Best of Both Worlds: Combining APDL with ANSYS Workbench for Structural Simulations

Using ANSYS Workbench techniques with APDL delivers the most benefit from ANSYS structural mechanics solutions.

By Pierre Thieffry, Lead Product Manager, ANSYS, Inc.

Whether you are a longtime ANSYS Mechanical APDL user wondering, "What is so good about ANSYS Workbench that everyone suggests I use it?" or an ANSYS Workbench user thinking, "I can't find the button, so how can I use this feature?" you can make better use of ANSYS mechanical solutions by combining all of the available technologies. Doing so will allow you to achieve process compression while performing more-advanced simulations.

APDL — ANSYS Parametric Design Language — is the primary language used to communicate with the ANSYS Mechanical APDL solver. This scripting language can be used to automate common tasks or even to build a parametric model. APDL encompasses a wide range of other features, such as if-then-else constructs, do-loops, and vector and matrix operations.

The ANSYS Workbench platform allows users to create new, faster processes and to efficiently interact with external tools such as CAD systems. It provides a foundation for easy multiphysics simulation and enhanced company-wide communication of simulation results. Those performing a structural simulation use a graphical interface (called the ANSYS Workbench Mechanical application) that employs a tree-like navigation structure to define all parts of their simulation: geometry, connections, mesh, loads, boundary conditions and results.

By utilizing the ANSYS Workbench platform, the user saves time in many of the tasks required to perform a simulation. The bidirectional links with all major CAD systems offer a very efficient way to update CAD geometries along with the design parameters. The meshing tools available in ANSYS Workbench are based on some of the best algorithms ANSYS has developed — for example, structural users can now benefit from technologies that once were available only to CFD users, and vice versa. The ANSYS Workbench Mechanical application has proven to be a very efficient way to quickly set up mechanical analyses. Automated contact detection is probably the most noticeable advantage as compared to the long-standing ANSYS Mechanical APDL interface.

ANSYS Workbench includes automation capabilities in addition to those provided by APDL. For example, when looking at design variations, a user can easily perform geometric variations by directly modifying the parametric CAD models. In the case of coupled analyses such as thermal–stress or prestressed modal analyses, all data shared among the various simulations are handled automatically, allowing the user to focus on designing a better product rather than having to worry about file management.

As an ANSYS Workbench Mechanical application user, you may

Figure 1. APDL commands for defining additional material properties

wish to perform tasks that are not natively available. For example, consider that you want to use a material model not available from engineering data, to use a contact option not available from the menus, or even to perform a more-complex task, such as a submodeling analysis. The solution is to insert APDL `Commands` objects to send additional instructions to the solver. When you click on the `Solve` icon, the ANSYS Workbench Mechanical application adds these APDL commands to the input file for the ANSYS Mechanical APDL solver. The `Commands` objects (also referred to as snippets) can be included at the `Geometry` level for modifying element types or material properties (Figure 1). At the `Connections` level, additional contact options or non-linear spring definitions can be introduced. Commands inserted under the analysis branch provide the flexibility to add pre-processing actions, loads and boundary condition definitions, or solution settings. When used under `Solution`, commands allow accessing the APDL post-processors and could be used, for example, to perform complex operations on results or to export data to an external tool. `Commands` objects can be edited manually, or existing macros can be directly imported. The `Named Selections` in ANSYS Workbench Mechanical are a key companion feature to `Commands` objects, as they are converted to nodal or element components that can be referenced.

If you are an ANSYS Mechanical APDL user and your company has created and validated APDL macros for many years, these scripts can be reused directly or with slight changes. The current version of the ANSYS Workbench project schematic contains a Mechanical APDL system that can be linked to a structural analysis (Figure 2). You can then use

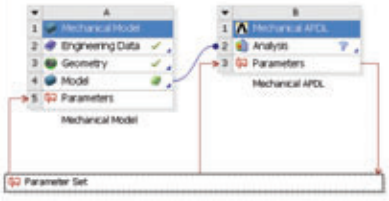

Figure 2. Applying APDL scripts to a base ANSYS Workbench model

macros to operate on the model or the results of the ANSYS Workbench Mechanical analysis. The ANSYS Mechanical APDL system will accept a list of APDL files to be executed sequentially on any data that is provided by the upstream analysis. Furthermore, you can define input and output parameters from the variables of your APDL script and combine them with other parameters in the project. For example, APDL variables can be mixed with CAD dimensions to perform design variations. All standard solver options, such as job definition and licensing options, are defined with

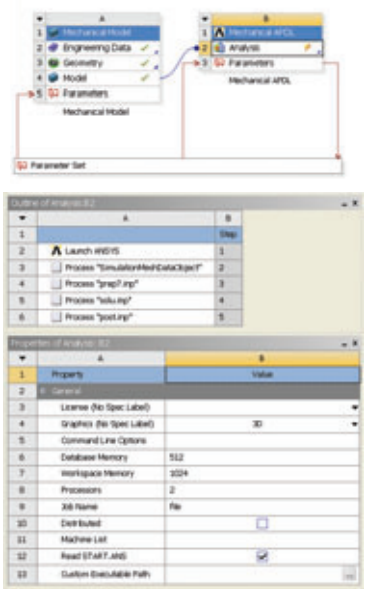

Figure 3. Multiple scripts applied to a model and solver option definitions

the ANSYS Mechanical APDL system (Figure 3).

It is likely that users will not be able to convert all their existing procedures at once, but you can start with the processes that will most benefit from the advantages of the ANSYS Workbench platform. And if you have an entire procedure scripted in APDL that does not require any user interface, you probably don't need to use ANSYS Workbench for that specific case.

ANSYS Mechanical APDL users may want to take a look at the benefits of the ANSYS Workbench platform to see how much time can be saved on geometry import and modifications, meshing, contact detection and general model setup. If you are an ANSYS Workbench user and have found you need functionality that is not available as a button or menu, then learning APDL is definitely worth investing the small amount of time that can result in huge dividends. ■

Are you talking to me?

The chances of jamming this blender with oranges are probably fewer than jamming it from crosstalk emitted by other electronics all around us.

Electromagnetic Interference. EMI. A prime suspect in recent auto and aircraft safety issues. And the hidden culprit in what may be the most daunting engineering challenge of our time.

Today's most innovative and quality-driven companies are just beginning to understand the domino effect of modern products bursting with ingredients. Electronics and a cacophony of other interconnected systems and physical components each one talking over the other.

This new era of smart products doesn't seem so smart if you try to design things the same old way. Product design leaders rely on ANSYS engineering simulation software for analyzing the interplay of electromagnetic emissions, structural mechanics and fluid dynamics.

ANSYS is the true test of product readiness. Go to www.ansys.com to learn more.
Or call us at 1 (866) 267 9724. *Smart products require smart design tools.*

ANSYS®