IDAC Help File

Convergence and Relevance in ANSYS/DesignSpace

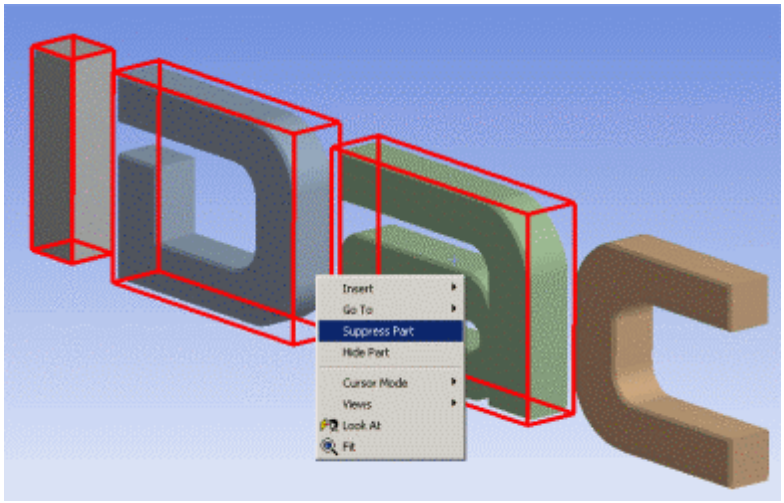
Author : Tim Atchison

This IDAC Tutorial discusses improving your results in ANSYS/DesignSpace through the use of relevance and convergence controls. With or without the advanced meshing controls available in some licenses of DesignSpace, sensible use of convergence controls can give you accurate and efficient results.

The example below uses DesignSpace, but the issues are relevant to ANSYS analysis in general. If you would like to step through the tutorial for yourself, you can download a DesignSpace database (unmeshed and unsolved, 350 KB) from the IDAC website.

Getting started

I've provided a sample IDAC database that is going to be variously squashed, heated and shaken over a few more tutorials. For now, we just need the "C" however so we can suppress the other parts. The easiest way to do this is make sure your select filter is set to "Part" mode then drag your mouse over the unwanted parts. Once they are highlighted, right-click and choose "Suppress Part".

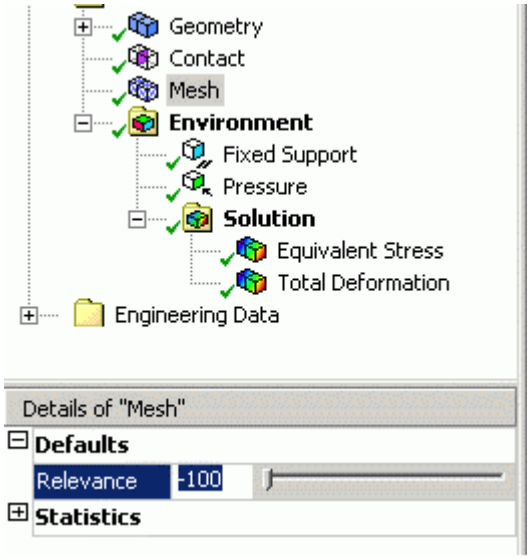


Now that shameless bit of self-promotion is out of the way, we are right to begin. We will look first at the uses (and limitations) of Relevance.

To load our model we will just fix the lower cross-section face of the "C" and put a pressure of 1 Pa on the top cross-section face.

Relevance

The term "Relevance" comes from assembly analysis. It is a tool that DesignSpace provides to let you quickly change the mesh size in different parts - ie you tell DesignSpace a part is more relevant by giving it a higher relative Relevance number (from -100 to +100). It responds by giving that part a finer mesh than other parts.

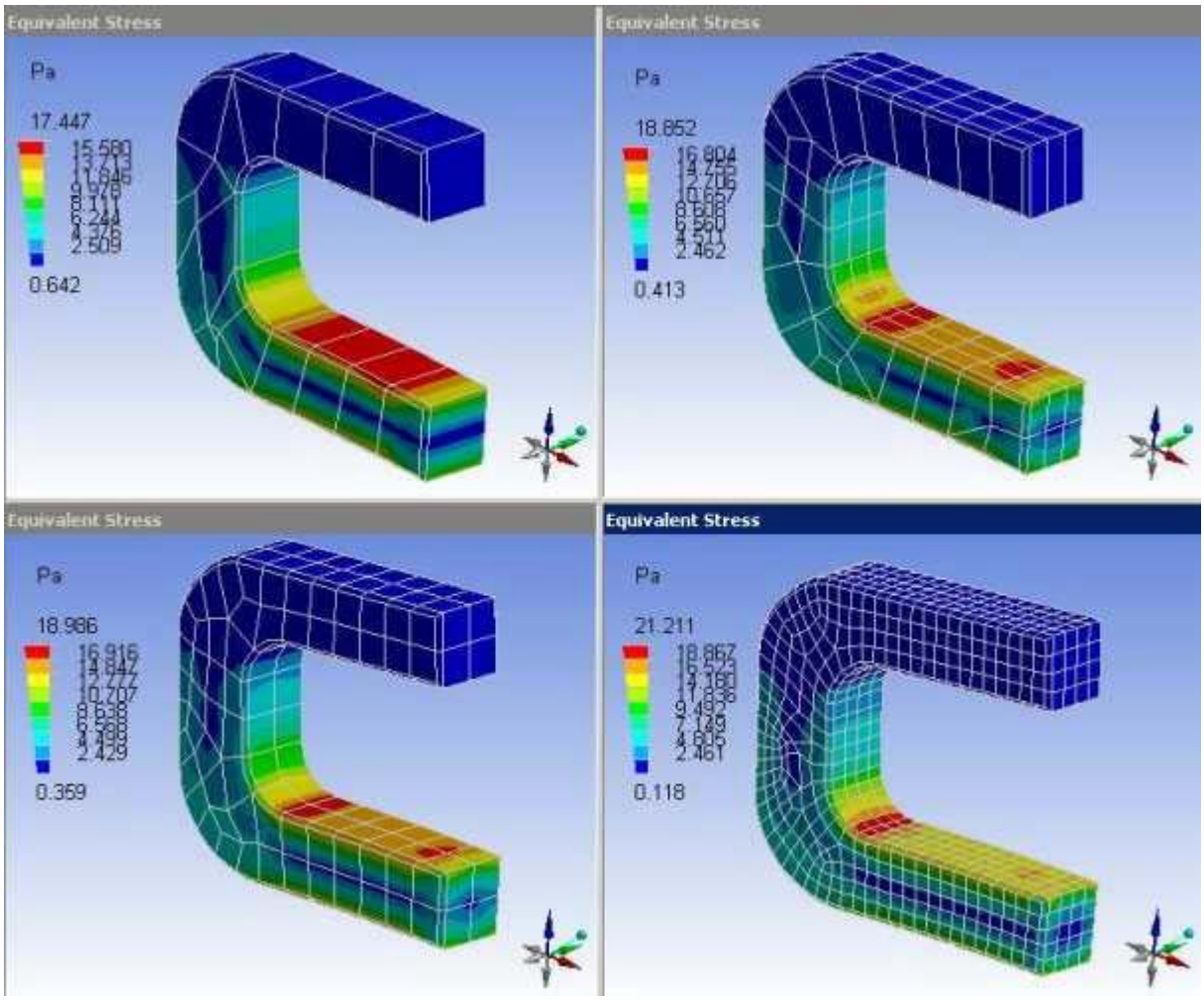


This is useful but there are limitations.

- You aren't controlling the actual mesh size - it is a relative size control compared to other parts
- More importantly, it gives you a general increase in mesh density. If you are only interested in one section of your model you are increasing your runtime enormously without necessarily getting sufficient accuracy in the bit you want.

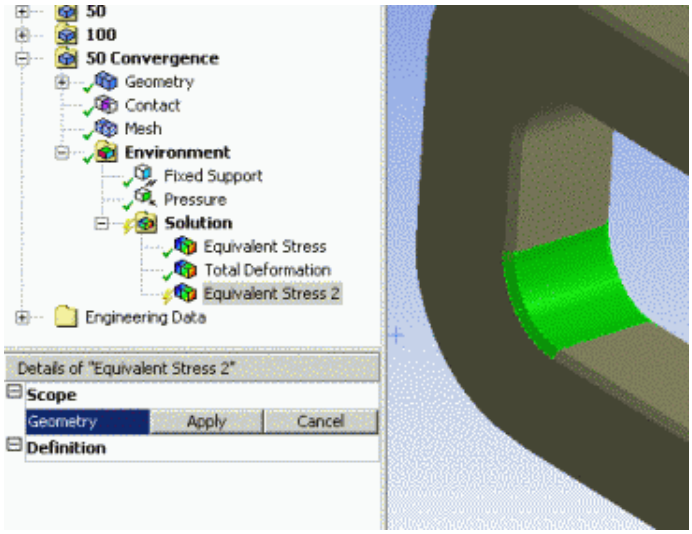
Take a look at the results below. I've run 4 models with Relevance settings of (clockwise from top left) -100, 0, +50 and +100. You can see the stress result become more sensible as mesh density increases but when we hit the limit of +100 Relevance our maximum stress isn't exactly converging on a figure compared to the previous result. How can we be confident that this is the correct answer? We can't just yet.

It is worth noting after you solve, that the displacement results for these 4 models barely differ. If you are only after displacement then a relatively coarse mesh will often give sufficient results. It's only when you need to capture more varying stress or thermal gradients for example that you need to provide a finer mesh.

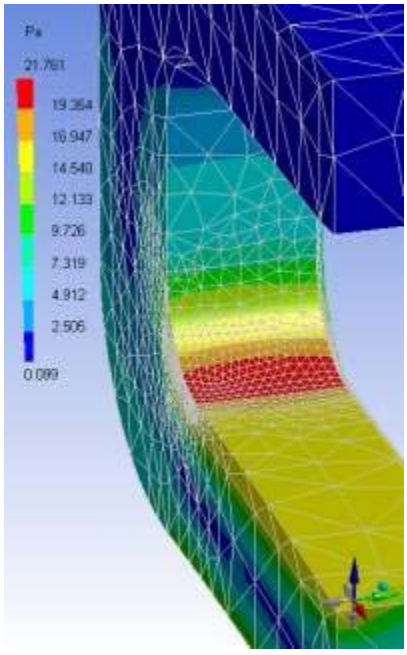


Convergence

Convergence is a tool that gets around the problem we have just encountered. We get to tell DesignSpace the bit of the model we are most interested in along with our acceptable level of accuracy. It then tries to do the rest. To use convergence, insert a new equivalent stress result into your Solution branch. Before doing anything else, switch to Surface select mode and pick the area you are interested in - in our case the lower blend of the "C" where the high stresses are occuring. Go to the Details window of the stress result and click on the text in the Scope section. It will change to an "Apply" button which you then pick.



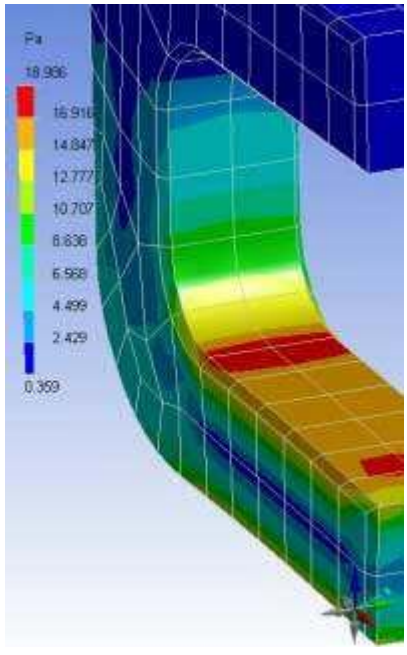
This will just give us a result for that patch. More importantly, if we now insert a Convergence control onto this result (right-click > Insert > Convergence) then DesignSpace will concentrate its remeshing and accuracy checks on this patch alone - saving us from remeshing an entire model to a level we don't need.

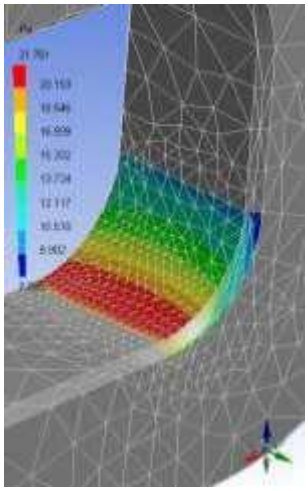


This is the result of a convergence control. We started with a +50 Relevance mesh and asked DesignSpace to keep remeshing the blend area until the stress results converged within 2% (slightly unrealistic but stay with me on this one).

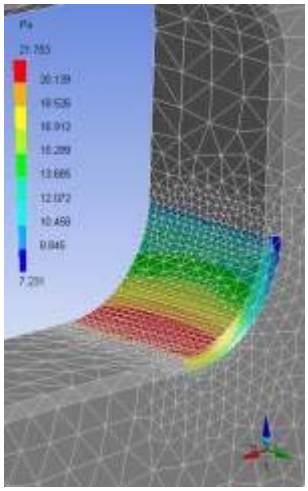
DesignSpace solves once, checks the maximum stress in the region, remeshes and solves again and then compares the new maximum stress. Once this difference falls below 2% it stops.

There are a few things worth noticing. The mesh has shifted to a tet mesh in order to allow free remeshing but all the remeshing has been concentrated in the zone we picked. Another excellent use for this is to avoid pointless remeshing of singularity regions by Scoping the results elsewhere. The maximum stress result is also significantly higher than the initial one.

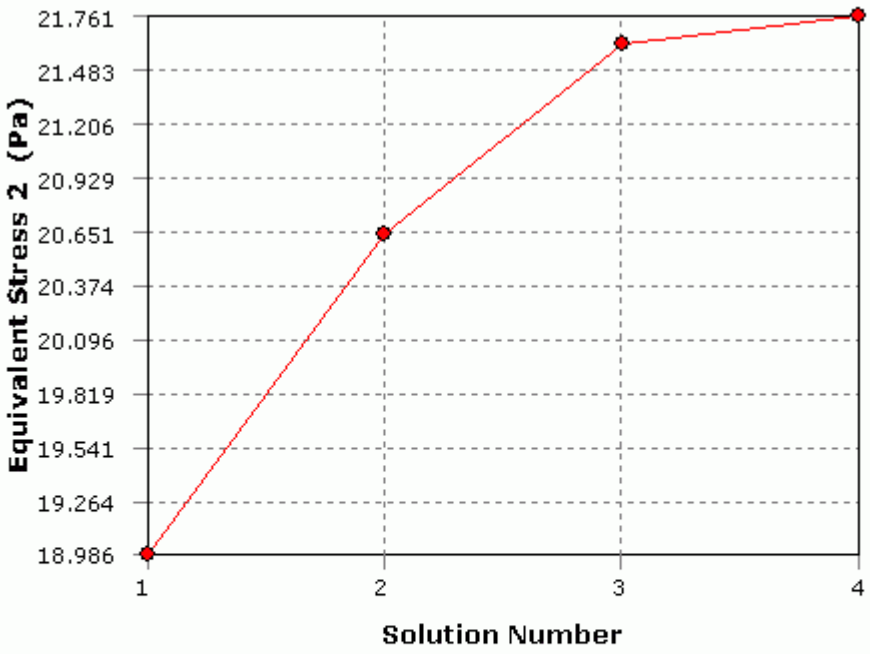




Another point worth noting is that where you start from affects where you finish. These are both 2% convergences on Equivalent Stress but the result on the left started from +50 Relevance whereas the one on the right started from +100 Relevance. Look fairly similar but notice how the mesh on the left is being forced to blend from relatively large to quite small in a short distance. If you insert the same Convergence on an even coarser mesh, say 0 Relevance, there is a chance you will just get a meshing error and no solution at all.



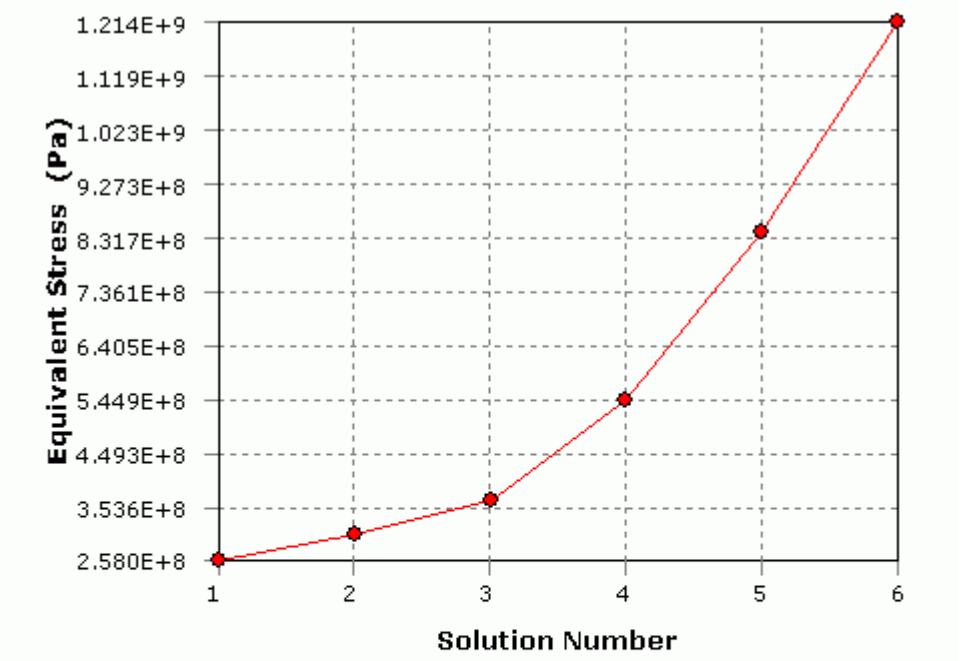
DesignSpace gives you valuable feedback on your results. The graph to the right shows us the changing maximum stress result with each remeshing operation, along with the new node count. You can see that you never get something for nothing - we are confident our stress result is showing a converging behaviour but our mesh size (and therefore runtime) has increased dramatically. A fair price to pay though.



	Equivalent Stress 2 (Pa)	Change (%)	Nodes	Elements
1	18.986		1461	232
2	20.637	8.3364%	6425	3922
3	21.616	4.6322%	13202	8645
4	21.761	0.6670%	30722	20996

Convergence can't avoid standard FEA pitfalls however. Insert a convergence on an area exhibiting singularity behaviour (a zero radius internal corner in this case) and it will simply diverge.

To get around a situation like this you would either have to model a realistic radius, scope the result elsewhere to converge on a more sensible stress or just accept that your results in that region are invalid.



	Equivalent Stress (Pa)	Change (%)	Nodes	Elements
1	2.5803e+008			
2	3.0509e+008	16.7160%		
3	3.6566e+008	18.0600%		
4	5.436e+008	39.1390%		
5	8.3947e+008	42.7850%		
6	1.2142e+009	36.4910%	19819	12515

Summary

Convergence will let you concentrate your computer resources on the result that interests you. While relevance will give you improved meshes within limits, convergence will keep working to the level of accuracy that you dictate.

IDAC provides expert training, support and consulting in the use of ANSYS products. If you have found this tutorial to be valuable and are interested in more of our services, then please check our training, support and consulting pages.