

The Focus



A Publication for ANSYS Users

Contents

Feature Articles

- [Tet-meshing: ANSYS vs. ICEM CFD](#)
- [ANSYS Resources on the Web](#)

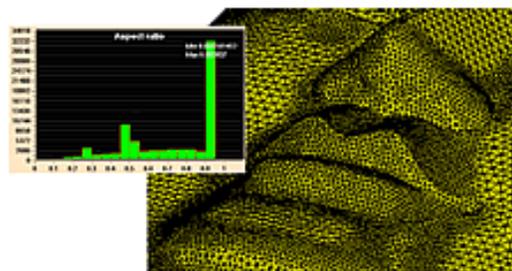
On the Web

- [ModelChecker \[PDF\]](#)
- [ANSYS Users' Conference 2002](#)

Resources

- [PADT Support: How can we help?](#)
- [Upcoming Training at PADT](#)
- [About *The Focus*](#)
 - *The Focus* Library
 - Contributor Information
 - Subscribe / Unsubscribe
 - Legal Disclaimer
- [Next in *The Focus*](#)

Tet-Meshing: ANSYS vs. ICEM



... Also

- [ModelChecker for ANSYS](#)
- [Resources on the Web](#)



ANSYS
Support Distributor

© 2002, by Phoenix Analysis & Design Technologies, Inc. All rights reserved.

The Focus



A Publication for ANSYS Users

Tet-meshing: ANSYS vs. ICEM CFD

by [Joe Woodward](#), PADT

Update (May 2003): AWE Offers Improved Solutions

With the release of ANSYS 7.0, many of the meshing issues raised in this article are now addressed with new functionality found in the ANSYS Workbench Environment (AWE). This modern pre- and post-processor delivers on the promise of robust and associative CAD to FEA connectivity. As PADT has used these new tools over the past 8 months, we have found that AWE produces good quality tetrahedral meshes on almost any geometry. The best part is that these new tools are part of the ANSYS we all know and love, so you can have your cake and eat it too. There is no need to use a separate software package.

Even more features will be available with the 7.1 release. Please see future issues of *The Focus* for details.

Introduction

This article is intended to introduce you to ANSYS's new product, ICEM CFD 4.2. Now don't let the name fool you. ICEM is much more than a CFD tool. It is a high-powered mesh generation tool that can create hex, shell, and prism meshes as well as tetrahedral meshes, which is the focus of this article. ICEM also has direct CAD interfaces to many of the standard CAD packages and direct geometry reader for many others. These features will be highlighted in future articles in *The Focus*. Since you are most likely already familiar with ANSYS, the majority of this article will describe ICEM and its functionality.

In this article, we compare the tetrahedral meshing capabilities of ICEM CFD and ANSYS 6.0 on large non-symmetric models with relatively small features. The sample case in Figure 1, constructed in Pro/Engineer, is a 5" cube with a 0.050" curving groove on one side and three holes of varying size and depth. The purpose of the exercise is to mesh the block with as few elements as possible to keep the model size down, while still resolving the small features. In order to make the test case more like CAD geometry of a real part, some normal features such as splinter surfaces, like in Figure 2, were included.

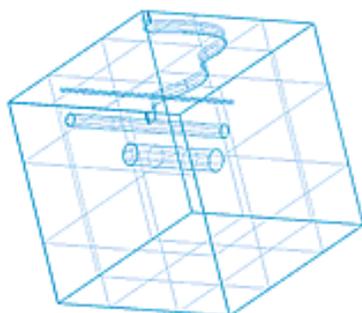


Figure 1. Test Case Model

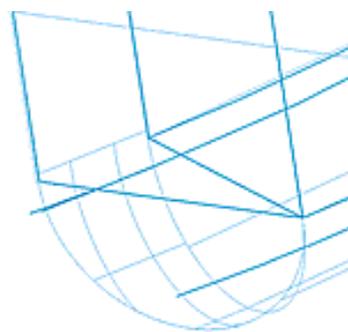


Figure 2. Surface Splinter

The Focus



A Publication for ANSYS Users

ANSYS / ICEM Similarities and Differences

Before we begin meshing the part, I would like say of few words about the similarities and differences in the two programs. Both ANSYS and ICEM have the ability to group features such as areas and line, or surfaces and curves as they are called in ICEM, but in ICEM, the use of families is much more intertwined with the overall process. ANSYS allows you to select entities and group them into components, but only entities of a common type are allowed in any one component. ICEM on the other hand starts by putting everything into at least one family and then you can add families and move entities around. The Display Menu (Figure 3), that is always present in ICEM, allows you to quickly pick families for display and operations purposes. The families in ICEM can also hold more than one type of entity. Any entities that are created from other entities that are in a family automatically get put into that family, too.

Another similarity is that in both ANSYS and ICEM, element sizes are assigned to entities (areas, lines, etc.), but there is a difference in how the sizes are defined. In ANSYS, sizing is done as an absolute size or number of elements that will be on an entity. In ICEM, an absolute value is specified for a reference size and all other sizes are set as a multiplier on the reference size. This makes it extremely convenient when the model is scaled, since you only have to change the reference size. It is inconvenient, however, if want to keep the same element sizes around a feature while changing the overall reference size, since you then have to recalculate the multiplying factor for that feature. ICEM also allows you to view all the sizes that are applied to the families in a single table, as shown in Figure 4. ICEM has Natural Sizing which is analogous to the Smart Sizing in ANSYS, but that is not discussed in this article. ICEM can also show the size of the elements with respect to the geometry before the mesh is calculated. Figures 5 and 6 shows the ANSYS and ICEM element sizing. ANSYS will show the number of elements along the lines if the element size has been specifically set for the lines, but the size and shape of the elements as they transition away from the lines is unknown until meshing is initiated.

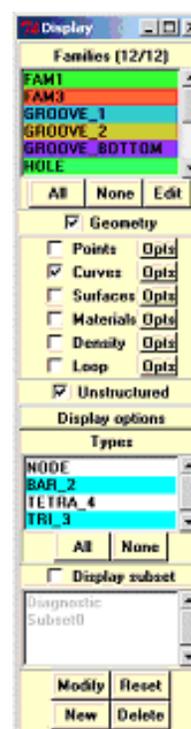


Figure 3. ICEM Display Menu

Family	Size	Height	Ratio	Min Size	Max Deviation	Width	Int Wall	Split Wall	Fix for Adaptation
GROOVE_1	0.05	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
GROOVE_2	0.05	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
LEVEL0	1	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
SURFS	0.025	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
HOLE	0.15	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
HOLE2	0.1	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
HOLE3	0.05	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
GROOVE_BOTTOM	0.0125	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Show size params using ref size

Apply Dismiss

Figure 4. ICEM Family Mesh Sizing Dialog

The Focus



A Publication for ANSYS Users

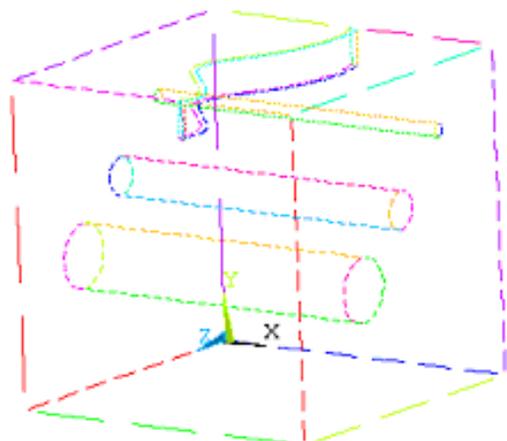


Figure 5. ANSYS element sizing

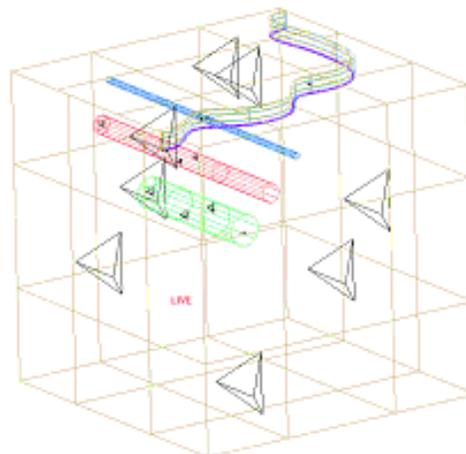


Figure 6. ICEM element sizing

The greatest difference between the ANSYS and ICEM CFD tetrahedral-mesh generation is in the algorithms used for meshing. In ANSYS, the mesh starts at the lines and is then advanced into the areas, and then into the volumes from the areas. This is called a paving algorithm. As seen in the example below, this often results in poorly shaped elements where two greatly dissimilar meshes converge, and the remaining elements try to fit to both mesh sizes. ICEM on the other hand, uses an OCTREE-based meshing algorithm. This means that ICEM fills the volume with elements of the largest size that you have specified and then searches for geometry that specifies a smaller size. If a smaller size is required, the larger elements are split into two. This continues until all the sizing requirements are met. This allows for a smooth transition from coarse to fine mesh size. It also allows for another main difference, and one of ICEM's most useful tetrahedral-meshing abilities, the ability to ignore small features.

CAD geometry often includes many small surfaces that are not part of the physical geometry of the part, but are merely the result of the mathematical formulations. With such surfaces, shown in Figure 7 as an ANSYS area, ANSYS must resolve each area that is created from the otherwise continuous patchwork of surfaces. This often causes major complications in meshing parts of relatively simple topology. To avoid this, ICEM allows the unnecessary curves to be deleted, as in Figure 8, leaving curves only where there is a topological feature that needs to be captured. This helps avoid high element counts from refinement around unnecessary features and poor element quality due to surface splinters. ICEM can extract all the curves or only those curves that are created when two surfaces meet at some angle that is greater than the tolerance set by the user. This will usually produce all the curves necessary for a given topology, without making curves that outline each individual surface.

The Focus



A Publication for ANSYS Users

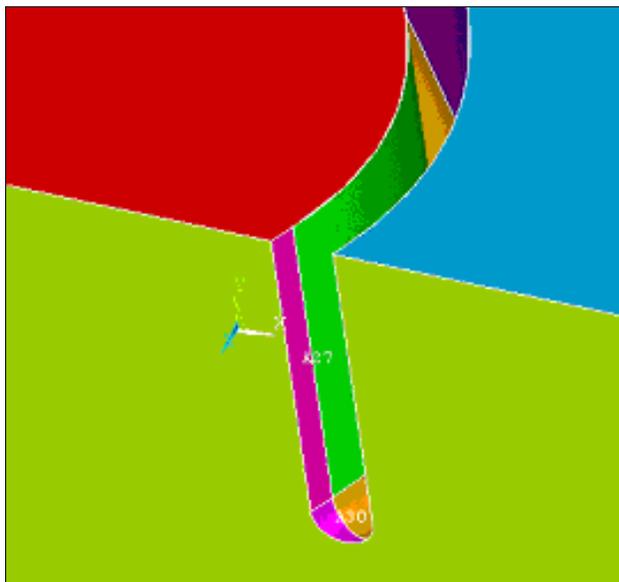


Figure 7. ANSYS areas

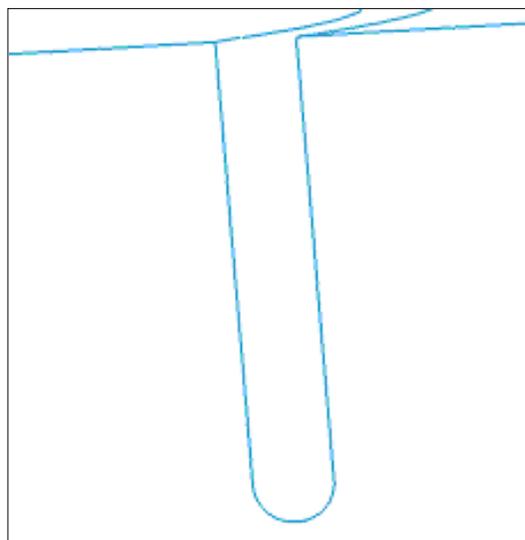


Figure 8. ICEM uses only necessary curves

ICEM Meshing Procedure

1. **Import geometry.** An IGES file was created from Pro/E model so that both programs would start with the same file.
2. **Create curves using the Build Topology function.** As discussed earlier, this creates curves only at edges above the given tolerance.
3. **Organize surfaces and curves into Families.**
4. **Specify a material point.** This tells ICEM which space you want to mesh, inside or outside the surfaces. We pick inside for structural. CFD analysis often meshes outside, around an object.
5. **Set reference size.**
6. **Set element sizes on Families.** At this point the sizes can be shown as in Figure 6 and adjusted.
7. **Calculate the mesh.**
8. **Check the mesh quality and Smooth the mesh where needed.**
9. **Coarsen or refine the mesh if desired.** In this case, the Coarsen function was used to reduce the element count from 390,678 elements to 139,754 elements. You can specify which Families not to coarsen.
10. **Output the mesh for ANSYS.** Boundary conditions can be applied in ICEM, and the defined Families are translated to components within ANSYS.
 - Total approximate time: 2 hours, 10 minutes
 - Resulting Mesh: 139,754 elements
 - Minimum Element Quality: 0.310

The Focus



A Publication for ANSYS Users

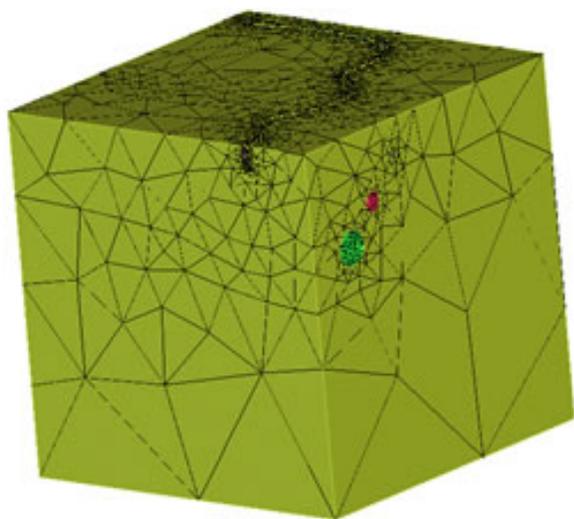


Figure 9.

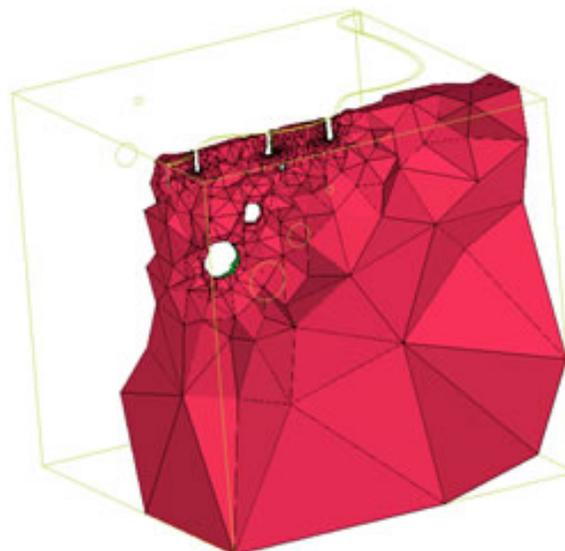


Figure 10.

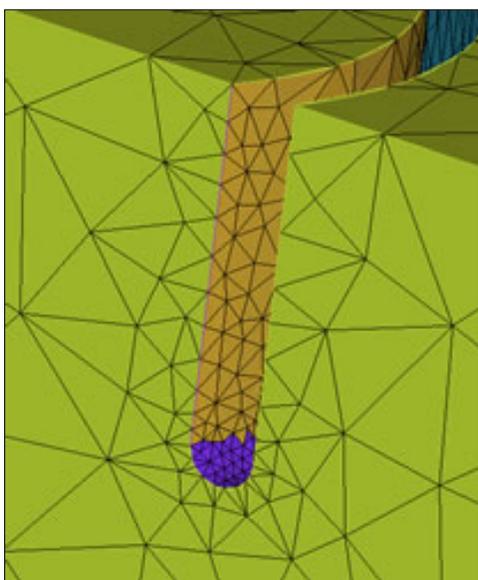


Figure 11.

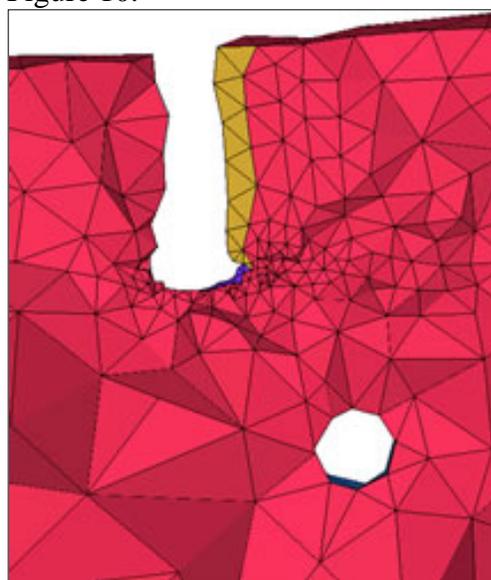


Figure 12.

ANSYS meshing Procedure

As the saying goes, There are many ways to mesh a model. As an experienced ANSYS user, I tried to start with the simplest approach using the default values, as many users would tend to do. I then worked in more complicated techniques. Given time, better meshes could be achieved, but I felt that this was sufficient for the exercise.

1. **Import Geometry.** This is the same IGES file that was used for ICEM.
2. **Set element types.** Shells as well as solid elements were defined in order to mesh areas before meshing the volume.
3. **Turn Smart-sizing off.** This helps for the area meshing.
4. **Select areas of the groove and the holes separately and set element size with AESIZE and LESIZE.** The element sizes are set to match the ICEM model.

The Focus



A Publication for ANSYS Users

5. **Mesh the areas of the groove and the holes.**
6. **Turn Smart-sizing on. SMRT,6**
7. **Mesh the volume.** At this point, several errors occurred. After using AMESH on all the areas, an error in the geometry was discovered. Figure 13 shows that the groove did not cut through the back surface of the block. ICEM had no problem, so I did not think to look for such things.

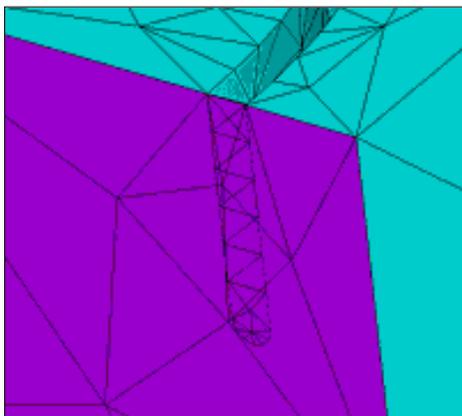


Figure 13. Area covering end of groove

The original geometry is unmeshable in ANSYS and is then fixed in Pro/E.

8. **Repeat steps 1 to 7 with the newly corrected geometry from a new IGES file.** This time, the following error was encountered (see Fig. 14).



Figure 14. Memory Errors

9. **Set Smart-sizing to a coarser setting. SMRT,8**
10. **Mesh Volume.** Same error.
11. **Increase element size with ESIZE.**
12. **Mesh Volume.**
13. **Clear area meshes.** Needed in order to do Boolean operations on the volume.
14. **Use the workplane and the VSBW command to cut volume.** Sliced into eight volumes.
15. **Reselect and reassign element sizes to the new areas of the groove and holes.**
16. **Re-mesh the areas of the groove and holes.**
17. **Set element size with ESIZE.**

The Focus



A Publication for ANSYS Users

18. **Set Smart-sizing.** SMRT,6
19. **Mesh the volume with the holes and the groove.** Shape errors occurred.
20. **Repeat Steps 15 to 19 several times, varying the element sizes.** Get something that works.
21. **Mesh the remaining volumes one at a time.** The same error as in Step 10 occurs if more than one volume meshed with each VMESH command.
22. **Clear the Volume mesh.** Try to improve mesh quality.
23. **Set mesh option manually with MOPT command.** Change the expansion and transition settings.
24. **Re-mesh all the volumes and clear area meshes.**

The mesh in Figures 15 and 16 resulted from Step 21, with the default settings for the mesh options set by Smart-sizing. There were no shape errors, but there were a lot of shape warnings.

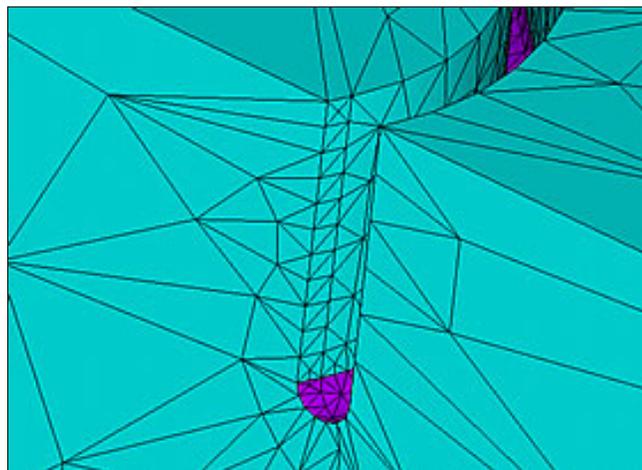
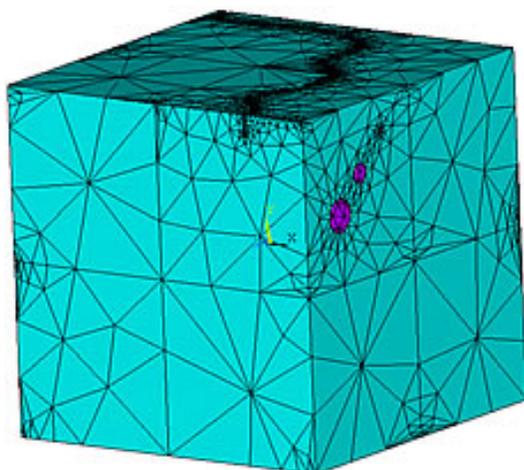


Figure 15. ANSYS Mesh with Smart-sizing Figure 16. Large Transition Rate

The final mesh used for comparison with ICEM is shown in Figures 17 and 18. This mesh resulted from turning off Smart-sizing and then manually setting mesh options for the expansion and transition rates with the MOPT command.

- Total approximate time: 8 hours
- Resulting Mesh: 113,362 elements
- Minimum Element Quality: 8.6×10^{-4}

The Focus



A Publication for ANSYS Users

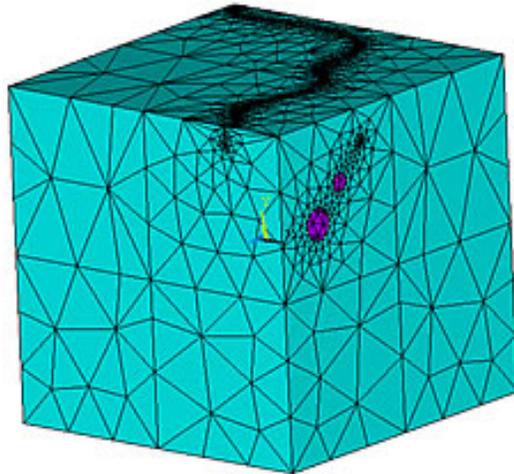


Figure 17. Final ANSYS Mesh

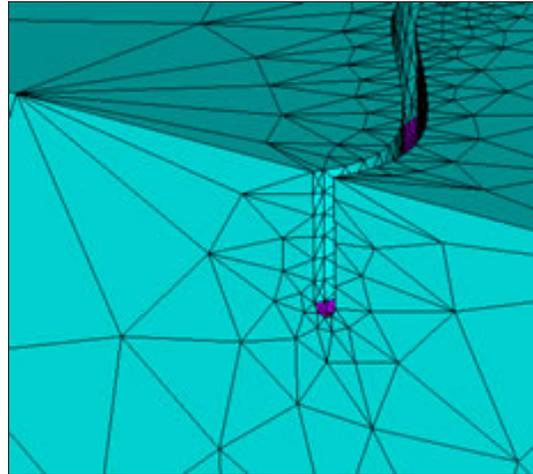


Figure 18. Final Trainstions

ICEM Mesh Editing Features and Tools

The quality metrics given above come from the standard ICEM shape-checking algorithm. ICEM can also check for minimum angle, aspect ratio, warping, and many other characteristics. ICEM also presents this information in a useful and convenient manner. A histogram shows the number of elements that have a given quality value. This allows the user to see the distribution of the element quality and how many elements are on the borderline of the established quality cutoff limits. The plot in Figure 19 was obtained by writing out the ANSYS database with the CDWRITE command and then reading the resulting file into ICEM.

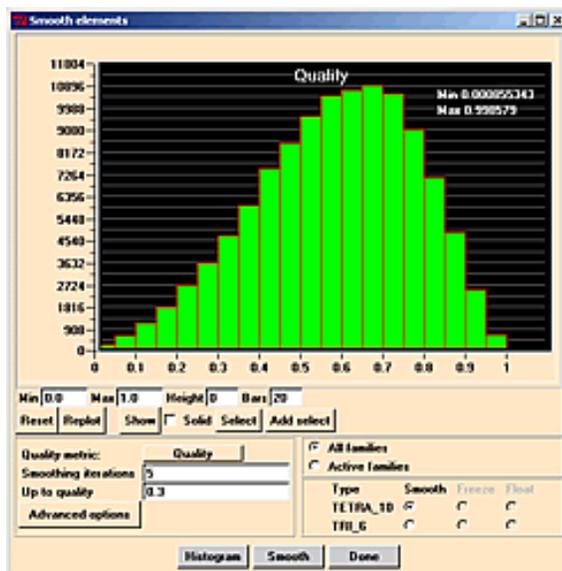


Figure 19. Quality of ANSYS Mesh

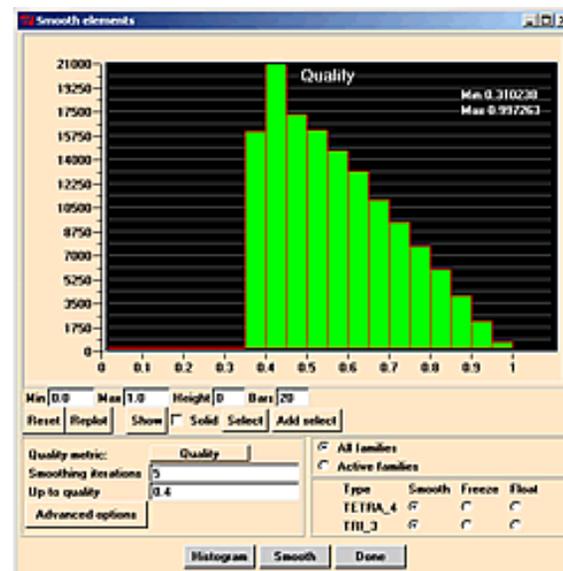


Figure 20. Quality of ICEM Mesh

The Focus



A Publication for ANSYS Users

Conclusions

ICEM CFD is a powerful addition to the ANSYS suite of tools for Finite Element Analysis. It has many more features and mesh-editing capabilities than I have had time to cover. Functions like single-node-merging, element deletion, and cyclic symmetry enforcement are just a small part of the overall functionality.

Of course, meshing capabilities depend greatly on the type of geometry being meshed, and this particular case was built to give ANSYS problems. ANSYS has excellent capabilities and works very well with most models and features. There are just some things where a different approach works better, and in those cases, it is always good to have more tools in your toolbox. To learn more about ICEM CFD, visit the [ICEM CFD](#) web site, or contact [Joe Woodward](#) at PADT.

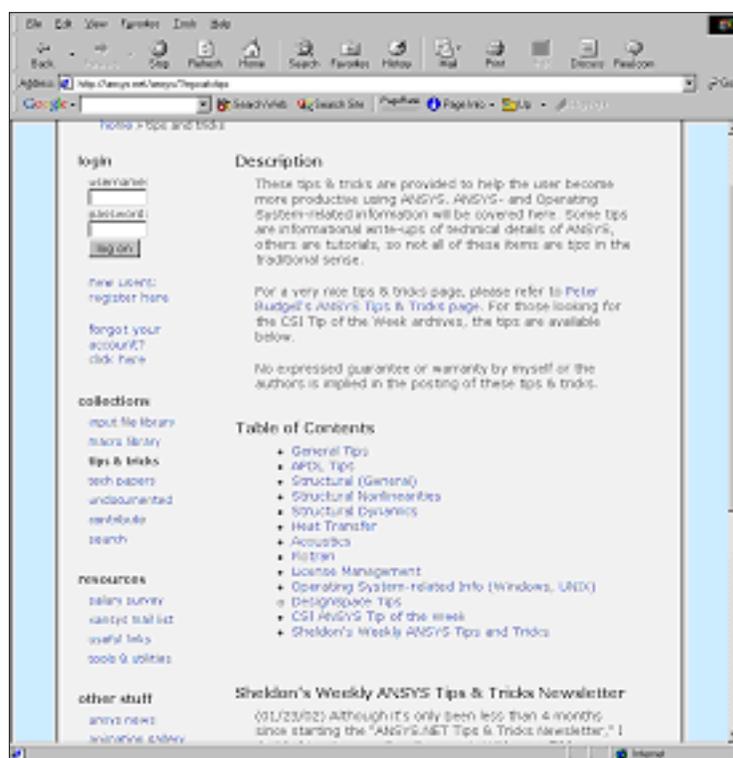
The Focus



A Publication for ANSYS Users

ANSYS Resources on the Web

There are numerous resources on the internet for ANSYS users. The best place to start is, of course [ANSYS.NET](http://www.ansys.net), a spectacular website built and maintained by Sheldon Imaokas. If you are not familiar with his site, then if you remember only one thing from this article, remember this: **www.ansys.net**.



Sheldon's Web Page: Macros, Tips & Tricks, Tech Papers, Undocumented Commands, Surveys, News, Online Resource List, and more!

Some Stuff Not Found on ANSYS.NET:

ANSYS Knowledge Base and FAQ:

http://www.ansys.com/services/service_support.htm

Service Packs / QA Notices / New Features / Tech Releases:

<http://www1.ansys.com/customer>. *Note:* You have to be a paying TECS customer to use this resource (meaning you have a current license agreement that includes technical support).

The Focus



A Publication for ANSYS Users

Hardware Benchmarking:

<http://www.padtinc.com/support/benchmark>. Benchmarking of hardware on ANSYS-specific problems. Review others or perform your own.

Peter Budgell's Page:

http://www3.sympatico.ca/peter_budgell. Tips, Modeling Issues, and more.

Tutorials: (in rough order of quantity)

<http://www.mece.ualberta.ca/tutorials/ansys>. This is the gem.

<http://www.me.umn.edu/courses/me5348/ansys.html>

<http://www.engr.uky.edu/~jrbake01/ansystutor.html>

http://www.me.cmu.edu/tutorials/ansys_56/introduction.html

Materials Data:

<http://www.matweb.com>. This is the jackpot for materials searching. If you don't have it bookmarked or memorized, you will soon...

<http://ansys.net/ansys/?mycat=matl>. Sheldon has captured the significant ones we're aware of.

Online ANSYS Community / E-Mail Board:

<http://groups.yahoo.com/group/xansys>

The Focus



A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

Contributor Information

Please don't hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.