The International
Journal of Modern Engineering
A Leading Journal For Modern Engineers
Volume 7, Number 2, Spring 2007

Application of ANSYS Structural™ in Structural Analysis and Design

Ali Mehrabian, Ph.D., M.ASCE
mehrabia@mail.ucf.edu

College of Engineering and Computer Science
University of Central Florida
Orlando, FL 32816

INTRODUCTION

Today ANSYS is viewed as a "household item" in many design and research institutions here in the United States and around the world. It is regarded by many engineers and researchers as a modern, robust, accurate, and visually sensible tool to provide solutions for many engineering and scientific problems. By many accounts, ANSYS is also heavily used as an effective teaching tool to demonstrate the robustness and the computational capabilities of the finite element method, both in the classrooms and corporate training sites. Reviewing the entire family of ANSYS products can be the subject of many reviews and can not be presented here. One of the members of the family of the ANSYS products is ANSYS Structural™, the subject of this software review here. This review is intended to be brief, and it only highlights some of the key features of the software. This article is based solely on the authors' professional experience with using this software. Any findings and opinions reported here are those of the author and do not necessarily reflect those of the software's manufacturer(s) and their distributor(s).

GENERAL CAPABILITIES

ANSYS Structural™ is a finite element-based tool that provides a powerful structural simulation software package. The author has used the software in his professional work to perform static, dynamic, nonlinear, and linear structural analysis and design. These are only a few capabilities. The complete list of the capabilities is too long to list here. The reader is advised to refer to the ANSYS website with the URL given at the end of this article. ANSYS Structural™ provides for a variety of studies including rigid or flexible dynamics, from steady state thermal analyses to coupled thermo-mechanical transient studies. Based on the author's experience, it has the capabilities to deliver the highest-quality, most-reliable structural simulation results available in the industry. ANSYS Structural™ is fully complemented by nonlinear elements, nonlinear and linear material

constitutive laws, and plastic material models. It allows simulation of many different types of structures, from the most intricate of structures to the complicated assemblies, by using nonlinear contact functionality. By offering users an intuitive, tree-structured GUI for easy definition of even the most intricate material models, it provides a choice of iterative and direct solvers for optimal results. This is achieved by using supplementary solvers that may be added via the Parallel Performance for ANSYS module.

CAD SYSTEM INTEGRATION

Without any translations, no IGES, and no middle geometry format, ANSYS can use the existing native CAD geometry directly. For over a decade, ANSYS has had native, bi-directional, integration with the most popular CAD systems. Furthermore, it provides integration directly into the CAD menu bar making it simple to launch the ANSYS simulation directly from the user's CAD system. A sample figure for compatibility demonstration is presented in Fig. 1.

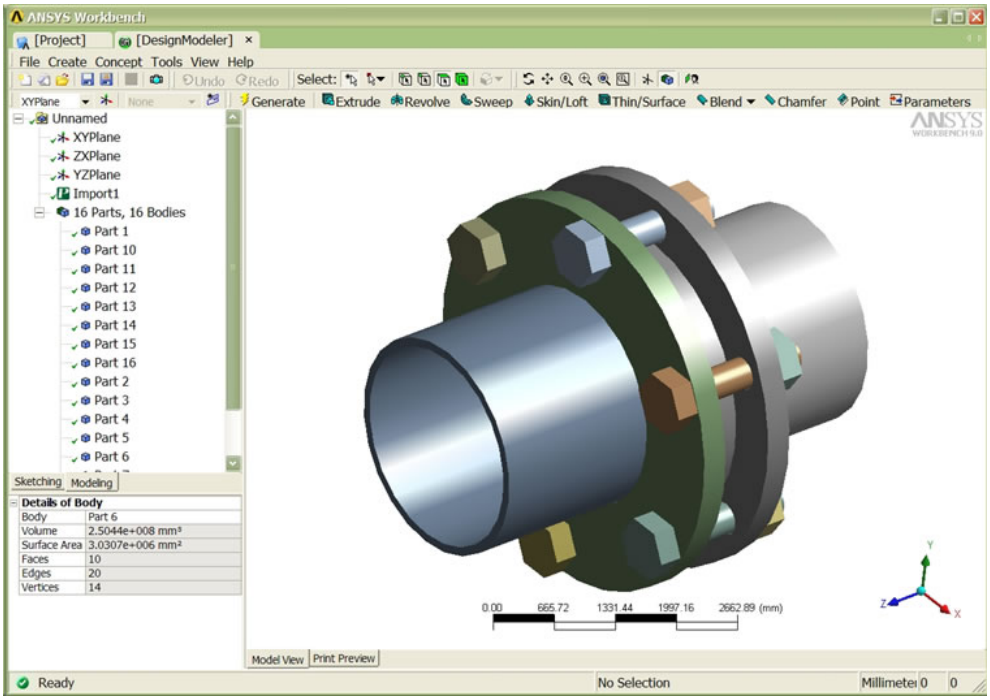

Figure 1: Bidirectional CAD and Geometry Editing

MESHING APPLICATION

The Meshing Application in the ANSYS Workbench environment provides the user with a variety of meshing methods available from the ANSYS meshing software portfolio. On a per-part basis, the Meshing Application can assign different meshing methods. An example is a swept or extruded mesh (prism or hexahedral elements) in a long pipe. The meshing methods available include swept meshing, advancing front with inflation meshing (AFI), patch-independent meshing, hex-dominant meshing, and a number of

others. The meshing application provides a number of centralized services such as virtual topology, which can be applied for all meshing methods. For 3-D meshing, the mesher provides for automatic surface and volume meshing. In regions of high surface curvature, the mesh can be automatically refined in order to maintain the necessary geometric resolution. This control, which can be applied globally and overridden locally, greatly reduces the need for manual specification of mesh spacing. 3-D proximity detection automatically refines the mesh in areas where geometric features are in close proximity and result in small spaces within the volume. This enhances robustness and reduces user input. Figure 2 demonstrates a snap-shot sample of the automatic meshing with manual option screen.

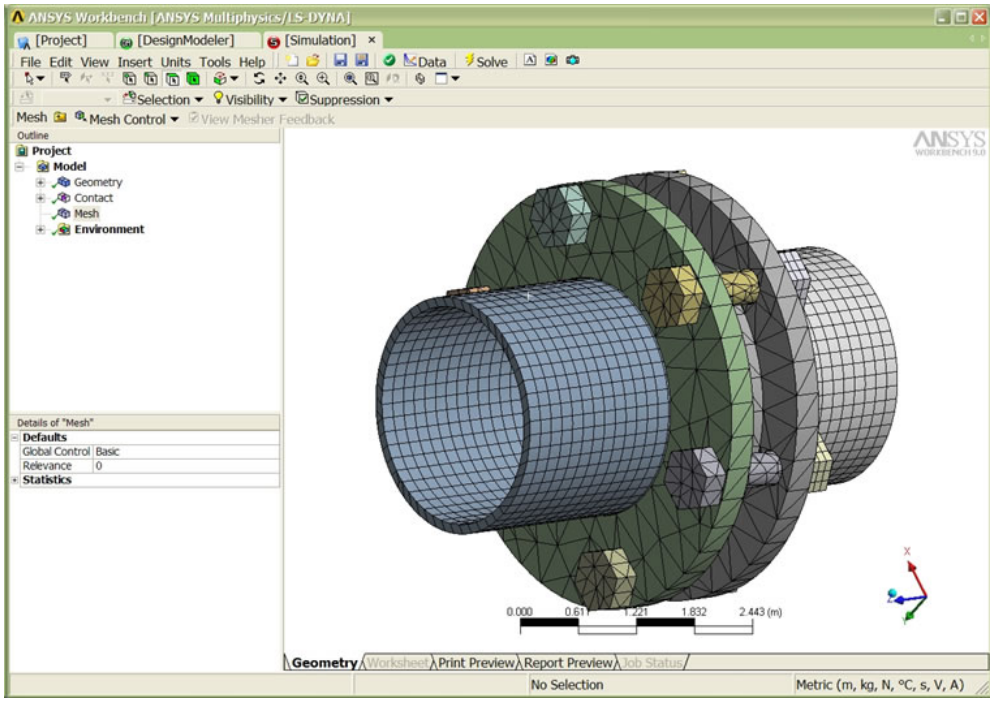

Figure 2: Automatic meshing with Manual Option

PARAMETER AND DIMENSION CONTROL

The ANSYS Workbench Environment uses a unique plug-in architecture to maintain compatibility and association with the CAD systems for solid and surface models. This allows the user to make design changes to the CAD model under consideration without having to reapply loads and or supports. The user can either pick the CAD dimension to change directly, or enhance the design iterations with the Parameter Manager. The ANSYS Parameter Manager in Workbench provides an easy way to set up multiple design scenarios by simply filling out the Parameter Manager spreadsheet and automatically updating the geometry, running multiple solutions and allowing a more efficient simulation.

UNIQUE REPORT GENERATION TOOL

ANSYS Structural™ robustness allows the user to customize the simulation reports by clicking on the Report Preview tab and completing a report setup page. This page includes provisions that the user can specify for a title page, summary and introduction, body content (including reporting individual environments, even for Solution Combinations), appendices, and images. Information on the report page is saved with the .dsdb file. The report is then generated by clicking on the Generate Report button. In addition to using the report setup page, the user can further tailor the final report as customized table characteristics and the maximum number of digits using the Simulation Report settings in the Options dialog box. The Tree Outline can also be customized and renamed. In ANSYS Structural™ it is very well possible to create different editions of Report in order to support another language or to customize report contents without altering the default report files. Figure 3 is a snap-shot of a sample Report Preview screen adopted from ANSYS website.

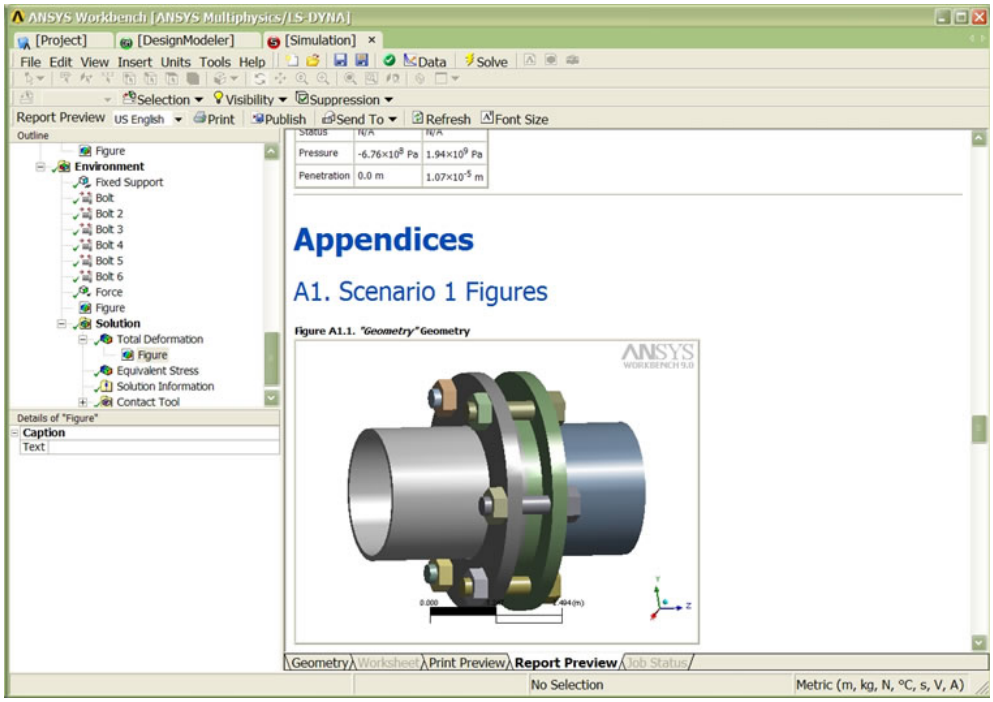

Figure 3: Unique Report Generation Tool

PRODUCT DEVELOIPMENT AND INFORMATIOM

According to the company, “product development is a constantly evolving process and down the road, if you realize the need for additional simulation capacity, ANSYS Structural is easy to upgrade.” Other family of ANSYS products offers powerful tools for both structural and thermal simulation, as well as many other types. Much more information about the ANSYS family of products can be found at the company’s website at http://www.ansys.com via the internet.