MACHINE DESIGN

 Machine Design (ISSN 0024-9114) is published semimonthly by Penton Media, Inc., 1100 Superior Ave., Cleveland, OH 44114-2543. Copies not qualified for domestic requester circulation: one year, $105; two years, $165.

Penton

Penton Media, Inc.

Easily made errors mar FEA results

Even before you mesh a part, you may have introduced the potential for erroneous stresses and deflections.

Paul Kurowski
President
Design Generator Inc.
London, Ontario, Canada

Every day brings news of powerful analysis software for shortening development cycles. Trying to keep pace with such progress makes it easy to forget that all those great programs provide accurate results only when properly used.

One of the stumbling blocks on the road to quick and correct FEA results includes idealization errors, those that come from simplifying the real world. They are common yet often unrecognized and dangerous. In this discussion of them, lets use the term finite-element method (FEM), the foundation of FEA, because it emphasizes the underlying numerical method.

It's also useful to briefly review the four steps present in any FEM project. Step one transforms boundary conditions, material properties, and geometry into terms acceptable for analysis. Simplifications are almost always necessary, but they introduce idealization errors. Some are benign but others are hazardous. And don't be too smug if you're using "advanced" software because idealization errors have nothing to do with mesh, elements, or the type of solver used.

Mathematical models formed by simplifications and containing idealization errors then take the form of differential equations. These are usually too difficult to solve analytically, so solvers use an approximate numerical method. Most often we choose the FEM, which has dominated engineering analysis because of its adaptability and numerical efficiency. FEM requires splitting the continuous model into discrete regions or elements. Call this step two, another operation with opportunity for errors.

Step three, the solution, leaves little opportunity for user intervention. The software can introduce numerical round-off errors, but recent programming minimizes their impact.

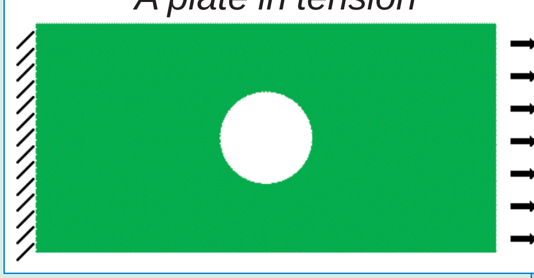
And in step four, after solving a model, you apply results to the design. At that time, it's important to recall the assumptions made during the simplifications and meshing because they have influenced the results.

The boxes that follow show several particular idealization errors that are introduced in step one. And the article avoids referencing FEM-specific issues, such as elements and meshing, as much as possible. However we must touch on the convergence process and the idea of degrees-of-freedom in FEM models because we'll use those as tools to expose the problems of idealizations. To illustrate the convergence process, the examples that follow are meshed and solved with a program that uses p-elements, but the problems they illustrate apply to all FEM-based analysis or any other kind of numerical analysis for that matter.

THE PROBLEM WITH BUILT-IN SUPPORTS

A classic problem involves finding the maximum principal stress in a plate with a center hole. Two-dimensional plane stress is assumed. Results show the highest stress equal to 377 MPa, quite close to 370 MPa predicted by the analytical solution. But 377 MPa is *not* the highest maximum principal stress in this model. The highest one is actually infinite. Why? Because the formula that predicts 370 MPa assumes tension is applied to both sides of the plate, while our model shows the left vertical edge rigidly constrained. The constrained edge tries to shrink sideways under the tensile load, the effect of Poisson's ratio.

A plate in tension



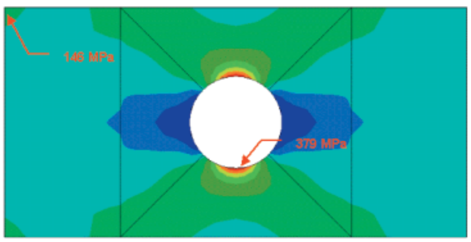
The tensile strip with a hole looks innocent enough. It's 200 × 100 mm, and 10 mm thick. The hole is 40 mm diameter. Load is 100,000 N in tension and the material has a modulus of 200,000 MPa and Poisson's ration of 0.27. Results are in following images.

But built-in supports prevent the edge from "shrinking." Therefore, the mathematical model, as shown in *A closer look at the corners* predicts infinite stresses in both corners. (Infinite stress may also be called singular stress.)

Large elements, such as those in *Results for the test plate*, let the FEM model overlook high stresses altogether. Even though each iteration adds degrees of freedom (dof) to the model, there are still not enough dof to detect those very localized stresses in the two corners. A convergence of the highest principal stress in the chart *Convergence for the plate model refers* to stress at the hole, not to corner stresses, and provides another example of how corner singularities go undetected.

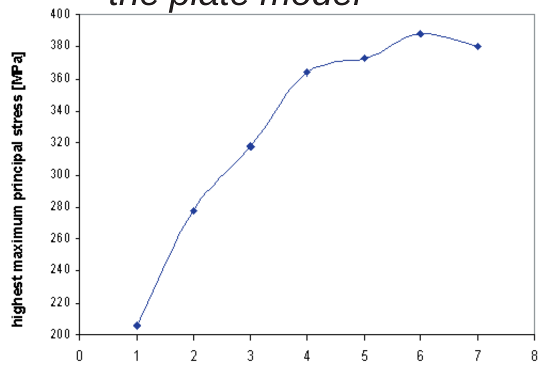
A closer look at the corners more clearly shows that placing small elements in corners and using nonadaptive convergence, reveals high corner stresses. And the graph *Convergence in the corner* shows how stresses there grow with each iteration. It starts high from the beginning. This is because small elements can detect stress concentrations even at low p (polynomial) levels. Perhaps most interesting is that stress shows no signs of convergence. Each consecutive iteration simply produces higher stress.

Results for the test plate



Results for the plate show a high stress at the hole boundary and in the corners. The 146 MPa is high enough to warrant further investigation. A closer examination will reveal that the 146 MPa found there is meaningless.

Convergence for the plate model



A plot of the highest maximum principle stresses for the plate show stresses around the hole.

A DOMINATING FAILURE MODE

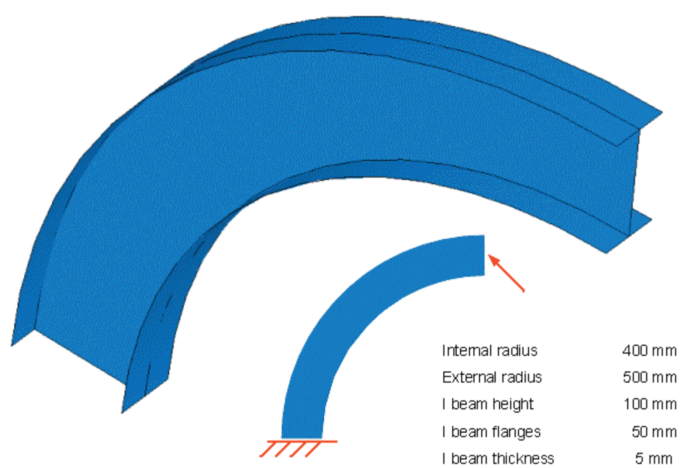
Sometimes it's easier to think of safety in terms of stress levels while forgetting other issues. For example, while analyzing *A curved I-beam* it is easy, yet deadly, to forget that buckling, not the stress, will define the structure's safety.

With a little work, we could fill this entire issue with examples of modeling errors introduced during the idealization process. Indeed, reducing 3D models to 2D representations, beam and shell modeling, defeaturing and geometry clean up, all that is done to simplify models and allow meshing. Each process abounds in traps awaiting an unsuspecting user.

Modeling errors originate from incorrect mathematical models. Some modeling errors, such as singularities, can be revealed (but not cured) using the FEM-based convergence process. Most remain hidden. The only defense is full understanding of the analyzed problem.

A curved I-beam

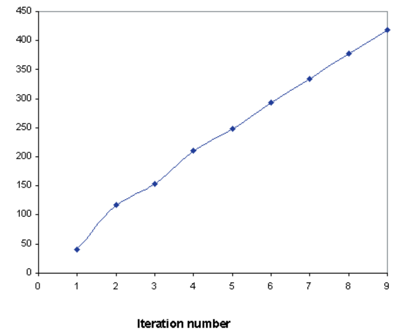
Buckling is not obvious, but it's the dominating mode of failure for the curved beam.



So you might ask: Which model is correct, *Results for the test plate* or *A closer look at the corners*? Answer: Both are correct if we are interested in stress around the hole, but neither one is correct if we are interested in corner stresses because those are singular, a condition that causes ever increasing stress while adding more dof. Consequently, the model can't be used for finding corner stresses. Discretization (step two) just masks singularities introduced in step one, which become visible only by looking for convergence — one way to spot singularities. It would be easy to produce results showing 1,000 or 1 million MPa stress. Just make the corner elements small enough. The same won't happen around the hole. Stress in singular locations (corners) returned by the discrete FEM models are purely discretization dependent and therefore completely unreliable.

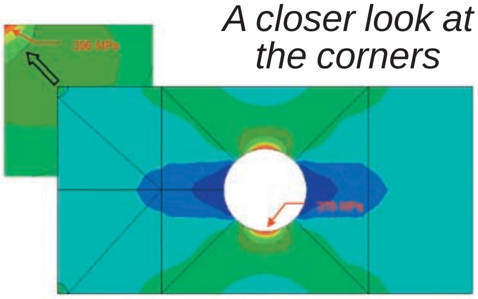
We should emphasize that our plate with the center hole illustrates benign idealization errors introduced in by a rigid support. We had to hunt for singularities and use the trick of very small elements and non-adaptive convergence to reveal them. Benign singularities are common and practically unavoidable. Even the following test bracket is not free from them. In practice, we either don't notice pesky stress concentrations or learn to ignore them, but it is still worthwhile to remember about occasional troubles caused by Mr. Poisson and his ratio.

Convergence in the corner



The highest maximum principal stress in the plate shows no signs of converging to a finite value. Each subsequent iteration produces higher stress.

A closer look at the corners

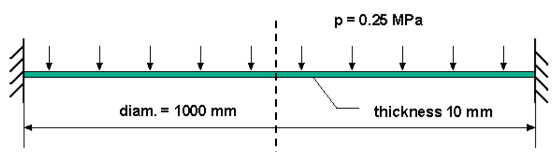


The square area behind the plate shows a magnified corner element. It shows a high stress hidden in the results of the previous contour plot of the plate. Fortunately, this singularity is benign and, most often, we can just ignore it.

THE HAZARD OF MEMBRANE STIFFENING

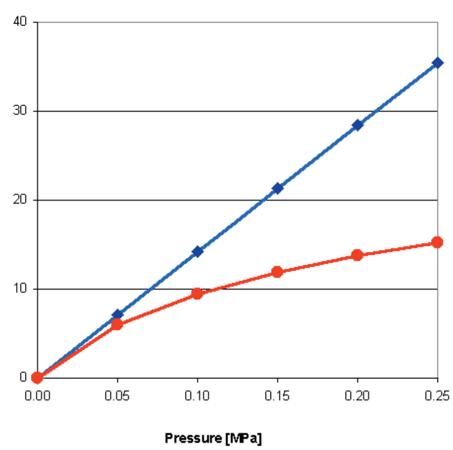
The next objective is to analyze a deflection in the model in *A flat, round membrane*. Deflections won't be more than 20 mm, which is small compared to the 1,000-mm diameter. So we run a linear-static analysis which assumes small displacements, another grave error. Results show a maximum displacement of 35.5 mm, much more than from a physical test. What went wrong?

A flat, round membrane



The membrane is supported around its diameter. The material has E = 73,000 MPa and ν = 0.33

Deflection per theory



The red graph assumes a large-deformation solution while the blue graph assumes small-deformation theory.

Small-displacement theory assumes membranes don't change stiffness as they deform and so it accounts only for the initial bending stiffness. However, the material under deformation acquires "membrane stiffness", which adds to the original value for bending stiffness. As a result, the overall stiffness increases as the membrane deforms. A nonlinear-geometry analysis, also called large-deformations analysis, is required even though the displacement magnitude seems small. The difference between results returned by linear and nonlinear models appears in the red and blue graph.

SHARP RE-ENTRANT EDGES HINT AT STRONG STRESS SINGULARITIES

Idealization errors are not always mild. They often lead to hazardous situations. For example, find the maximum principal stress in a simple 2D plane-stress model for an L-shaped bracket. The *Maximum principal stresses* seem acceptable. As expected, the highest stress is in the corner. But is the maximum-principal stress really equal to 79 MPa? Examining the chart *Stress convergence in the L-shaped bracket* hints that something is wrong. Stress values climb with each iteration and this time we don't have to hunt diverging stress, we just use a standard adaptive-convergence process. To find out what is going on, do as before: place smaller elements in areas of interest. *A closer look at the bracket* shows a maximum stress of 415 MPa and again the convergence curve relentlessly keeps climbing. Can we say which result better approximates reality: 72 MPa or 415 MPa? No, they are the same — they're wrong. That's because the correct FEM model created in step 2 is based on the wrong math model created in step 1. The area of interest holds a singularity so no meaningful results can be produced.

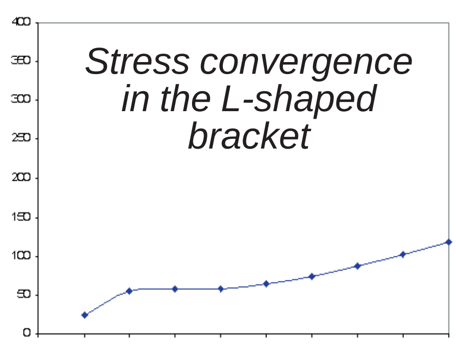
As revealed by the convergence process, reducing element size and upgrading element order lets us chase infinity — 415 MPa is just as far away from infinity as 79 MPa. This time we can't ignore the singularity and still produce meaningful results because we are interested in stress in the location where it's singular.

The remedy is to change the model. One way is to add a fillet, which is always present in real parts anyway. You can also avoid stress singularities by using a different material model, for example the elastic-plastic model instead of a linear-elastic material. The elastic-plastic material model would put an upper bound on stress, and instead of producing meaningless high stress, a plasticity zone would be formed.

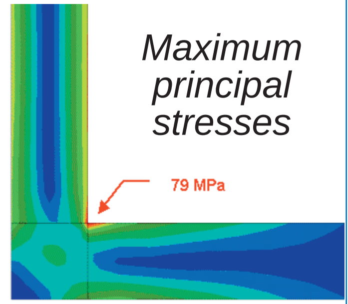
Geometric details, such as a fillet, are frequently difficult to mesh. A technique called defeaturing removes such offending geometry to simplify meshing. Defeaturing, however, can be dangerous. Take the *Support bracket* for example. If the indicated fillet is removed, the model can still be used for deformation or modal analysis. It doesn't have a displacement singularity. Displacements are still finite despite the sharp edge at the base of the center bosses. But removing a fillet creates a sharp re-entrant edge constituting a stress singularity which makes it improper and potentially dangerous for stress analysis. Remember, stress results are meaningless around the sharp reentrant edges (the arrow). This is the area of most engineering interest in the bracket. We can produce stresses as low or high as we want by manipulating elements size and order. Try it yourself.

Convergence should determine whether or not the results are significantly discretization dependent. Only when results converge can we use them with confidence to make design decisions. The L-shape bracket demonstrates the opposite: results are entirely discretization dependent and completely meaningless.

The highest maximum principal stress in the L-shaped bracket shows no sign of converging.

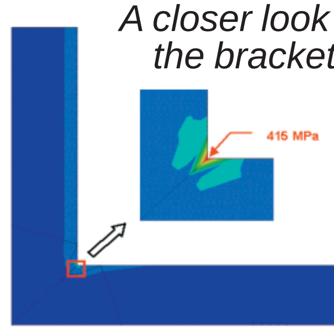


Maximum principal stresses



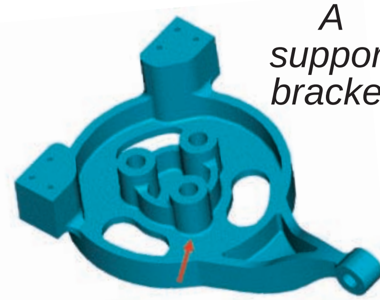
Stress results look plausible and might fool us into believing that 79 MPa is a good answer.

A closer look at the bracket



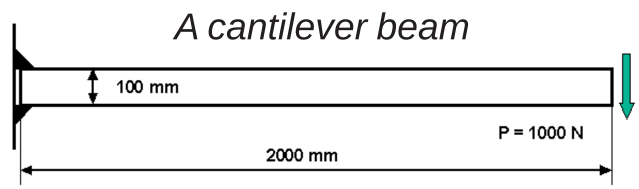
With small elements placed around the sharp inside corner, the solver now finds much higher stress. But is this result closer to the truth than that provided in the previous model?

A support bracket



It may be tempting to remove fillets at the base of the center bosses for easier meshing, but results could be either meaningless or dangerous if fillets are in the area of interest for stress analysis.

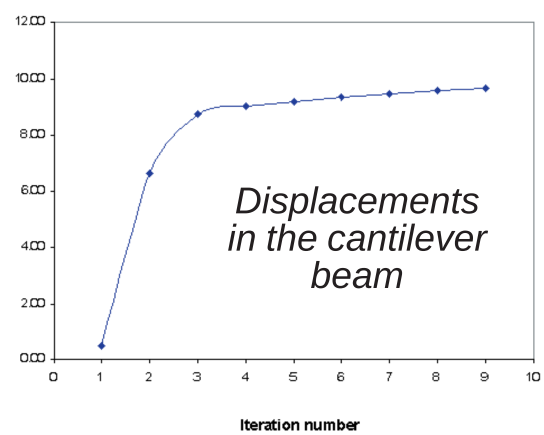
DISPLACEMENT SINGULARITIES



The cantilever beam has a material E = 200,000 MPa, ν = 0.27, and it's 20 mm thick.

Singularities can also affect displacements. Imagine a beam in bending supported at one end by two welds, as in *A cantilever beam*. The objective is to calculate stresses and deflections. Model geometry and boundary conditions lend themselves to a 2D plane-stress representation. Because the weld is small in comparison to the overall beam, we decide to model them as point supports (a bad mistake) surrounded by small elements to more precisely capture the local stress distribution.

A stress convergence curve (not shown) reveals a problem: The curve is not converging. Clearly, this model is not useful for stress analysis because stresses are discretization dependent, that is, they are singular. Notice again that the error in definition of the math model is revealed by the convergence curve. Looking at results of just one single run would have been misleading. So a useful stress analysis is out of the question. But can we use the model for deflection analysis? *Displacements for the cantilever beam* looks okay, and the maximum deflection is just above 8 mm as predicted by beam theory. However, a closer examination of the graph reveals an unnerving fact: Its curve is not converging. Instead, it's slowly but surely increasing. How is that possible? Point supports at the corner show stress that tends to infinity. In fact, the strain also tends to infinity. After all, it is proportional to stress. (The linear relation between stress and strain is expressed by Hooke's law, or $\sigma = E\epsilon$) With strains tending to infinity, displacements have no choice but to follow.



Beam displacements are diverging to infinity, an indicator of a singularity.

The model in *A cantilever beam* is completely wrong. Point support is a cardinal sin of FEM modeling. So why do all FEM programs include point supports? Because they can be used to restrict rigid body movement — when a support generates zero reaction. Point supports are also useful in beam-element models.

One can conclude that both types of modeling errors (sharp re-entrant corners and point supports) originate from an improper definition of the mathematical model upon which the FEM model is constructed. But the errors have nothing to do with FEM. We committed them in step one *before* FEM even entered the scene. So the problems can't be fixed using FEM. Convergence studies, however, can identify the benign from the severe.

STEPS IN FEM PROJECT

Step 1 Simplifications of reality lead to a mathematical model and introduce idealization errors.

Step 2 Replacing a continuum with a set of discrete regions (meshing) introduces discretization errors.

Step 3 The solution of a discrete system introduces numerical errors.

Step 4 Analysis of results introduces interpretation errors.

A FINAL WORD ON CONVERGENCE

The convergence process adds degrees of freedom to the FEM model to see how results change. Degrees of freedom are added either by using more elements (mesh refinement, called h convergence), or by using higher-element orders (p-convergence).

Convergence should demonstrate that results converge to a finite value and, therefore, are not significantly dependent on the choice of discretization. If the process shows that results are significantly discretization dependent, then the results are unreliable.