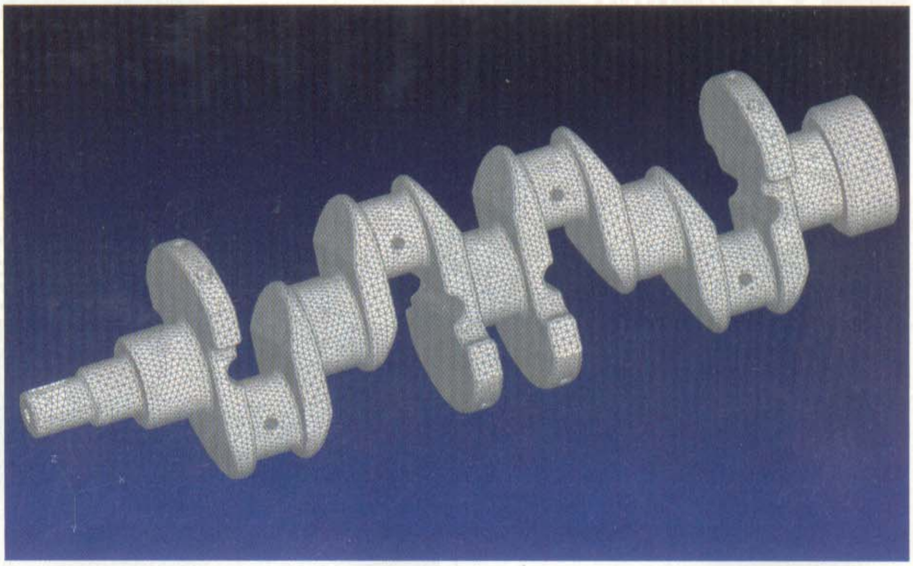
ELEMENTS OF Finite Element Analysis



Noran Engineering's NEiWorks meshed this crankshaft model with TET10 elements for more than 1 million degrees of freedom. Its fine mesh density works well for complex shapes, offering accurate results.

Proper methods of meshing structures and models is key to accurate analysis.

This is the first installment of a series exploring where real-world, hands-on FEA is headed.

Tony Abbey

When I joined the UK aircraft industry in the mid 1970's the standard method of interacting with a computer was rudimentary. We used punch cards, each containing a line of data. We assembled the cards into a deck, forming the FE (finite element) analysis data and analysis job control definition. The deck was fed into the jaws of the mighty IBM mainframe.

My first major analysis, a composite taileron with a titanium shaft under static loading, had an element budget of around 1000 elements using beams, bars, shells, and solids. The budget was based on the available memory of the IBM mainframe and keeping the job turnaround to 2 or 3 days.

Today, many of my clients are using 2 to 3 million elements to model a single component. FE analysis has become simpler to set up and execute, but FEA techniques have become more sophisticated and more powerful. Sophistication has brought a wide range of specialized techniques such as nonlinear contact, rotor dynamics, random analysis, etc. More power means bigger models.

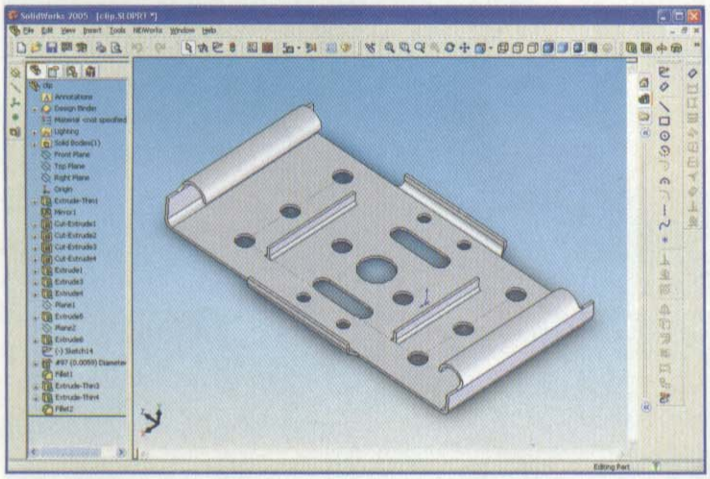


Figure 1: This simple stiffening plate is shown in NeiWorks and needs to be meshed for analysis. The user of the embedded SolidWorks FEA program has a choice between using fast TET meshing, or the more accurate, but time consuming, plate meshing technique.

So, with all the increasing power and sophistication, let's pause and ask ourselves, where is FEA heading? To begin our investigation, we'll start our series looking at some common meshing techniques and the merits of the elements they produce.

BIG MODELS MEAN GIGA DOF

Bigger is better is a well-known cliché. In FEA this equates to the number of elements in the model. This is sometimes a valid approach, but can also be an unnecessary drain on resources. In a worst case, it can give bad answers.

So, how can a model with many elements give worse answers than one with fewer elements? The answer lies in the approach to representing the actual physical model with finite elements—the process known as idealization.

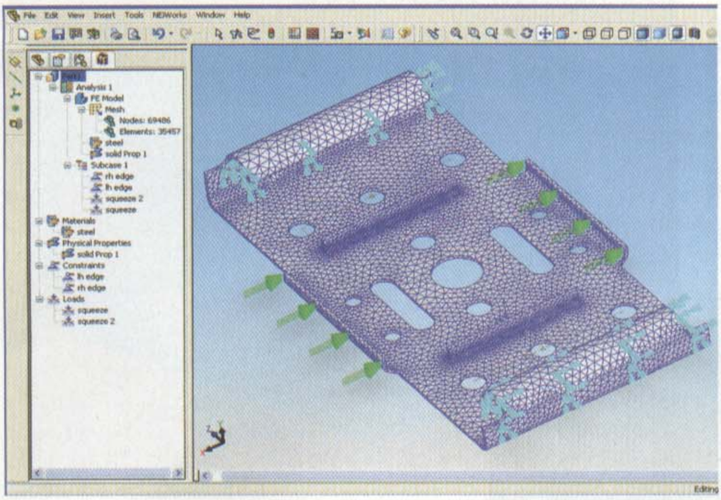


Figure 2: This TET mesh of the stiffening plate already has an element count up around 100,000. With a thin structure like this object, plate element meshing would probably make more sense to achieve better accuracy.

Consider a simple plate-like structure (*see Figure 1, page 22*). The fastest way we can mesh this is with a tetrahedral (or TET) element mesher. A TET element is a pyramid shape that lends itself well to meshing awkward shapes. Assuming we use a higher-order TET, one with mid-side nodes, then the results will be, in general, adequate for solid body type regions.

On the other hand, remember engineers' theory of bending from college? With thin plate structures, we really need a linear stress variation through the depth of the plate. A TET mesh may give a stress distribution very different from a smooth linear variation.

A TET mesh is often very irregular in pattern with big variations in element shape. A pure equal-angled TET gives the most accurate answers. As we distort from this, then numerical integration problems creep in and we start to get inaccuracies. To compensate for limitations of the element we would want at least 3 elements through thickness.

However, we are often faced with the practical fact that we can only afford 1 or 2 elements through the thickness if we want to keep the element count below a million or so (*see Figure 2, above*). In our NeiWorks example—a product of Noran Engineering of Westminster, California—the element count is already up

A Brief History of FEA

By the time the first practical semblance of modern computing methods arrived in the 1950's, the aircraft industry already had a long tradition of solving structural problems using simultaneous equations. So, it's no surprise that methods of solving by computer quickly emerged.

However, the drive to automate this process by being able to describe the equations so that general structural problems could be solved was much slower in coming. An early false start came out of a very logical approach: Stress engineers are most interested in stresses. Stresses are derived from forces. So, it seemed sensible to solve the structural equations for forces using flexibility matrices. However, it proved impossible to set up a general problem definition and solution method with forces as the unknown.

There was a big step forward when it was realized that solving for displacements using stiffness matrices was very practical. Soon after, the idea of defining the structural equations via the concept of elements connecting nodal degrees of freedom evolved. This provided a convenient way of visualizing equations. Instead of pages of math, we have a nice physical analogy.

Opinions on who first coined the term "finite elements" differ on both sides of the Atlantic, and mathematicians claim, as ever, that they invented the idea many years earlier. However, the terminology and the basic idea has remained with us ever since.

—TA

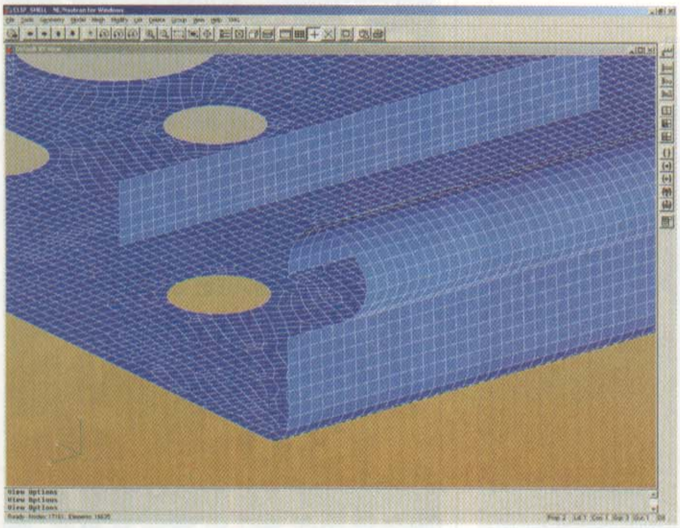


Figure 3a: Here's a region of the plate shown in NEiModeler meshed with plate elements, giving excellent overall accuracy.

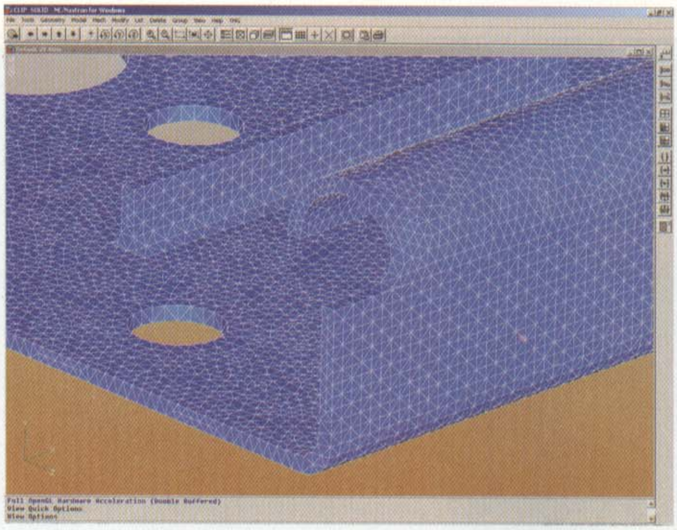


Figure 3b: TET meshing of the part means overall accuracy will suffer due to few elements through its thickness, but detail can be assessed in complex regions.

around 96,000.

An alternative to TET meshing is to use a dedicated plate element, designed to represent thin shell behavior exactly. This will give excellent results and the number of elements necessary will drop dramatically (*see Figures 3a and 3b, above*). Figure 3a shows a detail of the part meshed using plate elements, Figure 3b shows the same region using TET elements. In both cases, NEiModeler was used.

The net result is that we may throw a lot of TET elements at the problem, but they are not always appropriate for the physics of the structural case. A simpler and much cheaper plate mesh will give good answers in this case. The downside of the simple mesh is that it may not be simple to set up. To extract one surface of the structural geometry may take some work in the preprocessor or the CAD program. In the part shown, 13 surfaces are used.

TETS, BRICKS, AND A HOLY GRAIL

The earliest types of solid elements were simple brick-like shapes (*see Figure 4, below*). In the FE world this is often referred to as a hexahedral (or HEX) element.

A basic HEX element has a node at each corner and can have a linear variation in stress in any of its principle directions. If we add mid-side nodes, then it becomes a "higher-order" element. This means the stress variations it can model increase to quadratic or even higher orders. The advantage is that where we have high stress gradients—due to local stress raisers in the structure shape, high local loading such as a jacking point, or very localized constraints such as bolts—then we can do a better job of representing this stress distribution in the FE model.

In the geometrically simple element, the TET shape can also have nodes at corners only, or have mid-side nodes in addition. However, mathematically and in terms of performance, TET is an inferior element to the HEX. The simple TET with only corner nodes is notoriously bad. In effect, it is a constant stress element. This makes it a very poor approximation to any stress gradient locally, but also makes the model overstiff, reducing deflections below correct values.

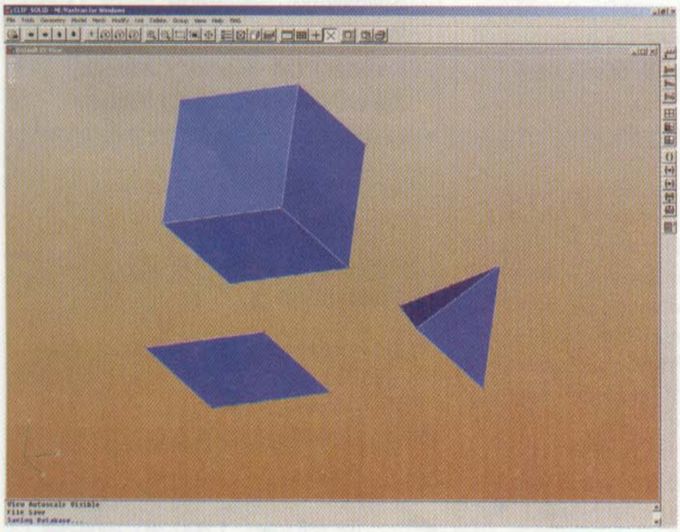


Figure 4: The three basic meshing element shapes are shown: (clockwise from right) TET, plate, and, one of the earliest, the HEX.

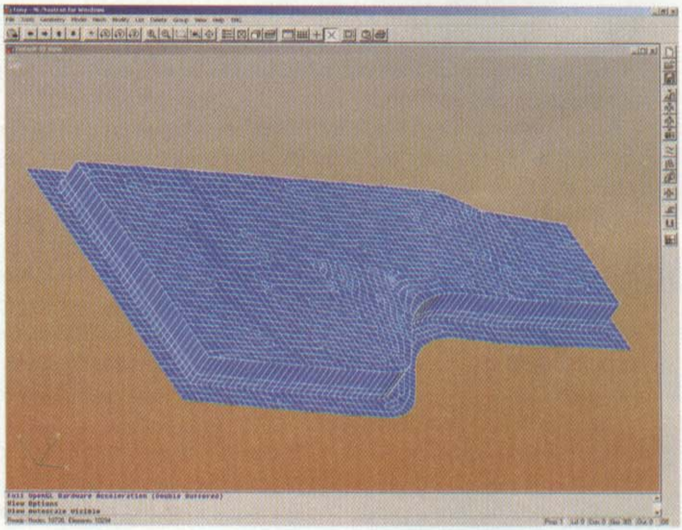


Figure 5: A honeycomb aircraft fuselage skin panel with facing skins meshed in plates and the core meshed with TETs using NEiModeler's extrusion methods.

Why use such a TET element? The answer is that it is straightforward to develop meshing algorithms with the ability to model any shape, and current meshing technology should give high-quality TET meshes.

Amazingly, there is no solution yet to the search for a truly automatic HEX mesher, although many researchers have tried to achieve this over the last 25 years. Several claim to have found this holy grail, but all include workarounds, such as local HEX-to-TET transition zones, or do not work for all arbitrary shapes.

Some regulatory authorities in the nuclear and other industries require that an FE model be exclusively HEX mesh. For very accurate results, such as required for fatigue or fracture mechanics solutions, it may be necessary to use a HEX mesh. In this case, the timescale involved increases dramatically—typically, several weeks to mesh a complex solid versus just a few hours to achieve a TET mesh.

A very useful alternative is to use a 2D meshing approach (*see Figure 5, page 24*). If the variations in the third dimension are simple lofting or sweeping shapes, then we can use a 2D surface mesh and extrude that into a 3D shape. The downside is that the structure is possibly degraded in shape and there is no longer an association between CAD geometry and mesh.

REAL-WORLD MODELING IS TOUGH

Verification manuals and introductory textbooks often use classical solutions to demonstrate the principles and accuracy of FEA. This is an important part of the QA process of any FEA code, but it can be misleading if you think there will always be a correct answer. Over the course of this series we will see many cases where modeling the real world with FEA can be tough. As in any form of engineering, approximations, interpretations, and judgments have to be made as to how the real structure will be fabricated, held, and loaded.

The basic rule is to always question the validity of your model. Are the deflections and stresses simulated of the order that were expected? If I switch off the contour smoothing, do I see the effect of bad meshing? Does the total load summation balance the net pressure loading that I applied?

These are a few of the topics we will explore in future articles. In the meantime, remember the bottom line of analysis: Your analysis is guilty until proved innocent.

Technical Manager **Tony Abbey** *works for Noran Engineering where he demystifies FEA for current users and builds solutions for the future. Send him your thoughts on this article via e-mail c/o de-editors @helmers.com.*

COMPANY INFORMATION

For additional information about meshing for analysis, go to the online version of this article at deskeng.com.

Noran Engineering
Westminster, CA
nenastran.com