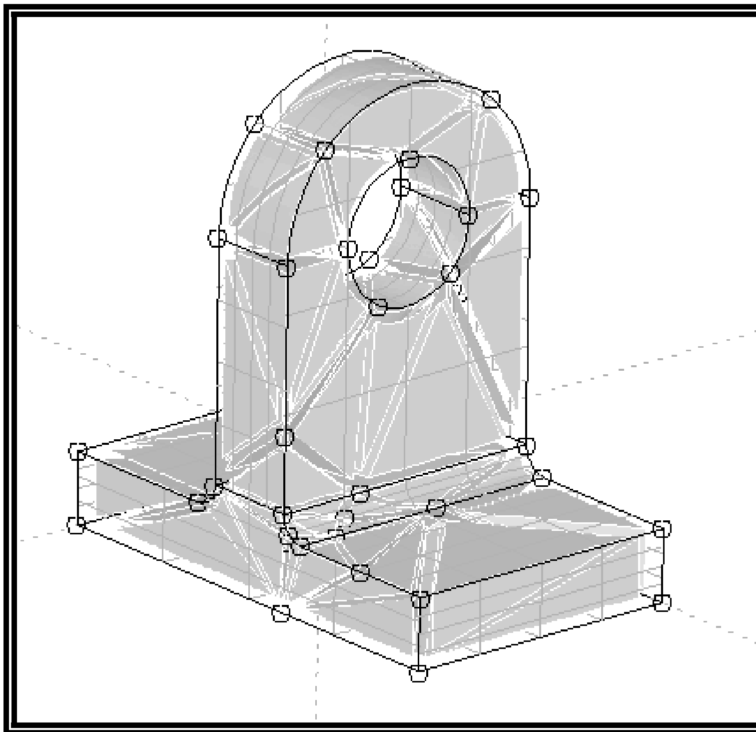
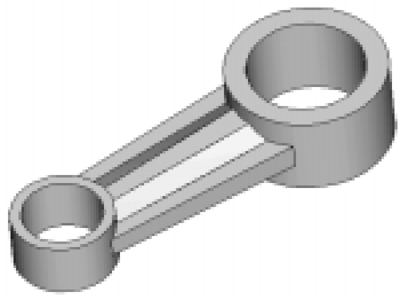
Pro/MECHANICA Tutorial
Structure
(Release 2000*i*)

A Click-by-Click Primer



Roger Toogood, Ph.D., P. Eng.
Mechanical Engineering
University of Alberta

SDC
PUBLICATIONS
www.SDCpro.com



Chapter 1 :

Introduction to the Tutorials

Synopsis:

An introduction to finite element analysis, with some cautions about its use and misuse; examples of problems solved with MECHANICA; organization of the tutorials; tips and tricks for using MECHANICA

Overview of this Lesson

- ♦ general comments about using Finite Element Analysis (FEA)
- ♦ examples of solved problems using Pro/MECHANICA Structure
- ♦ layout of the tutorials
- ♦ how the tutorial will present command sequences
- ♦ 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 design engineer's toolkit this century. 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 computation engines that are capable of handling the immense volume of calculations necessary to 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 Pro/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 procedures can be transferred to other FEA packages. Notwithstanding this desire, it should be realized that Pro/M is unique in many ways among packages currently available. Therefore, numerous topics treated will be specific to Pro/M.

MECHANICA is actually a suite of three programs: *Structure*, *Thermal*, and *Motion*. The first of these, Structure, is able to perform the following:

- ♦ linear static stress analysis
- ♦ buckling analysis
- ♦ modal analysis (mode shapes and natural frequencies)

This manual will be concerned only with the first of these. The remaining two 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 other two programs (MECHANICA/Thermal and MECHANICA/Motion) are used for thermal analysis and dynamic analysis of mechanical systems, respectively. These are planned to be the topics of further tutorials in the Click-by-Click series.

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 accuracy[1].

MECHANICA does not currently have the ability to handle non-linear problems, for example a stress analysis problem involving a non-linearly elastic material like rubber. New capability in 2000i, however, allows problems involving very large geometric deflections to be treated, as long as the stresses remain within the linearly elastic range for the material.

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. We have included a short overview of the FEA theoretical background, but want to emphasize that this is very limited in scope. Our attention here is on the use and capabilities of the software, not providing a course on using FEA, its theoretical origins, or the "art" of FEA modeling strategies.

Examples of Problems Solved using MECHANICA

To give you a taste of what is to come, here are three examples of what you will be able to do with MECHANICA on completion of these tutorials. The examples are a simple analysis, a parametric design study called a sensitivity analysis, and a design optimization. In MECHANICA's language, these are called *design studies*.

[1]This 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.

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), material properties 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 stress[2] 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 interested in the maximum Von Mises stress in the part, and the magnitude and direction of maximum deformation of the part.

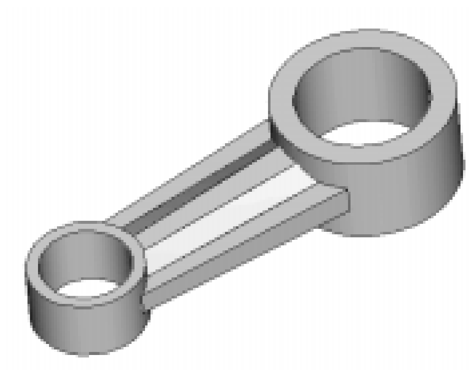


Figure 1 Solid model of a part

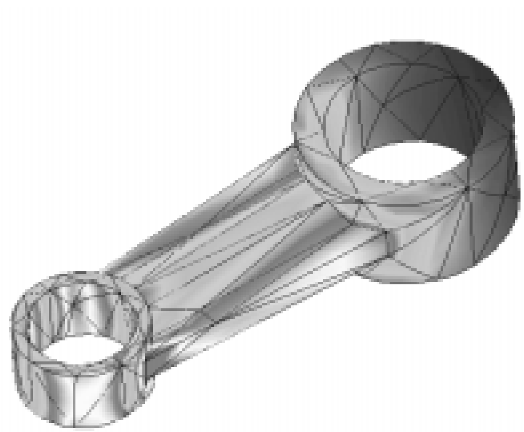


Figure 2 Von Mises stress fringe plot

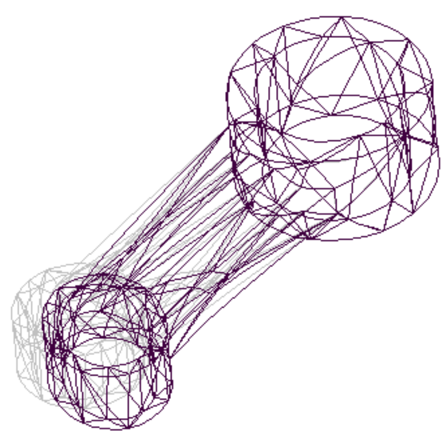


Figure 3 Deformation of the part

Example #2 : Sensitivity Study

Often you need to find out the overall effect 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.

---

[2] The Von Mises stress is obtained by combining all the stress components at a point in a way which produces a single 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.

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.

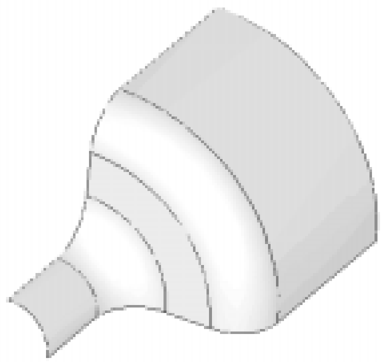


Figure 4 3D Shell quarter-model of transition between cylinders

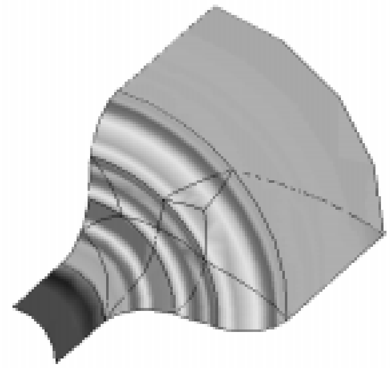


Figure 5 Von Mises stress in shell model

Figure 5 shows the contours of the Von Mises stress on the part. 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.

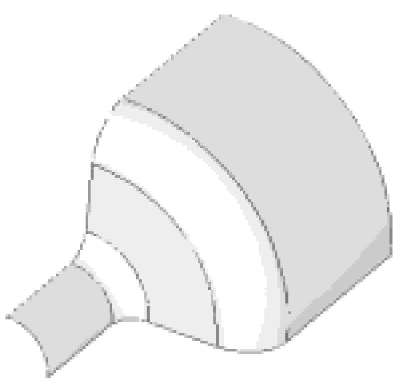


Figure 6 Minimum radius fillet

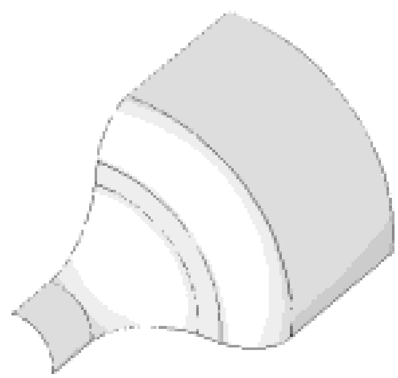


Figure 7 Maximum radius fillet

Figure 8 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.

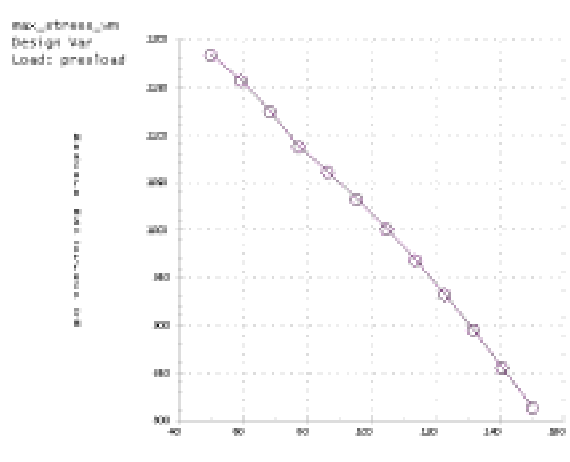


Figure 8 Variation of Von Mises stress with fillet radius in shell model

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). Pro/M 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 hole located on the plate centerline, 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?

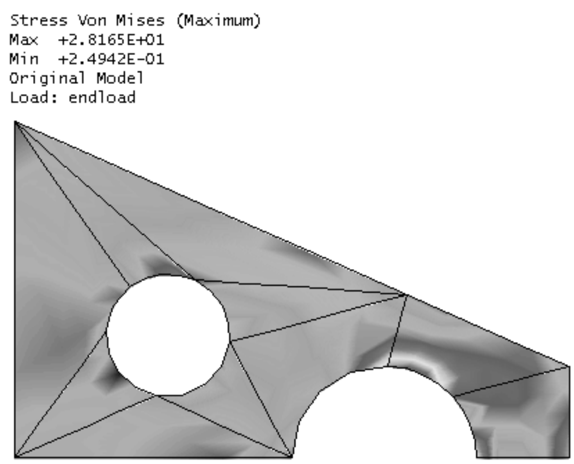


Figure 9 Initial Von Mises stress distribution in plate before optimization

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

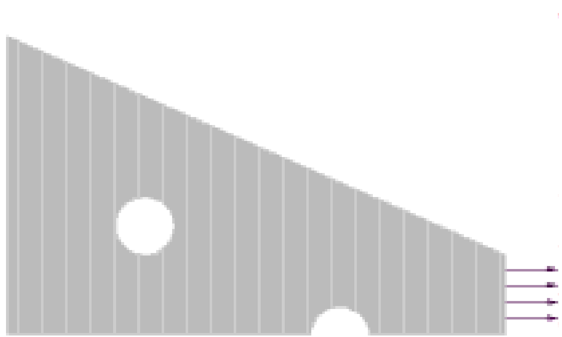


Figure 10 Minimum values of design variables

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 - note that this initially exceeds the allowed maximum stress, but Pro/M 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). Pro/M allows you to view the shape change occurring at each iteration.

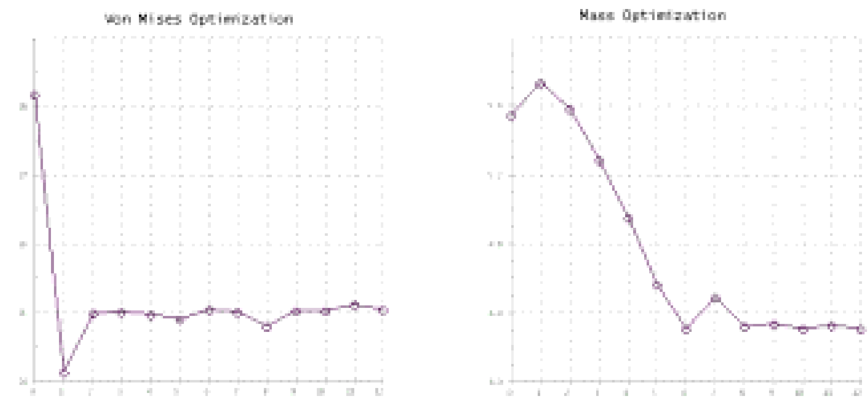


Figure 12 Optimization history: Von Mises stress (left) and total mass (right)

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

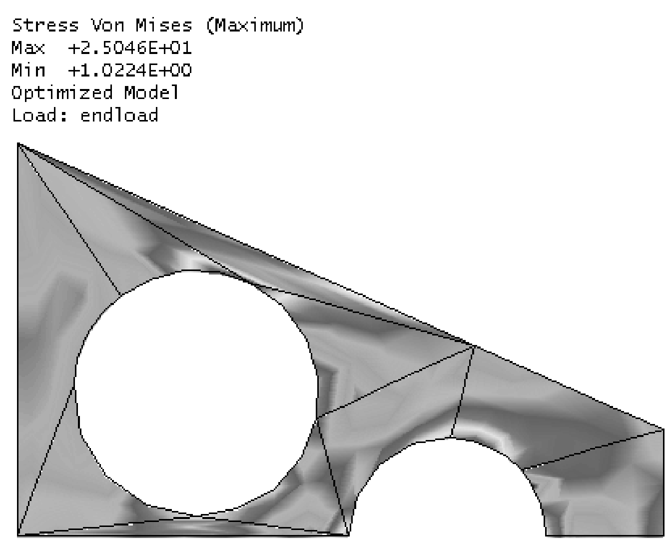


Figure 13 Von Mises stress distribution in optimized plate

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-wary users to dangerous and/or disastrous conclusions. Users might take note of the fine print that accompanies all FEA software licenses, 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 14 below. 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 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 Pro/MECHANICA. Readers who have already moved out along the "theory" or "experience" axes will have to bear with us - at least this manual should assist you in discovering the capabilities of the MECHANICA software package.

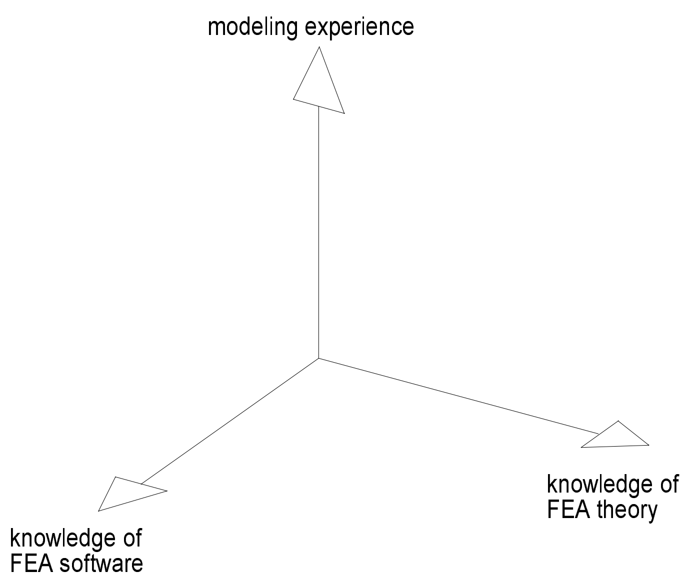


Figure 14 Knowledge, skill, and experience requirements for FEA users

In summary, some quotes from speakers at an FEA panel at a recent ASME Computers in Engineering conference should be kept in mind:

"Don't confuse convenience with intelligence."

In other words, as more powerful functions (such as automatic mesh generation) get built in to FEA packages, do not assume that these will be suitable for every modeling situation. 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 - an wrong answer in half the time is worse than no answer at all!

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.

Layout of this Manual

Running the Pro/MECHANICA software is not a trivial operation. However, with a little practice, and learning only a fraction of the capabilities of the program, you can perform FEA of reasonably complex problems. This manual is meant to guide you through the major features of the software and how to use it. The manual is not meant to be a complete guide to either the software or FEA modeling - consider it the elementary school of practical FEA!

Chapter 2 of the tutorial will present an overview of the theory and mathematics behind how FEA is implemented in MECHANICA. In particular, the origin and differences between h-code analysis and the p-code method in MECHANICA are discussed. The primary purpose of this chapter is to outline the main capabilities of MECHANICA as they apply to the design and analysis of mechanical parts. These include simple analyses, sensitivity studies, and parameter optimization. This chapter will basically introduce you to the terminology used in the program, and give you an overview of its operation.

Chapters 3 and 4 will present some commands for creating 2D geometry, and performing a simple analysis of a plane stress problem. Common methods of displaying results are shown. Some issues of modeling are discussed, such as symmetry. Several modeling pitfalls, which also occur in other model types (axisymmetric and 3D, for example) are investigated, and solutions proposed. At the end of each of these chapters, a number of additional exercises are presented. You should try to do as many of these as you can in order to build up your knowledge and repertoire of modeling scenarios.

Chapter 5 will introduce you to sensitivity studies in Pro/M and the definition of design variables. The context will be that of axisymmetric models using solid and shell elements. Some new geometry commands will be introduced as well. Chapter 6 is devoted to setting up an optimization. Here, the context will be simple plane stress and plane strain models. The use of a temperature load is introduced.

Our first major foray into 3D modeling comes in Chapter 7 where we will have a look at more geometry commands and models that utilize shell elements. Beams and frames are treated in Chapter 8, including distributed loads, shear and moment diagrams, and beam releases. Finally, solid models will be the topic of Chapter 9 where we will use three different models of the same part, created using different command variations.

Tips for using MECHANICA

In the design study examples, 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". 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 manual. 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 " | ". Sometimes, an option from a menu of buttons will be indicated with the menu title and option in parantheses. So, for example, you might see command sequences similar to the following:

Materials > Assign > Part > STEEL_IPS | Accept
Model > Geometry > Line > Two Points > Snap(Point)
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 carry out. 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 whenever ***Accept*** or ***Enter*** is required. The dynamic view controls are obtained by holding down the *Ctrl* key and dragging with a mouse button (left = zoom, middle = spin, right = pan). Users of Pro/ENGINEER 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 tools menus at any time. Finally, any command sequence in progress can be immediately terminated/aborted using the escape key, *Esc*.

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 a design study?
2. What are the three types of design study that can be performed by Pro/M?
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?

Exercises

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