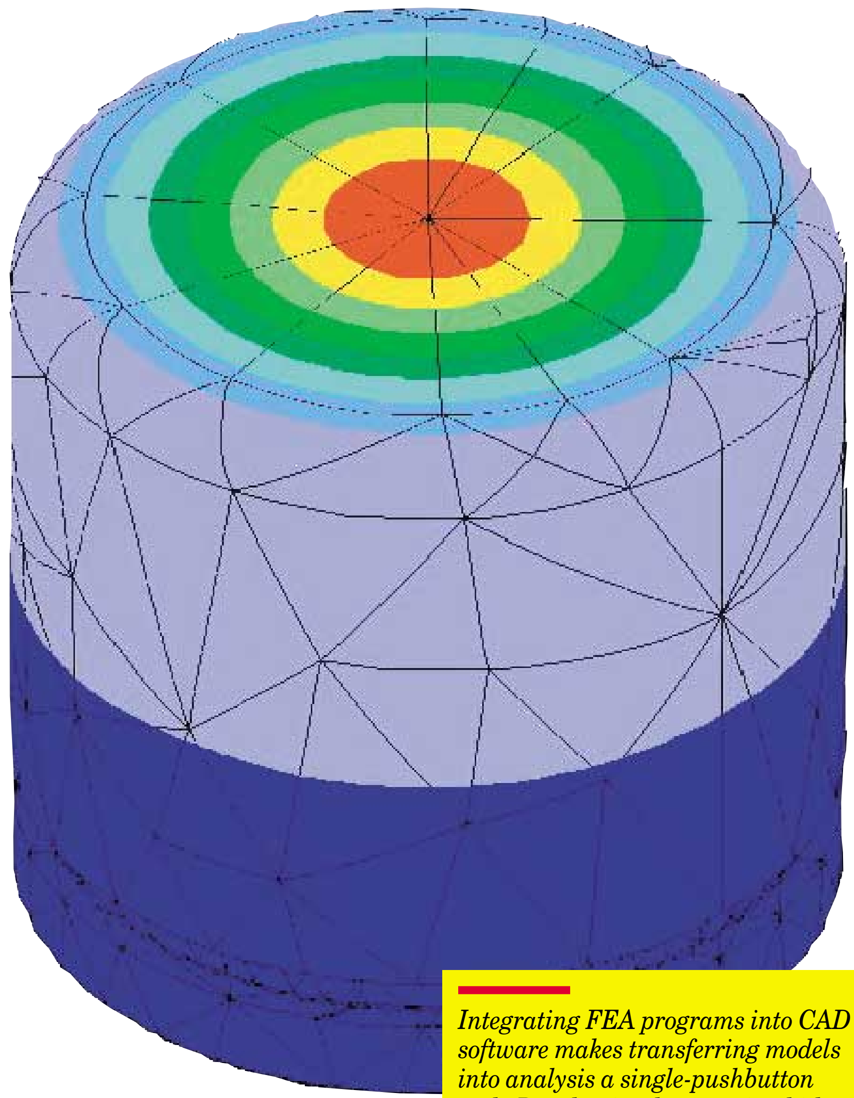
DON'T CHANGE THE MODEL TILL THE SIMULATION FINISHES

Companies that change model geometry while running FEA studies can wind up scrapping the modeling work when analysis uncovers surprises. There are ways around this dilemma.

PAUL KUROWSKI
President
ACOM Consulting
London, Ontario, Canada



A consultant on a recent project was asked for a finite-element stress analysis of a structural design when the CAD model was sufficiently complete. The analyst began the FEA work while the engineer continued developing the CAD model. Both thought that implementing a concurrent CAD-FEA process would shorten the product-development cycle. Both were wrong. Here's what happened.

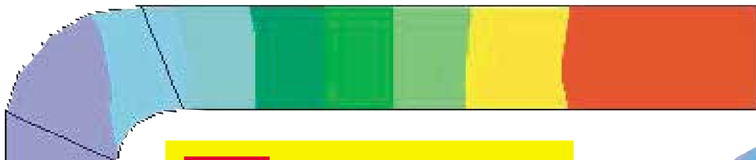
When the first simulation completed, the stress analyst presented the engineer with results that lead them to conclude a major redesign was needed to fix apparent problems. The redesign meant scrapping the engineer's work from the previous several days. What's worse, instead of being an isolated incident, the situation repeated itself several times as FEA kept revealing unexpected structural problems. The team finally decided they would not change the design while the consultant conducted stress analyses. After that, no more time was lost building uncalled-for geometry.

This may sound like heresy to concurrent-engineering advocates because it suggests placing design and FEA work in serial loops, doing one before the other, rather than conducting them simultaneously or on parallel paths. But there are other good reasons for doing so, especially with new designs.

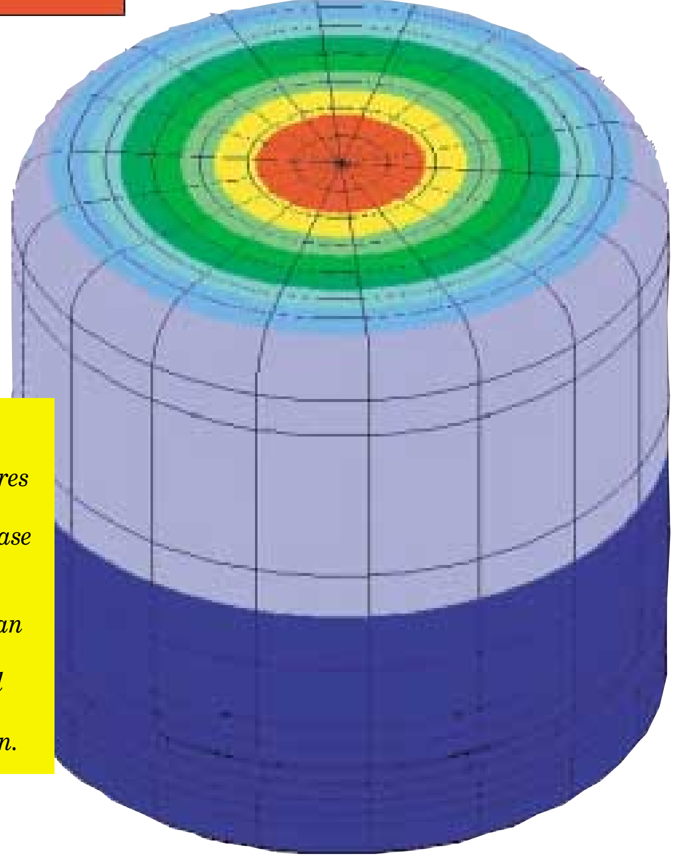
These are usually developed in CAD systems and include manufacturing

Integrating FEA programs into CAD software makes transferring models into analysis a single-pushbutton task. But the simple operation hides drawbacks. The geometry necessary for manufacturing turns into huge analysis models that can bite big chunks out of development schedules. The brake piston, for example, meshes with p-elements into a 2,714-element model, enormous by p-code standards. Fortunately, there are ways around long analyses.

You can find related information at: www.penton.com/md/bde/cad_cam/



One solution to the 2,700-element model is a 2D-stress analysis on half of its symmetric cross section. This model needs only 10 elements.



When the analysis requires 3D models, as would be the case for a modal study, the 2D-cross section can be rotated to generate a still compact 468-element version.

details. Call it CAD-specific geometry. Designs are then submitted to stress analysts who convert CAD-specific geometry into simulation-specific geometry for FEA. This means removing unimportant structural details. Further, analysts may have to idealize the CAD-specific geometry by using zero-thickness wall representations and meshing them with shell elements, or introducing a stick model for beam-element meshing. Simulation-specific geometry may also take advantage of symmetry, asymmetry, or cyclic symmetry, and include only a portion of the analyzed structure. Results should then be used to guide design modifications, which must in turn be verified by FEA, and so on.

HERE'S WHY

One might try to avoid reciprocations by using CAD-specific geometry for analysis. But several problems illustrate the sometimes significant differences between CAD-specific geometry and that needed for simulations. Although the problems have been solved with a p-element version of FEA software, the examples are not software specific and apply to working with any FEA program.

A brake piston, for example, and its hydrostatic-pressure loading make a simple model and seem to be a good candidate for using CAD geometry as it is for the FEA simulation. An FEA model has been created by meshing the geometry imported from a CAD system without any defeaturing.

In the case of the brake piston, the automesher was forced to consider all geometric details such as barely visible chamfers along the lower edge. This produced a huge model of 2,714 solid tetrahedral p-elements in what appears to be a simple part.

A better approach, however, creates FEA-specific geometry instead of using unaltered CAD geometry. In this case, one-half the cross section allows conducting a 2D axisymmetric analysis. The 2D model has only 10 elements.

One might argue it is not worth the time to prepare FEA simulation-specific geometry for a simple 2D model. The argument says just let the 2,700-solid-element model run for as long as necessary.

Using CAD geometry in an analysis might make sense when the design effort called only for one run. But an impending time crunch becomes more obvious when analysts are asked to study many design iterations, or optimize the piston geometry, or both, without first simplifying geometry.

The proposed 2D model is, of course, suitable only for 2D analysis. If the design team needs a modal analysis, for instance, then the study would require a 3D model. But even when analysis requires a 3D model, the 2,700-element version is too large. We would, instead, create a model by revolving the radial cross section used for the 2D analysis. The resultant model consists of only 468 semiautomatically created solid elements.

A radiator manifold, another example, is made of two thin-walled stamped parts soldered together. Meshing the CAD geometry as it is to produce an FEA model is out of the question because of its thin walls. Meshing thin sections with solid p-elements would make another large model. The thin walls would also prohibit meshing the model with h-elements because it would need several element layers across the wall to capture an accurate stress distribution.

The only practical way to construct

an FEA model is to generate mid-plane surfaces suitable for shell elements. The accompanying illustration shows it is enough to model only one-forth of the manifold using double symmetry of geometry, loads, and boundary conditions.

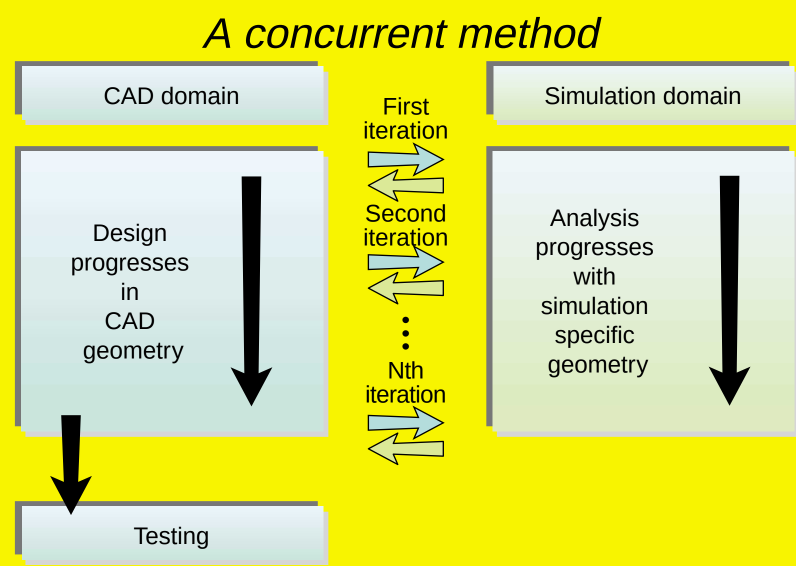
A MODEST PROPOSAL: START THE DESIGN WITH FEA SPECIFIC GEOMETRY

To avoid frequent "FEA round trips," such as those described at the outset, one must first acknowledge that analysis geometry is not the same as design geometry. Then we can question why an engineering department would start and maintain a design in the CAD domain, thereby requiring FEA-round trips. Is it tradition from a time when simulation tools such as FEA were not available? Perhaps. Even now, software such as FEA, CFD, and motion analysis are often thought of as add-ons to CAD, so it may seem natural to start new designs in the CAD domain.

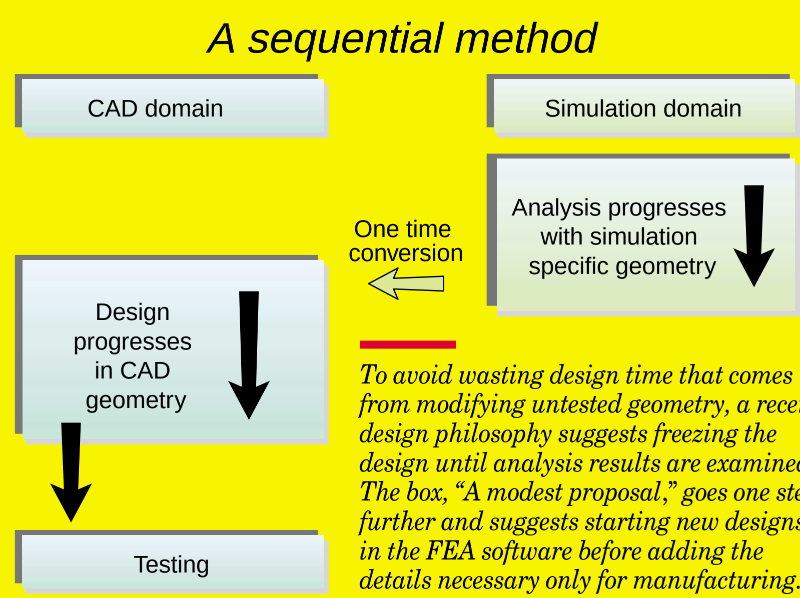
We propose to modify the traditional concurrent CAD-to-FEA process in which each is done simultaneously with its frequent back-and-forth cycles, into a simulation-driven, product-development process. This method would start in the FEA domain and stay there throughout all design iterations avoiding unnecessary FEA round trips.

Once reasonably sure that problems have been weeded out, the design can be converted into CAD-specific geometry. The conversion should occur only once.

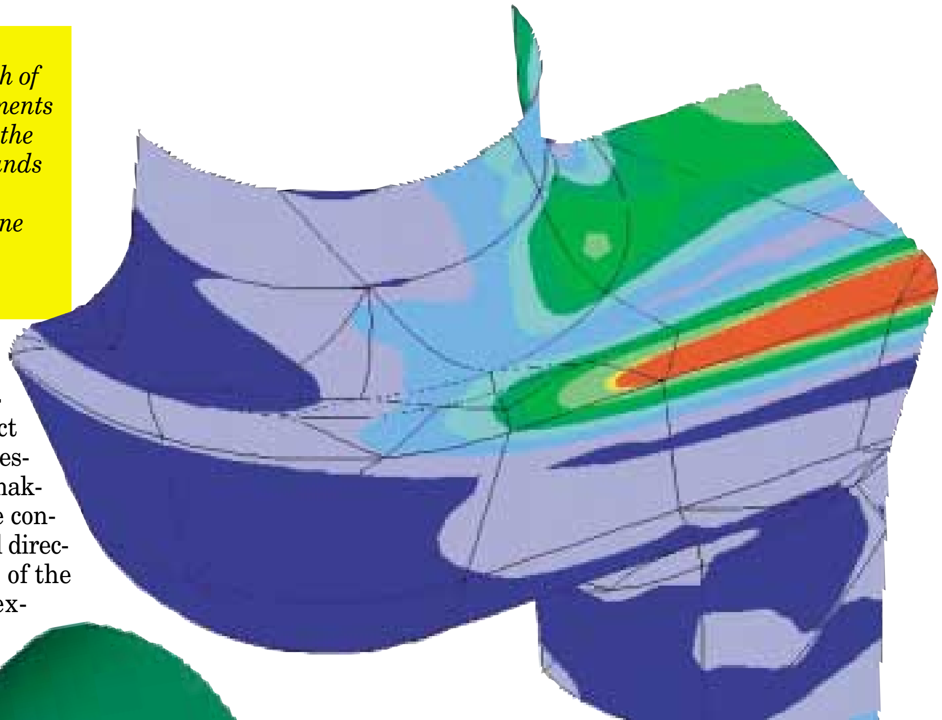
However, we do not propose to use FEA software for conceptual designing. While doing so is conceivable, engineers usually prefer to use CAD for building simulation-specific geometry while maintaining a high level of associativity between two models. When the simulation is complete, geometry can be easily modified with manufacturing-specific detail.



Each occurrence of FEA in the design process requires a shift from CAD-specific geometry to that tuned for analysis and back as illustrated in the diagram. Each cycle, however, consumes time and introduces a potential for errors. FEA results may also reveal shortcomings in the design geometry, requiring the scrapping of work that progressed in the mean time.



To avoid wasting design time that comes from modifying untested geometry, a recent design philosophy suggests freezing the design until analysis results are examined. The box, "A modest proposal," goes one step further and suggests starting new designs in the FEA software before adding the details necessary only for manufacturing.

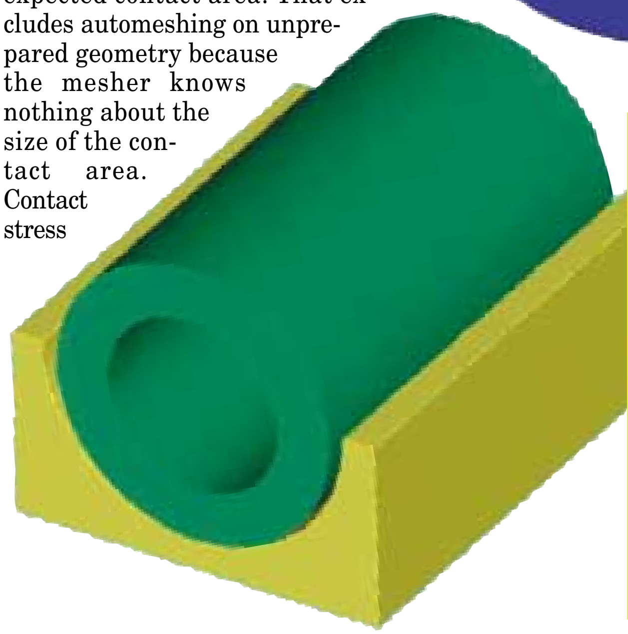


The model represents one fourth of a radiator manifold. Solid elements are out of the question because the thin wall would require thousands to mesh. The solution is to transform the model to midplane surfaces and mesh with shell elements, about 50 in this case.

A roller and concave support provide a model for contact stresses, and another meshing lesson. Correct results come from making sure the element size in the contact region is small enough in all directions in comparison to the size of the expected contact area. That excludes automeshing on unprepared geometry because the mesher knows nothing about the size of the contact area. Contact stress analysis should be run sequentially with several mesh refinements to show that results are not sensitive to smaller elements. So it's still a good idea to simplify the model, in these cases, to a 2D plane strain model.

Some FEA programs, such as Pro/Mechanica, issue a warning when elements are too large relative to the size of contact area and offer localized mesh refinement. Other programs without such features require that users make sure there are enough elements to properly model contact stresses.

Weldments provide other examples of CAD geometry that must be

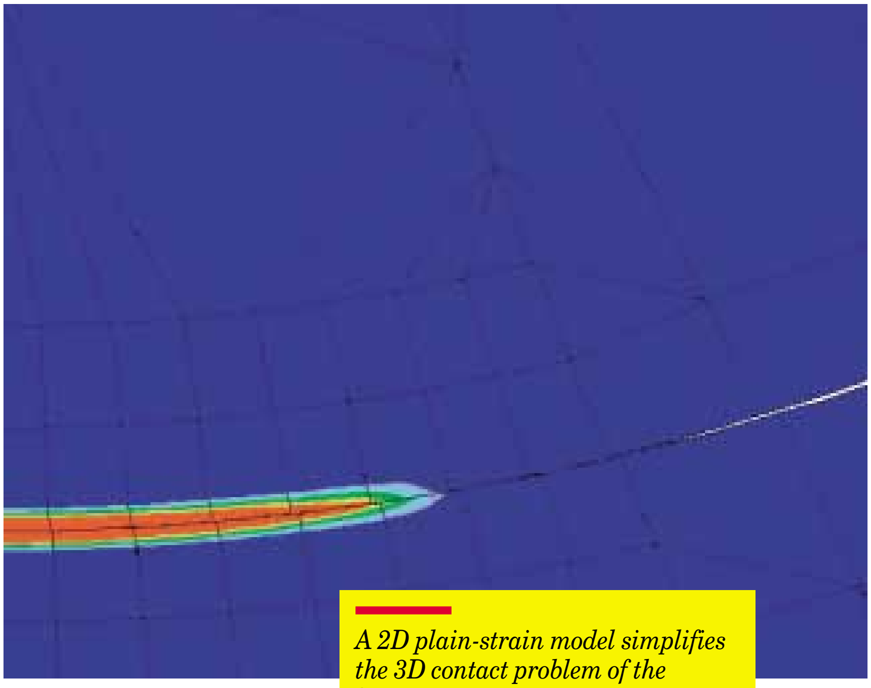


Working in 3D to examine contact stresses in a journal bearing becomes extremely difficult without special contact elements and a mesher to place them. Elements cannot be haphazardly placed, as an automatic mesher might do.

THE NARROW ADVANTAGE OF P-ELEMENTS

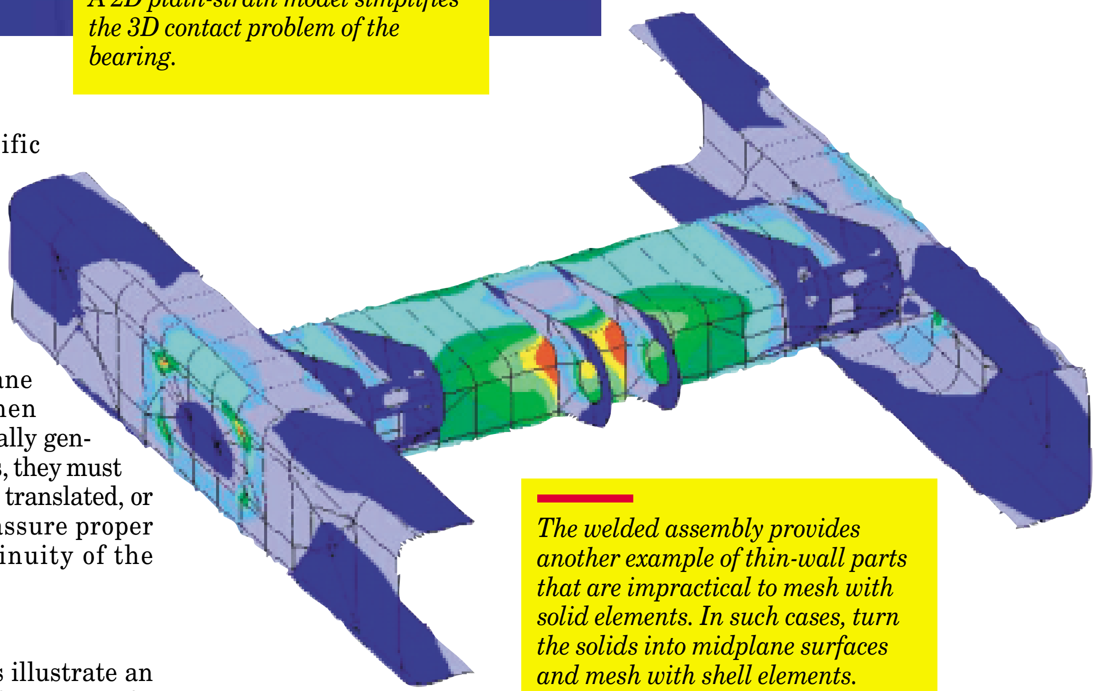
P version of finite elements, such as those used in Pro/Mechanica and other p-codes, use variable-order polynomial functions to describe the displacement field in each element. The required polynomial order comes after several iterations as the software compares results before and after p-order upgrade. The mathematical capability lets one solid p-element across a thin wall accurately model stress distributions, such as bending stress. Even though a solid element in such a case may not be the best modeling practice (shell elements will probably work better) it is possible to automesh thin-wall geometry with solid p-elements. The FEA model may be large and inefficient but will produce correct results.

H-elements use fixed-order polynomial functions (usually first or second order equations) to describe the displacement field in each element. Therefore, one element placed across a thin wall is not capable of correctly modeling stress patterns across the thickness. These need a few layers of h-elements which lead to enormous models. Automeshers, however, will often interpret a thin wall as a characteristic dimension for elements and place only one element across it. The mesh may look impressive but it violates laws of mechanics.



A 2D plain-strain model simplifies the 3D contact problem of the bearing.

turned into FEA-specific models. Meshing thin-wall solid CAD geometry is impossible using h-elements and impractical with p-elements. Shell elements are the right choice when meshing mid-plane surfaces. But even when software can automatically generate mid-plane surfaces, they must be worked on, extended, translated, or split, for example, to assure proper connections and continuity of the model geometry.



The welded assembly provides another example of thin-wall parts that are impractical to mesh with solid elements. In such cases, turn the solids into midplane surfaces and mesh with shell elements.

FEA STILL NEEDS CAD

The former examples illustrate an analyst's need for simulation-specific geometry. However, they do not minimize the importance of a CAD-to-FEA interface. It is almost always easier to rework CAD geometry, even extensively at times, than recreate it from scratch in an FEA program. So the most effective mode of operation is to produce FEA-specific geometry in CAD software that can quickly alternate between manufacturing specific, and idealized or FEA-specific geometry, preferably without losing associativity.

For those searching for a better CAD-to-FEA link, these capabilities in a design program can supplement the features of analysis software:

- A CAD system should create 100% of the geometry, both design and simulation specific.
- A CAD system should let users alternate between CAD-specific geometry and simulation-specific geometry.
- An analysis program should automatically mesh relevant geometry.
- Finite elements should map precisely to geometry. ■

WE WANT YOUR FEEDBACK.

Did you find this article interesting? **Circle 701**

Do you want more information on this topic? **Circle 702**

Comment via e-mail to ***mdeditor@penton.com***

What related topics would you like to see covered? What additional information on this topic would you find useful?