applied to a single place, or **Pressure Forces**, where the force is applied to a surface. In a Point Force, we need to specify the size of the force, and also the direction (eg, is it at 90 degrees to the point, or 60 degrees, etc?)
You may need to adjust the scale of colours in the resulting image so that you can see areas that are under stress (shown in **Red**) and areas that are not (shown in **Blue**).

The Stress Analyst would look at the results and from them could advise where changes to the object should be made—this could be done by adding a web, for example, thickening a component, changing the material to something that is stronger, or adding additional support. FEA is much cheaper than testing real objects (imagine testing a car crashing), and it is quick to change the object and run the simulation again.

Inventor FEA simulations can be set to show the **Mesh** that has been used (you can change the size of this if required), often triangles, and also to show any **Deflection** or bending/deforming that the object would suffer from the forces applied.

FEA for beginners video   What is Finite Element Analysis video