Introduction to Workbench in ANSYS 12.0

Introduction to ANSYS
Workbench 2.0

an ANSYS Tutorial

Click Here to Start Tutorial

Using Workbench in ANSYS 12.0

Topics Covered in this tutorial
- An introduction to the new Workbench window
- Adding systems to the Project Schematic
- Working through Analysis Systems
- Launching and working with Workbench applications
- Adding connected systems
- A first look at parameters and design points
- Customizing the Toolbox

ANSYS 12.0 is being delivered on a new framework, Workbench 2.0, which introduces a new way of working with your projects in schematic form.

This tutorial is intended for those who have used ANSYS tools in previous versions. We hope it will help you to quickly get started using ANSYS 12.0.

Welcome to ANSYS Workbench 2.0!

Click Here to Continue
If you are familiar with Workbench at ANSYS 11.0, you will be accustomed to interfacing with the Project Page to interact with your simulation files.

At ANSYS 12.0, the focus has been taken off of the simulation files and instead placed on the analysis workflow.

Let's take a look at the new Workbench window.

The Project Schematic is where we will see a map of our entire project. As you are about to see, the map will contain systems that describe the workflow and act as a point of entry into the ANSYS tools.
The Toolbox contains systems which we will use as the building blocks of our project.

To begin, drag the desired type of Analysis System from the Toolbox and drop it onto the Project Schematic.

Since we selected Fluid Flow (CFX), each cell of the system corresponds to a step in the process of performing the CFX analysis.

We start by preparing our geometry.
We use the geometry to generate a mesh.

We setup the physics of the problem.

We run the problem in the solver to generate a solution.

And we post-process the solution to gain insight into the results.
From the very beginning, we know the sequence of steps required to complete the analysis.

ANSYS Workbench also provides visual indications of a cell's state via icons on the right side of each cell.

Let's go ahead and walk through the setup of this analysis.

To free up some screen space, we'll close the Toolbox for now.
We can save the project at any time.

In this case, we'll import a recently loaded geometry.

The thumbnail images shown on the right side of the screen wouldn't be displayed in the software; they're just used in this tutorial for illustrative purposes.

To edit the geometry, right-click the **Geometry** cell and select **Edit**.
The geometry is loaded in DesignModeler. Once the geometry has been prepared for analysis, we can minimize or close the window.

Notice that we did not save our work in DesignModeler. ANSYS Workbench does all of the file management for us in the background.

The checkmark indicates that we can proceed to the next cell.

Right-click the Mesh cell and select Edit.
The geometry is loaded in the Meshing application. Once the mesh has been generated, we can minimize or close the window.

Right-click the Setup cell and select Edit.

The mesh is loaded in CFX-Pre. Once the setup is complete we can minimize or close the window.

The Mesh cell is now up to date.
The Setup cell is now up to date.

Right-click the Solution cell and select Edit.

The setup is loaded in CFX-Solver Manager and we run the solver to convergence.

The Solution cell is now up to date.
Right-click the **Results** cell and select **Edit**.

The analysis is complete. Let's take a look at the files that were generated.

The solution is loaded in **CFD-Post** where we can complete our desired post-processing.

Select **View > Files**.
Information about files is listed in the Files view.

Let's see how the project reacts to a change.

Right-click on the Geometry cell and select Edit.

In DesignModeler, we change the wing’s position.
The downstream cell states immediately change to reflect the upstream modification.

We can update each cell by right-clicking it and selecting Update...

Or we can click Update Project to update all of the cells in a single click.

ANSYS Workbench is updating the project in batch.
The project has been updated.

Now we'll expand the analysis by using the CFX solution as an applied load in a Static Structural analysis.

Right-click Solution > select Transfer Data to New > select Static Structural (ANSYS).

A new system has been added to the Project Schematic.
Each cell of the new system corresponds to a step in the process of performing the Static Structural analysis.

...and similarly, the systems are also listed in sequential order.

The cells are listed in sequential order...

The square link indicates that geometry is being shared with the first system...
And the circle link indicates that the solution is being transferred from the first system.

Within ANSYS Mechanical, suppress the fluid geometry...

Right-click the Model cell and select Edit.

Generate the solid mesh...
Select the surfaces on which the imported pressure will be applied...

Notice that all of the Static Structural cells are up to date.

And generate the solution and results.

If we were to change the geometry again, clicking Update Project would update the cells in all systems.
Parameters can be used to automate the process of investigating design alternatives.

Parameters are added within the associated applications.

After adding parameters, the Parameter Set bus bar appears in the Project Schematic.

Double-click the Parameter Set bus bar to control the parameters.
Each set of parameters is a row in the Table of Design Points. To start, we only have 1 design point...

But additional rows can be added to create new design points.

We can run through all design points by clicking Update All Design Points...

But for now we will Return to Project.
Now we'll expand the project by setting up a FLUENT analysis using the CFX mesh.

We have the option to construct our project using Component Systems which tend to be more task-oriented or solver-centric.

Drag-and-drop the FLUENT system onto the desired target location in the Project Schematic.

To start, we will show the Toolbox again.
Dragging the system onto the empty region will create a standalone system...

If you find that your schematic could be reused for other projects...

And dragging the system onto an existing system will share and/or transfer data from that system.

Right-click the Project Schematic and select Add to Custom.
Introduction to Workbench in ANSYS 12.0

Sample System has been added to the Custom Systems.

We also have the option to simplify the Toolbox by customizing the contents. To do so, click View All / Customize.

What have we learned?

- How to work with your Workbench projects
- How to add systems to the Project Schematic using the Toolbox and the context menu
- How to launch and work with Workbench applications
- How to add connected systems
- A brief introduction to working with parameters and design points
- How to customize the toolbox

Click Here to Continue