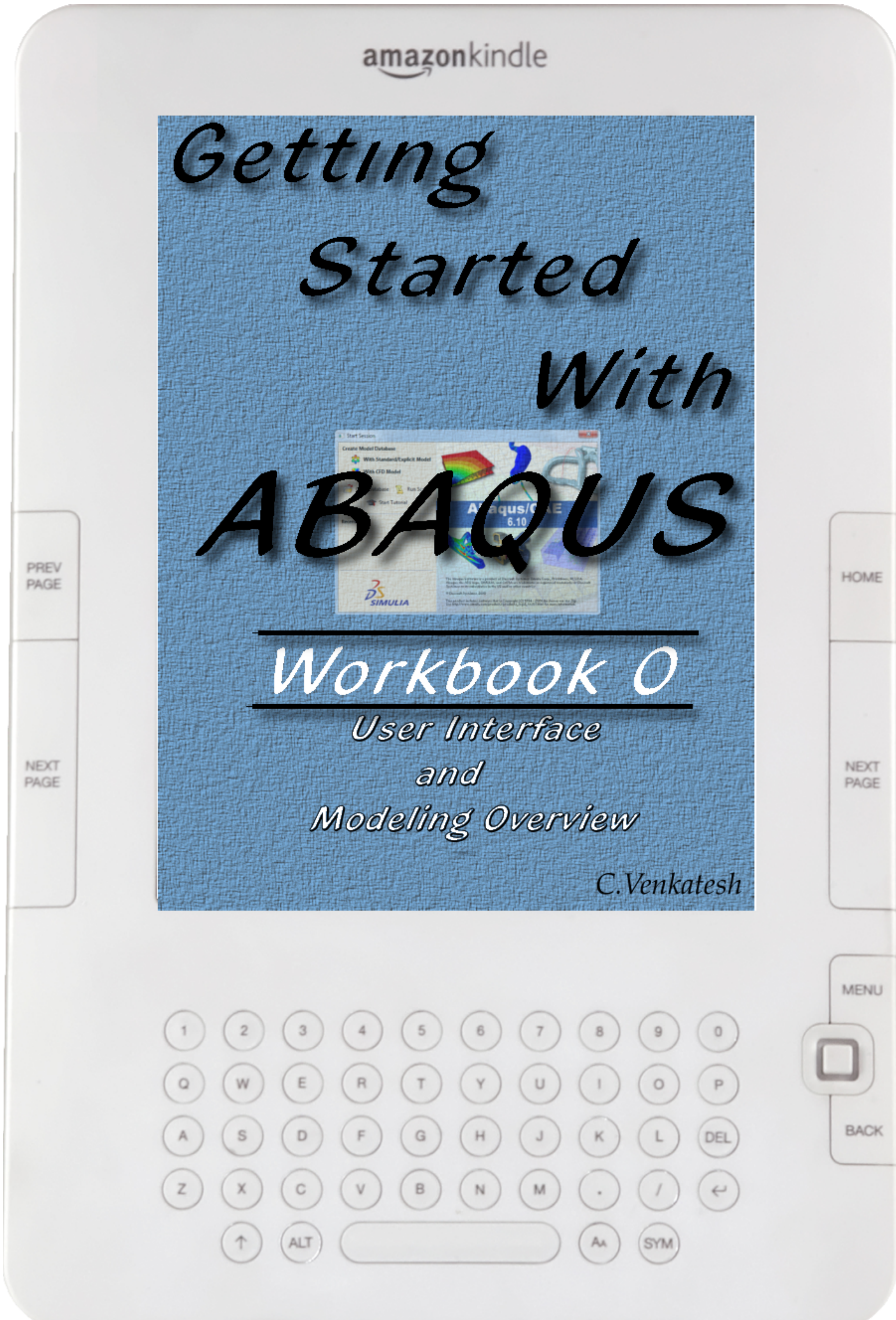
http://www.kfourmetrics.com/



KFourMetrics

PDF vs. Kindle

While the book itself is designed for Amazon's Kindle, this PDF "preview" is intended for readers who haven't used Kindles. If you already have a Kindle, try using the "mobi" preview file (**here**), or "look inside" the book at Amazon.com (**here**).

The reading "experience" can be markedly different. In the first place, what you see on the PDF is not quite what you will see when you read the book using either a Kindle or a Kindle-app for the device of your choice.

Will this matter? If you haven't used a Kindle yet, and are used to working with PDFs on a laptop or desktop computer, it might. Consider these comments from two readers of one of the books in this series:

From a reader who prefers PDFs on a PC:

> "[It] was not as convenient as a continuous PDF. Normally (as far as possible) it is best to have the picture and explanation in the same page. So I had to do back and forward many times to read about the image I see in a page. I saw a lot of white space which I think is better to use in some way or the other."

From a reader who used the Kindle app on an Android device and on a PC:

> "I think the Kindle for PC is a little inconvenient since the pics/text are reformatted depending on the screen size. [F]or instance, if I expand the Highlight/Notes Column at the left side of the screen, the images would get a little scattered. Looks far better on smaller screens (read the doc on a 4.3" screen mobile phone, looks great)"

This PDF is laid out to resemble what you can expect to see on a Kindle, but does nothing to reproduce the effect of changing the font size, zooming, etc. that you can do with Kindle or Kindle-apps on different devices. If you want to experiment with the Kindle-experience, you can download the relevant software from Amazon.com – you do *not* need to purchase or own a Kindle.

Prefer a PDF?

We currently sell this book only on the Kindle. If you're interested in another format (PDF, other eBook, etc.) **drop us a line** and we'll see what we can do.

http://www.kfourmetrics.com/

TABLE OF CONTENTS

* * *

KFourMetrics

http://www.kfourmetrics.com/

Getting Started With Abaqus

Workbook 0

User Interface and Modeling Overview

by

C.Venkatesh

KFourMetrics

Preface

These Workbooks consist of a series of examples presented as a mixture of text and images. They're intended to be used along with the "theory" book in this series - **Getting Started with Abaqus - Essential Theory** - and are therefore deliberately low on text and rich on images. The workbooks are self-contained, but you may want to read the "theory" if you're new to the finite element method itself. The other Workbooks are listed in the references section at the end of this book.

The general presentation approach I have used is to write a few lines of text then follow these up with one or more images. This approach works on all Kindles, even the ones with the smallest black-and-white screens. It certainly looks nicer on color devices (such as *Kindle for PC*).

Abaqus' interface, of course, looks best on a large screen - and Kindles, of course, have small screens. The convenience of buying and reading a book on the Kindle is of no use if the images are scaled down to the point of illegibility. To get around this, each scaled-down "large image" is immediately followed by a "clipped" image of the highlighted area. This seems like the best solution to the problem. It does take you an extra click to move to the next page, but is worth the added legibility.

Why not drop the larger images and retain only the "zoomed in" ones? Remember that if you're reading this book, you're new to *Abaqus/CAE*. And one of the problems with using a new software interface is that you can't always locate that pesky menu. The larger image, then, serves to show you where the command / menu is located on the overall screen. The zoomed-in image makes it a little more legible.

In some cases, though, you will find (at least on the 6" Kindles) that you will need to "zoom-in" even on the zoomed-in image. To do this move the Kindle's cursor to the image and click on it for the Kindle to rotate the image by 90-degrees. Then use the Kindle's **Back** button to restore the earlier view and orientation. I've tried to preempt this by flipping some images in the book itself, which means you have to rotate either the Kindle or your neck, but I figured this is easier than the clicks-to-zoom-in-and-back. It's harder to rotate a desktop or laptop computer, of course, so this does make things a little worse on those devices.

Finally, the Kindle lets you select your font size. At larger font sizes, the captions for some images spill over to the next page - and in some cases, are entirely on the next page. Please keep this in mind when reading the book.

http://www.kfourmetrics.com/

I recommend that you page through the sets of images more than once, then return to the preceding text and re-read it. This is not a very easy way to read on the Kindle, which is designed more for sequential paging back and forth. I've used a number of hyperlinks to make it easier to jump around between pages. You will probably use the Kindle's "Previous / Next" section controls a lot more than with a "serial" book.

Your goal, of course, is not just to reproduce the illustrated procedures on Abaqus. You should be able to extend the documented procedures to your own scenarios. Note that all the examples presented in this series can be worked out using even the student version of Abaqus.

The models used in this book can be downloaded from www.kfourmetrics.com – email the author if you have any trouble finding the files or using them.

1: Some Definitions

These are intentionally brief, and are included as quick reminders on some common terms that you'll encounter as you work with Abaqus. If you're new to the finite element itself, chances are you'll find these terms incomplete. To fill the gap, look up either the "theory" book that accompanies these Workbooks or one of the references listed at the end of this book.

The Importance of Modeling

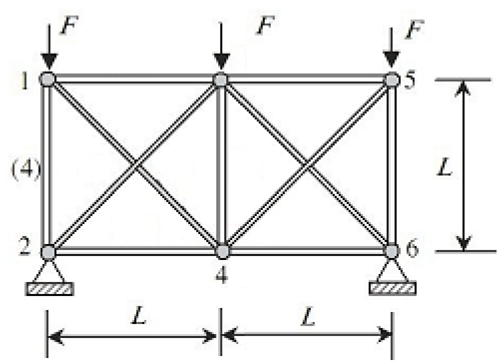
Remember that there are many, many methods to model systems.

Most undergraduate courses do not cover continuum mechanics based approaches. Coursework usually covers only discrete methods (of force equilibrium) and basic stress-strain equations without covering the differential-equation forms. In order to use Abaqus' documentation, it will help if you draw the equivalence and the difference between the two: Abaqus' documentation uses differential equations extensively.

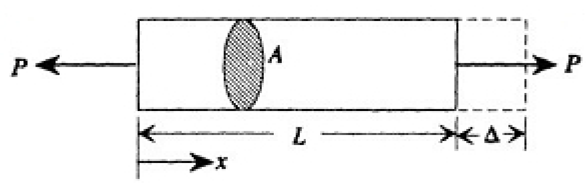
FEA is a way to construct mathematical models in which differential equations represent the behavior of the system. To get to grips with analysis, you need to understand at least some of applied mathematics and computer science, in addition to the relevant engineering discipline.

The next images show 3 possible models of a typical "framed" structure. The first model is of the complete frame. The second is of a subsystem – a single bar. The third is of the region around a riveted joint.

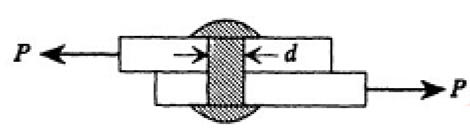
Which is correct?



The "system" - the complete frame



A "sub-system" - a single bar



Another "sub-system" - the neighborhood of a riveted joint

The answer of course, is that none of them is “correct” in the absolute sense. It depends on what you’re interested in.

Next, since we are viewing the FE method as a way to solve differential equations, it's useful to remember (for any given physics) the variables of state, the quantity that is conserved (the flux), the conservation law and the constitutive (i.e. material behavior) law.

Physical Problem	Conservation Principle	State Variable	Flux	Material Modulus	Source	Constitutive Equation
		u	σ	k	f	$\sigma = -ku'$
Deformation of an Elastic bar	Equilibrium of Forces (Conservation of linear momentum)	Displacement	Stress	Young's Modulus	Body Forces	Hooke's Law
Heat Conduction in a rod	Conservation of Energy	Temperature	Heat flux	Thermal Conductivity	Heat Sources	Fourier's Law
Fluid Flow	Conservation of linear momentum	Velocity	Shear stress	Viscosity	Body Forces	Stokes' Law
Electrostatics	Conservation of electric flux	Electric Potential	Electric flux	Dialectric permittivity	Charge	Coulomb's Law
Flow through porous media	Conservation of mass	Hydraulic head	Flow rate	Permeability	Fluid Sources	Darcy's Law

From "Finite Elements: An Introduction (Volume 1)" by E.B.Becker, C.F.Carey and J.T.Oden

Equivalence between different "physics"

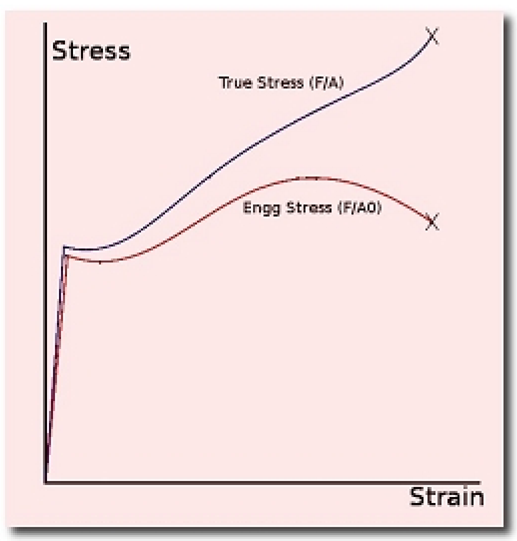
The next few pages contain some definitions related to stress and strain. The nomenclature can easily lead to confusion: for instance, what's the difference between tensile strength, yield strength and ultimate strength? And what in the world is a Proof Stress?

If you receive a problem statement in engineering, it's a bad idea to work under the assumption that all parties agree on all nomenclature. It's far better to have your interpretations of available data confirmed.

Tensile stress: "the tensile force per unit area of the original cross section within the gauge length, carried by the specimen at any given moment".

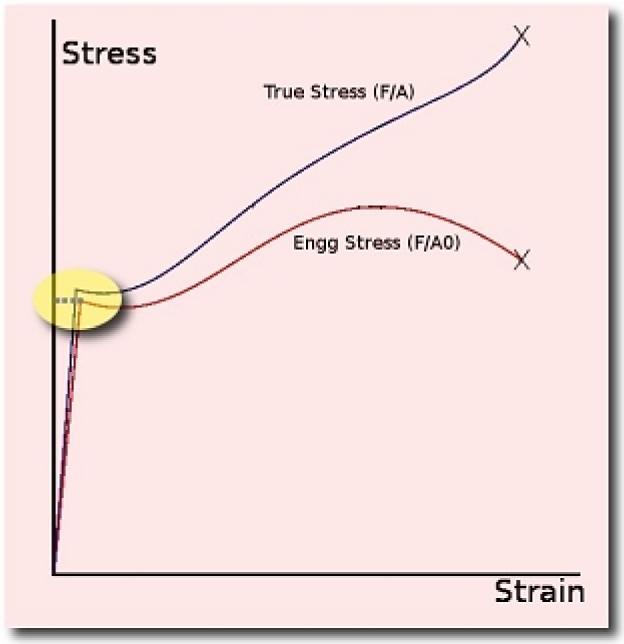
Remember the difference between true and engineering stress: the former uses the "instantaneous" cross-sectional area while the latter uses the initial area.

Subsequently we will see how to convert data into forms that are consistent with Abaqus.



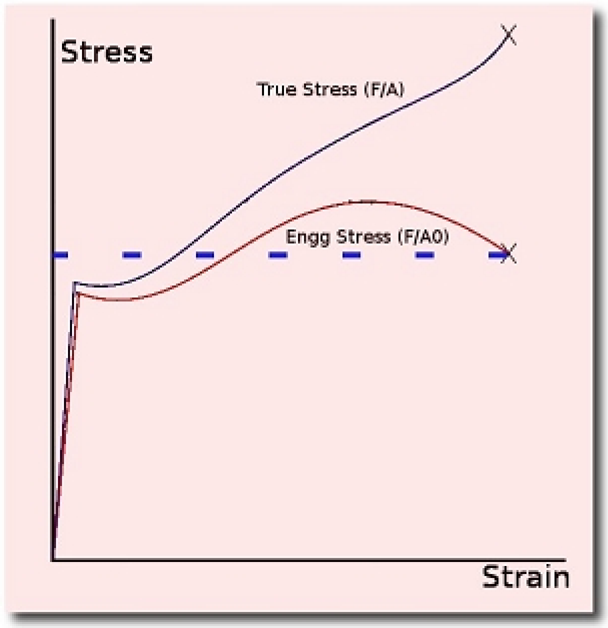
True and Engineering stress-strain curves

Tensile Stress at Yield (Yield Stress): “the first stress at which an increase in strain occurs without an increase in stress”



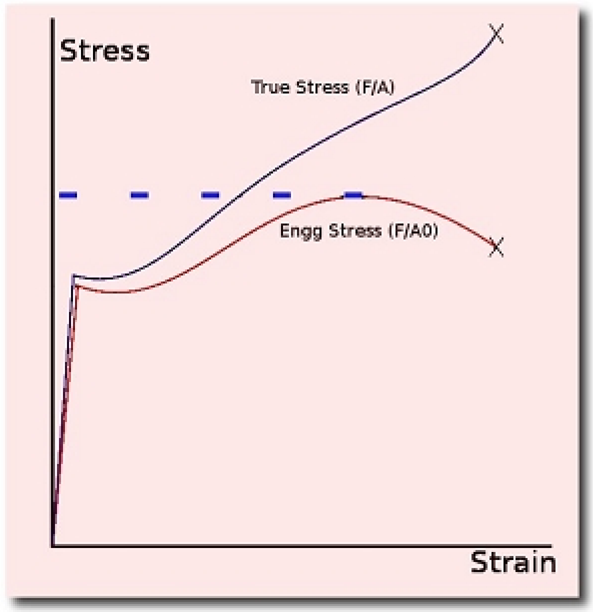
The yield point - yield stress

Tensile Stress at Break (ultimate stress): “the tensile stress at which the test specimen ruptures”.



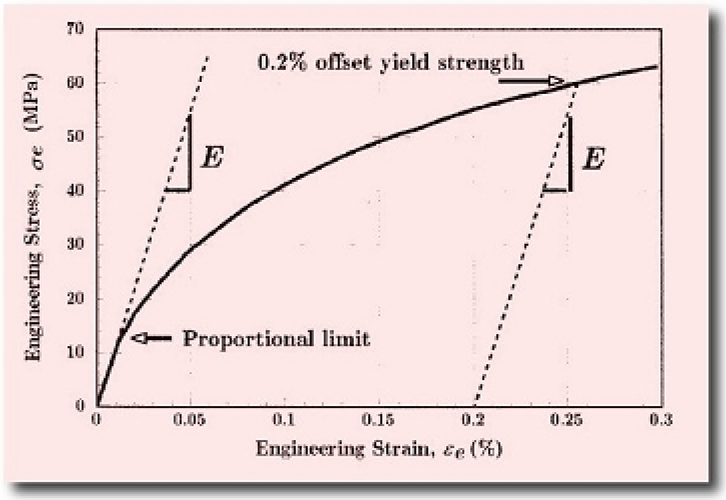
The point of rupture - ultimate stress

Tensile Strength: "the maximum tensile stress sustained by the test specimen during a tensile test"



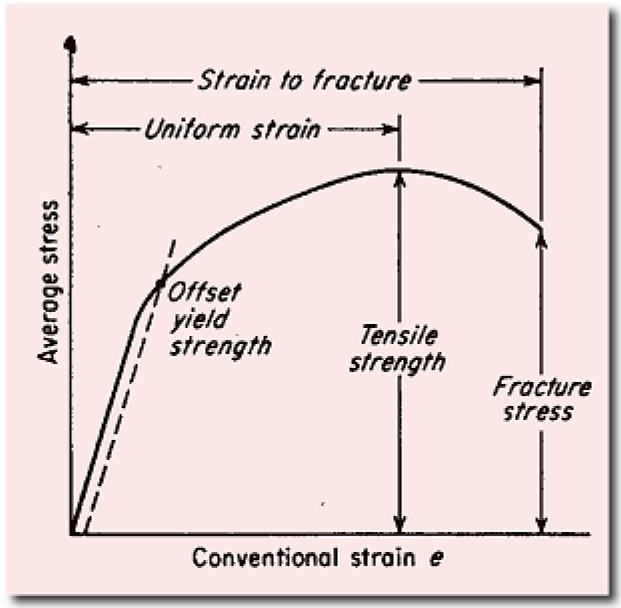
Maximum stress sustained - **not** the same as the permissible stress!

Tensile Stress at x% strain: "the stress at which the strain reaches the specified value x expressed as a percentage. It may be measured, for example, if the stress-strain curve does not show a yield point. In this case x must be defined either in the relevant product standard or as agreed upon by the interested parties"



Permissible (or "offset") stress

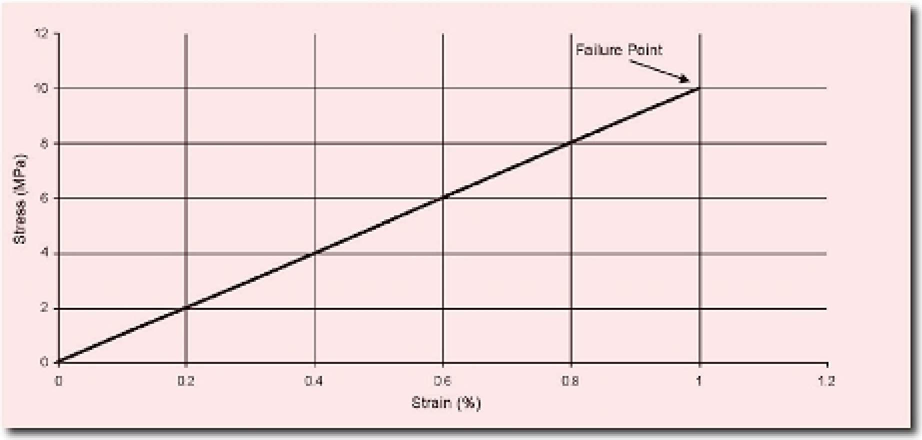
0.2% proof stress: Not all materials show a definite yield point, so an “offset” is often used to define the yield strength. An offset of .2% strain is widely accepted.



"Proof" stress

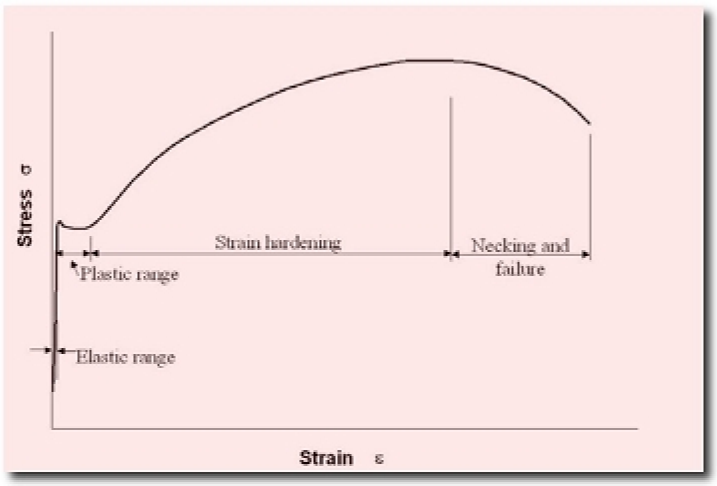
Now a quick recap of the difference between elastic and plastic materials.

This is the stress-strain curve for a **perfectly elastic material**:



Perfectly elastic material. Note the absence of a yield point.

Next, look at the curve for a carbon-steel. Only the initial part is close to a "perfectly elastic" material.



A more familiar stress-strain curve. Do you think the y-axis is engineering stress or true stress?

Discontinuities in data and their impact on the FE Solution

When we draw free-body diagrams, we frequently use representations without being fully aware of the implications. For instance, the use of point-loads, step-loads, and sharp corners is widespread. This can hurt the FE method. Why?

Because discontinuities in functions are inherently troublesome from a calculus-perspective (and we have chosen to view FEA from a differential-equation perspective): such functions may not be defined “properly”, may not be differentiable, may not be integrable, etc.

The nomenclature of continuous functions is a subject in its own right: see http://en.wikipedia.org/wiki/Smooth_function#The_space_of_Ck_functions if you’re interested in this aspect.

Here’s the minimum you should know:

- A function is **continuous** at any point if the left limit = right limit = value of function at that point
- A differentiable function is one whose derivative exists (it may, of course, be differentiable only within a given domain)
- Not every continuous function is differentiable, though every differentiable function is continuous
- A function’s **degree of continuity** is C_i if its "i"th derivative can be evaluated
- If it is $\mathbf{C_0}$, that means the function is continuous
- If it is $\mathbf{C_1}$ that means the function’s derivative is continuous
- The **Jacobian Matrix** is used to represent the partial derivatives of functions that have more than 1 independent variable - the determinant of the matrix (often referred to as "the" Jacobian) is used to check whether the function is differentiable.

If your data is discontinuous, you should view the results in the immediate neighborhood with some suspicion - and should probably "smear" (or average) the result over the immediate neighborhood. Unfortunately, there's no ready formula to define the "immediate neighborhood". Most engineers rely on St.Venant's principle, which is valid only for linear problems.

Common Errors in FE Models

Before we start on the modeling exercises, remember that your objective is to build an "efficient" model. In other words, a high-quality model is one that is "efficient". By this yardstick, how can you measure the quality of the model?

On the one hand, you want to minimize the time and cost: of building mode and of evaluating the "solution".

On the other hand, you want acceptable accuracy: that is, similarity to the real-world situation.

Also, you want to minimise the sensitivity to errors in data. If you can't do this, you should at least be able to quantify the sensitivity.

The most common errors in building FE models?

- Too many elements = Too much time to solve
- Too few elements = Not representative of physics
- Wrong elements = Not representative of physics
- Wrong elements = Incorrect assemblage of elements, which in turn leads to the wrong equilibrium
- Misrepresentation of conditions on boundary = Wrong loads, restraints, initial conditions

Consistent Units

Finally, Abaqus is units-agnostic. You **must** ensure that various data are consistent – size of geometry, elastic constants, forces, density, etc.

This is probably the most common error beginners make, and it's pernicious because problems in dynamics behave differently than problems in statics. We won't go into it here: the "theory" book contains more details, as does the Workbook on Mode Shapes.

But please do pay attention to this aspect when building models for Abaqus. If it's hard to teach an old dog new tricks, it's even harder to "unlearn" a bad habit. It's far, far better to get into the habit, right from the start, of using consistent units consistently.

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2/in^4

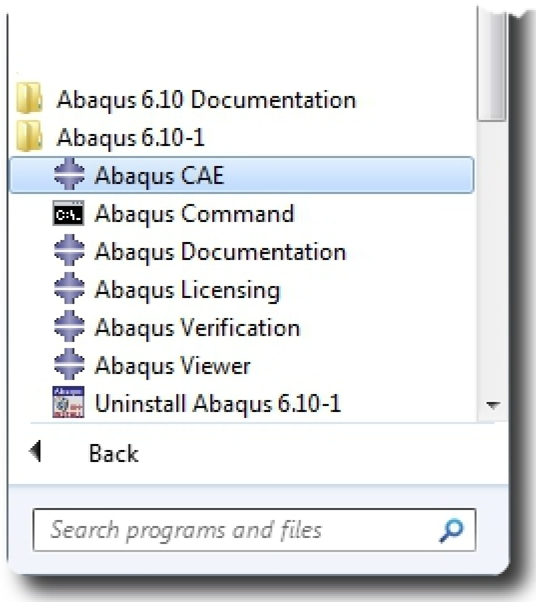
Some options for consistent units

That's it for this brief review of some concept. For more on "theory", see the companion book **Getting Started with Abaqus - Essential Theory** or one of the references at the end of this book.

Now, onto *Abaqus/CAE* itself.

2: Start Abaqus and set the "start in" folder

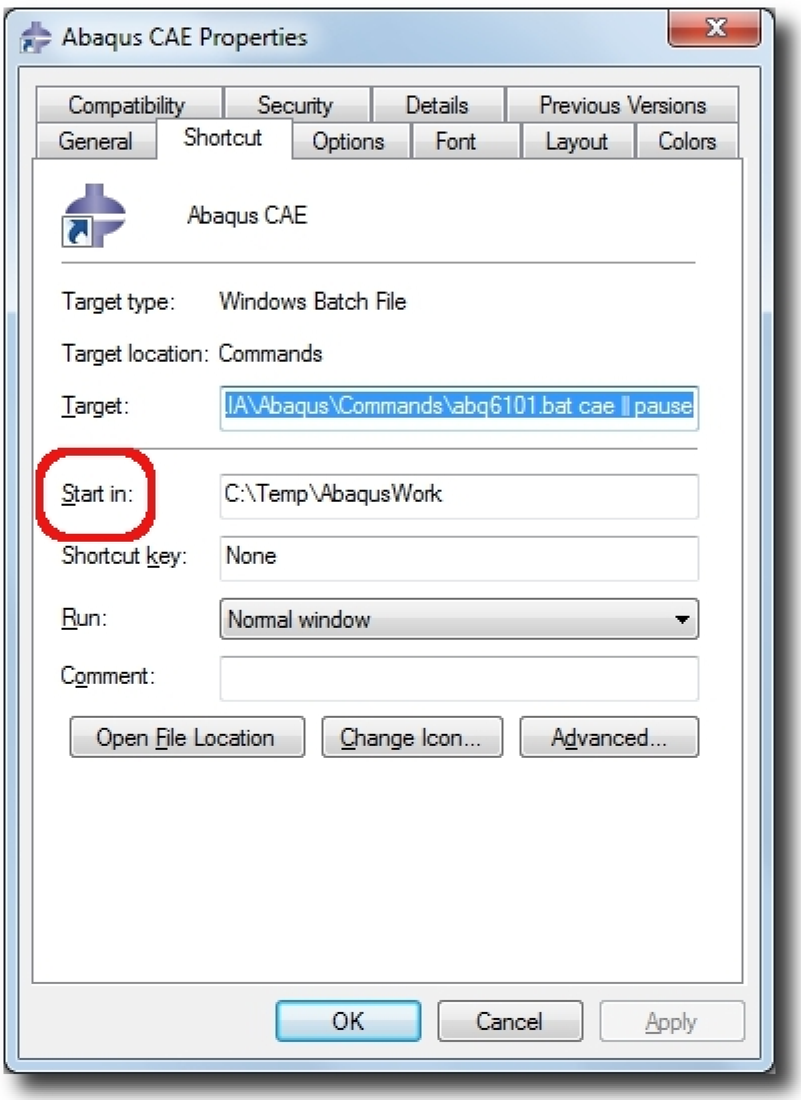
To start Abaqus, use your Start menu. On Windows 7, this is what you should see:



The "start" menu on Windows 7

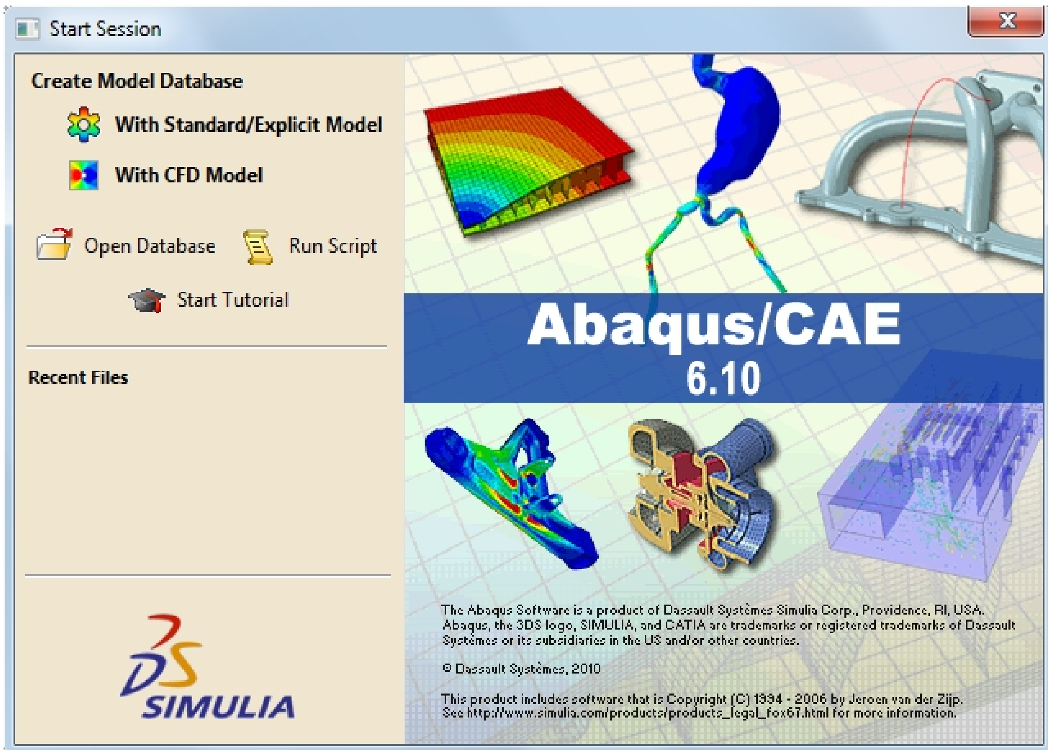
To change the “default” folder where *Abaqus/CAE* stores its various files, right-click on the shortcut, select its "Properties", and change the “Start In” folder.

It's a good idea to do this since, as we'll see later, FE analyses can create a pretty large number of files.



The "start-in" properties of the shortcut

You’ll know that Abaqus has “started” when you see the screen below:



The "splash" screen

3: The User Interface

The principal objective of these Workbooks is to show how to go from an assembly to the simulation results.

In this Workbook we will cover the overall modeling procedure and the logical organization of the user-interface of *Abaqus/CAE*. In the other Workbooks, we will go over analysis-specific modeling procedures, review the organization of files, and look at post-processing and error estimation.

Recall that the steps involved in creating a finite element model are

- Specify the region (or domain) of space (and time, if things change with time) that's of interest
- Create an FE model to reflect the behavior of regions in the domain by selecting appropriate elements, materials and sections
- Apply loads and restraints
- Solve
- Interpret and report

Should you do the first task (creation of the domain) within *Abaqus/CAE*, or create it using a CAD application?

That’s debatable.

The answer depends on the problem at hand, the other software available to you, the time available and your skill / comfort with *Abaqus/CAE* and any other software available. Depending on the situation, you will have to choose between the following:

- create it entirely within *Abaqus/CAE*
- import the “final” CAD data (i.e. ready to mesh) to *Abaqus/CAE*
- import CAD data and modify in *Abaqus/CAE* till it's ready-to-mesh

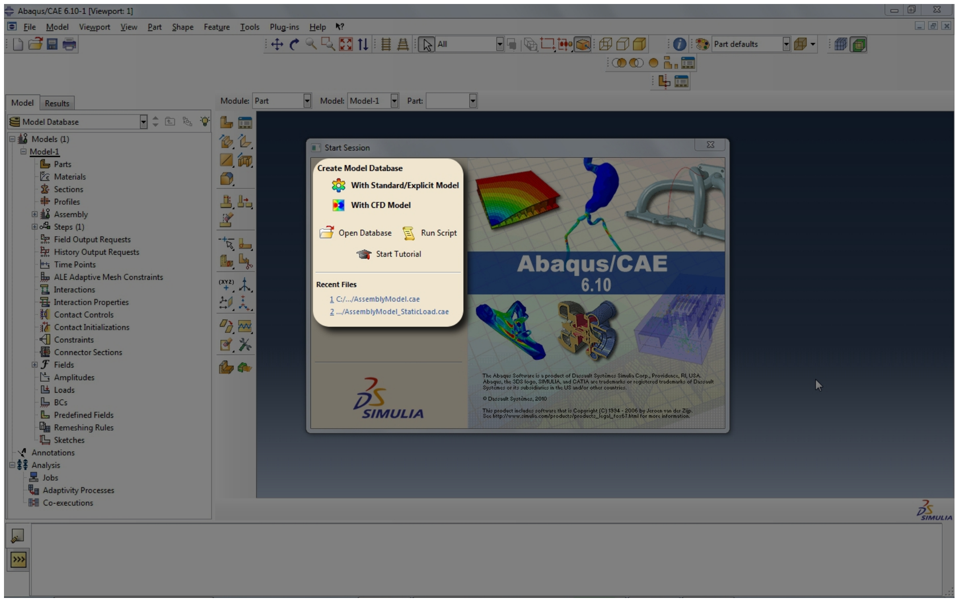
Create / Retrieve files

When *Abaqus/CAE* starts, the first thing you do is choose whether to create a new file, open an existing file, or select a recently opened file. Of course, once a file is open (whether it’s a new file or an existing file) you can “import” another file into it if you wish.

The "splash" screen that you see when you start Abaqus, shown in the next image, contains icons that you can use to either create a new database (i.e. file) or retrieve an existing one.

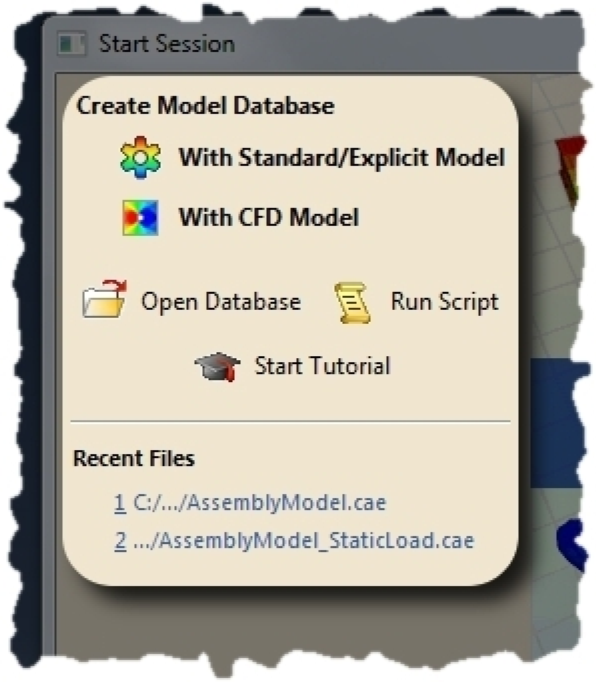
As the next close-up image shows, there are other options too - you can run a "script" (a program that's somewhat like a macro - see the references at the end of this book for more information on scripts for Abaqus), or you can start doing an "in-built" tutorial.

Note that the create-database option itself is really 2 options: create a file using Abaqus/Standard or Abaqus/Explicit, or using Abaqus' CFD capabilities. If you're not clear which you should use, look up the "theory" book in this series (**Getting Started With Abaqus - Essential Theory**) for some guidelines and advice.



Options on the "splash" screen

Close-up image on the next page



To select an option, click on the corresponding icon or link

Close-up of image on the previous page.

http://www.kfourmetrics.com/

KFourMetrics

References

A First Course in Finite Elements
J.Fish, T.Belytschko, John Wiley and Sons

This is a comprehensive text-book and is a good addition to your reference-shelf. Chances are you won't read it completely unless it's a text-book for a course you're doing. But if you do choose to pursue a career in FEA, chances are very good that you **will** end up reading every bit of it sooner, rather than later.

An Analysis of the Finite Element Method
W.G.Strang, G.J.Fix, Wellesley-Cambridge Press

While not directly related to Abaqus, this is an outstanding book by an outstanding teacher (you can, and should, view Dr.Strang's video lectures off the MIT website). Immensely readable, invaluable, well worth the price of purchase and the time to read it.

Python Scripts for Abaqus. Learn by Example
G.Puri, Wellesley-Cambridge Press

Considering I'm writing for Abaqus-beginners, why am I recommending a book on scripting (i.e. customising or programming) Abaqus? The book does require that you know a fair deal about Abaqus. However, the author has created some nice videos on some common applications. You could do worse than check these out.

The 3DS "Community" - http://iam.3ds.com

Licensed users can access paid support, but how about academic users or others? This is a recent (June 2011 onwards) attempt by Dassault and / or Simulia to address such needs. You can use any email address to create an account, then (in theory) tap into the wisdom of the crowds. Once you login, your browser is redirected (at the time this book was published) to https://swym.3ds.com.

There are some tutorials, etc. so perhaps you'll find what you want. But I suspect the password-protected approach is too heavy-handed for light-fingered internet users

The Yahoo-group for Abaqus

This is a relatively lively (by CAE standards) group, averaging a few posts a day. It doesn't seem to have any affiliations with Dassault, but does have some pretty erudite members. However, on the downside, many of the questions tend to be advanced: well beyond the profile of the intended reader of this book. Also, perhaps because of their erudition, some of the gurus can be quite acid at times. "Stupid" questions are sometimes snapped at, sometimes ignored. But if you do your homework **and** establish this fact when you ask your question, chances are you'll get an answer.

Finally, there are the other books in this **Getting Started With Abaqus** series (clicking on a link takes you to the corresponding Amazon page):

- Essential Theory
- Workbook 1: Linear Static Analyses and Basic Mesh Generation
- Workbook 2: Thermal and Thermo-mechanical Analyses
- Workbook 3: Mode Shapes
- Workbook 4: Non-linear Analyses
- Workbook 5: Explicit Analysis

Copyright

KFourMetrics

About the Author

C.Venkatesh lives in Hyderabad, India. His email address is
venkatesh@kfourmetrics.com