Finite Element Analysis

Creating a mesh model.

We are now ready to build the computer model and apply the loads and restraints. Remember that numerically the analysis model comprises only the elements of material connected to each other at the nodes - the mesh of elements.
Its structural behaviour is governed by the individual element mathematical shape functions and material properties.

The solid model has been converted to the mesh model - this isn't a solid model with a mesh on it - the mesh of elements is the model. Let's see how Lewis created the model.
Well this is a solid model with 6 degrees of freedom, so we always tend to use tetrahedral elements with ten nodes and that basically tries to minimise some of the errors you can get using the lower order elements, the four-noded ones, and the reason we use those rather than bricks is to try and pick up for the, some of the 3D features that are on this component. The mesh itself, as we are about to see here, has around about 72,000 elements and you can see on the screen there that we've obviously tried to refine in the areas of interest for stress and that will obviously give well in excess of 150,000 degrees of freedom for the model. …and restraints on this model are effectively split into two parts: one is on the bearing side of the axle if you like, the left hand side on the screen; and the other is the right and between those two is the flange, the bearings are loaded using pressure distributions which I can try and display a little bit now. See the red arrows on the screen, the one's that apply the loads to the bearings and those are split into two axes, sort of longitudinal and vertical. We can apply those separately to give our resultant radial load and then one of the constraints is applied to the flange, which I can try and bring up now also. As you can see again one of them is axial (if I can just show that a little clearer just here), one of them is an axial constraint and the other one is radial and as well as that there are the loads on the opposite side of the hub, one of which applies the hub pre-load from the nut, which is shown of the FE which I need to actually display separately. Just hang on second, I can show that. All those markers on there show the elemental loads that are applied to represent the hub pre-load from the nut, and if we zoom back out again, you can also see for this load case where the load is applied for the actual wheel applied load. Now that appears to be in the middle of space but in reality there is an element there that connects it to the outboard side of the flange here on the screen, which effectively loads it according to the stiffness of the nodes that it attaches to, so it is relatively conservative in quite a few ways. And then finally there is a constraint which is applied again to the flange, which basically stops rigid body motion of the component off into space, which I think we should also be able to show here. It's just a single node which constrained here basically to stop the component sort of flying off into space; it's the last constraint of the 6 degrees of freedom we needed to constrain.

Well after you've set the model up the next stage really is to verify that all the resultant loads you have applied to the model are correct, and as you'd expect, so you'd need to have some kind of method of checking that before you proceed so you end up trying to show all the different markers that you have to try and display, effectively to make sure the forces are applied in the right directions and that the overall resultants are correct. Check that your boundary conditions fully constrain all 6 degrees of freedom otherwise you will be wasting your time when you press the solve button. So in the case of this component it will probably take a few minutes to solve but we're going to skip passed that piece because I already have the results here.

Just to summarise the key points of the model:It comprised 72 000 elements of 10 noded tetrahedral type giving a total of 150 000 degrees of freedom overall considering all the possible element displacements.

Remember that the solver produces the resulting displacements of all the element nodes that can move in response to the applied loads. The element strains and stresses are then calculated using the material properties;

Lewis refined the model mesh, that is increased the mesh density in areas of interest or likely high values and gradients. He made sure that all the whole model 6 degrees of freedom were accounted for by the boundary conditions and loads. And he double checked that the correct resultant loads were applied.

There were two lots of restraints and loads each side of the central flange; On the bearings he put the reaction pressure loads; On the bearing side of the flange the surface was restrained in the axial direction to react the axial thrust; A further restraint on the flange edge ensured that the whole body model didn't zoom off in other directions into virtual space when the loads were applied; The wheel nut force was applied as a uniformly distributed load effectively pulling the hub from the flange; The main lateral load case side load was applied using a special element to connect it to the flange; Note that Lewis shows the load condition upside down on the screen which is of no matter to the computer of course;

Once he was confident that the model was checked, loaded and restrained correctly, it was time to press the solve button. Years ago a model of this size would have taken all night to solve, but now it only takes a few minutes. It is absolutely essential therefore to double check that the results are reasonable and valid.