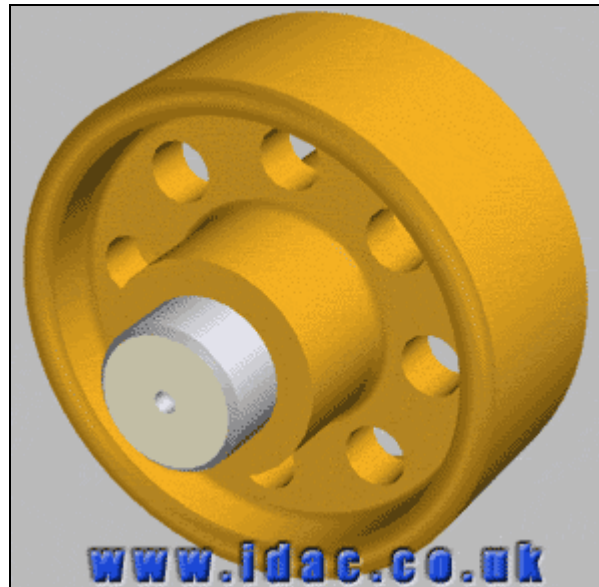


IDAC Help File

Preparing models with AGP (Analysis Geometry Processor)

Author : Tim Atchison



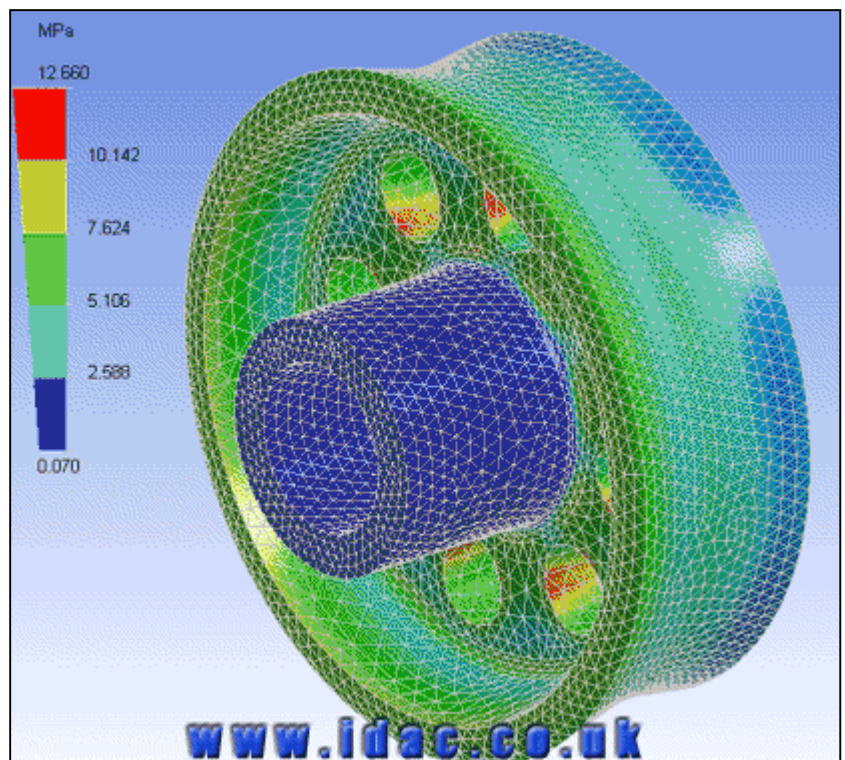
The idea of having a master CAD model that in turn feeds drawings, rapid prototyping, analysis and more is a nice one in theory. What most designers and analysts discover soon enough is that a fully featured CAD model often contains far more information than the analysis warrants – or can handle. Combine this with the fact that CAD users somewhat understandably dislike having to create a defeatured copy of their parts just to keep analysts happy and the idea of an intermediate product starts to make sense. ANSYS Inc have attempted to address this issue with the Analysis Geometry Processor (AGP).

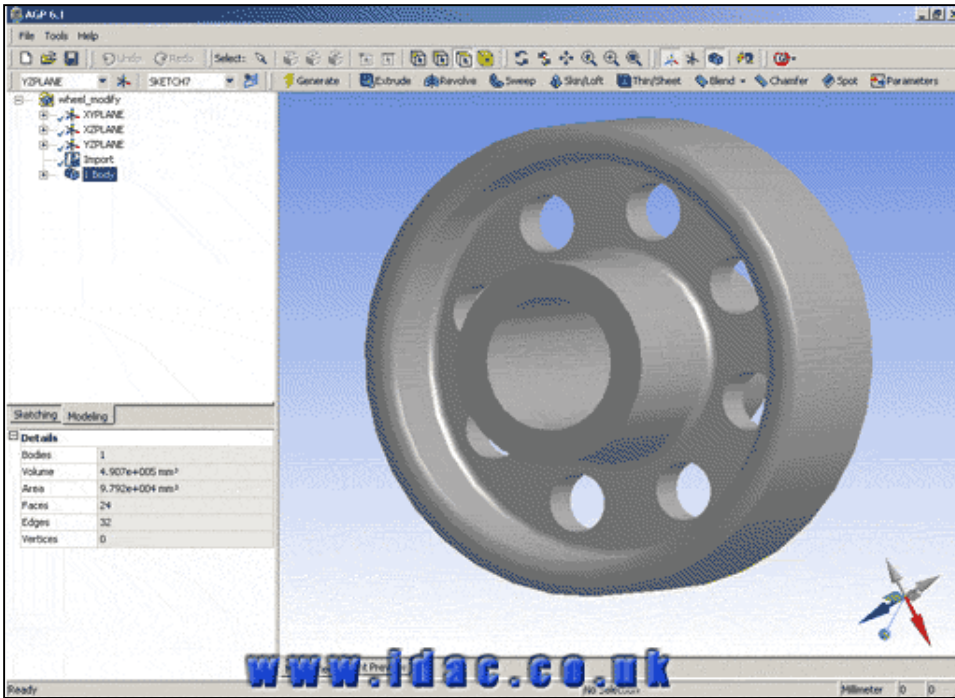
Current Model

In this help file we will show how AGP can allow us to take advantage of symmetry and hex meshing in the wheel model shown. We will also use it to apply a boundary condition on a segment of the model that isn't available otherwise.

The model itself is a wheel created in an external CAD system then saved in parasolid format. AGP has basic modelling tools to create something of this level of complexity but is better suited to modifying existing CAD geometry. If you want to create a complex CAD part then use a dedicated CAD system.

The loading conditions in this case are a radial fix on the inner bore, an axial fix on one end of the bore and a constant angular velocity. The results from a 360 degree model are perfectly valid but the node count is a little high.





Preparation in AGP

We start to clean things up by bringing the parasolid file into an AGP session. You can see that AGP has the same look and feel as DesignSpace – both are built on the AI*Workbench platform so if you are familiar with one it doesn't take long to get into the other.

Although a little fuzzy in this image, you should be able to pick a few solid modelling tools across the top – Extrude, Revolve, Loft etc. Our feature tree to the left only shows our three default orthogonal work planes at the moment. As we add features they will appear in here.

An introduction to solid modelling techniques is a little beyond the scope of this document but for those who are unfamiliar, it is slightly different to the way ANSYS approaches things. There is no “bottom-up” approach of building keypoints etc into higher order entities. Similarly, booleans have little relevance. The general approach usually follows three steps, described here for creating a rectangular prism:

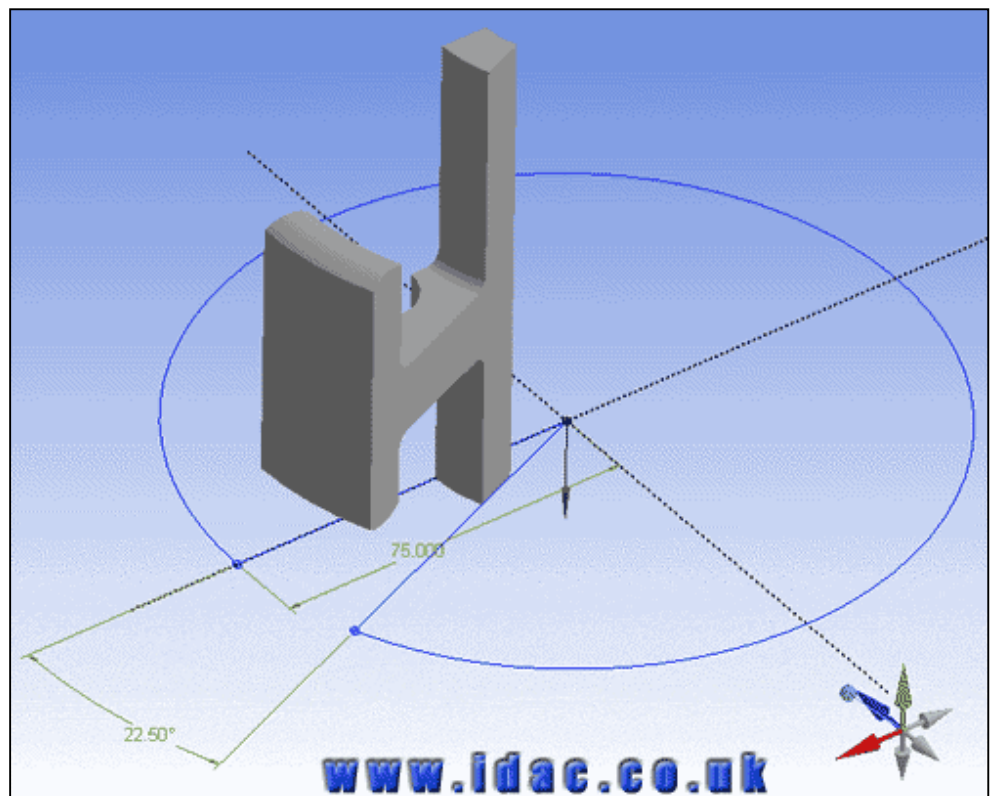
- Choose a plane to draw a 2D sketch on
- Draw a “profile” on this plane (in our case we draw the cross-section of our prism)
- Turn this 2D profile into a 3D feature (in our case by Extruding it in a straight line)

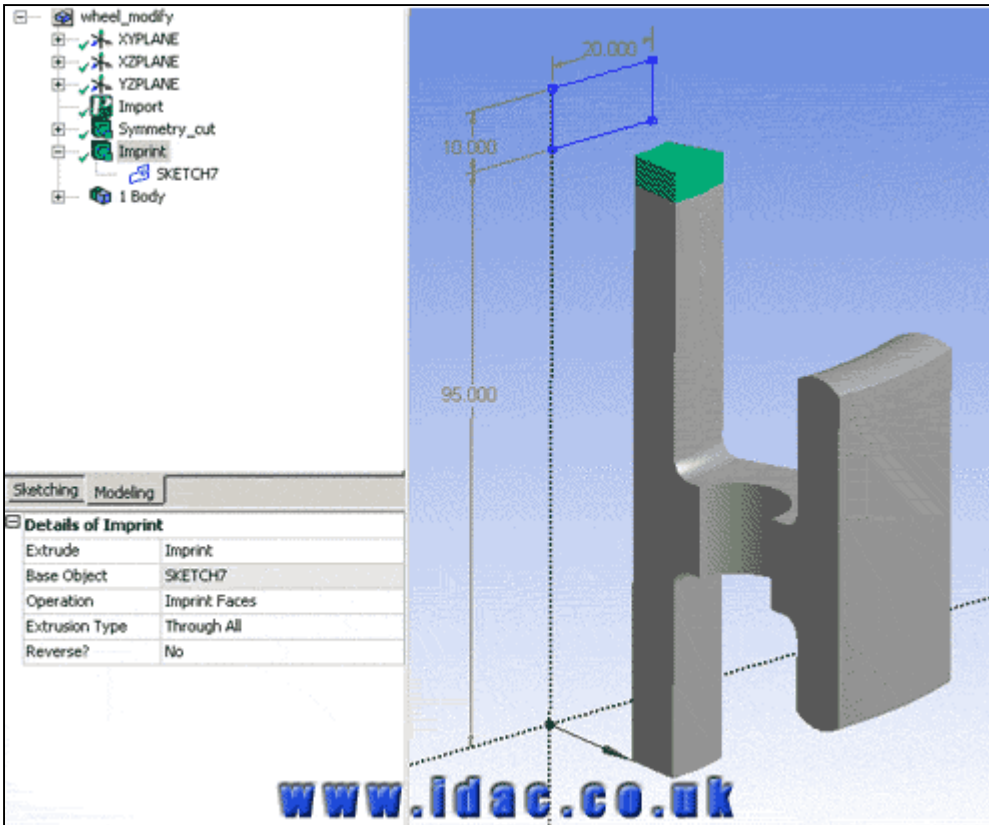
Obviously the final step could be anything we like. We could revolve our profile into a 3D ring, sweep it along a line, loft it towards another 2D profile to create a transition section and so on. Once you have basic features completed, secondary features such as fillets, chamfers, thin walls and so on are just added on. This is all parametric as well – we can change and update dimensions at any time.

Reduce to symmetric part

Back to our model though. The first feature we will add to our imported part is a cutout. Our geometry and loading both display symmetric behaviour so we can slice our model into the 1/16th segment shown and get the same results as the full model.

This picture shows the three steps. I have chosen a default plane (the black triad) to draw a 2D profile. This is a circle that encompasses all but 22.5 degrees of my model. I then “extrude” my profile upwards to remove anything inside my profile, leaving the segment I want.





Create loading area

Next we will create an area at the top of our model for applying a boundary condition later. AGP will let you split a surface into smaller segments without dividing the volume below it – thus letting you apply loads on what would be otherwise unbroken faces.

Again I draw a 2D profile and choose an Inprint feature (you can see this in the feature tree and the Details window) and extrude the profile through the model.

Notice our feature tree is updating and now has a record of our import, original cut and now this imprint. I can go and edit these at any stage.

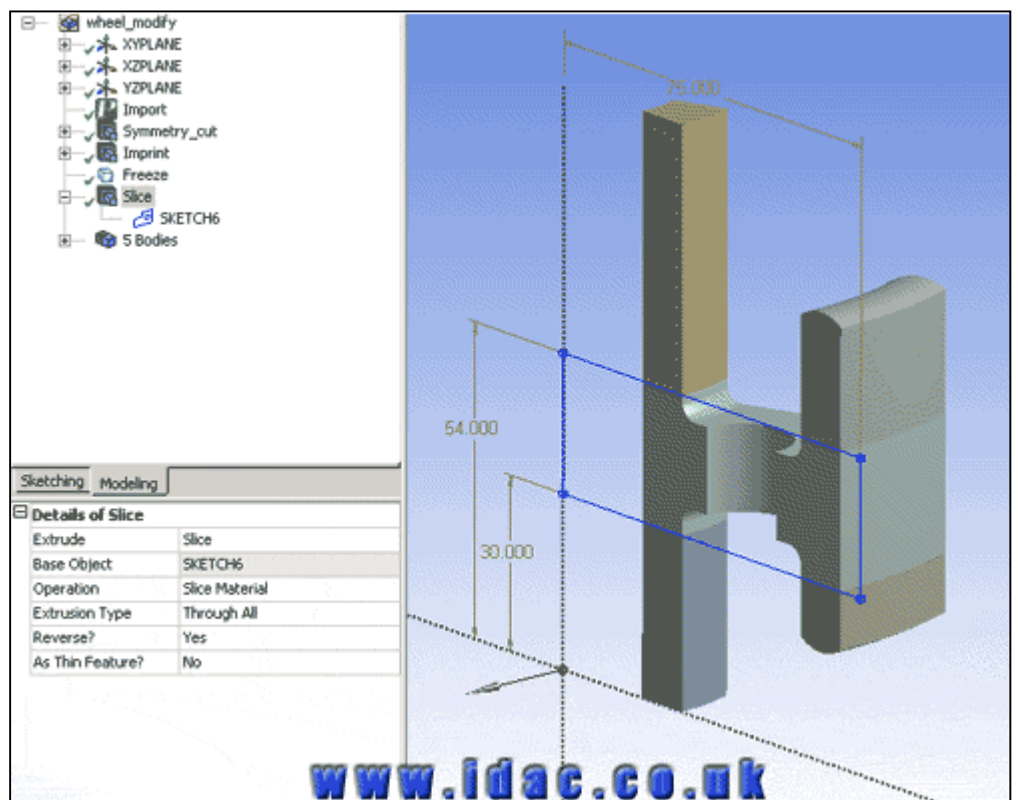
We could take this model into DesignSpace or ANSYS now but we will include one more step. With the model in its current state our mesher is only going to be able to use tet elements. To get the benefits of hex elements (their smaller node count is most important in our case) we can split our model into “sweepable” regions where possible and just leave the minimum possible for tet meshing. DesignSpace will automatically apply bonded surface contact at these interfaces so its not quite as clean as a transition mesh but is acceptable in this case.

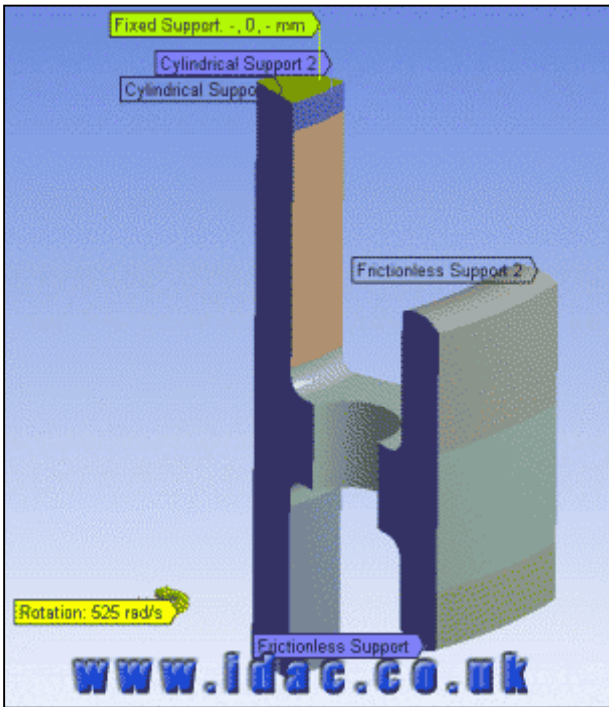
Divide for hex meshing

The process should be getting familiar by now. I pick the vertical plane, enter into Sketch mode and draw a shape that slices the model the way I want.

This shape could have any general topology – the only important bits here are the two horizontal lines that divide the model into 5 bodies.

Once this is done I have the option of linking straight into DesignSpace, writing an ANSYS Neutral File (.anf), or parasolid. In our case we will link into new DesignSpace scenario.





Apply symmetry loading

Once we are into DesignSpace, the loading process is straight forward. As we have a solid model, we represent planar reflective symmetry on the cut faces by applying a "Frictionless Support". We take advantage of our newly available cut face up the top to apply a radial clamp (we couldn't do this in the original model as there was no area there). We fix the inner face radially, apply an axial fix up the top and put on a rotational velocity.

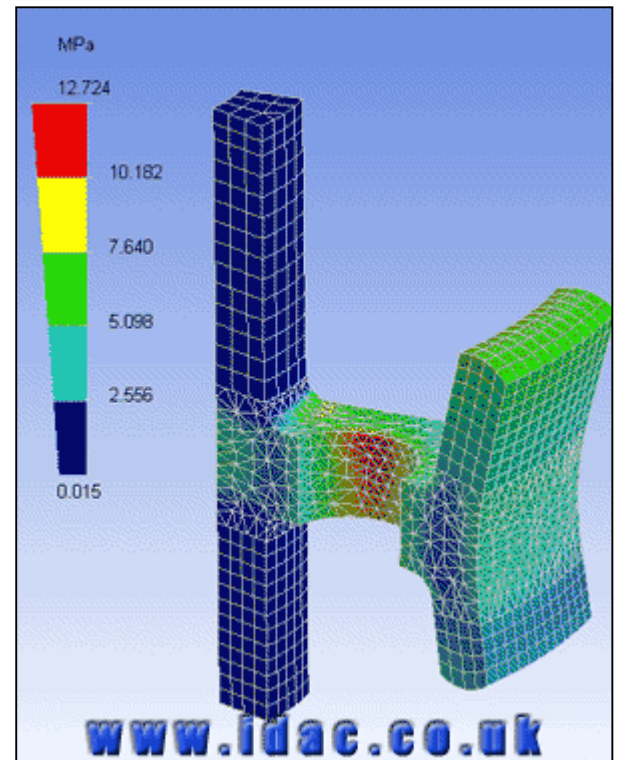
The usual advantages of symmetry now apply. We can define a relatively fine mesh compared to our initial model and still end up with a node count significantly below that of the full model.

The contact is assigned automatically. We just need to make sure that our mesh size is fine enough to give a smooth transition of stress values across the interface.

Results

The resulting mesh and Equivalent Stress results are shown to the right. DesignSpace has been able to hex mesh the four outer volumes and has confined the tet mesh to the middle. Our stress results match those for the full model nicely (12.72 MPa here, 12.66 MPa in the original model) as do our displacements. The stress contours appear to be smooth across the bonded interfaces so our mesh size is adequate.

The node count for the symmetry model is around 18,000 nodes whereas the full model – with a coarser overall mesh – was around the 150,000 level.



Summary

AGP is designed to give analysts the ability to prepare models for analysis without interfering with the master CAD model. Based on solid modelling techniques, it is very easy to use and gives parametric control over every step you take. There are many features not covered in this tutorial – linking parameters to DesignSpace, mid-planing, spot-welding and more. If you have found this help file to be of use then visit our website (www.idac.co.uk) to see our full archive of tips and tricks for ANSYS analysis.