Introducing ANSYS Workbench 2.0

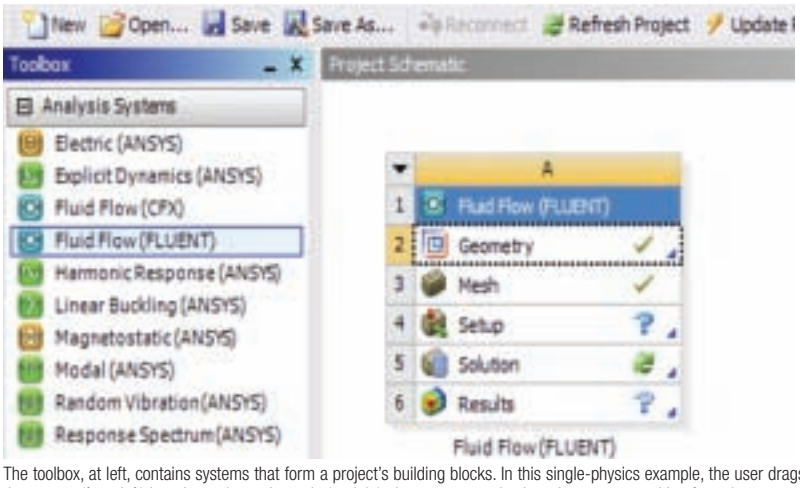
Proven simulation technology is delivered in a truly innovative integration framework.

ANSYS 12.0 delivers innovative, dramatic simulation technology advances in every major physics discipline, along with improvements in computing speed and enhancements to enabling technologies such as geometry handling, meshing and post-processing. These advancements alone represent a major step ahead on the path forward in Simulation Driven Product Development. But ANSYS has reached even further by delivering all this technology in an innovative simulation framework, ANSYS Workbench 2.0.

The ANSYS Workbench environment is the glue that binds the simulation process; this has not changed with version 2.0. In the original ANSYS Workbench, the user interacted with the analysis as a whole using the platform's project page: launching the various applications and tracking the resulting files employed in the process of creating an analysis. Tight integration between the component applications yielded unprecedented ease of use for setup and solution of even complex multiphysics simulations.

In ANSYS 12.0, while the core applications may seem familiar, they are bound together via the innovative project page that introduces the concept of the project schematic. This expands on the project page concept. Rather than offer a simple list of files, the project schematic presents a comprehensive view of the entire analysis project in flowchart form in which explicit data relationships are readily apparent.

Building and interacting with these flowcharts is straightforward. A toolbox contains a selection of systems that form the building blocks of the project. To perform a typical simulation, such as static structural analysis, the user locates the appropriate analysis system in the toolbox and, using drag-and-drop, introduces it into the project schematic. That individual system consists of multiple cells, each of which represents a particular phase or step in the analysis. Working through the system from the top down, the user completes the analysis, starting with a parametric connection to the original CAD geometry and continuing through to post-processing of the analysis result. As each step is completed, progress is shown clearly at the project level. (A green check mark in a cell indicates that an analysis step has been completed.)



The toolbox, at left, contains systems that form a project's building blocks. In this single-physics example, the user drags the system (from left) into the project schematic (at right), then sets up and solves the system, working from the top down through the cells in the system. As shown, the Fluid Flow system (at right) is complete through mesh generation, as shown by green check marks.

Passing files and data from one application to the next is managed entirely by the framework, and data and state dependencies are directly represented. More-complex analyses can be constructed by joining multiple systems. The user simply drags a new system from the toolbox and drops it onto the existing system in the schematic. Connections are created automatically and data is transferred behind the scenes, delivering drag-and-drop multiphysics with unprecedented ease of use.

The ANSYS Workbench environment tracks dependencies among the various types of data in the project. If something changes in an upstream cell, the project schematic shows that downstream cells need to be updated to reflect these changes. A project-level update mechanism allows these changes to be propagated through all dependent cells and downstream systems in batch mode, dramatically reducing the effort required to repeat variations on a previously completed analysis.

Parameters are managed at the project level, where it is possible to change CAD and geometry parameters, material properties and boundary condition values. Multiple parametric cases can be defined in advance and managed as a set of design points, summarized in tabular form

on the ANSYS Workbench project page. Design Exploration systems can be connected to these same project-level parameters to drive automated design investigations, such as Design of Experiments, goal-driven optimization or Design for Six Sigma.

In addition to serving as a framework for the integration of existing applications, the ANSYS Workbench 2.0 platform also serves as an application development framework and will ultimately provide project-wide scripting, reporting, a user interface (UI) toolkit and standard data interfaces. These capabilities will emerge over this and subsequent releases. At ANSYS 12.0, Engineering Data and ANSYS DesignXplorer are no longer independent applications: They have been re-engineered using the UI toolkit and integrated within the ANSYS Workbench project window.

Beyond managing individual simulation projects, ANSYS Workbench interfaces with the ANSYS Engineering Knowledge Manager (EKM) product for simulation process and data management. At ANSYS 12.0, ANSYS Workbench includes the single-user configuration of ANSYS EKM, called ANSYS EKM Desktop. (See sidebar.)

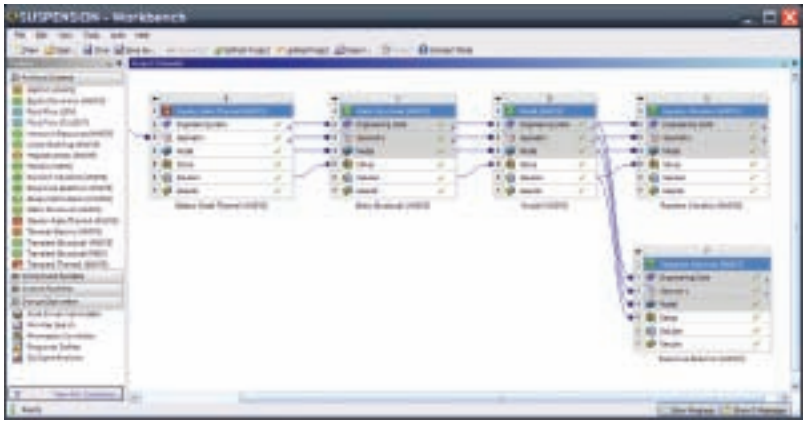
ANSYS Workbench 2.0 represents a sizable step forward in engineering simulation. Within this innovative software framework, analysts can leverage a complete range of proven simulation technology, including common tools for CAD integration, geometry repair and meshing. A novel project schematic concept guides users through complex analyses, illustrating explicit data relationships and capturing the process for automating subsequent analyses. Meanwhile, its parametric and persistent modeling environment in conjunction with integral tools for design optimization and statistical studies enable engineers to arrive at the best design faster. Looking beyond ANSYS 12.0, the ANSYS Workbench platform will be further refined: The aim is to deliver a comprehensive set of simulation technology in an open, adaptive software architecture that allows for pervasive customization and the integration of third-party applications. ■

Judd Kaiser, Shantanu Bhide, Scott Gilmore and Todd McDevitt of ANSYS, Inc. contributed to this article.

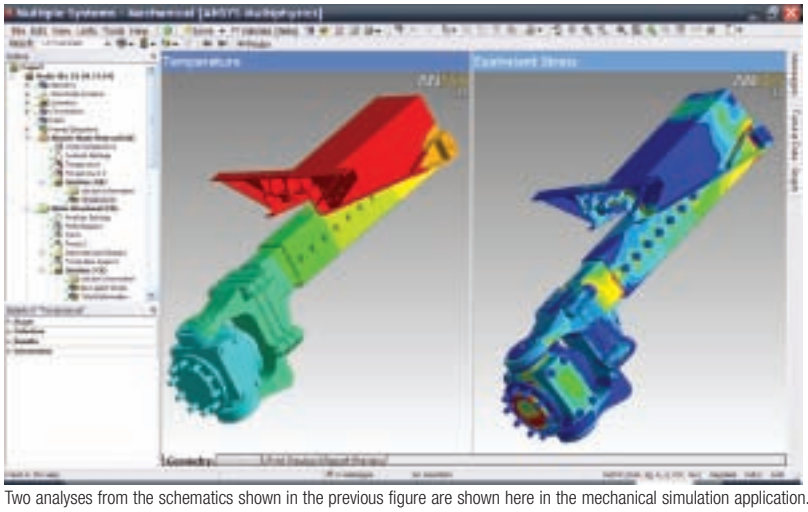
Managing Simulation Data

With the ever-increasing use of simulation, keeping track of the expanding volume of simulation data becomes more and more difficult. The need to be able to quickly locate information for reuse is paramount to increasing productivity and reducing development costs.

ANSYS EKM Desktop is a new tool, integrated in the ANSYS Workbench environment, that facilitates managing simulation data from multiple projects. ANSYS EKM Desktop is a single-user configuration of EKM that allows users to add files from any project to a local virtual repository. Simulation properties and other metadata are automatically extracted (or created) from files when added, and users can tag files with unique identifiers at any time. These attributes can all be used to search and retrieve files based on keywords or complex search criteria. Reports can be easily generated to allow efficient side-by-side comparison of the attributes of related analyses. Search queries and reports can be saved for later re-use. Files that are retrieved can be directly launched in their associated simulation application from within the ANSYS EKM Desktop tool.



More-complex analyses involving multiple physics can be built up by connecting systems. Data dependencies are indicated clearly as connections. State icons at the right of each cell indicate whether cells are up to date, require user input or need to be updated — for example, whether they are just meshed or fully solved.



Two analyses from the schematics shown in the previous figure are shown here in the mechanical simulation application. Launched from the schematic, individual applications may be familiar to existing users.