Finite Element Analysis as Generally Practised

Just about any FEA software will deliver excellent results, but only when it is driven correctly. The proliferation of CAD embedded FE modules and easy to use windows based systems has made FE very accessible to large numbers of designers, engineers and other users of questionable qualifications. The situation of ten or more years ago when the software was difficult to use, hardware resources were severely limited and only a handful of specialists had the necessary knowledge to run simulations has been turned on its head. Whilst those few specialists were able to build accurate and meaningful analyses, as mistakes were too costly to make in terms of run times and computer rental, the user today can run many simulations with a few mouse clicks paying scant attention to the validity of their results. With CAD embedded systems finite element analysis appears as just another bolt on module and the perception that all is well is enforced by the "what you see is what you get" manner in which other CAD modules are viewed. Rival software developers have competed with each other to produce the simplest and easiest user interface, but in doing so they have taken away from the users the necessary thought processes required to create a valid simulation.

The stark reality is that probably the majority of users are blissfully unaware that their analyses are invalid. Attend any software demonstration, study any tutorial or just watch over someone's shoulder and with near certainty you will witness a model that has loads applied at one end with the other end fully fixed. Thus all models are reduced to a cantilever (arguably a mathematical entity that doesn't actually exist). Simply reverse the clamped and loaded ends to see a completely different set of results! In all likelihood the software will perform some kind of error estimation and checking procedure and adaptively refine the mesh to reduce these errors. This helps to serve the false notion that the FE module is an expert system. The old adage "garbage in, garbage out" is as true today as it ever was.

Is this scare mongering? Unfortunately no, errors and mistakes that used to be made by only the novice user appear in the main stream at an alarming regularity. One does not have to search very hard to find examples of these in promotional documents, training manuals and web sites offering online analysis services. The "everything is a cantilever" approach is only one such error, there are many more.

The root cause of the problem is a fundamental lack of understanding of how to set up valid boundary conditions on a finite element model. Such models tend to use rigid body elements to spread loads out from a single point of application and are grounded to earth by rigid clamps covering a set of element faces.

Finite Element Analysis as it Should be Practised

Here are a few basic principles to achieve correct, accurate and realistic analyses when dealing primarily with static linear problems.

1.) Sketch a free body diagram of the component. Label all the applied loads. Check that the loading sums to zero in all directions. Similarly check that moments also equate to zero. Use this diagram as a guide in loading the FE model.
2.) Apply all loading as variable pressure distributions that accurately represent the contact with other parts in an assembly. This can be achieved using well proven empirical formulas. Never use rigid body elements to spread loads or any form of constraint equation as these are highly artificial and may severely corrupt results. It goes without saying that point loading must not be used either.
3.) Once all loads have been applied, check that a reasonable load and moment balance has been achieved. The perfect balance is impossible, but residual loads should be a tiny fraction of any of the applied loads.
4.) Only minimal supports must be used to ground the model. In 3D this is done using 3-2-1 supports (see below). Correctly applied the supports will not react any loads and thus not interfere in any way with the results.
5.) Mesh on a first pass should be fine enough to pick up concentrations. Subsequent runs with a finer and good quality mesh in hot spots should be applied until stress values converge (commonly called mesh convergence).

It is essential that CORRECT and REALISTIC boundary conditions be applied. Otherwise no matter how fine and good the mesh is, the results are quite simply wrong!

Balanced Loading and 3-2-1 Minimal Supports

To achieve accurate and realistic analyses using finite elements it is necessary to steer well clear of the artificial features available in most solvers like rigid body elements and multi-point constraints, which are often used to facilitate the loading and restraining of models. Rigid clamps also fall into this category, as any structure or component can never be fully restrained in reality. In short assume that every part of a model is free to deform.

Initially this may appear to some to be impossible. However the method employed to satisfy this goal is quite simple to follow.

Firstly, loads and moments applied to the model must be in complete balance and should be applied as variable pressure distributions that simulate contact with other components.

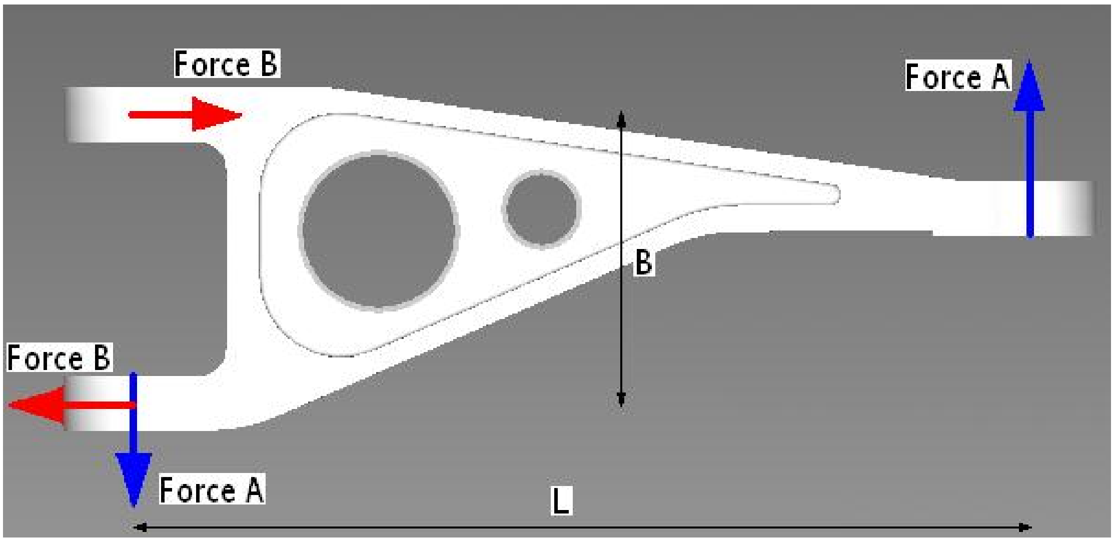
Secondly, support the model with 3-2-1 minimal supports. This involves choosing three reasonably well separated points that define a plane (i.e. not three points in a straight line!) Any convenient plane will do, but for the sake of argument let’s use the XY plane. The first point is restrained in all three translational directions.

Now any single object has six degrees of freedom, three in translation and three in rotation, commonly known as rigid body motions since no internal strain energy is involved. What this first point of restraint does is reduce the number of remaining degrees of freedom to the three rotations. The second point is carefully chosen at an X offset from the first and thus shares the same Y and Z coordinates. The second point is restrained in the Y and Z directions only. There now remains just a single unrestrained freedom, rotation about the X axis. The third point is restrained in the Z direction only and thus knocks out this final freedom.

Of course there are many variations that can be used instead, a similar approach can be applied to any of the global planes, or if there is no convenient global plane available then a local axes system will suffice.

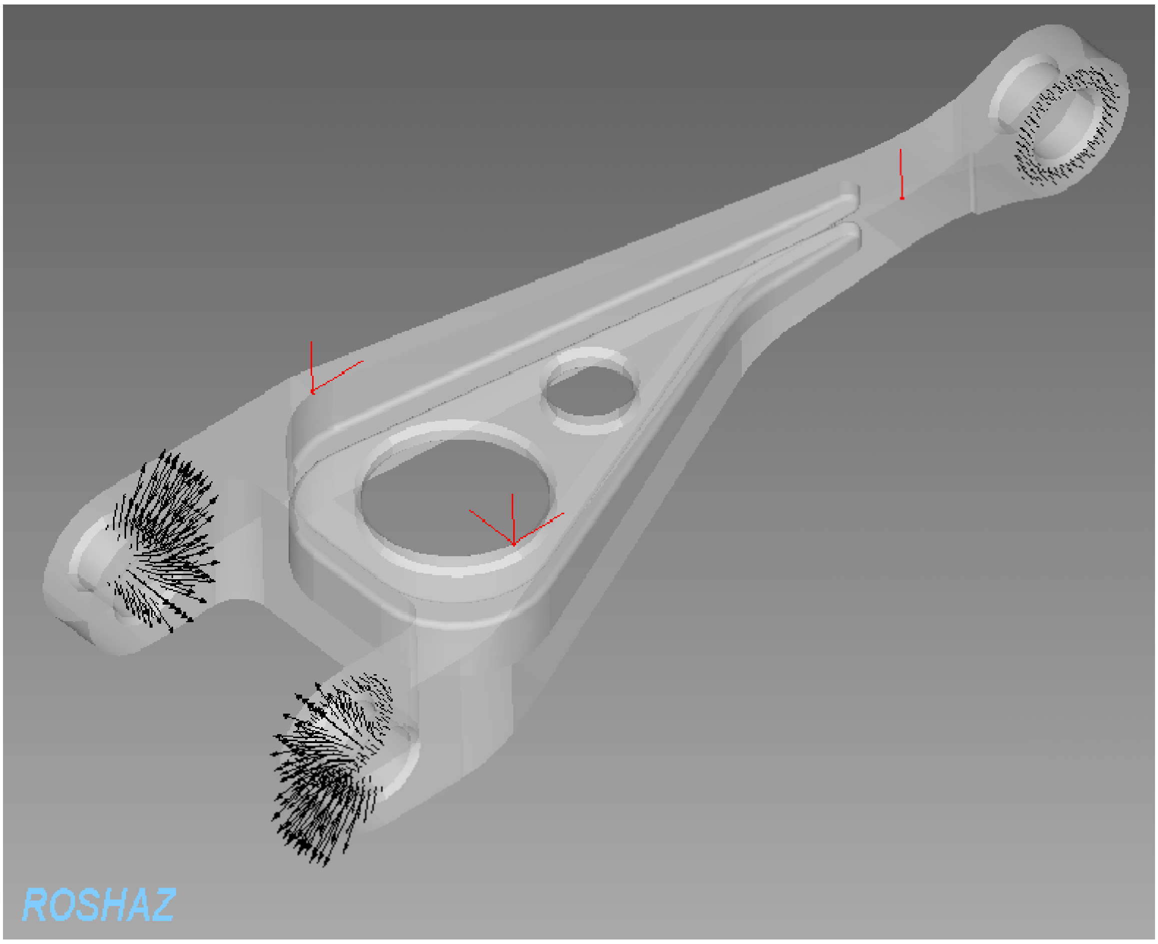
However it is achieved the number of supports in a 3D model must equal six for a minimal support condition, any less and the structure is under supported and is insoluble, any more and the structure is over constrained.

When used correctly, the supports prevent rigid body motion without applying any restriction to the deformation of the part and thus will not react any load so long as a fully balanced set of loads and moments has been applied.

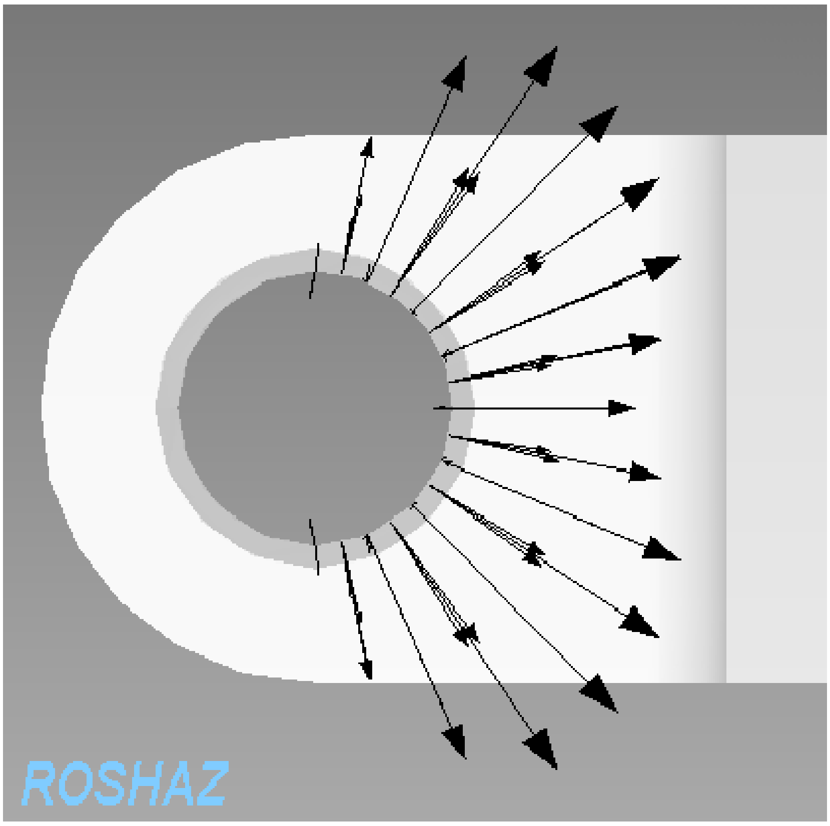


Free body diagram of a torque link showing a balanced set of loads.

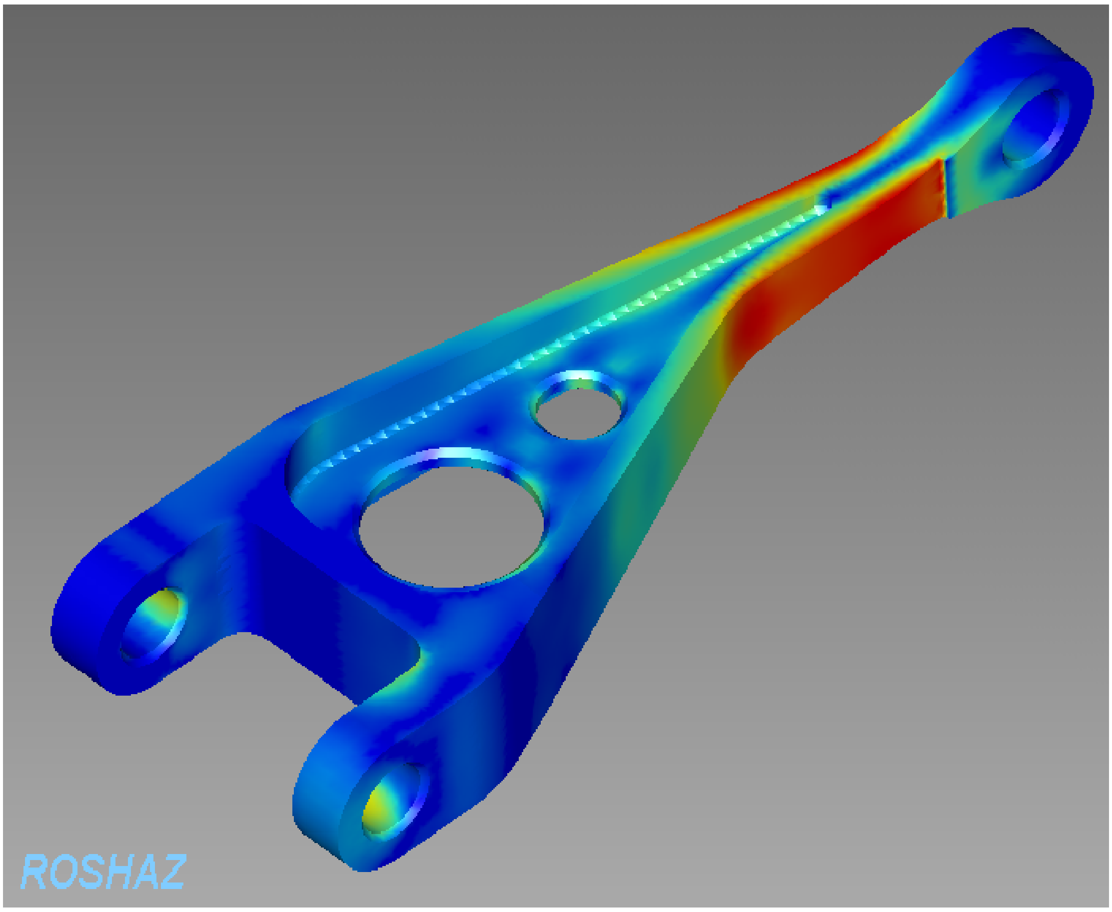
Force B * distance B = Force A * distance L



Loads applied to the model as an equivalent set of nodal forces shown in black and 3-2-1 minimal supports shown in red.



Gencoz pressure distribution used to apply lug loading, which is an empirical formula that has excellent correlation with actual measurements.



Von Mises stress fringe plot. Note that the three nodes used by the 3-2-1 supports have had no effect on the stress contours.