Avoiding pitfalls in FEA

The uninitiated FEA user is likely to get wrong answers from blindly applying the analysis. Here’s a laundry list of what could go wrong and how to make it go right.

### How elements add accuracy

- **P**
  - 27-element model
  - 75-element model

**Never assume**

- A structure’s behavior can be predicted by dividing it into small regions, or finite elements, and summing their individual responses.

This misconception implies that if you divide a structure into a finite number of elements and sum their responses, your results will be as good as without discretization. In fact, representing a structure with finite elements lets the software find an approximate description of structural behavior with limited information, not because elements have simple shapes and analyze easily. When users prepare the data set required by a FE program, they create a discrete mathematical model of a continuous structure and arbitrarily impose a stress pattern in each element. It is a major challenge for any FEA user to model a continuous structure with a continuous-stress distribution by using finite elements in which the pattern of stress distribution is arbitrarily assumed.

**First-order elements produce constant stress in each, represented in the image by the height of each bar. The stair steps in the shaded images after analysis shows that more elements produce a lesser height difference between adjacent elements. Also, higher stresses appear in the 75-element model, an indicator of better accuracy.**

The illustration *How elements add accuracy* shows a quarter of a hollow plate modeled with a different number of first-order quadrilateral membrane elements. Symmetry boundary conditions simulate the presence of the remaining three quarters of the plate loaded horizontally in tension.

First-order elements used in this example impose uniform stress within each element. The illustration’s shaded images use bar height as a stress indicator. The figures show that a finer mesh yields higher stress values and probably better represents a real continuous stress distribution. Novice users may ask: Is a finer mesh really better, is it

**PAUL KUROWSKI**

President
ACOM Engineering
London, Ontario, Canada
good enough? And, how do we know what mesh is good enough? These questions lead to the next problem: error analysis.

- **Computerized finite-element analysis is sufficiently accurate to make error analysis unnecessary.**

Unfortunately, error is inherent in FEA methods. It originates from arbitrary assumptions required to represent the stress distribution within elements. The magnitude of error depends on how well the discretized finite-element model approximates the real stress distribution. Convergence analysis, an indispensable part of any FEA project, estimates error. Few good engineers accept quantitative results without an error estimation. Yet FEA results are commonly produced without error estimates, despite errors occasionally exceeding 100%.

To illustrate error-analysis principles, the table *Examining convergence error* shows results from different models of the same structure. Element size is decreased by about half with every model. Without several runs, the error in the initial mesh is unknown.

An alternative to refining mesh for convergence analysis is automated convergence analysis offered by p-code programs such as Rasna's Mechanica. In these, element sizes remain the same but element orders change while an algorithm compares results from the two last iterations and upgrades element order where required to meet user-specified accuracy. Stress-distribution patterns within p-code elements are calculated, not arbitrarily assumed. The pattern may be different along different edges of the same element and can change according to the shape function obtained during calculations. Iterations continue without user intervention until the convergence error becomes lower than a user-defined value or until calculations hit the highest possible element order. This process is called p-convergence.

The illustration, *A plate of p elements*, shows that the model needs only four large elements because their order can be upgraded as required to reach the user-specified convergence error. Accompanying graphs also detail p-convergence. One shows a monotonic increase in global strain energy with each iteration. The other shows changes in maximum Von Mises stress. The global strain energy plot is smooth and well-behaved so it's a convenient criterion for convergence analysis. Von Mises effective stress is a local measure and generally shows "kinks" during the convergence process making it a poor candidate for a criterion of convergence analysis.

It's interesting to note that FEA always underestimates global strain energy. The reason is that by modeling a structure with finite elements, the real continuous stress distribution is replaced with a discrete distribution modeled by finite elements. Consequently, we impose extra constraints on the structure in the form of constant stress values in first-order elements or a linear stress distribution across second-order elements. As users add elements to an h-code mesh, or a p-code upgrades element order in p-element models, these extra constraints become less significant because the model approaches a continuous structure.

To avoid a confusion of terms, it's important to note that convergence error is not the same as solution error. Convergence error refers to a percent difference between the last two iterations. Solution error, on the other hand, refers to the difference between the solution derived from the model at hand and a hypothetical model with an infinite number of elements.

Convergence error and solution error relate only to the idealized FE model and not the real structure. Finite-element representations of the real structure (beams, plates, solids, idealized geometry, idealized material properties, idealized loads, and supports) introduce another error. It depends on modeling practice and cannot be found with FEA techniques. This third error can only be estimated by comparing FEA predictions with results of other analytical or experimental methods of stress analysis.

**Hazards of meshing**

- **Structures are best represented in finite-element analysis by solids be-**
cause they model actual geometry most accurately.

This misconception probably originates from efforts to promote integrated solid modeling and finite-element software. The drawback to solids is that a good solid mesh requires many elements, and that means huge and expensive models.

FE models derived from CAD solids look deceptively accurate. But meshing of solid models often produces poor meshes with many degenerated elements. Also, automatically generated solid meshes require a higher number of elements to properly model the real stress distribution. For example, a rib in bending needs several elements across the thickness.

What happens is that analysts are tempted to model the geometry in great detail only to find that the model is too large to solve. Error analysis is difficult because the model is so large it cannot be further refined, making convergence analysis nearly impossible. An analyst may then be forced to reduce the number of elements, degrade element order, and accept degenerated elements to the point that the model becomes incapable of representing the real stress distribution. A good example would be modeling a thin plate in bending with one first-order element across the thickness.

In truth, geometric accuracy must be maintained only for features important to the structural analysis. In many cases, plate or beam elements better and more efficiently represent the geometry.

Because solid models have excessive detail for FEA, users must simplify them before the models are used in analysis. That means deleting small nonstructural features and taking advantage of symmetry when possible to make the analysis more efficient. The problem is that defeaturing a CAD model is so time consuming it's often faster to rebuild the geometry in the FEA system. For efficient interfacing, CAD solids should be constructed with FEA in mind. However, that usually takes more time and must be carried out by CAD operators trained in FEA.

- Users should model a structure with a coarse mesh first to find stress concentrations, then refine the mesh as required.

While technically correct, the statement misses an important point. In h-code software, even the initial coarse mesh still must be fine enough to detect areas with stress concentrations, despite approximations introduced by constant or linear stress distributions within each element. When the initial mesh is too coarse, stress concentrations do not show up giving users the false impression that the analysis is complete.

P-codes, on the other hand, are capable of upgrading element order as necessary to satisfy the requirements of convergence analysis. Mesh coarseness is not as critical an issue in this technology.
- Higher order elements are more accurate than first-order elements.

Sometimes, yes. But accuracy or lack of it cannot be assigned to the element. Models made with elements of any order may or may not be accurate. It depends on how well the real stress distribution is represented with the given mesh. Higher order elements converge faster and models usually need fewer of them. However, by themselves, they are not any more or less accurate than lower order elements.
- Use the same mesh to analyze different load cases.
For the plate of p elements, strain energy practically converges on about the ninth iteration. For p elements, strain energy is often a better indicator of convergence than stress values.

Von Mises stress calculated from p elements often shows "kinks" as polynomial order of element shape function increases, making it a poor candidate for convergence analysis.

ELEMENT ORDER NOT AN EASY CALL

Using first-order and then second-order elements to model stress distributions is comparable to calculating the area below the curve in the drawing. How element order affects accuracy. Rectangular bars, analogous to first-order elements, produce reasonably accurate results only when many narrow bars are used. Many tall thin rectangles also correspond to using a highly refined mesh on a model.

Bars with tapered tops that nearly match the curve produce a more accurate area estimation. These tall thin polygons approximate the exact solution with fewer elements. Similarly, in reference to meshed FE models, meshes made of second-order elements may be less refined than first-order element meshes and converge on the exact solution more quickly. Their only drawback is that they require longer solution times. Consequently, selecting an element order may be guided by asking how much accuracy can you afford?

A good model involves a fine mesh in areas of stress concentration and a course mesh in the remainder of the model. To produce a good mesh, a stress distribution should be known beforehand. Of course, stresses are not known before meshing but we still need a good understanding of the stress pattern before analysis. In h-codes, initial meshes must be based on educated guesses and then improved during convergence analysis by refining the mesh where required. In p-codes, initial meshes generate more easily because the program upgrades the element order automatically as required to satisfy specified convergence criteria.

In many cases the same mesh may be good enough when loads are not too different, but the same mesh will never be optimal for more than one load case. Each load case may result in significantly different stress distributions so that the model requires a customized mesh for each load case, refined where stress concentrations are expected, and coarse in areas with low stress gradients.

Users often look to automatic mesh generators to solve meshing problems. But such generators lack the engineer's intuition for stress distributions so they generate infinite elements based on geometry alone. Often, they pick a characteristic dimension of a structure, such as rib thickness as element size, and place one element across. A dangerous situation may develop when modeling, for example, a rib in bending...
DEALING WITH INHERENT ERROR

An accurate finite-element analysis requires measuring and evaluating several different error types. For example, a convergence error of 10% means the results before and after mesh refinement differ by 10%. If convergence is taking place, then refining the mesh in high stress areas and running the solver again will produce a result that differs between the current and previous run by less than 10%.

Solution error estimates the difference between the results from an FEA run and, ideally, the exact results. The latter would come from a hypothetical model with an infinite number of infinitely small elements. To estimate a solution error, one has to assess the rate of convergence and predict how results might change in the next few iterations, as if they were performed.

Numerical error in FEA results is the round-off error accumulated by the solver. Usually it is very low and can be ignored. Some FEA codes provide measures to assess this error.

Modeling error comes from FEA having to work on an idealized model rather than an exact representation. Idealizations introduce unavoidable error which can be reduced by good modeling practice, but never eliminated. Modeling error cannot be estimated using any FEA technique. Rather, it should be assessed using other analytical or experimental methods of stress analysis.

Putting error in perspective

Solution of the infinite finite-element model — unknown but estimated

<table>
<thead>
<tr>
<th>Mesh number (iteration number)</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convergence criterion</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Max. Von Mises stress</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
</tr>
<tr>
<td>Convergence error</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>for third mesh</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solution error</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>for third mesh</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

with first-order elements. Almost no stress will be found in the location where the real structure is highly stressed. Furthermore, automatic meshing often produces severely degenerated elements, incapable of producing reliable results.

- Averaging stresses produces better results because it reduces the coarseness introduced by dividing a structure into finite elements.

Actually, stress averaging over a span of elements provides visually pleasing results but does nothing to improve accuracy. In cases of high stress gradients, as near the supporting edge of the disc in the accompanying photograph, stress averaging masks the highest stresses and provides misleading results.

Comparing averaged to nonaveraged stresses sometimes provides a measure of solution quality. When the averaged stress differs "too much" from nonaveraged values for the same location, it means that large stress gradients occur over the span of two elements. The mesh there should be refined. However, it is difficult to put a quantitative mea-
Finite Element Analysis

Stress averaging looks good but may be misleading. Subjecting the rim-supported disc to a pressure load produces the highest stresses around the support. Red indicates the highest-stress section in the disc. When stresses are averaged, as in the lower disc, stress concentrations disappear from the image.

Converging on a Better Answer

The most accurate FE answer is one that has converged on the nearly exact solution. But finding it is entirely too time-intensive. The better answer, guided by convergence analysis, may be the one that’s good enough.

A method of convergence analysis, however, changes with technology. For example, h convergence, or convergence in h-element codes, is based on decreasing element size, either globally or locally, where high stress gradients are expected. Element order stays the same throughout the process. When no singularities exist, such as point loads or point supports, then stress or other quantities chosen as convergence criterion (i.e., strain energy, strain, displacement) converge to the exact solution of the hypothetical, infinitely fine model. Final element size depends on the required convergence error. h-convergence requires user involvement.

Convergence in p-element software is based on increasing the polynomial order for functions that calculate stress distributions within elements. h elements use first and second-order equations (higher orders are rarely used) to model displacement patterns within elements. But p-element software modifies equations about up to the ninth order. Element size stays the same. The final element order depends on a required convergence error. The p-convergence analysis is performed in automated iterations.

P-h convergence is a combination of both p and h methods and should be done when p-convergence reaches the highest available element order and still fails to produce less than a user-specified error. Then, the element size should be reduced and the p-convergence process repeated.

sure on this kind of error estimation.

* FEA is the best method of stress analysis offering the most comprehensive results.

Not always. When backed by impressive graphics, FEA results offer a deceiving level of detail. It’s easy to forget that these approximate results need an error assessment. FEA is also an expensive method. In many cases it’s analysis overkill that slows down the design process. Other methods, such as calculating stresses from equations in Roark’s Formulas for Stress and Strain or physical experiments, can be faster, less expensive, and more reliable.

A maze of analysis

FEA is full of traps waiting for inexperienced users. These snares are the result of oversimplification, visually attractive computer graphics, and reluctance to question results produced by a high-tech tool. Here are a few of the most common ones.

The virtual reality trap lets analysts become so involved in FEA that they may forget about the link between the real structure and model. Then FEA becomes a task for its own sake.

The time trap snaps shut when the modeling process takes so much time and effort that little is left for other duties such as result analysis, alternative-design analysis, writing reports, documentation, backups, or sharing results with designers.

We-use-FEA-on-everything trap snare users when its results are considered such a high assurance of sound structural design that it is performed on simple parts that could be easily handled by handbook formula. Combine this trap with poor modeling techniques and we have designers left with a false sense of trust in their design.

The best way to successfully apply FEA is to combine basic knowledge of theory with practical experience and sound engineering judgment. Don’t be discouraged from using FEA. Like any other advanced and powerful tool, it needs a skilled user who understands its advantages and its limitations. Only well-trained individuals can turn it from an expensive toy into a productive engineering tool.