

FEA Mesh Architecture

-
- *SolidWorks*
 - ✓ **COSMOS**
 - *PDM*
 - *Rapid Prototyping*
 - *Office Productivity Tools*
-

This article is a grab-bag of tips and background info having to do with the FEA Mesh. I started out intending only to review my previous article on "FEA Volume Meshing", to make sure it was up-to-date with Solidworks2008. But as I did so, I realized that first article was purely geometric - controlling the size, shape, and number of FEA elements.

As your use of FEA becomes more advanced, you start needing information that is more about Architecture - how does Cosmos represent remote loads, contact elements and gaps, convert CAD bodies into element groups, etc? Which element types are compatible across different types of studies, etc. If you've ever gotten an error message beginning with, "No Element Group has been assigned..." then this KAP's Corner is for you.

About KAP

Keith A. Pedersen, CAPINC Principal Engineer, SolidWorks Elite Applications Engineer

Keith Pedersen has a BSME from Clarkson College and an MSME from Boston University. After a stint at General Electric in Burlington, VT, Keith was the lead Applications Engineer for Advanced Surfacing products for Matra Datavision USA, including EUCLID-IS, UniSurf, and STRIM. He joined CAPINC in 1998 to support advanced surfacing applications in SDRC I-DEAS and joined our SolidWorks group one year later. Keith has extensive industry and consulting experience in non-linear Finite Element Analysis and Computational Fluid Dynamics in addition to surfacing applications. He is a Certified SolidWorks Professional (CSWP) and certified to train and support COSMOSWorks.

About CAPINC

CAPINC has been in the top 5 VARs in North America for customer satisfaction since SolidWorks began surveying customers. In 2006, CAPINC was awarded number one in North America. CAPINC provides outstanding support for SolidWorks, COSMOSWorks®, and PDMWorks® Enterprise customers throughout New England.

Your Source for CAD Excellence: Products, Training and Service

Updates for 2008

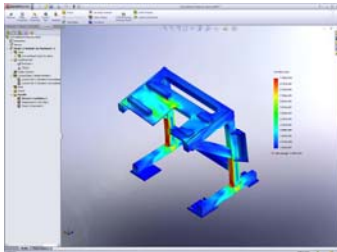
In my prior article, I offered work-arounds to two fundamental limitations of the mesh control. The first, that a Local Mesh Control can only create elements smaller, never larger, than the Global element size. Both my discussions with Cosmos folks, and my own tests, indicate that this is still true.

The second limitation that I discussed was that a Local Mesh Control would affect only the distribution of nodes along model Edges, and across model Faces, but not the size of Tet elements as they develop into the interior of a volume. Up till Cosmos2006, at least, the volume meshing algorithm was controlled only by the Global Mesh setting. The best example of how this could work against you would be if you had a large, blocky model, and wanted to use a large Global mesh size, but there existed a thin-plate region where you wanted to apply a smaller local mesh. In such an area, you would want to assure more elements in the thru-thickness direction. But if you applied a local mesh control to the faces of this region, you would get a high density of small elements on the outer faces, but still get large, and/or high-aspect, elements connecting the top and bottom faces thru the interior. Just the opposite of what you wanted! Is this still true in 2008?

I have been told by Solidworks people that, indeed, this situation has been amended, and that now if you apply a local mesh control to an entire solid body, the mesh size will apply throughout the thickness of the volume, not just at the surface. I've conducted mesh tests to try to verify this. First off, I've discovered that the interior meshing algorithm has definitely improved. This is great news. I had simple test-cases back in 2005 and 2006 that clearly proved that the interior tet development did not respond to local mesh control, and these same test cases now produce much, much better results.

However, I can't prove that the interior nodes are placed to adhere to the (Volumetric) local mesh size. The situation is messier than that. In fact, you can set up a model with a small local mesh size on all faces, and compare this to the same mesh size applied in one selection to the solid body, and you often get the exact same number of elements. And those elements DO get larger as they progress into the interior. So the (volumetric) mesh control is not really the source of the improvement. What has changed, then? It seems that the change is not specific to controls applied to Volumes, but in fact the mesh algorithm has changed for ALL local mesh transitions. My best guess is that the geometric growth ratio is now being respected for the creation of every Tet, whether on the surface or in the interior.

The BONDED contact now gives better results for models with different mesh densities between bonded faces - especially important for an H-adaptive mesh.



In some ways, this is a more valuable improvement than simply enhancing a volume-specific control, because Cosmos2008 is now producing meshes with much, much smaller, more consistent Aspect Ratios. So, life got better. I'll keep researching this issue of how to get a constant-size local mesh control in the interior of a solid, so watch this space for updates.

The Material/Bodies folders can now be updated!

This is big news. This will remedy a hotline call that I have been answering perhaps 2 or 3 times a month. When you create a new study, Cosmos analyzes the assembly (or part) and takes a head-count of all the distinct parts, and the Bodies in each part. (Cosmos does not distinguish between a multi-body part, or an assembly, to the FEA everything just becomes a 'body' which gets assigned to an element group). The problem occurred if, after creating the study, you went back to edit the assembly, and added or removed components; or especially, you replaced one part for another, or created a SPLIT inside a part file that resulted in a change in the number of solid bodies.

In any of these cases, it was very difficult to get Cosmos to re-examine the body-count and update the folders where you apply Material properties. Thus, you would have a material Element group for a body that no longer existed, or perhaps a certain component in the assembly that simply would not mesh. Symptoms from this point on could vary. Sometimes you would try to RUN the study, and get an error, "Unable to apply materials to element group number xxx". Sometimes the error message came when you tried to mesh, and would get "A element group is not defined". Or, you would get no error message at all, but you would see certain bodies/parts that simply would not mesh, even if all their neighbors did.

KAP's Tip:

Spread the word: Use "Update All Folders" each time you Add, Remove, Replace, or Split solids.

The solution in Cosmos 2008 is that you can now right-mouse-click over the folder for a study, and select the new command "UPDATE ALL FOLDERS". Why couldn't they simply make this function automatic? Oddly enough, it WAS once automatic, or at least tried to be. In SolidWorks 2003 and before, an Assembly would automatically update its Cosmos Element groups each and every time you activated the window, or changed to a different configuration. Performance was terrible. Then in v2004, (I think), they tried to make the Update more discriminating - and created the situation where it was too easy to 'fool' the assembly, (especially when SolidWorks added multi-body part files

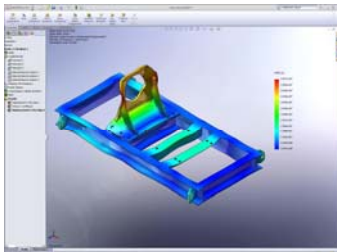
in 2005). So the current solution requires an informed user, but has a minimal performance hit.

Coupled studies!

There are a number of times in Cosmos where you can run a compound study. That is, you can use the output of one or more studies, as the input data for starting another study. This can come in very handy, but you have to keep a mental road-map of what element types are supported for each type of study. This is because it usually requires that the 2 (or more) studies must share a common mesh. In some cases, this limits the sort of loads and boundary conditions you can apply common to both, and in some cases, there is a work-around to get two studies to share a common mesh.

For this discussion, keep in mind that by "identical", I don't just mean that the two meshes should 'look' the same. Whenever you create Gap or Contact conditions, the program always nags you to re-generate the mesh. Why? Because a Contact condition is represented by creating additional mesh elements that connect the surfaces in question. Think of these elements as tiny, invisible springs with conditional stiffness. Your total number of mesh Nodes don't increase, but the number of mesh Elements do, when you add Gaps or Contacts.

COSMOSWorks 2008 can now analyze problems undergoing vibrations for their deflections and reactions.

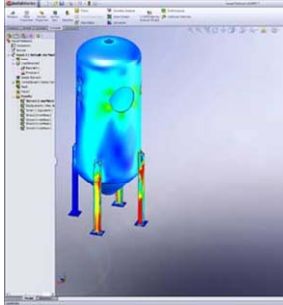


Thermal Stress Example:

IF you have a part subjected to heating or cooling, so that it develops temperature gradients, you run a Thermal analysis. Then you can identify the result of this Thermal study as an input to a Stress study, to measure the thermally-induced stress and strain. BUT - the two studies must have a common mesh. What if the Stress study also needs to apply Contact conditions, or Gap elements, (for a shrink-fit, for example)? You cannot create these kind of mesh elements in the Thermal study, so the two meshes will not be identical. What to do?

In this case, you apply a work-around. You set up the Thermal study, but do not mesh or run yet. Then set up the Stress study, and Mesh it. Finally, drag-n-drop the Stress mesh onto the Thermal study's mesh folder, and then run the two studies. This works because the Thermal study accepts (and then ignores) the gap elements for the Contact condition, even though these elements are not native to a Thermal study.

Pressure vessel design study helps users to combine the effects of multiple independent loading scenarios, such as dead weight, internal pressure, thermal, or seismic loading.



The reverse trick will not work, however. What if your Thermal study wants to represent the effects of a Contact Resistance between two bodies? You apply Contact resistance by using the Gaps/Contacts folder, but the problem will then no longer be valid for feeding a Stress study. Stress studies will neither accept, nor even ignore, a Thermal Contact element in the mesh, and so (to my knowledge) you currently cannot use thermal Contact resistance to formulate a thermal Stress problem.

Fatigue Analysis Example

This is another example of a compound analysis. Let's say you have an assembly that is repeatedly subjected to two load conditions. For example, I recently analyzed a landing gear component that undergoes a fairly low-level Compressive stress with a very high number of cycles, corresponding to rolling down a bumpy runway. But it also experiences a much larger, but less frequent, torsional load each time the nose-gear touches-down on landing. A Fatigue analysis will allow you to compute the cumulative damage from these combined loading conditions.

In my case, I applied the torsional loading by way of a Remote Load, (this is the 'cleanest' way to apply a pure torque to a model without having to compute and apply opposing loads on paired faces). Each of my load-condition studies ran great, but the Fatigue study would not. This was how I learned that a Remote Load / Remote Mass actually adds invisible connector elements to the mesh. The two studies had different numbers of mesh elements - and so the Fatigue study saw them as apples and oranges.

In this case, the work-around was very simple. I copied the Remote Load to both studies. Then in the Compressive load case, I changed the magnitude of the torque to be vanishingly small, so that the necessary mesh elements were present, but had no real effect on the solver results. I believe that most of the Connectors, (Like Springs, Links, and Rigid Connectors) create their own mesh elements as well, and so could require this same work-around if they are not present in all the load condition studies.

Pressure Vessel Studies

Cosmos 2008 introduces a new study type, the Pressure Vessel Design study. This also allows you to create a number of linear Stress studies to represent simple load cases, and then combine them. Again, the combination of the effects of 2 or more studies requires the identical mesh for each study.

Distributed loads - Half? Doubled?

KAP's Tip:

*In CosmosWorks,
multiple face
selections for a Force
are additive.*

Here's another oft-repeated hotline question. You have a net force load of, say, 100 lbs, that you want to apply to a pair of faces. If you create a separate Force boundary condition for each face, then clearly you would have to divide the force by half, and each face would need to be loaded to 50 lbs. But what if you select both faces in the same dialog for creating a single Force input? Perhaps this is a surprising answer, but you must still divide the load by half, because Cosmos will apply the specified force to EACH selected entity. It will NOT divide the force into the number of selected entities.

Fortunately, this rule does NOT apply to a Remote Load. If you input a 100lb Remote Load and select two (or more) faces, the load will be distributed over their net area. I don't know why this disparity exists between directly applied loads and remote loads, but it has come in handy on many occasions. Also, if you perform CFD analysis for CosmosFloWorks, you know that this application will also distribute boundary conditions over multiple faces. So if you jump back and forth between these two programs, be alert!

Summary

O.K., that's the news with Mesh structure as of Cosmos 2008. Some of our customers have not migrated up from the 2007 version yet, (especially the larger sites), so I will keep my original 2007 article on our web site for some time. In general, instead of overhauling and re-publishing articles, I'll just make sure they are explicit about what version is covered. If you come across corrections or updated information to any of these articles, I welcome your comments at: Support@capinc.com, subject line: KAPs Corner