Mesh Discretization Error and Criteria for Accuracy of Finite Element Solutions

Chandresh Shah
Cummins, Inc.

Abstract

Any finite element analysis performed by an engineer is subject to several types of errors that can compromise the validity of the results. These errors can be broadly classified under the following categories: 1) user error - incorrect usage of FE software or input by the FE analyst, 2) errors due to assumptions and simplifications used in the model and 3) errors due to insufficient mesh discretization. User errors can be prevented by developing and utilizing a comprehensive pre and post processing checklist and by appropriate training in the basics of finite element analysis and usage of FE software. Errors due to modeling assumptions and simplifications can be alleviated by adding complexity to the model so that it better represents the physics of the problem being analyzed. Errors due to the inadequacy or coarseness of the mesh are often overlooked by the analyst. These errors due to mesh discretization can be fixed by evaluating the quality of the mesh and by developing and utilizing criteria that characterize the accuracy of the FE solution. This paper describes the source of mesh discretization error and presents several criteria that can be used by an FE analyst to evaluate the accuracy of the FE solution.

Introduction

Any finite element analysis performed by an engineer is subject to several types of errors that can compromise the validity of the FE solution. The errors can be broadly divided into three categories. The simplest type of error is user error. This can be as simple as a typo in say, a material property or load specification or it could be incorrect usage of the FE program by the analyst. User errors can be prevented by developing and utilizing a comprehensive pre and post processing checklist that verifies that all input quantities to an FE model match the intention of the analyst. Improper usage of the FE program can be prevented by obtaining a basic understanding of FE theory and appropriate training in the usage of the FE software being used.

The second source of error in the FE solution is the error introduced due to the assumptions and simplifications made in the analysis. A finite element model is a mathematical representation of a real life component or system that is being analyzed. A completely accurate representation of the physical model may lead to an extremely complex mathematical model that may be hard to solve with the available hardware and software resources. Simplifications in geometry have to be made to keep model sizes and run times manageable. Loads are often not accurately known and are obtained from computer simulations that may themselves be approximate in nature. Boundary conditions may need to be approximated as well to keep the modeling effort and model size simple. Finally, material properties used in the analysis may not be accurately known. If these assumptions and simplifications are excessive, the results obtained from the finite element model may be inaccurate. Increasing the complexity of the FE model such that it better represents the physical model being analyzed can alleviate these types of errors.

The third source of error is due to insufficient mesh discretization of the finite element model. If the overall mesh is too coarse, the model will not capture the stiffness of the component. If the mesh in the areas of high stress is too coarse, the gradients in stress will not be accurately predicted and the resulting stresses and strains will not be accurate. This type of error is often overlooked by the analyst and is the primary subject of this paper. The errors due to mesh discretization can be fixed by evaluating the quality and adequacy of the mesh and by developing and utilizing criteria that characterize the accuracy of the FE solution.

Mesh Discretization Error

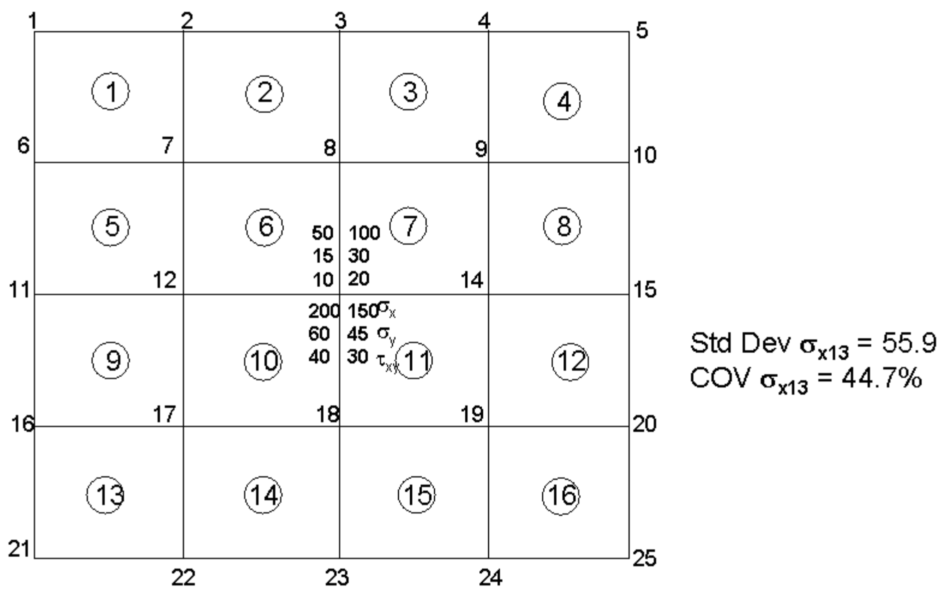
The primary unknown in conventional finite element analysis is the displacement at every node in the model. The finite elements used in the model have shape functions associated with them that characterize the resulting displacement field of the physical model. The principal of minimization of total potential energy is used to obtain the equation in the form of: [K]{x}={F}, where

[K] = global stiffness matrix

{x} = displacement vector

{F} = force vector

This equation is solved for the unknown displacement vector {x}. The stresses and strains are then obtained as the first derivative of these displacements. So while the displacement field in the finite element model is continuous, the stress field in the model is discontinuous. The stress at a node, as printed from ANSYS, is the average of the stresses from all the elements attached to that node. This introduces an error in the magnitude of stress at a node and is referred to as mesh discretization error. The phenomenon of mesh discretization error is graphically illustrated in figure 1. The coarser the mesh, the greater the potential for this error.



σ_{x13} = (50+100+150+200)/4 = 125; σ_{y13}=37.5; τ_{xy13}=25

$\{\Delta\sigma^6_{13}\}^T = \{125\ \ 37.5\ \ 25\}^T - \{50\ \ 15\ \ 10\}^T = \{75\ \ 22.5\ \ 15\}^T$

Figure 1 - Mesh Discretization Error

Basic FE theory also suggests that for elements that pass the patch test, as you keep refining the mesh, in the limit, you approach the exact state of stress. Hence, for a very fine mesh, the contribution to stress at a node from all elements attached to it will be the same, exact value. However, rarely do we have the luxury of repeated mesh refinement - hence we cannot see the convergence of stress to the exact value with increasing mesh density. If the mesh is too coarse and the stress gradient too high, the stress result may not be accurate. It is for this reason that the analyst should look at mesh discretization error and estimate the error in FE solution.

ANSYS Error Estimation Tools

ANSYS software provides the analyst with several tools to evaluate the mesh discretization error and assess the accuracy of the FE solution. Directly available from ANSYS are the following quantities:

1) SERR : error energy in each element (obtained from PRESOL or PLESOL commands)
2) SDSG : absolute value of the maximum variation of any nodal stress component in each element (obtained from PRESOL or PLESOL commands)
3) SEPC : percentage error in the energy norm for the selected set of elements (obtained from the PRERR command)
4) SMXB and SMNB : maximum and minimum stress bounds (obtained from a stress plot, PLNSOL command)

The explanation of each of these quantities is described in detail in the ANSYS manuals (ref. 1). Figures 2, 3 and 4 are excerpts from the ANSYS manual that illustrate and define each of these quantities. Appropriate usage and manipulation of these error estimation tools can be done to develop quantitative criteria to evaluate the accuracy of FE solution.

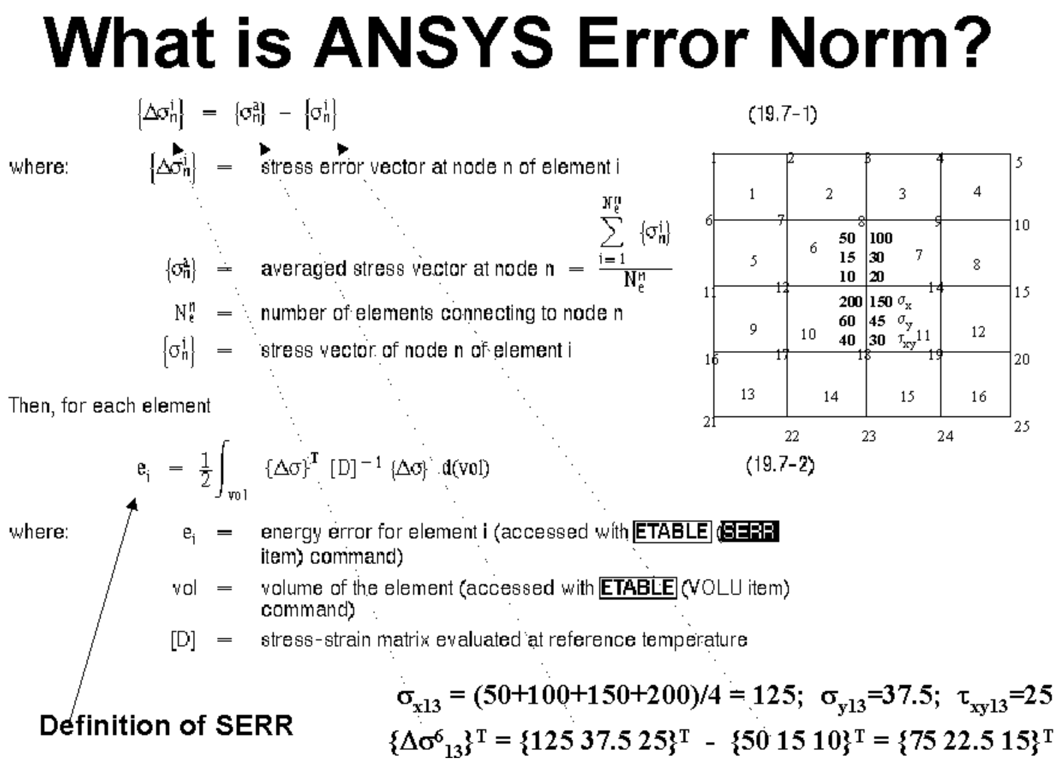


Figure 2 - ANSYS Error Estimation Tools - Definition of SERR

$\{\Delta\sigma\}$ = stress error vector at points as needed (evaluated from all $\{\Delta\sigma_n\}$ of this element)

The energy error over the model is:

$$e = \sum_{i=1}^{N_r} e_i \qquad (19.7\text{-}3)$$

where:

e = energy error over the entire (or part of the) model (accessed with ***GET** (SERSM item) command)

N_r = number of elements in model or part of model

The energy error can be normalized against the strain energy.

$$\boxed{E = 100\left(\frac{e}{U + e}\right)^{\frac{1}{2}}} \quad \textbf{Definition of SEPC} \qquad (19.7\text{-}4)$$

where:

E = percentage error in energy norm (accessed with **PRERR**, **PLDISP**, **PLNSOL** (U item), ***GET** (SEPC item) commands)

U = strain energy over the entire (or part of the) model (accessed with ***GET** (SENSM item) command)

$= \sum_{i=1}^{N_r} E_{ei}^{po}$

E_{ei}^{po} = strain energy of element i (accessed with **ETABLE** (SENE item) command) (see Section 15.16)

Figure 3 - ANSYS Error Estimation Tools - Definition of SEPC

Definition of SMXB and SMNB:

$$\sigma_j^{mnb} = \min\left(\sigma_{j,n}^a - \Delta\sigma_n\right) \qquad (19.7\text{-}5)$$

$$\sigma_j^{mxb} = \max\left(\sigma_{j,n}^a + \Delta\sigma_n\right) \qquad (19.7\text{-}6)$$

where min and max are over the selected nodes, and

where:

σ_j^{mnb} = output quantity VALUE (printout) or SMNB (plot) for nodal minimum of stress quantity

σ_j^{mxb} = output quantity VALUE (printout) or SMXB (plot) for nodal maximum of stress quantity

j = subscript to refer to either a particular stress component or a particular combined stress

$\sigma_{j,n}^a$ = averaged stress quantity j at node n

$\Delta\sigma_n$ = root mean square of all $\Delta\sigma_i$ from elements connecting to node n

$\Delta\sigma_i$ = maximum absolute value of any component of $\{\Delta\sigma_n^i\}$ for all nodes connecting to element (accessed with **ETABLE** (SDSG item) command)

Definition of SDSG

Figure 4 ANSYS Error Estimation Tools - Definition of SDSG

Figure 4 - ANSYS Error Estimation Tools - Definition of SDSG

Finite Element Accuracy Criteria

The following set of three quantitative criteria may be used as a starting point for mesh discretization error analysis. These criteria address the global versus local mesh discretization errors and also take into account the fundamentals of FE theory.

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

```
allsel    ! Select the entire model

prerr     ! Print error norm - should meet criterion # 1A
```

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%

```
nsel,s,node,,nn    ! Select the node with the highest stress of interest

esln               ! Select all the elements attached to this node

prerr              ! Print error norm - should meet criterion # 1B

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
```

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.

Criterion # 1C: In the local area of high stress, the averaging of stresses from the elements attached to a node must have a coefficient of variation of the dominant stress less than 7%

```
nsel,s,node,,nn    ! Select the node with the highest stress of interest

esln               ! Select all the elements attached to this node

nsle               ! Select nodes attached to the currently selected set of elements

esln               ! Select the second wave of elements

presol,s,comp      ! Print elemental stresses at each selected node
```

As explained in figure 1, the stress at any node is the average of stresses from all the elements attached to that node. The standard deviation of these stresses from elements attached to the node is also calculated. The coefficient of variation is then calculated by dividing the standard deviation by the nodal (mean) stress. This COV must be less than 7% to meet criterion # 1C. An external program (or an ANSYS macro), is used to calculate the mean and standard deviation of the dominant stress component in the currently selected set of nodes. This criterion addresses the fundamental principles of finite element analysis. As you keep refining the mesh, in the limit, the contribution of stress from all the elements attached to a node should approach the same value - hence the coefficient of variation of the stress should approach zero.

These three set of criteria adequately address the errors associated with mesh discretization at the local and global level and also address the fundamentals of finite element theory. These criteria can be used to evaluate the accuracy of the FE solution and can guide the analyst towards mesh refinement if further accuracy is desired.

There are several other criteria that can be developed from some of the other ANSYS error estimation tools. Two additional criteria are listed below.

Criterion # 2: The difference between the dominant stress component and its' bound (as calculated by ANSYS) in the local area of high stress must be less than 7%

nsel,s,node,,nn	! Select the node with the highest stress of interest
esln	! Select all the elements attached to this node
calculate % error	! % Error = (SMXB - SMX)/SMX
nsle	! Select nodes attached to the currently selected set of elements
esln	! Select the second wave of elements
calculate % error	! % Error = (SMXB - SMX)/SMX

% Error must be less than 7% to meet this criterion.

Criterion # 3A: The RMS value of the ratio SDSG/S_{eqv} in the entire model must be less than 15%

This criterion uses the ratio of the absolute value of the maximum variation of any nodal stress component in an element (SDSG) to the von Mises stress in that element. This ratio is calculated for every element in the model and an RMS value is calculated from that. The RMS value must be less than 15%. This criterion is similar to Criterion # 1A and attempts to address the quality of the global mesh density.

Criterion # 3B: The ratio of SDSG/S_{eqv} in the local area of high stress must be less than 10%

This criterion addresses the quality of the mesh in local areas of high stress and must be applied to all such areas in the model.

It should be noted here that the quantitative numbers used here for each of the criteria are based on personal experience. These values should be modified by analysts to suit their own set of problems and needs. The values specified in this report may be used as a starting point to develop a knowledge base for individual set of problems and modified to better reflect the field that the analyst is working in. These criteria can be very useful in comparing analysis performed by different analysts on similar components. Stresses between the two models can be compared if application of these criteria give "similar" results in both models. These criteria can be easily implemented by developing ANSYS macros that use APDL. Such macros can produce tabular and graphical output that is easy to understand and visualize.

Example Problem

The finite element analysis of a diesel engine connecting rod is used to illustrate the various criteria described in the section above. Figure 5 shows the finite element model of a connecting rod with three different mesh densities. The area of high stress is at the intersection of the oil rifle with the small end of the rod. The intent of the analysis is to accurately predict the magnitude of the principal stress at the drill intersection. Figure 6 shows the plot of maximum principal stress for each of the three meshes. Figure 7 lists the output of a macro developed to apply criterion 1. The results for each of the three criteria are tabulated in table 1 below.

Criterion	2.5 mm Mesh	2 mm Mesh	2.5 mm+ Mesh
Max Principal Stress	364 Mpa	381 Mpa	385 Mpa
Criterion # 1A	10.8 %	10.7 %	10.6 %
Criterion # 1B	20 %; 24.4 %	17.6 %; 21.2 %	2.9 %; 12.3 %
Criterion # 1C	12.9 % (9.2% - 28.8%)	9.1 % (8.7% - 23.1%)	1.4 % (0.8% - 7.7%)
Criterion # 2	21.2 %; 28.3 %	17.6 %; 21.3 %	2.9 %; 16.4 %
Criterion # 3A	13.5 %	11.9 %	11.8 %
Criterion # 3B	11 % - 53.5 %	14.6 % - 38.9 %	1.6 % - 8.8 %

Table 1 - Finite Element Solution Accuracy Criteria Results

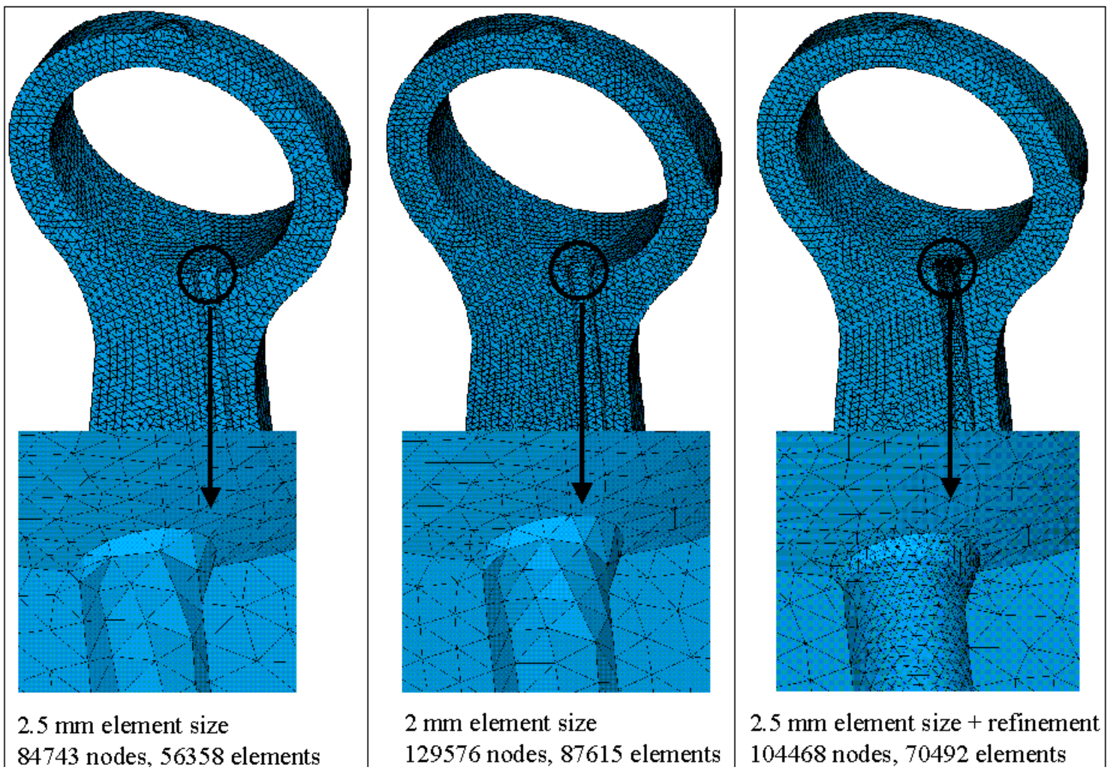


Figure 5 - Finite Element Model of a Connecting Rod

Maximum Principal Stress under Assembly + Firing Conditions

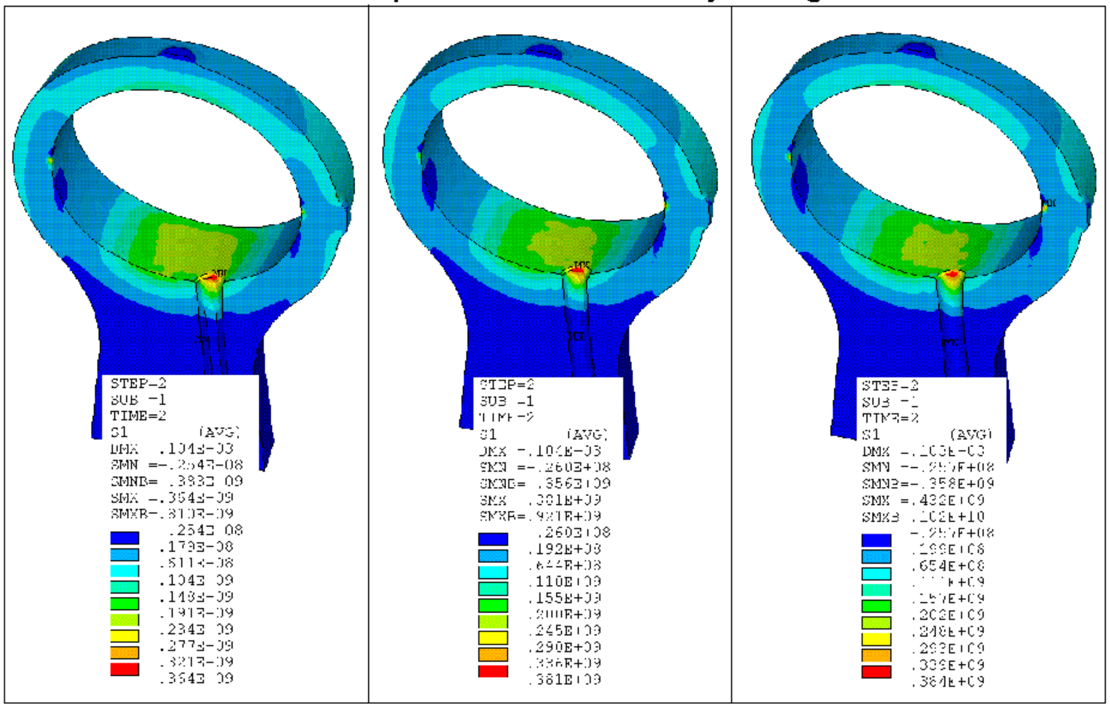


Figure 6 - Maximum Principal Stress in the Connecting Rod

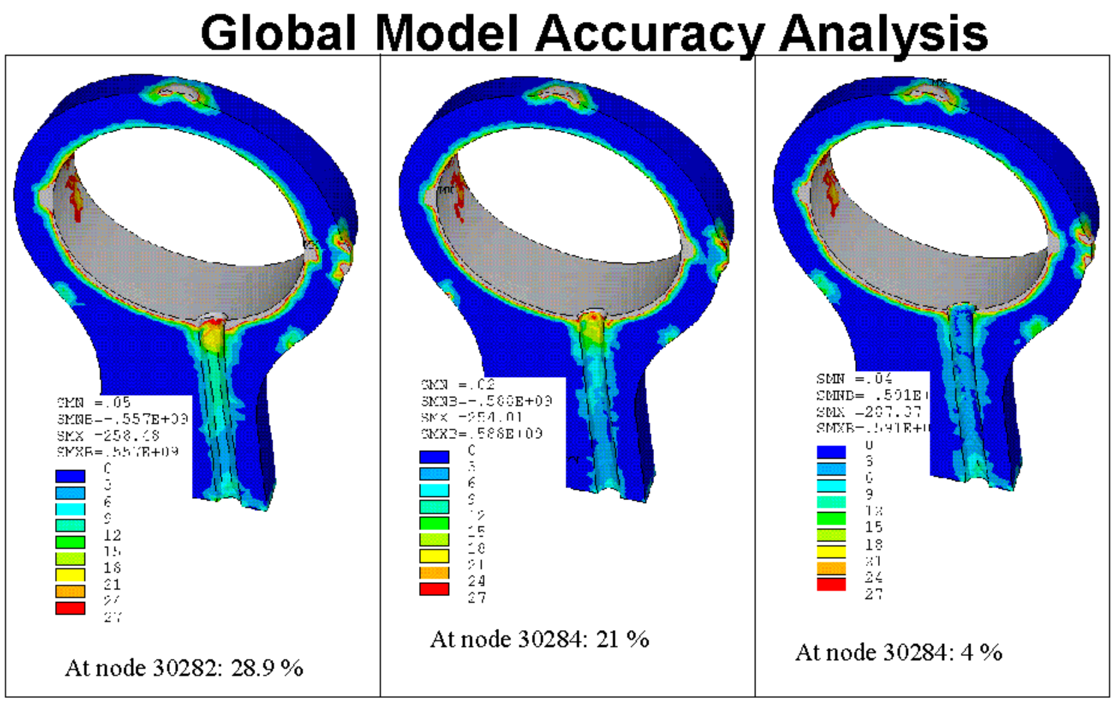
Accuracy of FE - Approach # 1

```
Criterion # 1A : SEPC for the Global Model = 10.7838130.
Criterion # 1B : SEPC in the Local Area of Interest = 20.0100369.
Criterion # 1B : SEPC for the 2nd Wave Around the Node = 24.3659972.
Criterion # 1C : COV of Elemental Stress Values.


Node No.= 1133; No.of Occurrences = 8.
Mean Stress, Std Dev and COV(in %) for SXX, SYY, SZZ, SXY, SYZ, SXZ.
     0.2417E+09 -0.1466E+09 -0.1522E+08 -0.1217E+09 -0.4078E+08 -0.1754E+08
     0.4262E+08  0.7319E+08  0.2792E+08  0.6670E+08  0.3455E+08  0.4988E+07
     0.1763E+02 -0.4992E+02 -0.1834E+03 -0.5481E+02 -0.8473E+02 -0.2844E+02

Node No.= 1162; No.of Occurrences = 8.
Mean Stress, Std Dev and COV(in %) for SXX, SYY, SZZ, SXY, SYZ, SXZ.
     0.2107E+09 -0.1959E+09  0.2366E+08 -0.1168E+09 -0.1862E+08 -0.8567E+08
     0.5316E+08  0.1055E+09  0.3386E+08  0.4174E+08  0.2704E+08  0.1731E+08
     0.2523E+02 -0.5387E+02  0.1431E+03 -0.3575E+02 -0.1452E+03 -0.2021E+02

Node No.= 30280; No.of Occurrences = 10.
Mean Stress, Std Dev and COV(in %) for SXX, SYY, SZZ, SXY, SYZ, SXZ.
     0.1454E+09 -0.2578E+08  0.1365E+09 -0.4409E+08 -0.8819E+07 -0.1407E+09
     0.4180E+08  0.4466E+08  0.2977E+08  0.2196E+08  0.1264E+08  0.1190E+08
     0.2876E+02 -0.1732E+03  0.2181E+02 -0.4981E+02 -0.1433E+03 -0.8454E+01

Node No.= 30282; No.of Occurrences = 13.
Mean Stress, Std Dev and COV(in %) for SXX, SYY, SZZ, SXY, SYZ, SXZ.
     0.3413E+09 -0.1344E+08  0.1358E+08 -0.9053E+08 -0.1268E+08 -0.2174E+08
     0.4396E+08  0.1853E+08  0.1934E+08  0.1008E+08  0.9783E+07  0.4063E+08
     0.1288E+02 -0.1379E+03  0.1424E+03 -0.1114E+02 -0.7715E+02 -0.1869E+03
```

ETC……

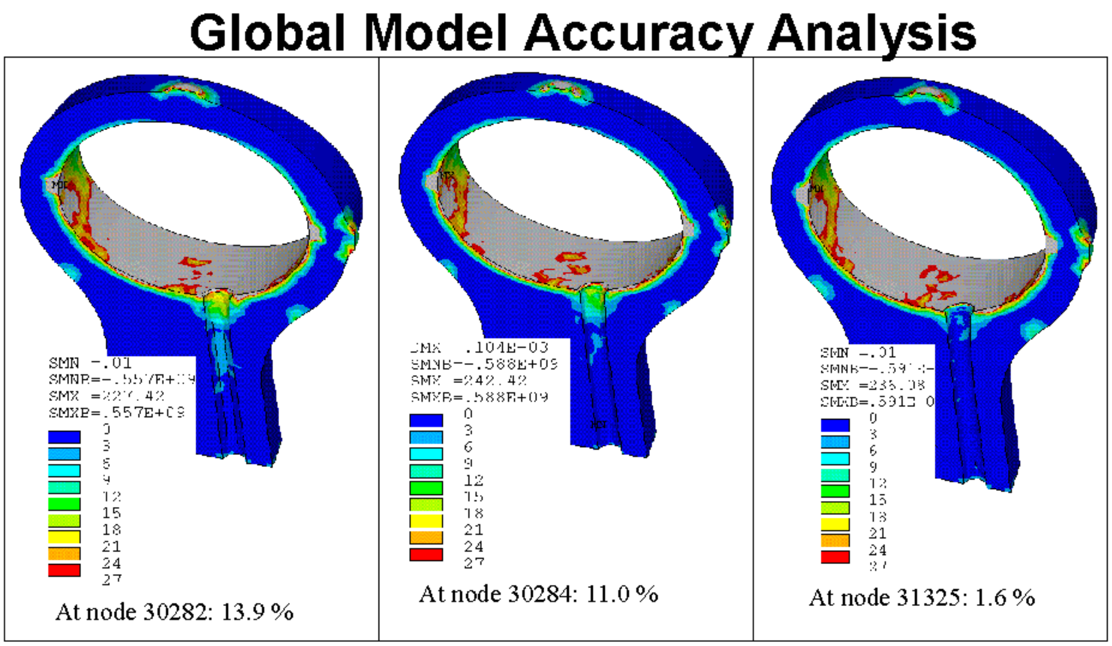
Figure 7 - Output from an ANSYS Macro to Implement Criterion # 1

For a three-dimensional model, every node will have six component stresses. Applying criterion #1C to each node in the model will result in six coefficient of variation values corresponding to each of the six components of stress. The six component stresses and their corresponding coefficient of variations may be combined in a linear or quadratic fashion to obtain a single coefficient of variation number for that node. Figures 8 and 9 show this calculation in a graphical format. These plots show areas of poor mesh resolution where an increase in mesh density would improve the solution. Figure 10 is yet another way of presenting the raw stress versus coefficient of variation data for every stress component at every node in the finite element model. The intention of this plot is ensure visually that the high tensile and high compressive stresses at the tail ends of the graph have low coefficient of variation values.



$$\text{Ave. Linear COV} = \frac{|\sigma_x|*COV_{\sigma x} + |\sigma_y|*COV_{\sigma y} + \ldots\ldots}{|\sigma_x| + |\sigma_y| + \ldots\ldots}$$

Figure 8 - FE Model Accuracy Analysis - Linear COV



$$\text{Ave. Square COV} = \frac{\sigma_x^{2} * COV_{\sigma x} + \sigma_y^{2} * COV_{\sigma y} + \ldots\ldots}{\sigma_x^{2} + \sigma_y^{2} + \ldots\ldots}$$

Figure 9 - FE Model Accuracy Analysis - Square COV

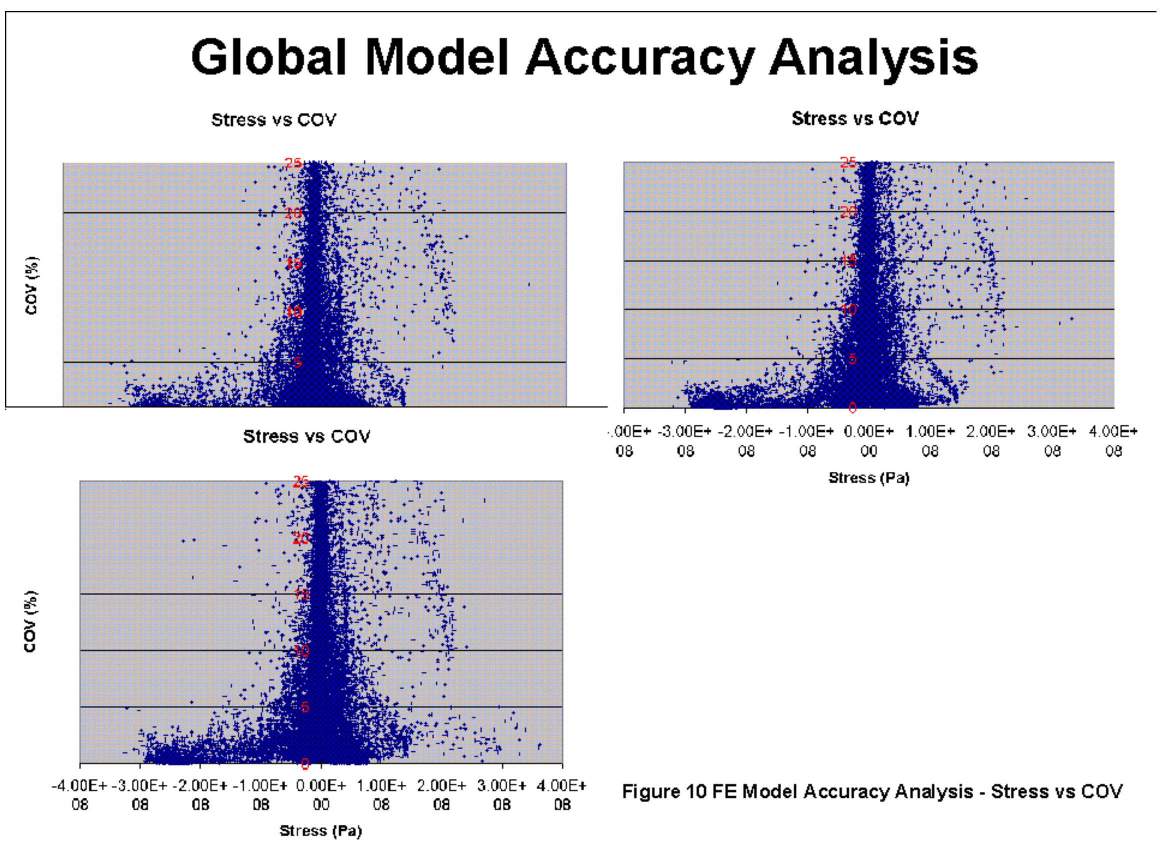


Figure 10 - FE Model Accuracy Analysis - Stress vs COV

Figures 11 and 12 show similar plots for criterion 3.

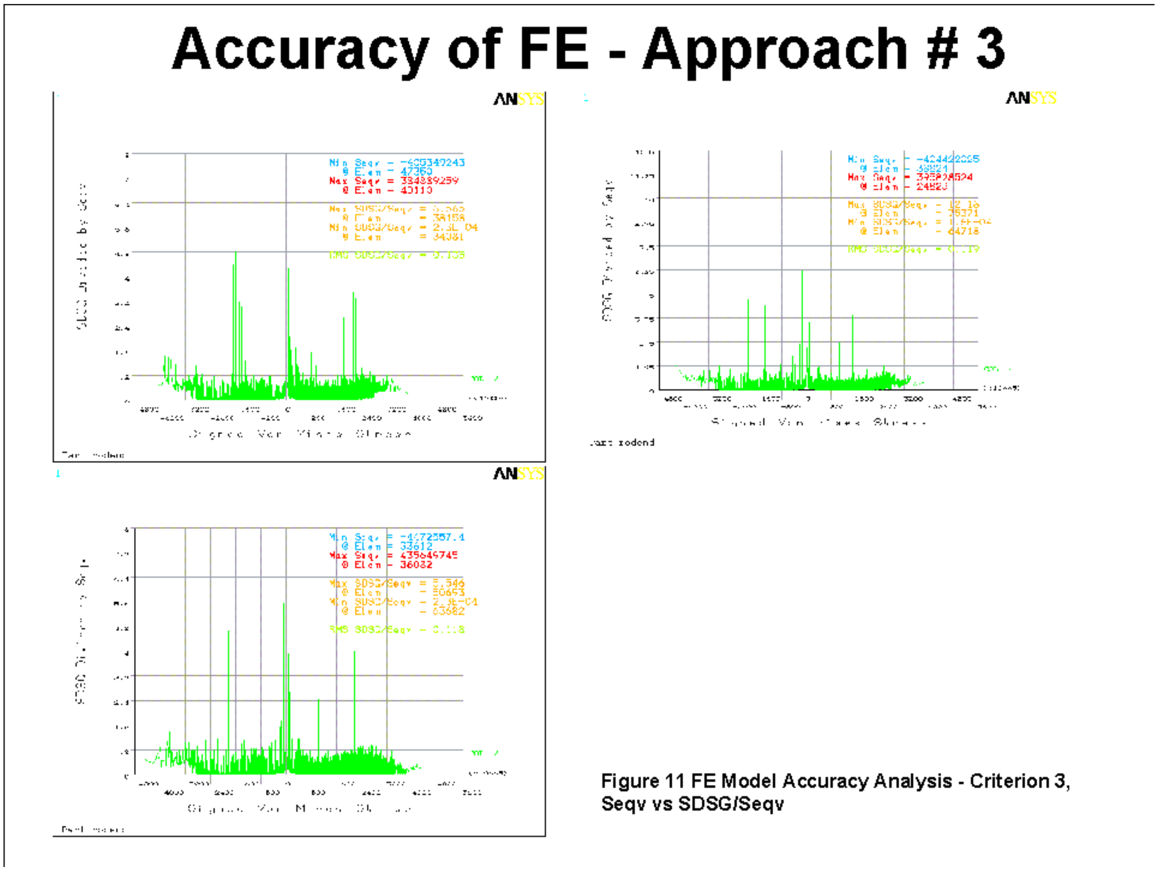


Figure 11 - FE Model Accuracy Analysis - Criterion 3, Seqv vs SDSG/Seqv

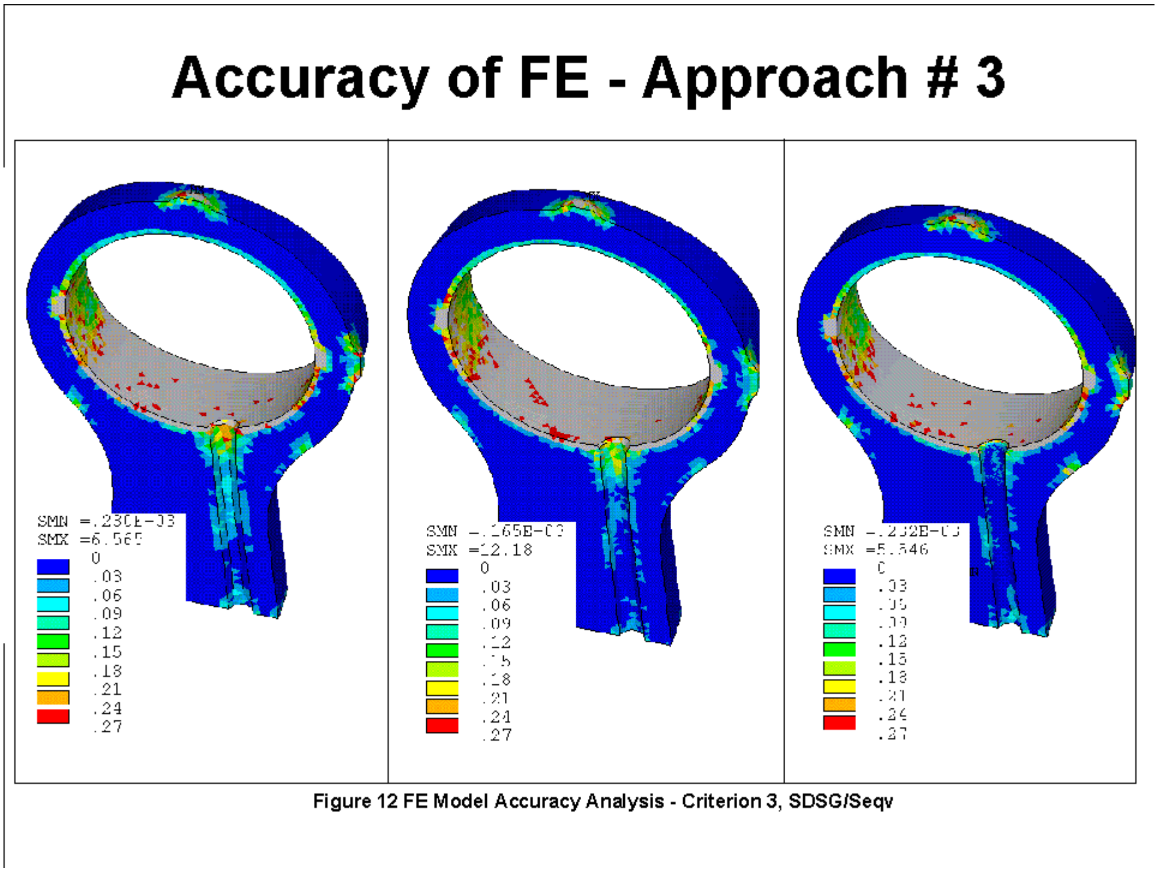


Figure 12 - FE Model Accuracy Analysis - Criterion 3, SDSG/Seqv

Conclusion

The three major sources of finite element solution errors have been described in this paper with a special emphasis on the error due to mesh discretization. A basic explanation of the mesh discretization error has been provided and ANSYS error estimation tools to evaluate the accuracy of the FE solution have been discussed. Three different sets of finite element solution accuracy criteria are then discussed in detail. These criteria are then implemented on a sample problem to show the efficacy of each criterion. Various graphical and tabular output formats are developed to easily understand the accuracy of FE solution.

References

1. ANSYS 5.7 Theory Reference