Understanding Accuracy and Discretization Error in an FEA Model

Jon Pointer

Woodward Governor Company

Abstract

The often-ignored topic of mesh discretization error is examined to identify a simple set of rules that the average user can enlist to determine solution accuracy. The sources of discretization error are explained, tools to quantify it are introduced and an example is given. The purpose is to bring an understanding of these issues and usable tools to the common user who is not proficient in the mathematical basis of FEA.

Introduction

How accurate is my solution? What is the error contained in my analysis? These questions relating to the accuracy of any given finite element analysis (FEA) are the most critical points that should be explored during the course of any modeling effort. "Without *some* indication (of accuracy), the solution is effectively worthless" (Burnett, 1987, pg. 55). Unfortunately, it has been the author's experience that the topic of accuracy is often overlooked in analysis reporting by end users and product training by FEA vendors. The focus is instead on capabilities, outputs and usability, with accuracy either not addressed or buried in a remote technical document that is beyond comprehension for the average engineer utilizing the FEA tool.

What is needed is a layman's discussion of what causes modeling errors and how those errors can be quantified to determine the accuracy of a solution. This paper strives to accomplish just this within the specific error subset of mesh discretization error. If I've got everything set up right, my loads and boundary conditions are correct, I'm using the correct element, my material properties are correct..... how do I know if I have arrived at an adequately fine mesh to properly capture the result I am interested in?

Sources of Error in FEA - Discretization Error

As described in other sources, FEA errors fall into three broad categories (Shah, 2002). These categories are as follows:

User Error - Incorrectly setting up the model due to inexperience.

Modeling Error - Incorrect representation of the real world event.

Discretization Error - Insufficient mesh density to properly capture the solution.

The primary focus in this paper will be discretization error. Here the user has set up the problem correctly, properly approximated the real world situation and entered all the correct information, but the density of the mesh is insufficient to capture the correct solution. Ansys and other FEA codes offer tools for the user to investigate how much error is being introduced by inadequate mesh density. Utilizing Ansys, the author will walk the user through the source of these errors, the tools available to measure them and a simple example. Note that others have presented these tools but the author has found these writings to be ineffective for the average engineer who is not trained in the mathematics of FEA.

What is Discretization Error?

The basis of FEA is taking a system governed by differential equations and partitioning it into regions (elements) that can individually be solved with simple linear equations. This partitioning creates a series of linear approximations that strive to map, as closely as possible, the true continuous solution. Using structural stress-strain as the primary example, the FEA code takes the mesh approximation and solves for the displacements at each node given the loading and constraints on the model. Next, as a secondary operation, the code approximates the stress contour in each element by looking at the relative displacement of the nodes of that element. In this manner a stress contour is determined for each element - an approximation of the stress throughout each individual element, including at the edge of the elements and the node locations. Given that this contour is approximated based only on the displacement of the nodes of the element in question, the stress contour will not necessarily be continuous from one element to the next. Essentially, this discontinuity of the stress contour from one element to another is the discretization error. As the size of each element is reduced towards zero this error goes to zero given that the increasing number of partitions is approaching the true continuous system.

What Do You Actually See in the FEA Post Processor?

Within an FEA package such as Ansys, the stresses (or other results of interest) are reported in several different forms. Typically the default is not to present the element stresses computed as described above. Instead, the postprocessor will average the stresses based on the surrounding elements so that the stress contour reported is continuous across elements. This is called an averaged stress plot (PLNSOL in Ansys). Alternately, packages can also plot a non-averaged stress plot where the stress reported in each element is only based on the displacement of its own nodes (PLESOL in Ansys). Again, as described above, as the element size approaches zero the discretization error will approach zero and these two stress plots will converge to the same display.

How Do I Assign Numbers To The Discretization Error?

Given the discussion above, each node will have x stress values associated with its location where x is the number of elements that share that node. Ansys Theory Reference, Section 19.7, calls each of these x stresses the stress vector of node n of element i.

$\left\{\sigma_n^i\right\}$ = stress vector of node n of element i.

If these x stress vectors are summed and divided by x, the number of elements sharing the node, the averaged stress vector of node n is computed.

$\left\{\sigma_n^a\right\}$ = averaged stress vector at node n.

Next for each element the stress error at each node is approximated by the following calculation.

$\left\{\Delta\sigma_n^i\right\} = \left\{\sigma_n^a\right\} - \left\{\sigma_n^i\right\}$ = stress error at node n of element i.

Therefore, if the mesh were fine enough such that two neighboring elements had perfectly continuous stress contours, the stress error at each node would be zero.

Ansys does not present this stress error at each node directly, instead it integrates the stress error over the volume of the element and reports an energy error for element i. It labels this element value SERR which can be accessed with either the PLESOL or PRESOL commands. Therefore, the goal of any analysis would be to drive the value of SERR for each element as close to zero as possible.

Additionally, Ansys sums the energy error for each element and reports a total energy error for the entire model. (SERSM accessed by the *GET command - *GET,*user_var*,prerr,0,SERSM). Next, Ansys normalizes the energy error against the strain energy and reports this as the percentage error in the energy norm. This is performed by the following equation:

$$E = 100\left(\frac{e}{U+e}\right)^{0.5}$$

where:

e = total energy error for the entire model.

U = strain energy over the entire model.

E = percentage error in energy norm.

E can be accessed by the PRERR command with the entire model selected and PowerGraphics turned off. Additionally, E can be reported for a subset of elements by selecting only those elements and then issuing the PRERR command. Again, the goal would be to drive the value of E, or the PRERR result, to zero.

How Do I Know When I've Reached The Correct Solution? What about Convergence Studies?

At this point the user has a mesh, he or she has verified loads, constraints, and modeling method and has accessed a few parameters that give an indication of the discretization error. So how does he or she know that the errors are acceptable and whether the particular value of interest is correct? Shah, 2002, has offered guideline criteria of limits for each of the discretization error parameters presented above based on experience. Therefore, the user can apply these rules and gain confidence that the correct solution has been reached. However, others (3, 5, 6, 7) assert that the only true indication of an accurate solution is to perform convergence studies by increasing the element count in the model and assuring that the result of interest is graphically converging to a stable value.

With regard to convergence, Burnett, 1987, offers the following summary.

1. If a model satisfies the completeness and continuity conditions (defined below), the energy of the entire model will converge to the exact solution as the size of the elements are decreased.
2. In a well-posed problem (defined below), convergence of energy will also result in convergence of a particular local result in the model.
3. Once convergence of the particular result is graphically established, the error in the final result can be approximated by the error between final two solutions generated in the convergence study.

In applying the convergence guidelines listed above, several qualifications must be presented.

1. To satisfy the completeness condition, the model must be such that as the element size approaches zero the stress in each element can approach a single value across the element. Therefore, elements containing a singularity condition will not meet this criterion because the stress at the singularity node will always tend towards infinity, creating large gradients across the elements containing that node.
2. The continuity condition is met by standard FEA formulation where the element shape functions ensure that the displacement solution is continuous across elements.
3. A well-posed problem must not have any singularity conditions in areas where stress is of interest. Interior corners and other stress concentrators must be adequately represented by the true radius to avoid singularities. Singularities will be apparent where the model does not converge, but instead diverges towards infinity, as the mesh density is increased.
4. The author wishes to point out the importance of "graphically" establishing convergence. Stress and other results parameters often wont converge smoothly towards a final value. Often they will oscillate or make large changes on top of a normal convergence curve. The user must make adequate runs of significantly changing mesh sizes to ensure that the solution is truly converging. Note that only making a small change in mesh size will potentially show a small change in the

result that may be misinterpreted as convergence. Mesh changes must be significant. A typical rule is to double the element density in the area of interest with each iteration.

5. As noted in Burnett, 1987, it is important that local mesh refinement is not too drastic. Item #1 in Burnett's list above is only theoretically valid if <u>all</u> elements are reduced in size uniformly. Therefore, when only a portion of the model is refined a finite error will be "trapped" in the unrefined elements and the solution will converge to some steady error instead of the actual solution. This error can be minimized by ensuring gradual transitions between mesh regions. Additionally, ensure that low mesh density regions do not contain large stress gradients that are not properly captured. In Ansys, note that *advanced options* can be selected when refining a mesh and the *depth of refinement* can be controlled. The author has found that this should be set to at least 2 in order to ensure smooth transitions.

Verifying Mesh Accuracy in an FEA Model

Given the items mentioned in the previous section and the fact that no single hard rule has been established in the industry, it is the author's opinion that the FEA user should do the following to ensure the correctness of an FEA result with regard to mesh density and discretization error.

1. Ensure that a convergence study has been performed in the area of interest with a minimum of three iterations. Plot the results to graphically verify convergence. Ensure that the x axis of this plot shows some indication of mesh density in the area of interest - number of elements on a curve, elements per unit length, etc. This is necessary to show true convergence over apparent convergence that is only due to a relatively small change in the mesh. Note that FEA vendors such as Ansys offer very helpful tools for automating this remesh and convergence process (9). However, by simply accepting convergence via a final convergence number without examining the convergence curve, changes in mesh density and the other factors listed below, the model could still contain significant errors.
2. Visually verify that the non-averaged and averaged results plots in the area of interest look very similar and show similar maximum values.
3. Perform Criteria 1A and 1B from Shah, 2002, shown in Appendix A.

A Simple Notched Beam Example

To begin, let's take a very simple notched cantilever beam model with the following parameters (see Figure 1).

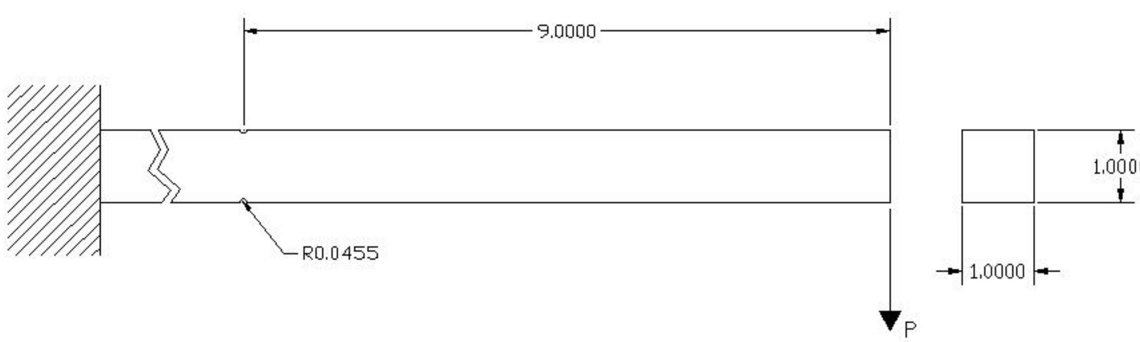

Figure 1. Notched Cantilever Beam Geometry

L = 9 in. (Length from notch to end)

B = 1 in. (base or width)

H = 1 in. (height)

r = .0455 in. (notch radius)

P = 1000 lbs. (load at right end)

The following can be calculated (ignoring the notch stress concentrator):

$$I = (B * (H - 2 * r)^3)/12$$
$$I = .06259$$
$$\sigma_o = (P * L * C)/I$$

where

C = 0.4545 in. (half the height at the notch)

Therefore, stress on the top of the beam in the horizontal direction at the location of the notch with no concentration factor would be:

$$\sigma_o = (1000 * 9 * 0.4545)/0.06259$$
$$\sigma_o = 65353.9\, psi$$

Next, according to Shigley, 1983, the stress concentration factor for the notch in this beam would be approximately 2.4. Therefore, the actual stress at the notch would be:

$$\sigma_a = \sigma_o * K_t$$
$$\sigma_a = 65353.9 * 2.4$$
$$\sigma_a = 156849.3\, psi$$

Now we will represent the same scenario in an FEA model (see Figure 2). Note that the 1000lb load has been divided into two 500lb loads on the two end keypoints.

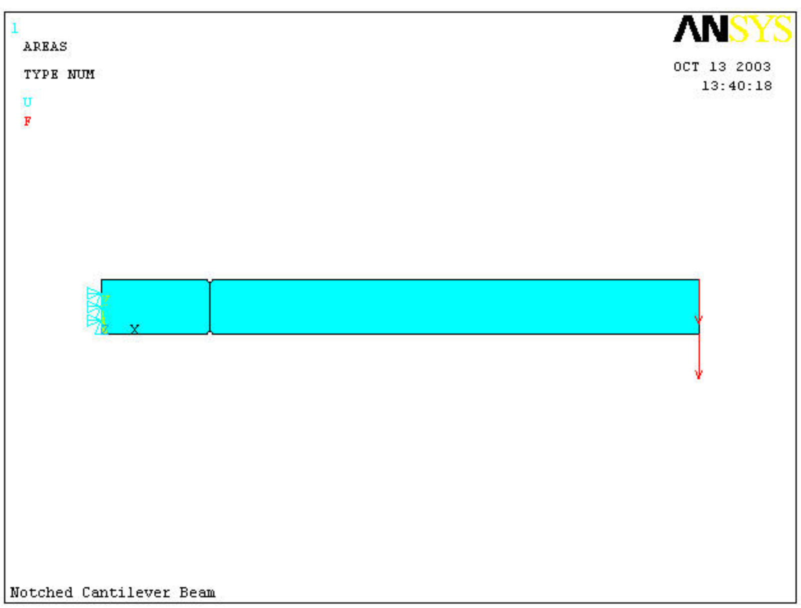

Figure 2. Notched Cantilever Beam Model

To highlight the discretization error, we will start with a very coarse mesh and then iterate to finer meshes. This has been done in two different scenarios to highlight possible error sources and the tools to find them.

Scenario One - User Controlled Mesh Parameters

In this scenario the number of elements around the notch and the density in the remaining model are completely controlled by the user. The mesh begins very course and then iterates to a more reasonable representations (see figures 3 and 4). The following table shows the values gathered from multiple runs of this problem.

Table 1: Solution Iterations for Scenario One

# Elements on Notch	Max Stress in X Dir (Averaged) (PSI)	Error to Previous Run	Max Stress in X Dir (Non-Averaged) (PSI)	Error Between Averaged and Non-Averaged	Criterion 1A	Criterion 1B first wave	Criterion 1B second wave
2	78611		78822	0.3%	9.26	11.08	13.45
4	86077	9.5%	86598	0.6%	6.3	8.36	12.97
6	96374	12.0%	96546	0.2%	4.8	7.19	14.2
8	104588	8.5%	104641	0.1%	4.1	3.67	13.81
10	112038	7.1%	112681	0.6%	3.35	2.9	13.41
16	124099	10.8%	124994	0.7%	2.38	3.8	8.35

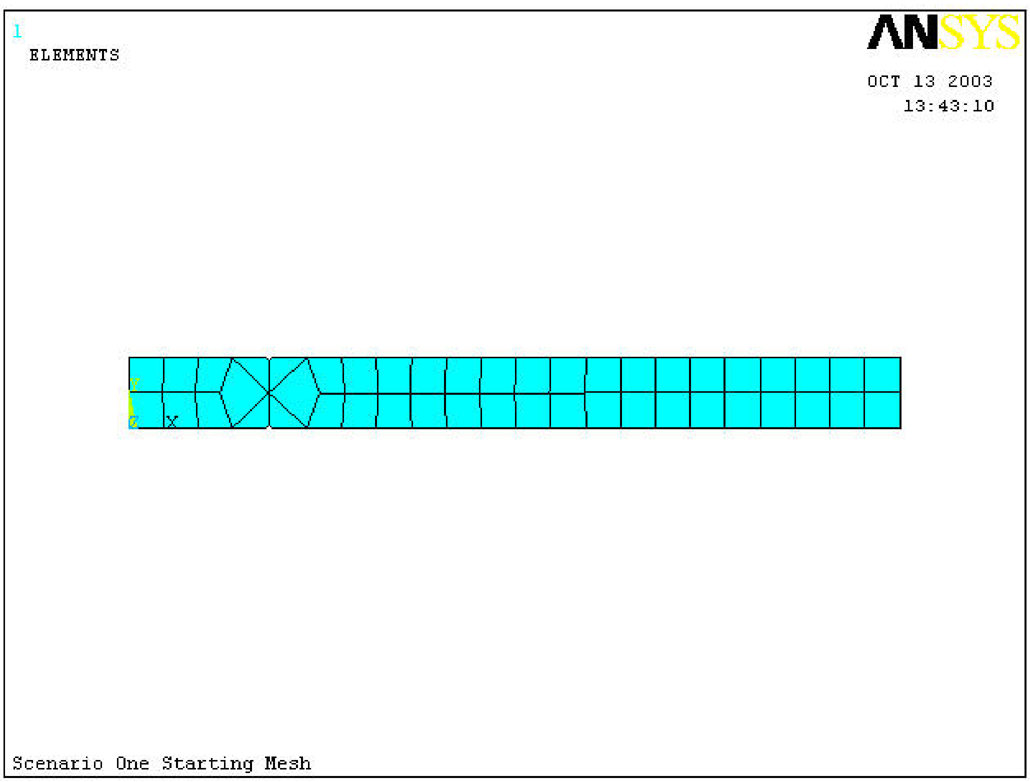

Figure 3. Scenario One Starting Mesh

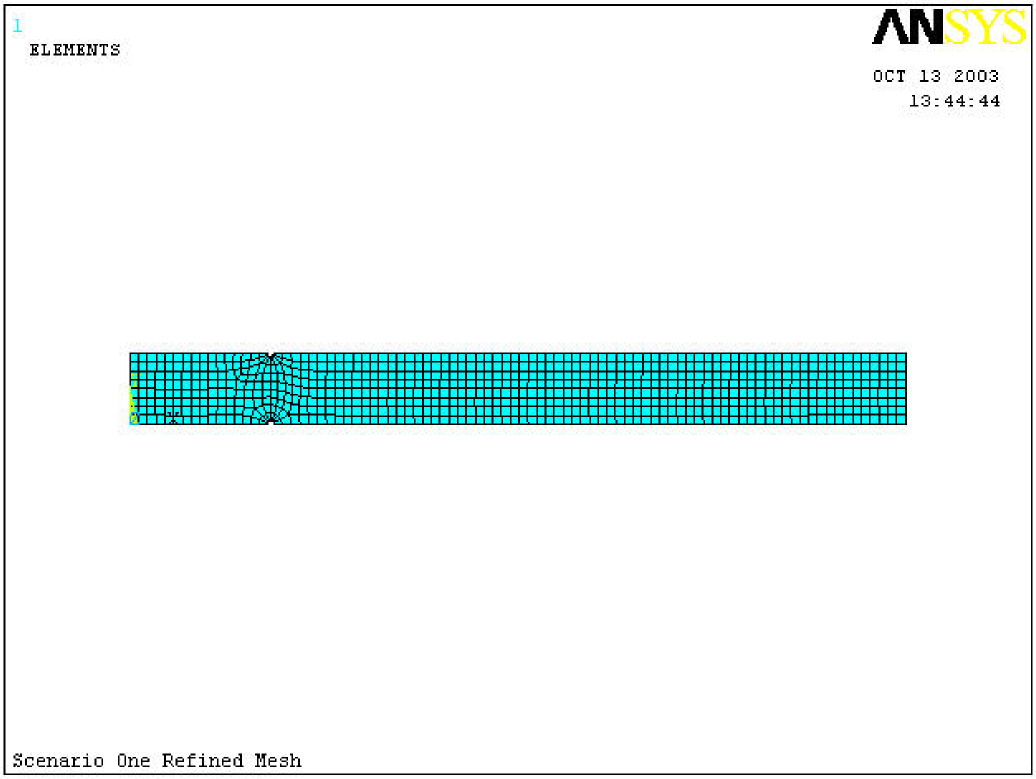

Figure 4. Scenario One Refined Mesh

Note that by the last iteration it would appear that all criteria have been met and the solution is correct. There is only a 0.7% error between the averaged and non-averaged stresses, they appear graphically equal and the criteria from 1A and 1B have been met. However, convergence must be verified graphically. Figure 5 shows a plot of the convergence from the table above. Note that the curve has not visually flattened out and that there is still a 21% error between the final average value of 124,099PSI and the previous hand calculations of 156,849PSI. Note that this method of meshing and iteration resulted in no mesh quality errors or warnings and met our main accuracy criteria, yet an error still exists in the model. This method of meshing was performed to illustrate a point. Often in a solid mesh there can exist very poorly shaped elements just below the surface although the number of elements along the surface appear adequate to ensure an accurate solution.

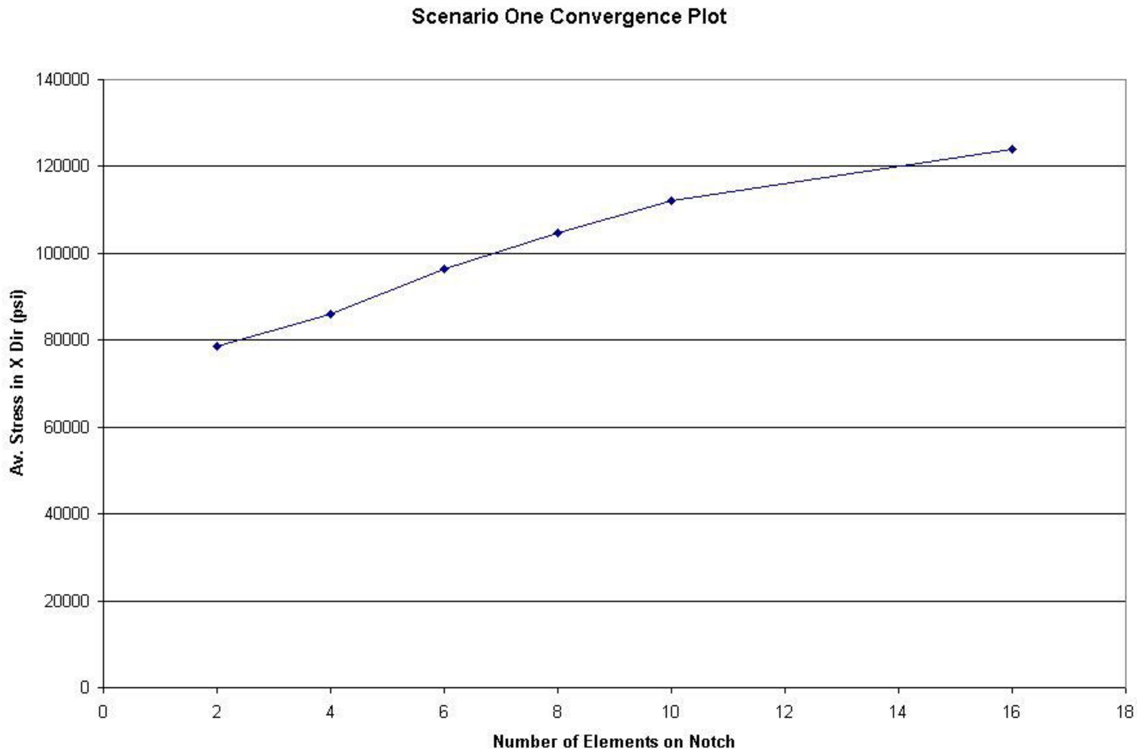

Figure 5. Scenario One Convergence Plot

Scenario Two - Meshing Controls and Smart Sizing

Next, the problem was meshed by specifying the number of elements desired around the notch and then utilizing smart sizing set to 6 to control the mesh in the remainder of the model. This resulted in a better quality mesh with aspect ratios closer to 1.0 and smoother transitions (see figure *Scenario Two Mesh* for an example of one iteration). The results of this scenario are shown below.

Table 2: Solution Iterations for Scenario Two

# Elements on Notch	Max Stress in X Dir (Averaged) (PSI)	Error to Previous Run	Max Stress in X Dir (Non-Averaged) (PSI)	Error Between Averaged and Non-Averaged	Criterion 1A	Criterion 1B first pass	Criterion 1B second pass
4	151406		151732	0.2%	6.51	21.42	15.55
6	151216	-0.1%	152596	0.9%	6.52	7.72	9.14
8	156084	3.2%	156715	0.4%	6.1	3.03	4.97
16	158531	1.6%	159513	0.6%	6.3	2.23	2.71

Note that in this example the final averaged stress is much closer to the predicted value and has less error to the last iteration. See Figure 6 for a comparison of the two convergence studies and how they compare to the hand calculations. Thankfully the auto-remesh and convergence capabilities offered by codes such as Ansys will help to keep some of these mesh quality related errors from occurring (9).

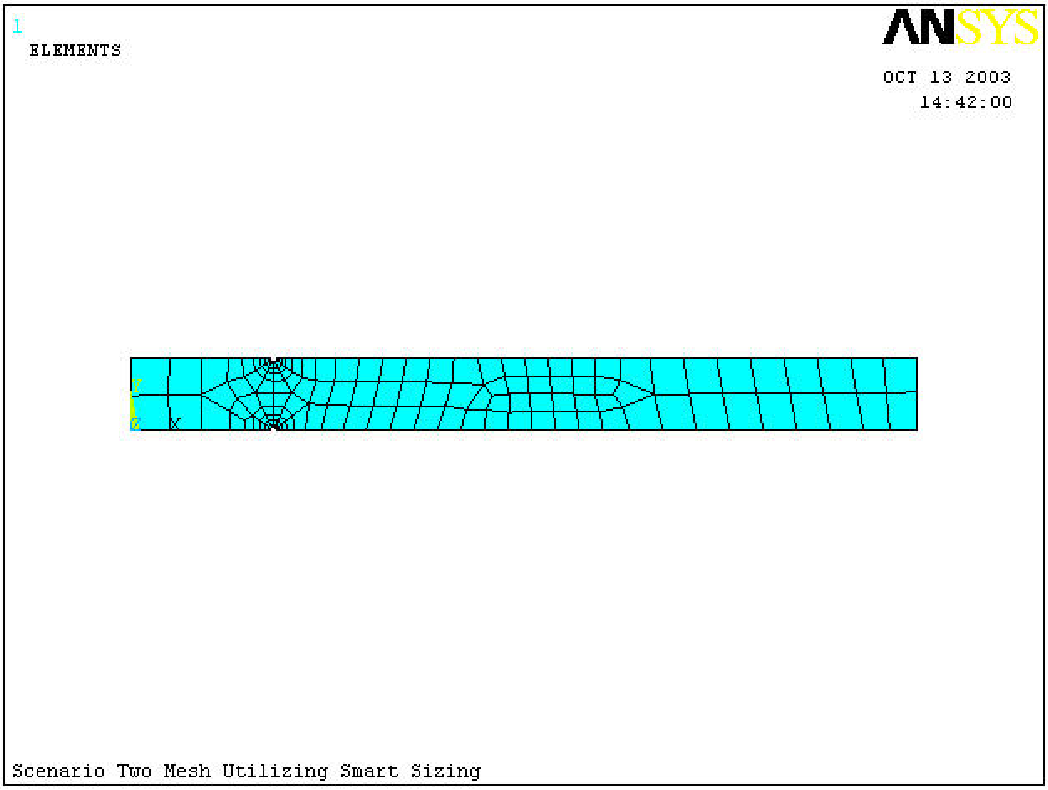

Figure 6. Scenario Two Mesh

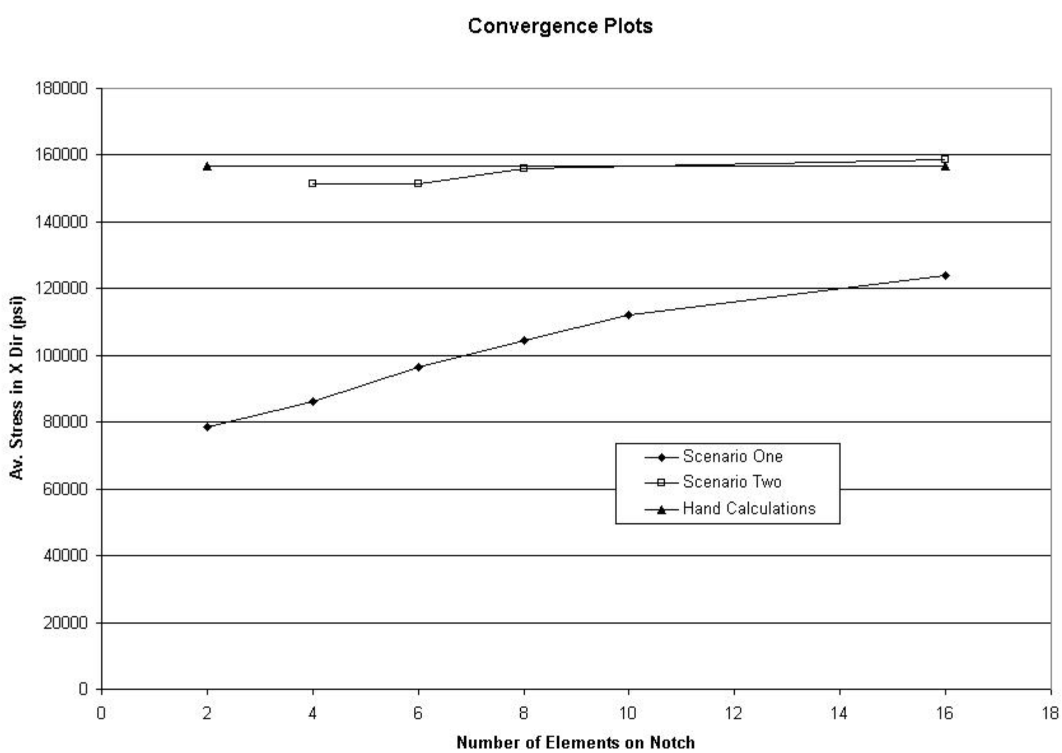

Figure 7. Convergence Comparison

Conclusions

Several important conclusions should be drawn from the two scenarios described above.

1. Without examining convergence a model can appear to contain adequate mesh density and have little error as shown in Scenario One. Here, if we were simply presented with the final solution we would note the error between the averaged and non-averaged stresses is low and that all of the accuracy criteria of Appendix A have been met. Additionally, the plots of averaged and non-averaged stresses show continuous and equal contours. Therefore, we would likely conclude that the solution is complete and contains minimal error. Only by plotting the convergence for multiple runs do we see that the model still contains a significant error caused by the poor quality of the mesh.

2. Convergence must be verified graphically. The 10.8% error comparing the last iteration of scenario one to the previous iteration would be judged acceptable by many codes utilizing default convergence criteria - even though the solution is still 21% off from the real value. Additionally, visual verification clears up questions regarding oscillation vs. convergence and true convergence vs. apparent convergence from too small of a mesh change. When utilizing automated convergence routines ensure that the number of elements in each refinement is understood.

3. Note that in Scenario Two the value for criterion 1A stays constant with each iteration while the other criteria drop and show convergence. Criterion 1A is met, but why does it stay constant and not decrease with increased mesh density? Remember that this value represents the percentage error in the energy norm of the entire model. Therefore, in this case there is a steady error trapped somewhere else in the model even though we are refining locally (as described in the previous convergence discussion). To illustrate this point, scenario two was further refined in the areas other than the notch and criterion 1A dropped significantly.

4. Criterion 1A can be an important indicator about the health of the overall model - other than the small, local area of interest. The author has experienced 3D models where there is graphical convergence, averaged and non-averaged results agree, criterion 1B is met, and yet criterion 1A is not satisfied. In some cases this has been an indication of another, yet unnoticed stress riser in the model that is not adequately captured in the mesh and is therefore causing a large gradient and error. This can even mask a stress that may exceed the initial stress of interest. Once corrected and refined in the newly discovered region, criterion 1A drops and both stress areas are captured correctly.

Given these points, it is important that all of the criteria listed in the Verifying Mesh Accuracy in an FEA Model section be demonstrated before judging a solutions complete. Otherwise, it can be shown that not meeting any one of the criteria can result in the presentation of a solution that is "effectively worthless" (Burnett, 1987, pg. 55).

Additionally, given point #3 of Burnett and the established correctness of scenario two above (given that it met all three of our tests in Verifying Mesh Accuracy in an FEA Model) the error in the solution can be approximated by the error between the last two iterations. The table shows this error to be 1.6%. Therefore, this would say that the real solution is within 1.6% of the final value of 158531PSI. The hand calculations showed a true value of 156849.3PSI. Therefore, 158531PSI is 1.1% off from the true value of 156849.3PSI and the error is less than 1.6% as predicted.

References

1. Shah, Chandresh, (2002). *Mesh Discretization Error and Criteria for Accuracy of Finite Element Solutions.* Paper presented at the Ansys Users Conference, Pittsburgh, PA

2. Zienkiewicz, O. C., and Zhu, J. Z., *A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis*, International Journal for Numerical Methods in Engineering, Vol. 24, pp. 337-357 (1987).

3. Burnett, David S., (1987). *Finite Element Analysis.* Reading: Addison-Wesley Publishing Company.

4. ANSYS 7.1 Theory Reference

5. UofA Ansys Tutorial. Retrieved September 25, 2003 from http://www.mece.ualberta.ca/tutorials/ansys/AU/Converge/Converge.html. University of Alberta (2001)

6. Rizzo, Tony. *Safety Tips and Techniques for FEA in Modeling Solids.* Retrieved September 25, 2003 from http://www.electronics-cooling.com/Resources/EC_Articles/JUN95/jun95_02.htm. Lucent Technologies.

7. Brundrett, Laurance. (2003) *Plate Stress Mesh Refinement*. Retrieved on September 25, 2003 from http://www.pveng.com/FEA/Plate_Stress_Mesh_Refinement.xls

8. Shigley, Joseph E., Mitchell, Larry D., (1983). *Mechanical Engineering Design Fourth Edition.* New York: McGraw-Hill Book Company.

9. Tristano, Chen, Hancq, Kwok. *Fully Automatic Adaptive Mesh Refinement Integrated Into the Solution Process.* Ansys Incorporated, Canonsburg, PA.

Appendix A - Accuracy Criteria from Shah, 2002

Finite Element Accuracy Criteria

Author's Note - The following criteria section is copied directly from Shah, 2002.

Criterion # 1A: The error norm of the entire finite element model must be less than 15%

Ansys Command	Resulting Action
allsel	! Select the entire model
prerr	! Print error norm - should meet criterion # 1A *(power graphics must be off)*

The intent of this criterion is to ensure that the mesh density used in the model adequately represents the global stiffness and displacements of the component (although the peak stresses may not be accurately captured). With the current hardware and software tools, this criterion should be easily met. It should also be noted that nodes and elements that have point loads and boundary conditions (causing stress singularities) must be removed prior to executing the "prerr" command.

Criterion # 1B: The error norm in the local area of high stress must be less than 10%

Ansys Command	Resulting Action
nsel,s,node,,nn	! Select the node with the highest stress of interest (*or select graphically*)
esln	! Select all the elements attached to this node
prerr	! Print error norm - should meet criterion # 1B *(power graphics must be off)*
nsle	! Select nodes attached to the currently selected set of elements
esln	! Select the second wave of elements
prerr	! Print error norm - should meet criterion # 1B *(power graphics must be off)*

This criterion addresses the quality of mesh in the local area of high stress. If multiple regions of high stress exist in the model, this criterion should be applied to each of those areas.