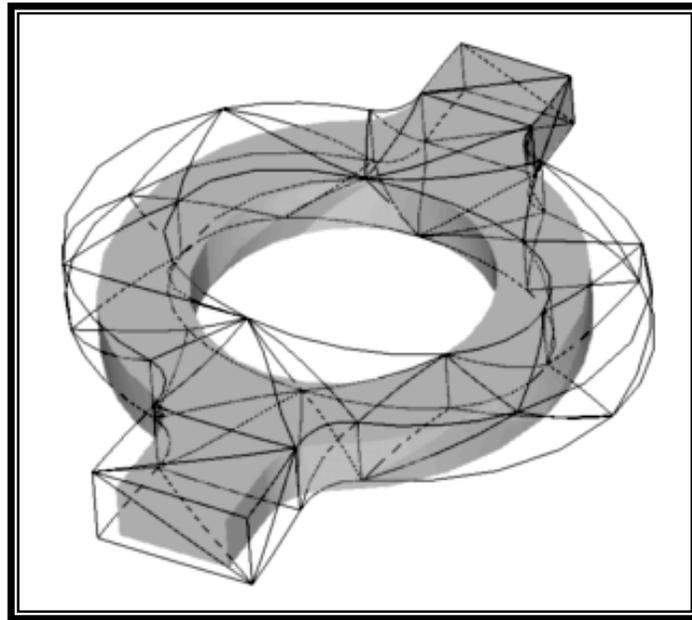


Pro | ENGINEER[®]
WILDFIRE[™] 4.0

MECHANICA Tutorial

(Structure/Thermal)

Integrated Mode



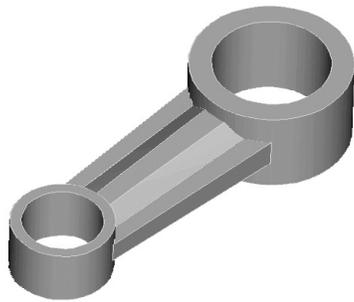
Roger Toogood, Ph.D., P. Eng.

SDC
PUBLICATIONS

Schroff Development Corporation
www.schroff.com

Better Textbooks. Lower Prices.

Updated for
**WILDFIRE
4.0**



Chapter 1 :

Introduction to FEA

Overview

This lesson will be used to get you set up for the rest of the tutorials. It will go over some basic ideas about FEA and what you can do with MECHANICA. The lesson is quite short, and will cover the following:

- ◆ general comments about using Finite Element Analysis (FEA)
- ◆ examples of problems solved using MECHANICA Structure
- ◆ how the tutorial will present command sequences
- ◆ some tips and tricks for using MECHANICA

Finite Element Analysis

Finite Element Analysis (FEA), also known as the Finite Element Method (FEM), is probably the most important tool added to the mechanical design engineer's toolkit in recent years. The development of FEA has been driven by the desire for more accurate design computations in more complex situations, allowing improvements in both the design procedure and products. The growing use of FEA has been made possible by the creation of affordable computers that are capable of handling the immense volume of calculations necessary to prepare and carry out an analysis and easily display the results for interpretation. With the advent of very powerful desktop workstations, FEA is now available at a practical cost to virtually all engineers and designers.

The MECHANICA software described in this introductory tutorial is only one of many commercial systems that are available. All of these systems share many common capabilities. In this tutorial, we will try to present both the commands for using MECHANICA and the reasons behind those commands, so that the general ideas might be transferred to other FEA packages. Notwithstanding this desire, it should be realized that MECHANICA is unique in many ways among packages currently available. Therefore, numerous topics treated will be specific to MECHANICA.

When operating in integrated mode with Pro/ENGINEER, MECHANICA is actually composed of two programs: *Structure* and *Thermal*. The first of these, *Structure*, is able to perform the following:

- ◆ linear static stress analysis
- ◆ modal analysis (mode shapes and natural frequencies)
- ◆ buckling analysis
- ◆ large deformation analysis (non-linear)

and others. This tutorial will be concerned only with the first two of these analyses. The remaining types of problems are beyond the scope of an introductory manual. Once having finished this manual, however, interested users should not find the other topics too difficult. The second program (*Thermal*) is used for steady state and transient thermal analysis. This program can pass information (computed temperature distributions) back to *Structure* in order to compute the associated thermal stresses. We will have a brief look at thermal analysis later in these lessons.

MECHANICA previously contained a third program, *Motion*, which was used for dynamic analysis of mechanisms. Some of the functionality of that program has been included within Wildfire itself (mechanism kinematics, with limited dynamic capability) with the appropriate license configuration (MDX - Mechanism Design Extension). Full dynamic simulation capabilities are available with Mechanism Design Dynamics (which requires appropriate licensing) which is also capable of passing information (dynamic loads) back to *Structure* in order to compute associated stresses. Due to space restraints, this tutorial will not cover any mechanism kinematics or dynamics.

MECHANICA offers much more than simply an FEA engine. We will see that MECHANICA is really a design tool since it will allow parametric studies as well as design optimization to be set up quite easily. Moreover, unlike many other commercial FEM programs where determining accuracy can be difficult or time consuming, MECHANICA will be able to compute results with some certainty as to the accuracy¹.

As of Wildfire 4.0, MECHANICA can handle problems involving a non-linearly elastic (hyperelastic) material like rubber. Problems involving very large geometric deflections (that is, geometric non-linearity) can also be treated, as long as the stresses remain within the linearly elastic range for the material. MECHANICA does not yet have the ability to treat problems where the material is loaded beyond its elastic limit (ie plastic deformation).

In this tutorial, we will concentrate on the main concepts and procedures for using the software and focus on topics that seem to be most useful for new users and/or students doing design projects and other course work. We assume that readers do not know anything about the software, but are quite comfortable with Pro/ENGINEER. A short and very qualitative overview of the FEA theoretical background has been included, and it should be emphasized that this is very limited in scope. Our attention here is on the use and capabilities of the software, not providing a complete course on using FEA, its theoretical origins, or the “art” of FEA modeling strategies. For further study of these

¹This refers to the problem of “convergence” whereby the FEA results must be verified or tested so that they can be trusted. We will discuss convergence at some length later on and refer to it continually throughout the manual.

subjects, see the reference list at the end of the second chapter.

Examples of Problems Solved using MECHANICA

To give you a taste of what is to come, here are four examples of what you will be able to do with MECHANICA on completion of these tutorials. The first three examples, all solved with *Structure*, are a simple analysis, a parametric study called a sensitivity analysis, and a design optimization. The fourth example is a thermal analysis where the temperature distribution is brought back into a stress analysis to compute thermally induced stresses.

Example #1 : Analysis

This is the “bread and butter” type of problem for MECHANICA. A model is defined by some geometry (in 2D or 3D) in the geometry pre-processor. This is not as simple or transparent as it sounds, as discussed below. The model is transferred into MECHANICA where material properties are specified, loads and constraints are applied, and one of several different types of analysis can be run on the model. In the figure at the right, a model of a somewhat crude connecting rod is shown. This part is modeled using 3D solid elements. The hole at the large end is fixed and a lateral bearing load is applied to the inside surface of the hole at the other end. The primary results are shown in Figures 2 and 3. These are contours of the Von Mises stress² on the part, shown in a *fringe* plot (these are, of course, in color on the computer screen), and a wireframe view of the total (exaggerated) deformation of the part (this can be shown as an animation). Here, we are usually interested in the value and location of the maximum Von Mises stress in the part, whether the solution agrees with our desired boundary conditions, and the magnitude and direction of deformation of the part.

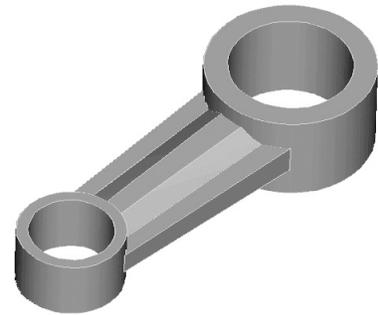


Figure 1 Solid model of a part

² The Von Mises stress is obtained by combining all the stress components at a point in a way which produces a single scalar value that can be compared to the yield strength of the material. This is the most common way of examining the computed stress in a part.

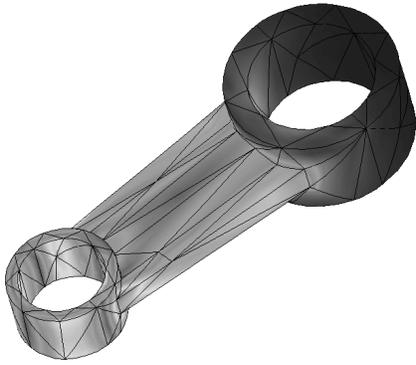


Figure 2 Von Mises stress fringe plot

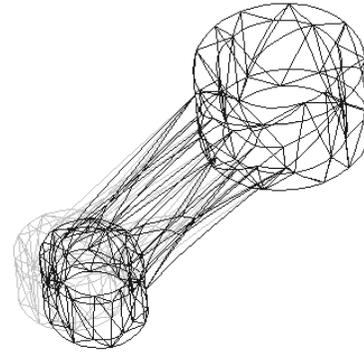


Figure 3 Deformation of the part

Example #2 : Sensitivity Study

Often you need to find out the overall effect on the solution of varying one or more design parameters, such as dimensions. You could do this by performing a number of similar analyses, and changing the geometry of the model between each analysis. MECHANICA has an automated routine which allows you to specify the parameter to be varied, and the overall range. It then automatically performs all the modifications to the model, and computes results for the intermediate values of the design parameters.

The example shown in Figure 4 is a quarter-model (to take advantage of symmetry) of a transition between two thin-walled cylinders. The transition is modeled using shell elements.

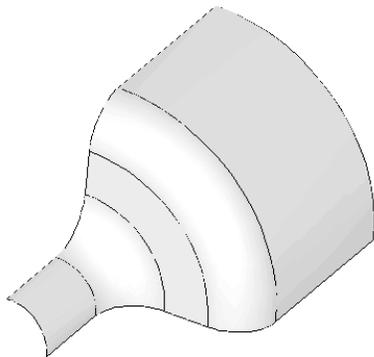


Figure 4 3D Shell quarter-model of transition between cylinders

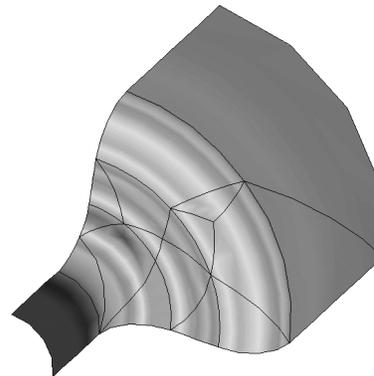


Figure 5 Von Mises stress in shell model

Figure 5 shows the contours of the Von Mises stress on the part produced by an internal pressure. The maximum stress occurs at the edge of the fillet on the smaller cylinder just where it meets the intermediate flat portion. The design parameter to be varied is the radius of this fillet, between the minimum and maximum shapes shown in Figures 6 and 7.

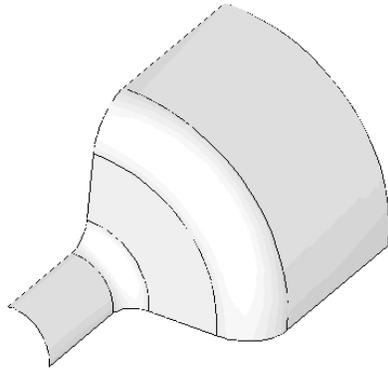


Figure 6 Minimum radius fillet

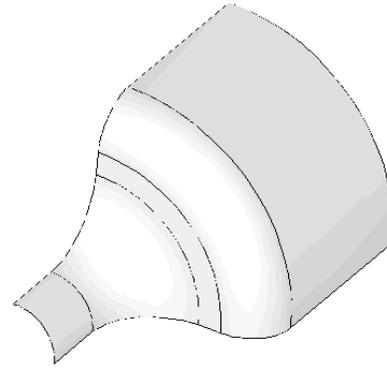


Figure 7 Maximum radius fillet

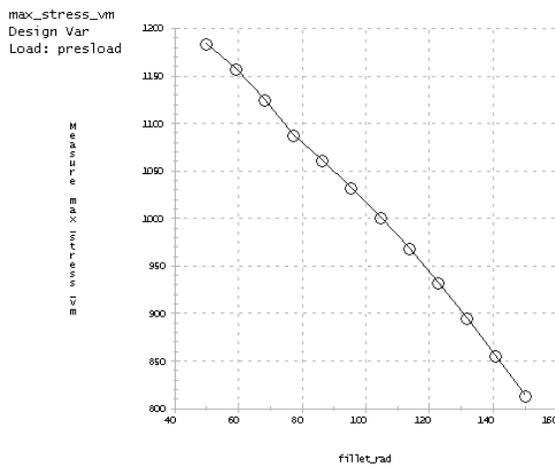


Figure 8 Variation of Von Mises stress with fillet radius in shell model

Figure 8 shows the variation in the maximum Von Mises stress in the model as a function of radius of the fillet. Other information about the model, such as total mass, or maximum deflection is also readily available, also as a function of the radius.

Example #3 : Design Optimization

This capability of MECHANICA is really astounding! When a model is created, some of the geometric parameters can be designated as design variables. Then MECHANICA is turned loose to find the combination of values of these design variables that will minimize some objective function (like the total mass of the model) subject to some design constraints (like the allowed maximum stress and/or deflection). MECHANICA searches through the design space (for specified ranges of the design variables) and will find the optimum set of design variables automatically!

The example shown is of a plane stress model of a thin, symmetrical, tapered plate under tension. The plate is fixed at the left edge, while the lower edge is along the plane of

symmetry. A uniform tensile load is applied to the vertical edge on the right end. The Von Mises stress contours for the initial design are shown in Figure 9. The maximum stress, which exceeds a design tolerance, has occurred at the large hole on the right, at about the 12:30 position. The stress level around the smaller hole is considerably less, and we could probably increase the diameter of this hole in order to reduce mass. The question is: how much?

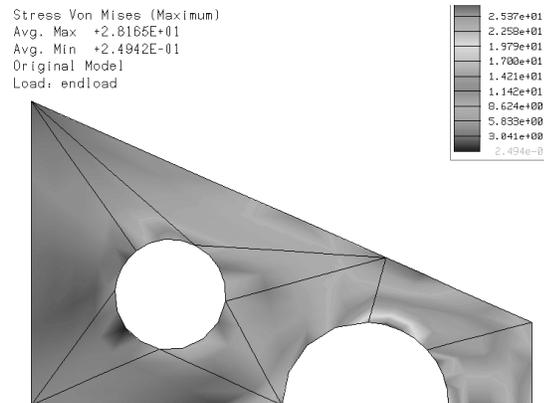


Figure 9 Initial Von Mises stress distribution in plate before optimization

The selected design variables are the radii of the two holes. Minimum and maximum values for these variables are indicated in the Figures 10 and 11. The objective of the optimization is to minimize the total mass of the plate, while not exceeding a specified maximum stress.

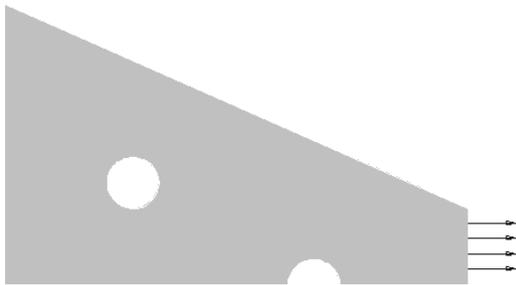


Figure 10 Minimum values of design variables

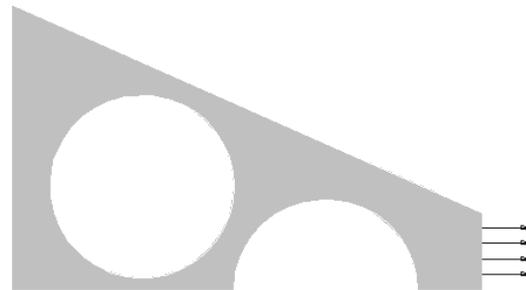


Figure 11 Maximum values of design variables

Figure 12 shows a history of the design optimization computations. The figure on the left shows the maximum Von Mises stress in the part that initially exceeds the allowed maximum stress, but MECHANICA very quickly adjusts the geometry to produce a design within the allowed stress. The figure on the right shows the mass of the part. As the optimization proceeds, this is slowly reduced until a minimum value is obtained (approximately 20% less than the original). MECHANICA allows you to view the shape change occurring at each iteration.

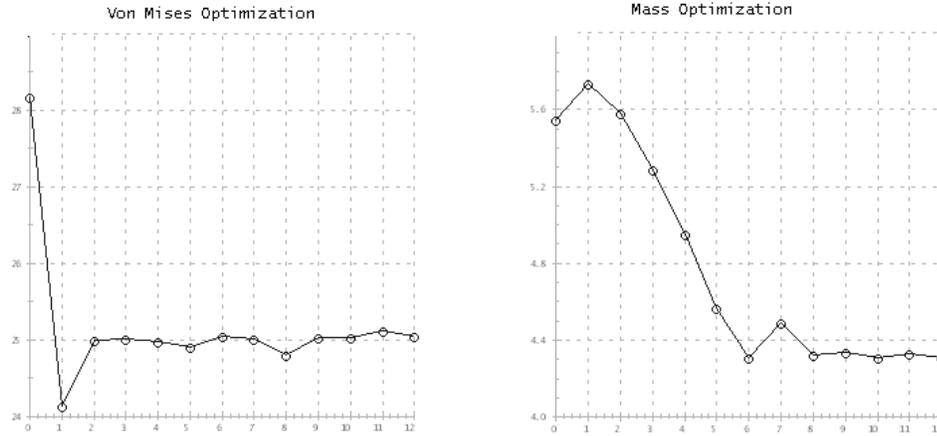


Figure 12 Optimization history: Von Mises stress (left) and total mass (right)

The final optimized design is shown in Figure 13. Notice the increased size of the interior hole, and the more efficient use of material. The design limit stress now occurs on both holes.

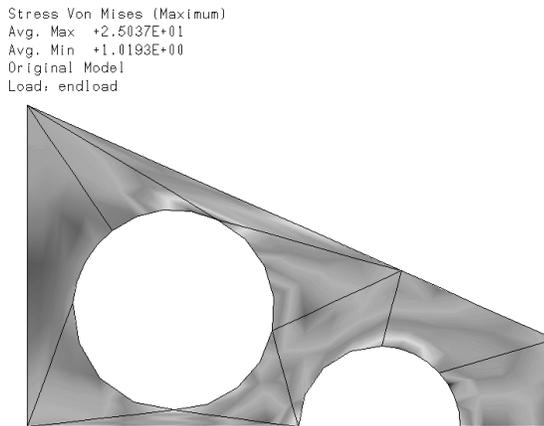


Figure 13 Von Mises stress distribution in optimized plate

In these lessons, we will explore variations of these three types of design study (simple analysis, sensitivity studies, and optimization). We will explore a number of different types of models while doing this (solids, shells, beams, plates, etc.).

Example #4 : Thermally Induced Stress

This example concerns the analysis of the body of a gate valve, shown in Figure 14. The model is first simplified using symmetry and removal of non-essential features (flange bolt holes, o-ring grooves, and some rounds). The resulting model is loaded as follows:

- an internal pressure that acts directly on the interior walls and produces a net upward force on the top flange (caused by the valve bonnet, not shown in the figure)
- temperature variation produced in the valve body due to the convective heat transfer caused by a hot gas flowing through the valve, plus heat lost to the environment.

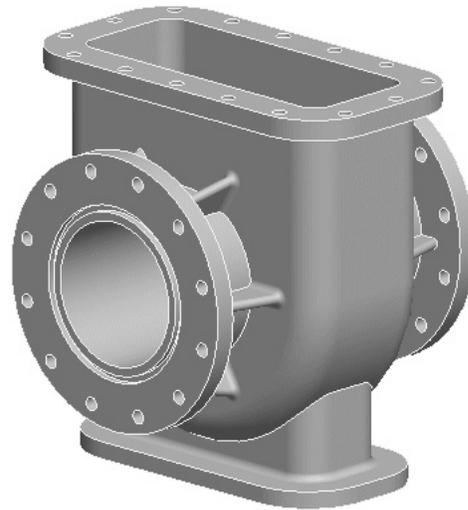


Figure 14 Valve body loaded with internal pressure and thermally induced stress

Figure 15 shows the temperature distribution on the model obtained with the thermal analysis. The thermal analysis uses the same mesh as the stress analysis, so that the temperature data can be fed back into Structure so that the stresses due to the temperature variation in the model can be determined.

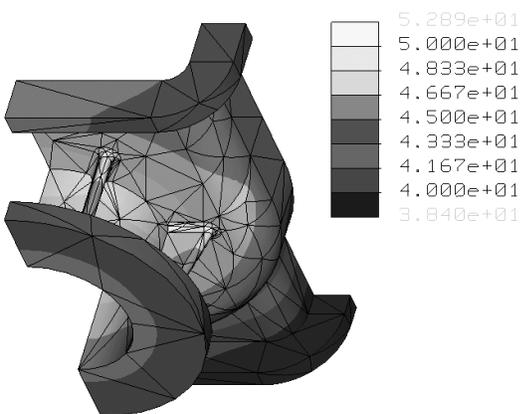


Figure 15 Temperature distribution in the valve body ($^{\circ}\text{C}$)

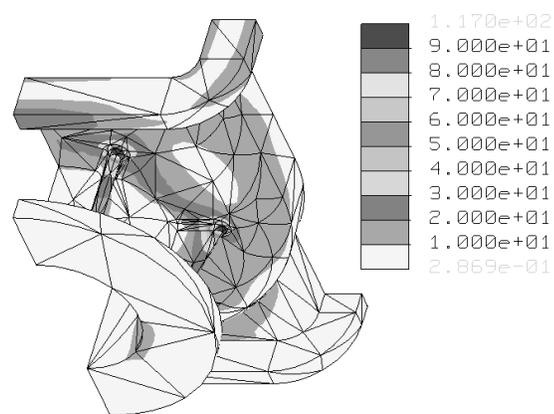


Figure 16 Von Mises stress resulting from combined pressure and thermal load

Figure 16 shows the final combined results of the thermal stresses and those due to pressure. Using superposition principles, it is possible to examine these separately to find out which one contributes most to the overall stress (it is the thermal stress in this problem).

This is a static analysis only. More complicated transient heat transfer models can be treated. Thermal model idealizations are available for 2D problems (analogous to plane stress and plane strain models), and beams and shells can also be treated. Some special care must be taken when dealing with convective heat transfer from these idealized geometries. Also, Thermal cannot presently handle radiation heat transfer problems directly.

FEA User Beware!

Users of this (or any other FEA) software should be cautioned that, as in other areas of computer applications, the GIGO (“Garbage In = Garbage Out”) principle applies. Users can easily be misled into blind acceptance of the answers produced by the programs. **Do not confuse pretty graphs and pictures with correct modeling practice and accurate results.**

A skilled practitioner of FEA must have a considerable amount of knowledge and experience. The current state of sophistication of CAD and FEA software may lead non-way users to dangerous and/or disastrous conclusions. Users might take note of the fine print that accompanies all FEA software licenses³, which usually contains some text along these lines: “The supplier of the software will take no responsibility for the results obtained . . .” and so on. Clearly, the onus is on the user to bear the burden of responsibility for any conclusions that might be reached from the FEA.

We might plot the situation something like Figure 17 on the next page. In order to intelligently (and safely) use FEA, it is necessary to acquire some knowledge of the theory behind the method, some facility with the available software, and a great deal of modeling experience. In this manual, we assume that the reader's level of knowledge and experience with FEA initially places them at the origin of the figure. The tutorial (particularly Chapter 2) will extend your knowledge a little bit in the “theory” direction, at least so that we can know what the software requires for input data, and (generally) how it computes the results. The step-by-step tutorials and exercises will extend your knowledge in the “experience” direction. Primarily, however, this tutorial is meant to extend your knowledge in the “FEA software” direction, as it applies to using MECHANICA. Readers who have already moved out along the “theory” or “experience” axes will have to bear with us - at least this manual should help you discover the capabilities of the MECHANICA software package.

³ Or, for example, see the Disclaimer at the front of this book.

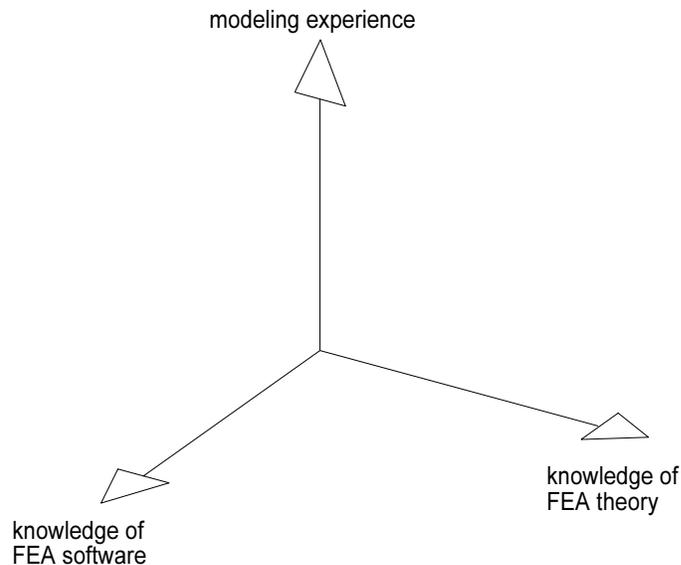


Figure 17 Knowledge, skill, and experience requirements for FEA users

Some quotes from speakers at an FEA panel at an ASME Computers in Engineering conference in the early 1990's should be kept in mind:

"Don't confuse convenience with intelligence."

In other words, as more powerful functions get built in to FEA packages (such as automatic or even adaptive mesh generation), do not assume that these will be suitable for every modeling situation, or that they will always produce trustworthy results. If an option has defaults, be aware of what they are and their significance to the model and the results obtained. Above all, remember that just because it is easy, it is not necessarily right!

"Don't confuse speed with accuracy."

Computers are getting faster and faster. This also means that they can compute an inaccurate model faster than before - a wrong answer in half the time is hardly an improvement!

and finally, the most important:

"FEA makes a good engineer better and a poor engineer dangerous."

As our engineering tools get more sophisticated, there is a tendency to rely on them more and more, sometimes to dangerous extremes. Relying solely on FEA for design verification might be dangerous. Don't forget your intuition, and remember that a lot of very significant engineering design work has occurred over the years on the back of an envelope. Let FEA become a tool that extends your design capability, not define it.

Tips for using MECHANICA

In the tutorial examples that follow, you will be lead through a number of simple problems keystroke by keystroke. Each command will be explained in depth so that you will know the “why” as well as the “what” and “how”. Resist the temptation to just follow the keystrokes - you must think hard about what is going on in order to learn it. You should go through the tutorials while working on a computer so that you experience the results of each command as it is entered. Not much information will sink in if you just read the material. We have tried to capture exactly the key-stroke, menu selection, or mouse click sequences to perform each analysis. These actions are indicated in ***bold face italic type***. Characters entered from the keyboard are enclosed within square brackets. When more than one command is given in a sequence, they are separated by the symbol ">". When several commands are entered on a single menu or window, they are separated by the pipe symbol "|". An option from a pull-down list will be indicated with the list title and selected option in parantheses. So, for example, you might see command sequences similar to the following:

*Materials > Assign > Part > STEEL_IPS | Accept
Analysis (QuickCheck)
Results > Create > [VonMises] | Accept*

At the end of each chapter in the manual, we have included some Questions for Review and some simple Exercises which you should do. These have been designed to illustrate additional capabilities of the software, some simple modeling concepts, and sometimes allow a comparison with either analytical solutions or with alternative modeling methods. The more of these exercises you do, the more confident you can be in setting up and solving your own problems.

Finally, here a few hints about using the software. Menu items and/or graphics entities on the screen are selected by clicking on them with the *left mouse button*. We will often refer to this as a ‘left click’ or simply as a ‘click’. The *middle mouse button* (‘middle click’) can be used (generally) whenever ***Accept, Enter, Close*** or ***Done*** is required. The dynamic view controls are obtained using the mouse as shown in Table 1. Users of Pro/E will be quite comfortable with these mouse controls. Any menu commands grayed out are unavailable for the current context. Otherwise, any menu item is available for use. You can, for example, jump from the design menus to the pulldown menus at any time. Many operations can be launched by clicking and holding down the right mouse button on an entry in the model tree or in the graphics window. This will produce a (context sensitive) pop-up menu of relevant commands.

Table 1-1 Common Mouse Functions

Function		Operation	Action
Selection (click left button)		LMB	entity or command under cursor selected
Direct View Control (drag holding middle button down)		MMB	Spin
		Shift + MMB	Pan
		Ctrl + MMB (drag vertical)	Zoom
		Ctrl + MMB (drag horizontal)	Rotate around axis perpendicular to screen
		Roll MMB scroll wheel (if available)	Zoom
Pop-up Menus (click right button)		RMB with cursor over blank graphics window	launch context-sensitive pop-up menus

As of Release 2001, Pro/E and MECHANICA incorporated a new “object-action” operating paradigm (as opposed to the previous “action-object” form). This means you can pick an object on the screen (like a part surface), then specify the action to be performed on it (like applying a load). This is a much more streamlined and natural sequence to process commands. Of course, the previous action-object form will still work. In this Tutorial, command sequences are represented at various times in either of the two forms. Hopefully, this will not get confusing.

So, with all that out of the way, let's get started. The next chapter will give you an overview of FEA theory, and how MECHANICA is different from other commercial packages.

Questions for Review

1. In MECHANICA-ese, what is meant by a “design study?”
2. What are the three types of design study that can be performed by MECHANICA?
3. What is the Von Mises stress? From a strength of materials textbook, find out how this is computed and its relation to yield strength. Also, for what types of materials is this a useful computation?
4. Can MECHANICA treat non-linear problems?
5. What does GIGO mean?
6. What three areas of expertise are required to be a skilled FEA practitioner?

Exercises

1. Find some examples of cases where seemingly minor and insignificant computer-related errors have resulted in disastrous consequences.