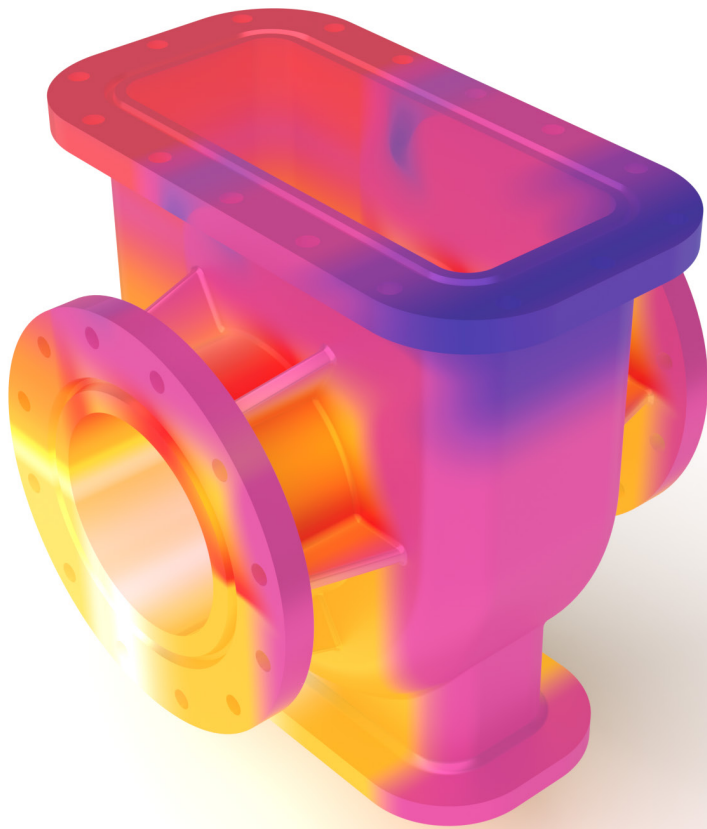


Creo® Simulate 6.0 Tutorial

Structure and Thermal



Roger Toogood, Ph.D.

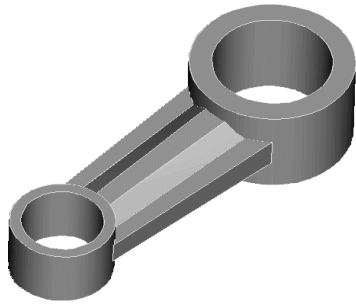
Visit the following websites to learn more about this book:



[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)



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 Creo Simulate. The chapter is quite short, and will cover the following:

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

Finite Element Analysis

Finite Element Analysis (FEA), also known as the Finite Element Method (FEM), is arguably the most important addition to the design engineer's toolkit in the last few decades. 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 use of simulation in design (FEA in particular) was 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. FEA is available at a practical cost to virtually all engineers and designers.

The Creo Simulate¹ software described in this introductory tutorial is only one of many commercial FEA 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 Creo 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 Creo is unique in many ways among software currently available. Therefore, numerous topics treated will be specific to Creo Simulate.

¹ This will be referred to as “Creo” from now on, unless reference is being made to other software in the Creo family, like Creo Parametric or Creo Direct.

Creo Simulate is able to perform the following *structural* analyses:

- ◆ static stress analysis for linear materials
- ◆ modal analysis (mode shapes and natural frequencies)
- ◆ buckling analysis
- ◆ fatigue analysis
- ◆ stress analysis of composite materials
- ◆ non-linear static stress analysis (hyperelastic materials and/or large deformations)

and others. This tutorial will be concerned only with the first two of these analyses, the remaining types of problems being beyond the scope of an introductory manual. Once having finished these lessons, however, interested users should be able to take on the more complex models with more confidence. Creo can also perform the following *thermal* analyses:

- ◆ steady state and transient analysis of problems involving conduction, convection, and radiation.

The results of the thermal analysis (computed temperature distributions) can be used to compute the associated thermal stresses. We will have a brief look at thermal analyses later in these lessons.

A previous incarnation (PTC's MECHANICA) contained a third program, *Motion*, which was used for dynamic analysis of mechanisms. Some of the functionality of that program has been included within Creo Parametric with the appropriate license configuration. Full dynamic simulation capabilities are available which allows passing information (dynamic loads) back to Creo Simulate in order to compute associated stresses. This tutorial will not cover any mechanism kinematics or dynamics.

Creo offers much more than simply an FEA engine. We will see that it is really an easily used but powerful 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, Creo Simulate will be able to compute results with some certainty as to the accuracy².

As mentioned above, Creo can handle problems involving non-linearly elastic (hyperelastic) materials like rubber. Problems involving very large geometric deflections (that is, geometric non-linearity) can also be treated. Creo can also treat problems where the material is loaded beyond its elastic limit (i.e. elastoplastic deformation) and therefore undergoes permanent, non-recoverable deformation.

² 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.

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 simulation but are quite comfortable with Creo Parametric (in particular the user interface). 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 the theoretical origins of FEA. For further study of these subjects, see the reference list at the end of the second chapter.

Examples of Problems Solved using Creo Simulate

To give you a taste of what is to come, here are four examples of what you will be able to do on completion of this tutorial. The first three examples are structural problems and include 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 : Stress Analysis

This is the “bread and butter” type of problem for Creo. A model is defined by some geometry (in 2D or 3D) in the geometry pre-processor (Creo Parametric). This is not as simple or transparent as it sounds, as discussed below. The model is transferred into Creo Simulate 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 surface of 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.

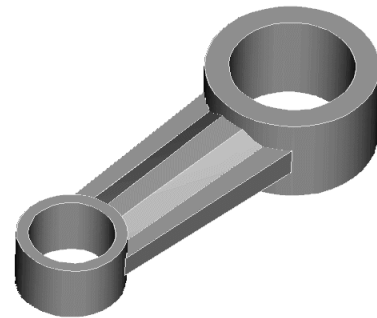


Figure 1 Solid model of a part

The primary results are shown in Figures 2 and 3. These are contours of the Von Mises stress³ 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.

³ 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 is compared to the yield strength of the material to determine failure. This is a common way in FEA of examining the computed stress in a part.

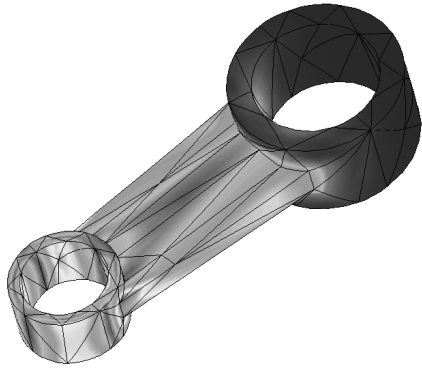


Figure 2 Von Mises stress fringe plot

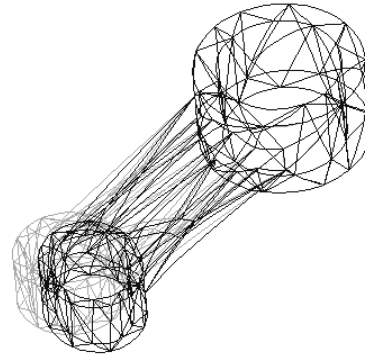


Figure 3 Deformation of the part

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. Creo 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 designated intermediate values of the design parameters. This is possible due to the tight integration with Creo Parametric, which is the geometry engine.

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.

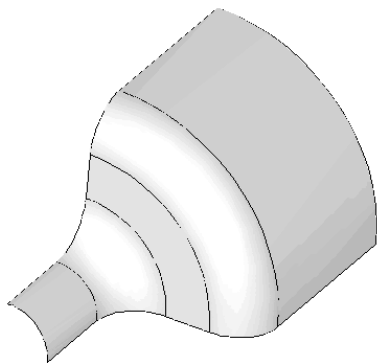


Figure 4 3D Shell quarter-model of transition between cylinders

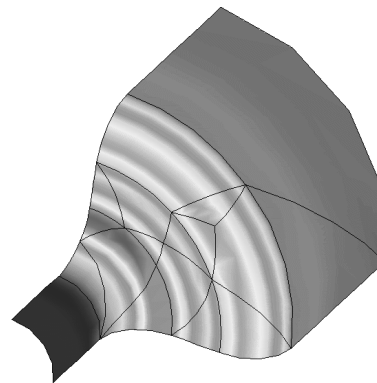


Figure 5 Von Mises stress in shell model

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.

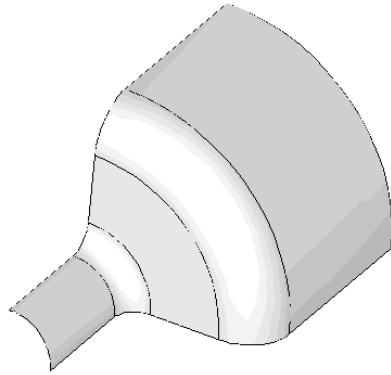


Figure 6 Minimum radius fillet

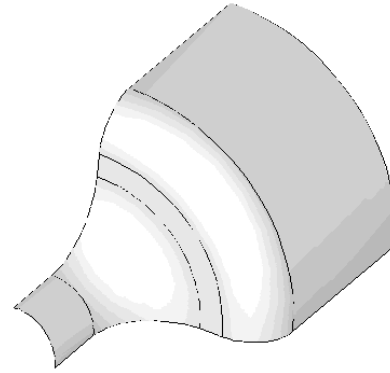


Figure 7 Maximum radius fillet

Figure 8 shows the variation in the maximum Von Mises stress in the model as a function of radius of the fillet. This graph is created automatically for a sensitivity study - a great time saver for the designer. Other information about the model can also be plotted, such as total mass or maximum deflection also as a function of the fillet radius.

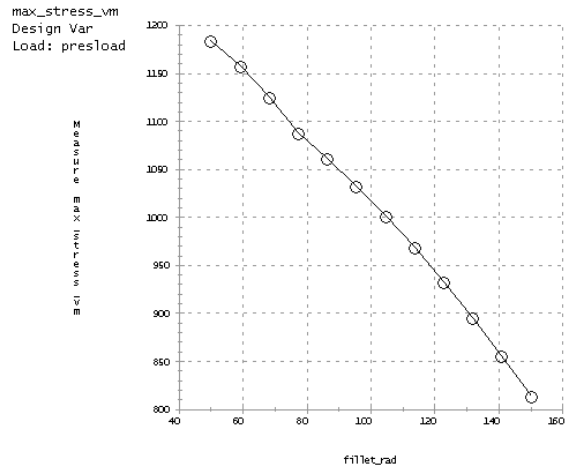


Figure 8 Variation of Von Mises stress with fillet radius in shell model

Example #3 : Design Optimization

This capability of Creo Simulate is really astounding! When a model is created, some of the geometric parameters can be designated as design variables. Then Creo 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). Creo searches through the design space (the specified ranges of the design variables) and will find the optimum set of design variables automatically!

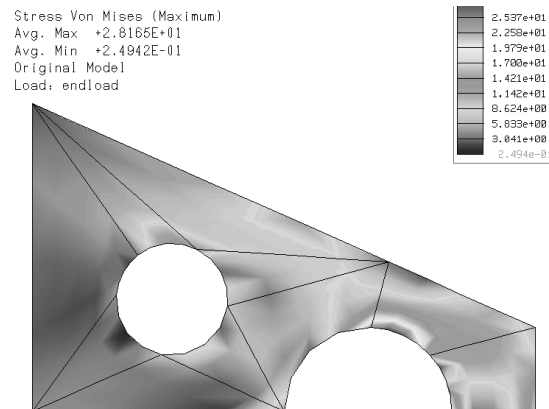


Figure 9 Initial Von Mises stress distribution in plate before optimization

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 constraint, 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?

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.

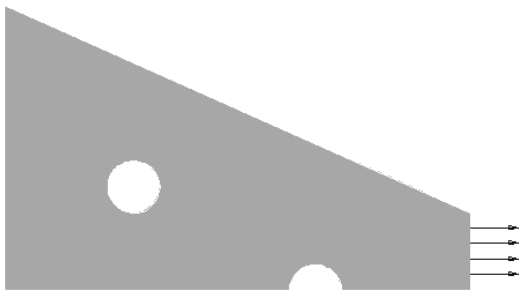


Figure 10 Minimum values of design variables

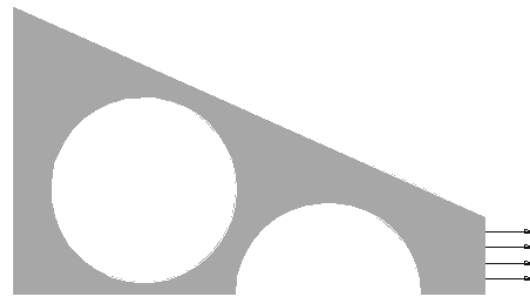


Figure 11 Maximum values of design variables

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 Creo Simulate 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). Creo Simulate allows you to view the shape change occurring at each iteration.

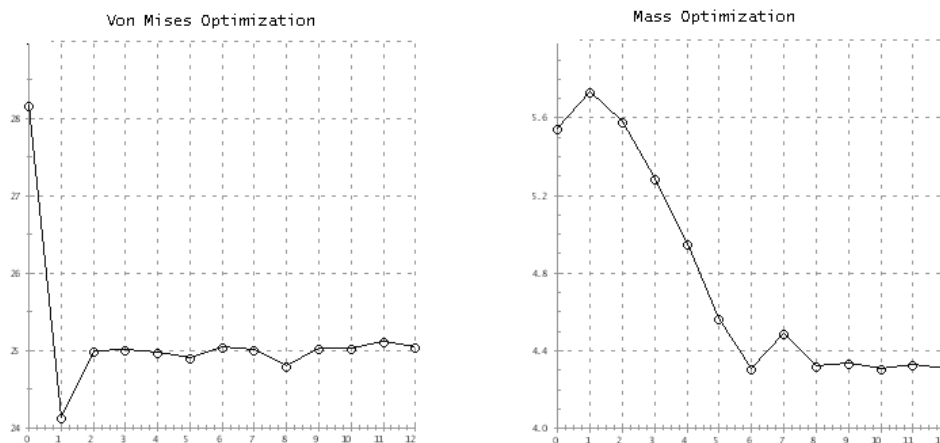


Figure 12 Optimization history: Von Mises stress (left) and total mass (right)

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.

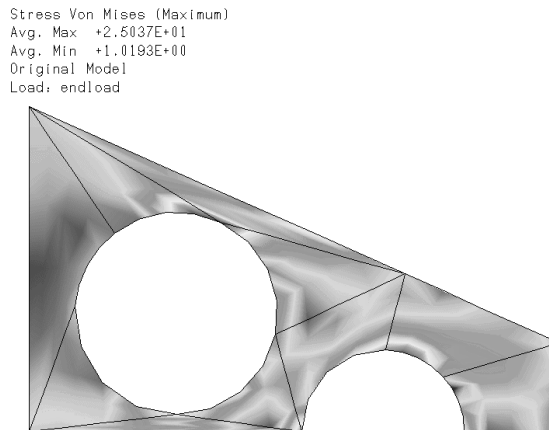


Figure 13 Von Mises stress distribution in optimized plate

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)
- temperature variation 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.

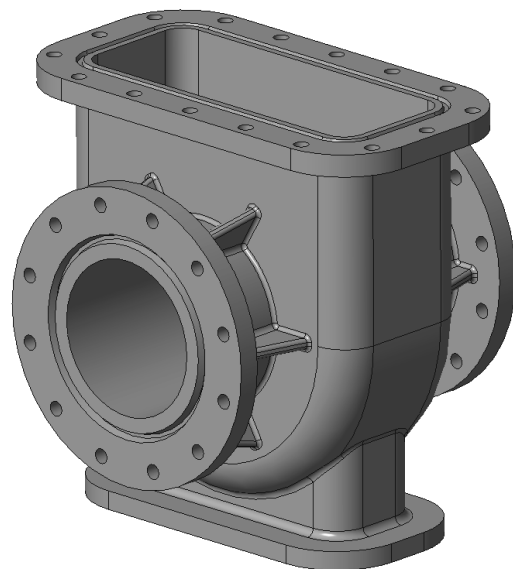


Figure 14 Valve body to be loaded with internal pressure and thermally induced stress

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 the stress analysis and the stresses due to the temperature variation in the model can be determined.

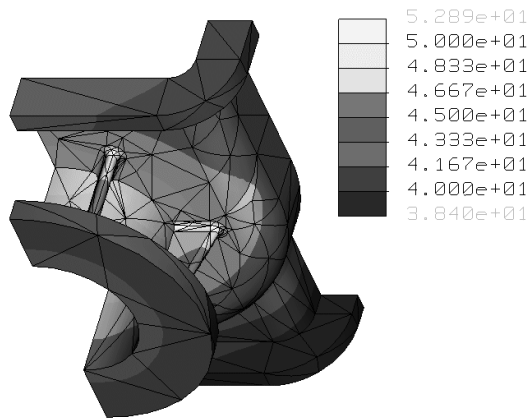


Figure 15 Temperature distribution in the valve body ($^{\circ}\text{C}$)

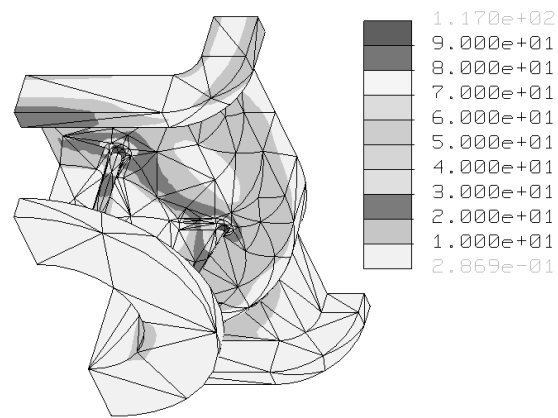


Figure 16 Von Mises stress resulting from combined pressure and thermal load

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. Although Creo can handle radiation heat transfer problems, that type of problem will not be dealt with here.

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 licenses⁴, which usually contains some text along these lines: “The supplier of the software will take no responsibility for the results

⁴ Or, for example, see the Disclaimer at the front of this book.

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. 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 three axes in 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 somewhat farther in the “experience” direction. Primarily, however, this tutorial is meant to extend your knowledge in the “Creo software” direction. Readers who have already moved out along the “theory” or “experience” axes will have to bear with us (we will not cover a lot of theory, and the problems we will look at are very simple) - at least these lessons should help you discover the capabilities of the Creo Simulate software package.

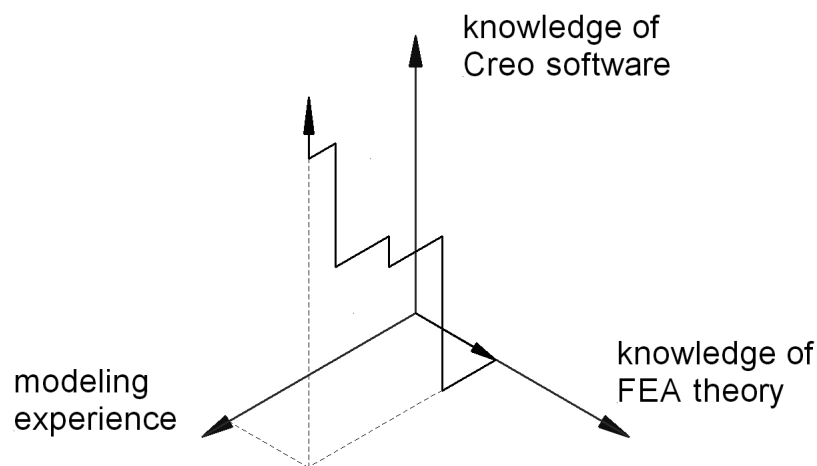


Figure 17 Requirements for Creo users and the Tutorial path

Some quotes from speakers at an FEA panel at an ASME Computers in Engineering conference in the early 1990s 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 and it is tempting to infer that

improved technology yields improved results. 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 is 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 extend your design capability, not define it.

Tips for using Creo Simulate

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 (and think about!) 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***.

Creo incorporates an "object-action" operating paradigm. 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.

The interface for Creo is based on a ribbon style. As mentioned above, it is assumed that users are familiar with the Creo ribbon interface style. Generally speaking, the ribbons are laid out to follow the work flow. At any stage in model development you will likely find all commands on the same ribbon. In the following, commands will be identified by group and command name (and sometimes icon), with new ribbons identified as required. Any menu commands grayed out are unavailable for the current context. Otherwise, any menu item is available for use.

As of Creo Simulate 4.0 and as part of the object/action command structure, whenever something is selected in the graphics window (geometric entity like surface, edge, or curve, or a modeling entity like a constraint or load icon) or model tree, a mini toolbar menu opens at the cursor location. This is context sensitive and complete enough that you may not need to use the ribbon at all.





Many operations can be launched by clicking and holding down the right mouse button (RMB) on an entry in the model tree or in the graphics window. This will produce a (context sensitive) pop-up menu of relevant commands.

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 “>”. Selections or settings made within a single menu are separated with “|”.

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, for those not familiar with the Creo interface, here are a few hints about using the mouse. 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 Creo Parametric will be quite comfortable with these mouse controls.

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 item selected	launch context-sensitive pop-up menus

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 Creo Simulate is different from other commercial packages.

Questions for Review

1. In Creo Simulate language, what is meant by a “design study?”
2. What are the three types of design study that can be performed by Creo Simulate?
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 Creo Simulate 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.
2. Find out what role computers played in the Apollo missions that landed the first astronauts on the moon on July 20, 1969. (Hint: a modern smart phone probably contains more computing power than was available to the design engineers of the day!)
3. a) Simple analytical models result in the following formulas for simple states of stress:

$$\sigma = \frac{P}{A} \quad \sigma = \frac{My}{I} \quad \tau = \frac{Tc}{J} \quad \tau = \frac{VQ}{It}$$

Identify each of these formulas and list the assumptions that were used in their derivation. For each formula, **which of these assumptions is most likely to be violated in a real-world problem?** Which assumptions do you think are also used in the finite element analysis of a 3D solid model?

- b) What is the formula for Von Mises stress? Where does this come from, and why are we interested in it?
4. What is meant by the *principle of superposition*, and what does it have to do with stress analysis? (We will need this in Chapter 4.)
 5. Find out something about the history of the development of FEA. When was it “invented” and what kinds of problems was it originally used for?