ANSYS®

12.1

12.1 RELEASE

ANSYS Workbench-Enabled Products

The ANSYS Workbench framework hosts the following software products and components:

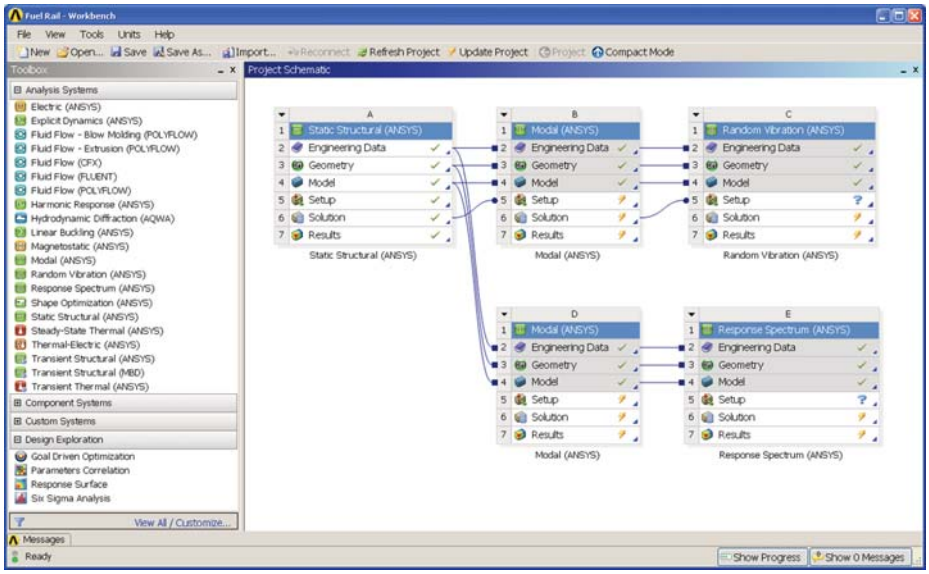
- Common Tools and Capabilities
  - ANSYS CAD connections
  - ANSYS® DesignModeler™
  - ANSYS Meshing
  - ANSYS® DesignXplorer™
  - FE Modeler
- Fluid Dynamics
  - ANSYS® CFX™
  - ANSYS® FLUENT™
  - ANSYS® Icepak®
  - ANSYS® POLYFLOW®
- ANSYS® Multiphysics™
- Structural Mechanics
  - ANSYS® Mechanical™
  - ANSYS® Structural™
  - ANSYS® Professional™
- Explicit Dynamics
  - ANSYS® Explicit STR™
  - ANSYS® AUTODYN®
  - ANSYS® LS-DYNA® (setup-only in ANSYS Workbench)
- Electromagnetics
  - ANSYS Emag
- Turbo System
  - ANSYS® BladeModeler™
  - ANSYS® TurboGrid™
  - ANSYS® Vista™ TF
- Offshore
  - ANSYS® AQWA™

Changing the Way Engineers Interact with Simulation

ANSYS® Workbench™ is the framework upon which the industry's broadest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user every step of the way. Even complex multiphysics analyses can be performed with drag-and-drop simplicity. With bidirectional CAD connectivity, an automated project update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers unprecedented productivity that truly enables Simulation Driven Product Development™.

The Next Generation of ANSYS® Workbench™

With the release of ANSYS 12, the underlying ANSYS Workbench framework has been reengineered. An innovative project schematic view transforms the way engineers work with simulation. Projects are represented graphically as connected systems in a flowchart-like diagram. At a glance, users can easily understand engineering intent, data relationships and the state of the analysis project.



The new project schematic view shows an overall view of the entire simulation project. Engineering intent, data relationships and the state of the entire project are visible at a glance, even for complex analyses involving multiple physics.

Working with the new project system is straightforward: simply drag the desired analysis system from the toolbox at left and drop it into the project schematic. Complete analysis systems contain all of the necessary components, guiding you through the analysis process as you work through the system from top to bottom.

ANSYS Workbench Journaling and Scripting

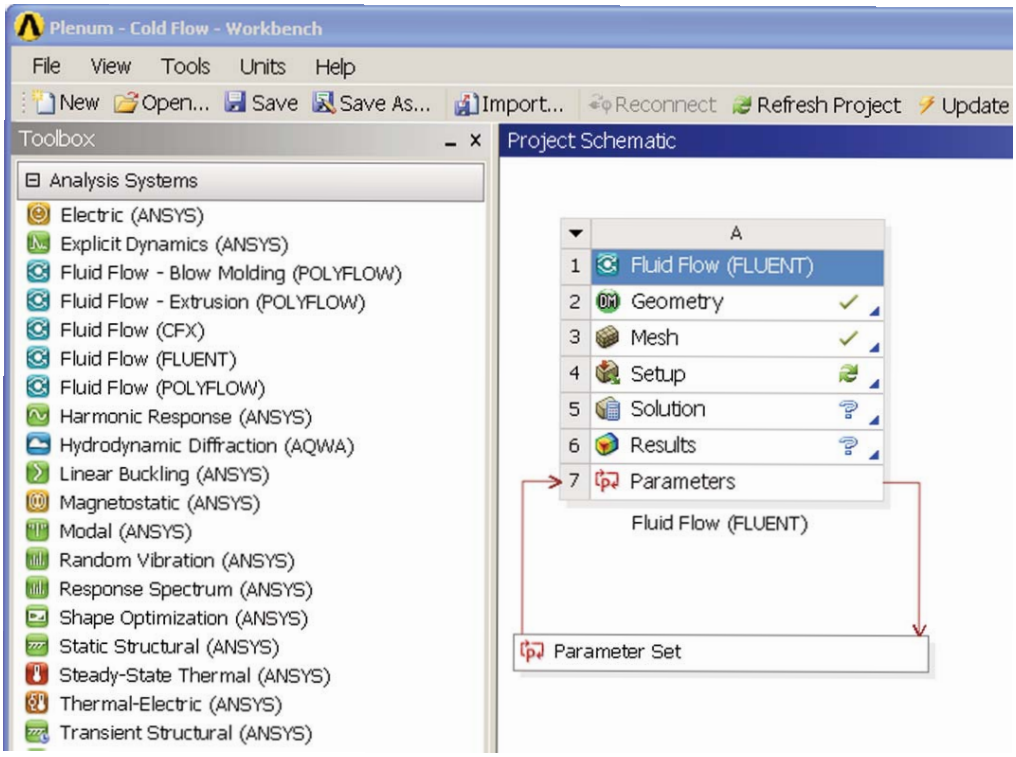
Operations executed within the ANSYS Workbench platform can be recorded to a Python®-based journal that can be replayed to easily recreate an ANSYS Workbench session. Users can modify the journal or write a new set of instructions to create a custom ANSYS Workbench script for automating repetitive tasks or performing complex operations. Journals and scripts can be replayed interactively or in batch mode when ANSYS Workbench is launched.

ANSYS Workbench scripting is based on the versatile, object-oriented Python scripting language, which provides a rich set of data types and standard libraries for efficient programming. Furthermore, the Python implementation used in ANSYS Workbench is well integrated with the .NET Framework (Windows®) and Mono (Linux®), providing interoperability with popular programs like Microsoft® Excel® to access ANSYS Workbench parameters, perform external calculations and drive the overall simulation process.

In addition to replaying journal files, ANSYS Workbench provides a command window to invoke individual commands. Keyboard shortcuts, command-completion and command-recall facilitate command window interaction and reduce tedious typing.

Many applications hosted in ANSYS Workbench support their own scripting languages (e.g., ANSYS® Mechanical™ APDL), and these application-specific commands can be embedded in an ANSYS Workbench script. This coordination between ANSYS Workbench and its applications provides comprehensive scripting support of the complete engineering simulation process.

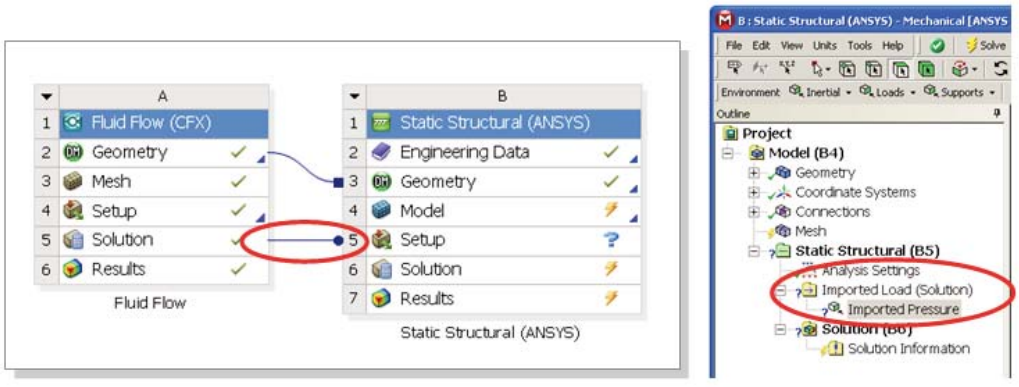
The entire process is persistent. Changes can be made to any portion of the analysis and the ANSYS Workbench platform will manage the execution of the required applications to update the project automatically, dramatically reducing the cost of performing design iterations.



After dragging the desired analysis system into the project schematic, work through the system from top to bottom to complete your analysis. Icons in the right of each cell clearly indicate progress. Here the analysis is complete through meshing.

Drag-and-Drop Multiphysics

The ANSYS Workbench platform has been engineered for scalability. Building complex, coupled analyses involving multiple physics is as easy as dragging in a follow-on analysis system and dropping it onto the source analysis. Required data transfer connections are formed automatically. As an example, consider the one-way fluid structure interaction (FSI) simulation shown schematically below.

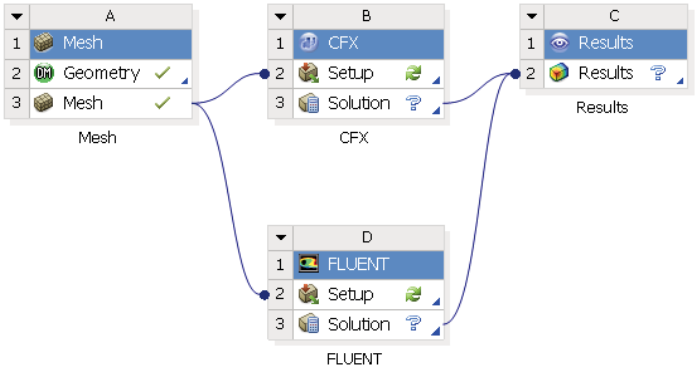


Drag-and-drop multiphysics: forming a link in the project schematic (at left) achieves data transfer between the different physics, and creates imported loads in the downstream simulation (shown inside the ANSYS Mechanical application at right).

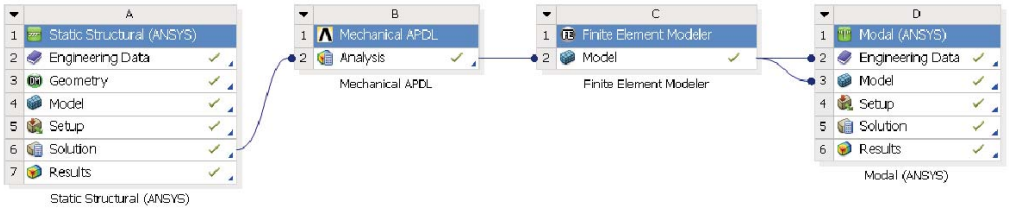
The ANSYS Workbench platform automatically forms a connection to share the geometry for both the fluid and structural analyses, minimizing data storage and making it easy to study the effects of geometry changes on both analyses. In addition, a connection is formed to automatically transfer pressure loads from the fluid analysis to the structural analysis.

Flexible Project Construction

Complete analysis systems are convenient because they contain all of the necessary tasks or components to complete start-to-finish simulations for a wide variety of physics. The project schematic has also been designed to be very flexible. You can connect component systems—task-oriented, "building block" systems—in a wide variety of ways to suit your analysis needs.



Here component systems have been used to represent a specific intent. A common mesh is used for analyses in ANSYS CFX and ANSYS FLUENT products, and the results are compared in the ANSYS CFD-Post tool using the Results system at right.



Chain systems together to achieve creative solutions to real-world engineering problems. Shown above is a static structural analysis that was performed to determine the deformation of a structure in response to applied loads. An ANSYS Parametric Design Language (APDL) script was used to export the mesh for the deformed shape to FE Modeler. FE Modeler was used to synthesize a geometry that was then used to perform modal analysis of the deformed shape.

Project-Level Parameter Management

As always, the applications hosted in the ANSYS Workbench environment support parametric variations, including CAD geometry dimensions, material properties, boundary conditions and derived results. Parameters defined within the applications can be managed directly from the project window, making it easy to investigate multiple variations of an analysis. From within the project window, a series of design points can be defined and executed to complete a what-if study with a single operation.

To fully leverage the power of parametric analysis, ANSYS DesignXplorer software extends the ANSYS Workbench platform to drive Design of Experiments, goal-driven optimization, min/max search or even perform Six Sigma analysis to investigate design robustness. All of this power is available across all applications, all physics and all solvers available within the ANSYS Workbench environment, including ANSYS Mechanical APDL.

ANSYS Workbench Features

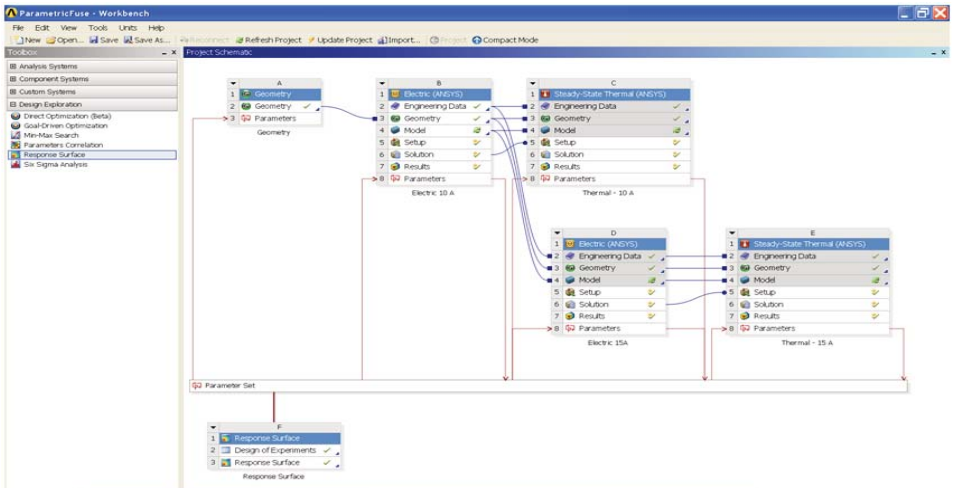
- Bidirectional, parametric links with all major CAD systems
- Integrated, analysis-focused geometry modeling, repair, and simplification via ANSYS DesignModeler
- Highly-automated, physics-aware meshing
- Automatic contact detection
- Unequalled depth of capabilities within individual physics disciplines
- Unparalleled breadth of simulation technologies
- Complete analysis systems that guide the user start-to-finish through an analysis
- Comprehensive multiphysics simulation with drag-and-drop ease of use
- Flexible components enable tools to be deployed to best suit engineering intent
- Innovative project schematic view allows engineering intent, data relationships, and the state of the project to be comprehended at a glance
- Complex project schematics can be saved for re-use
- Pervasive, project-level parameter management across all physics
- Automated what-if analyses with integrated design point capability
- Adaptive architecture with scripting and journaling capabilities and API's enabling rapid integration of new and third-party solutions

ANSYS DesignXplorer – Leverage the Power of Parametric Analysis

ANSYS DesignXplorer software, an extension to the ANSYS Workbench platform, allows users to take full advantage of any parametric analysis. ANSYS DesignXplorer technology provides a description of the relationships between the design variables and the performance of the product by using Design of Experiments (DOE) combined with response surfaces. DOE and response surfaces provide all the information required to take advantage of Simulation Driven Product Development. When performance variations due to design variables are known, it is easy to understand and identify all changes required to meet product requirements.

Once the simulation's responses are characterized, ANSYS DesignXplorer software supplies different types of optimization algorithms to identify good candidates for a given design. Several graphical tools are available to investigate a design: sensitivity plots, correlation matrices, curves, surfaces, trade-off plots and parallel charts with Pareto Front display, and spider charts. They can be used at any time during the development of the product without requiring additional simulations to test new configurations.

These capabilities are available across all applications and all physics. Also included are coupled analyses that allow users to perform design analyses by combining the performances from several different physics and finding the right trade-offs between potentially conflicting objectives. ANSYS DesignXplorer technology also offers probabilistic analysis techniques to determine the extent to which uncertainties in the model affect the results of an analysis and provide useful information in a Six-Sigma approach.



Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	P1	DS_ConnectorThickness	0.01	
4	P2	DS_ConnectorAngle	70	
5	P3	DS_ConnectorRadius	0.05	
6	P4	DS_ConnectorHeight	0.25	
*	New input parameter	New name	New expression	
8	Output Parameters			
9	P5	Electric Voltage (10 A) Maximum	0.10983	V
10	P6	Connection temperature (10A) Maximum	978.62	F
11	P7	Electric Voltage (15 A) Maximum	0.16475	V
12	P8	Connection temperature (15A) Maximum	2112.4	F
*	New output parameter		New expression	

The ANSYS Workbench platform manages parameters for all applications and displays them in a convenient table within the project window. This example studies the effects of varying four geometry dimensions on thermal and electric behaviors. ANSYS DesignXplorer software, fully integrated into ANSYS Workbench, automatically generates a response surface illustrating the effects of varying the geometric input parameters. Design of Experiments, goal-driven optimization, Six Sigma and other design exploration methods can be added with drag-and-drop simplicity.

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of our engineering simulation solutions, companies are transforming their leading edge design concepts into innovative products and processes that work. Today, 97 of the top 100 industrial companies on the "*FORTUNE* Global 500" invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world's most comprehensive multiphysics solutions to solve their complex engineering challenges. The engineered scalability of our solutions delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world's most successful companies turn to ANSYS — with a track record of almost 40 years as the industry leader — for the best in engineering simulation.

www.ansys.com

GSA Contract Holder

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

MKT0000550 12-09