

Learning to Work with ANSYS Workbench

Tutorials help both novices and experienced users understand the ins and outs of this simulation platform.

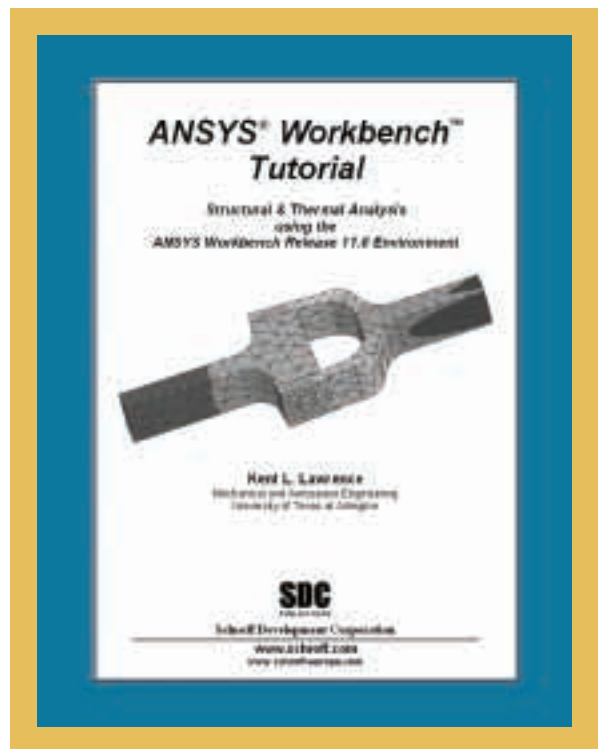
By Rajesh Bhaskaran, Director, Swanson Engineering Simulation Program, and Professor, Mechanical and Aerospace Engineering, Cornell University, New York, U.S.A.

In my work at Cornell University, I have been using traditional ANSYS Mechanical software for the last seven years to teach students the basics of finite element analysis (FEA) applications. Recently, a few passing encounters with the ANSYS Workbench platform — mostly through students returning from industry stints — piqued my interest in the sleek, modern interface. Two new books published by Schroff Development Corporation helped me cut my teeth on ANSYS Workbench release 11.0: *ANSYS Workbench Tutorial* by Kent Lawrence and *ANSYS Workbench Software: Tutorial with Multimedia CD* by Fereydoon Dadkhah and Jack Zecher.

ANSYS Workbench Tutorial

Lawrence's book kicks off with three chapters on solid modeling using the ANSYS DesignModeler package. The first few tutorials cover solid model creation through extrusion, revolution and sweeping of an L-shaped cross section. Modeling an assembly is demonstrated through the creation of a three-part clevis yoke assembly. It is well worth the time it takes to complete this tutorial successfully, since the book uses this model in a subsequent tutorial on simulation. The solid modeling portion of the book wraps up with exercises on parameters and surface/line models. The ANSYS Workbench interface simplifies working with computer-aided design (CAD) models created in other programs, and Lawrence briefly covers this topic in two separate places. I expect to be referring students to this coverage often.

FEA coverage begins in chapter 4 with an introduction to the ANSYS *Simulation* module. The book assumes that readers already know the fundamentals of FEA, and it launches straight into the tutorials. The first considers the classic problem of a plate with a central circular hole — a favorite of college professors. This tutorial solves a 3-D plate model with finite thickness using solid elements. Results are verified by comparing with published stress concentration factors and plotting estimates of the structural error, a very useful parameter.



The next tutorial considers the case of a square hole, as opposed to a circular one. The effect of the stress singularity at the corner of the square hole is explored through the automated mesh convergence capability in the ANSYS Workbench environment. This powerful feature successively refines the mesh until a selected solution quantity changes by less than a specified percentage. The author explains how to eliminate the stress singularity by adding a fillet at the concerned corner.

A subsequent chapter deals with the analysis of a cylindrical pressure vessel, an angle bracket and the clevis yoke assembly from the solid modeling section. The clevis yoke example introduces the use of contact capabilities.

Lawrence revisits the plate-with-a-hole, pressure vessel and bracket problems, and solves them using a 2-D or surface model. In each case, results for the solid and simplified models compare well. Other tutorials include analysis of natural frequencies, buckling, heat transfer and thermal stresses.

Instructions in the tutorials are terse, so the reader must peruse both the text and menu snapshots to be able to successfully complete the tutorials, as the two are not redundant. In some cases, the menus on my computer were slightly different from the ones shown in the book, even though I was using version 11.0. With some digging around, however, I was able to compensate for these differences and proceed with the tutorials.

ANSYS Workbench Software: Tutorial with Multimedia CD

While Lawrence's book focuses on ANSYS Workbench applications, Dadkhah and Zecher's book integrates discussion of basic finite element modeling concepts with tutorials using the ANSYS Workbench platform. Fundamental concepts considered include stiffness matrices, loads and supports, solid elements, 2-D/axisymmetric models, and mesh refinement/quality. The chapter on stiffness matrices analyzes a system with two springs, so shape functions are not covered.

Discussions about modeling concepts are accompanied by tutorials that focus on illustrating the concepts in ANSYS Workbench applications. For instance, discussion of mesh refinement and quality precedes a tutorial on using convergence to improve results for the normal stress in the fillet region of a T-section. This lets readers see the concepts in action in an actual FEA calculation.

Most tutorials deal with fundamental problems, such as a straight beam, plate-with-a-hole and cylindrical pressure vessel. This allows the authors to check their FEA results against hand calculations or handbook results, which they are very diligent about. This is a sound pedagogical approach to teaching the fundamentals to an FEA novice, and it will help instructors emphasize the importance of verifying FEA results.

Dadkhah and Zecher present tutorials on solid modeling using the ANSYS DesignModeler tool at the start of the book. This material covers some of the same territory as Lawrence's book. FEA chapters include tutorials on 3-D solid elements, plane stress/strain, shell elements, vibration (natural frequencies and mode shapes) and steady-state heat transfer. More advanced concepts, such as contact, assemblies and fatigue loading, are not considered. Overall, the focus is on understanding the fundamentals through solving canonical problems.

There are a few exercises at the end of each chapter that provide opportunities for further exploration and practice. These would work well as homework problems. The book comes with a CD containing videos of each tutorial in AVI

format, as well as model files and a material database used in some of the tutorials and exercises. The book ends with a short chapter on the role of FEA in engineering and issues related to its appropriate use. This chapter raises several issues worth chewing on, such as absolute versus comparative answers, establishing goals for the analysis, interpreting results and performing failure analysis.

In the educational setting, there are two important aspects in learning to use the ANSYS Workbench environment to obtain reliable engineering solutions. First, students need to develop the necessary skills to use the software interface in order to set up and solve a variety of engineering problems. Second, they need to understand the underlying concepts to apply the software correctly in order to obtain validated results. Either book will help students learn the interface, with Lawrence's book additionally addressing more advanced topics such as contact, assemblies and fatigue loading. Dadkhah and Zecher's book has more substantial discussion of the underlying concepts and, thus, is more suitable for the novice. Both books certainly are recommended additions to your engineering bookshelf. ■

For further information on the books *ANSYS Workbench Tutorial* by Kent Lawrence and *ANSYS Workbench Software: Tutorial with Multimedia CD* by Fereydoon Dadkhah and Jack Zecher, contact the publisher, Schroff Development Corporation, at 913.262.2664, or visit www.sdcpublishings.com.

