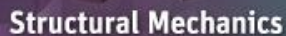


Lecture 2: Introduction to ANSYS Workbench

16.0 Release

A blue, wavy, translucent visualization representing fluid flow or dynamics, positioned on the left side of the central graphic.

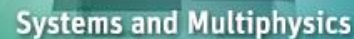
Fluid Dynamics

A purple gear with a glowing white center, positioned in the middle-left of the central graphic.

Structural Mechanics

A series of concentric green circles with a glowing center, positioned in the middle-right of the central graphic.

Electromagnetics

A stack of blue and black 3D cubes, positioned on the right side of the central graphic.

Systems and Multiphysics

Introduction to ANSYS Meshing

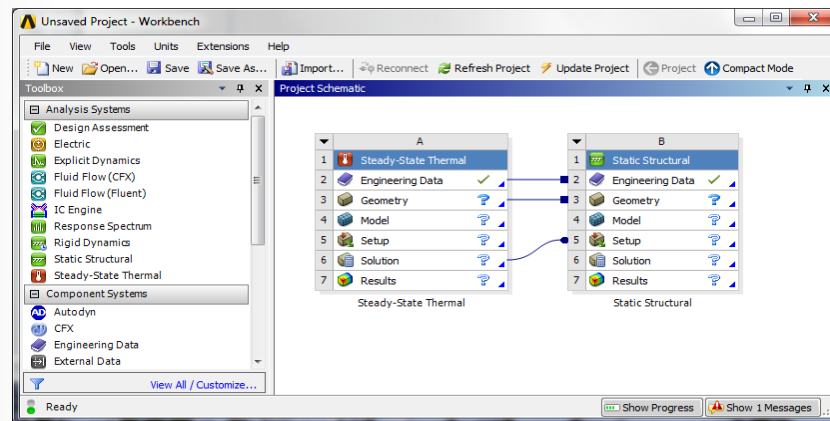
- The purpose of ANSYS Workbench
- Basic use of Workbench
- How to share data between applications
- Working with files, and archiving and restoring projects
- Performing parametric analyses

ANSYS Workbench is a project-management tool. It can be considered as the top-level interface linking all our software tools.

Workbench handles the passing of data between ANSYS Geometry / Mesh / Solver / Postprocessing tools.

This greatly helps project management. You do not need worry about the individual files on disk (geometry, mesh etc). Graphically, you can see at-a-glance how a project has been built.

Because Workbench can manage the individual applications AND pass data between them, it is easy to automatically perform design studies (parametric analyses) for design optimisation.

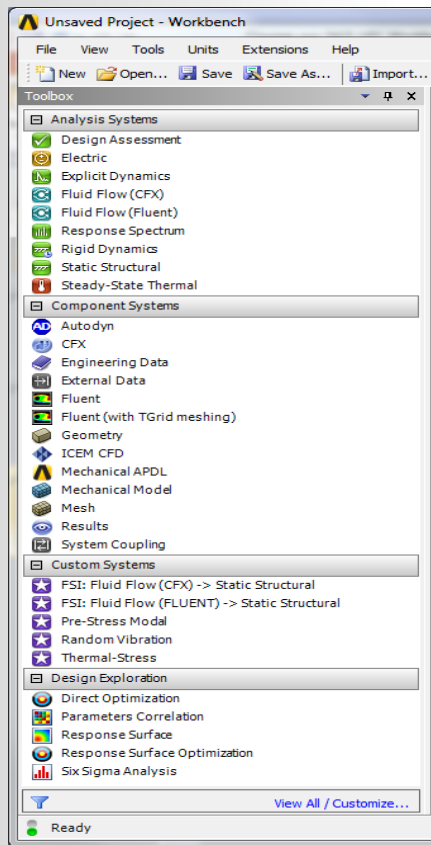


Workbench Overview

The options visible in the **Toolbox** show all the products (systems) you have licenses for.

TIP: If this list appears empty, you have a problem with your licensing!

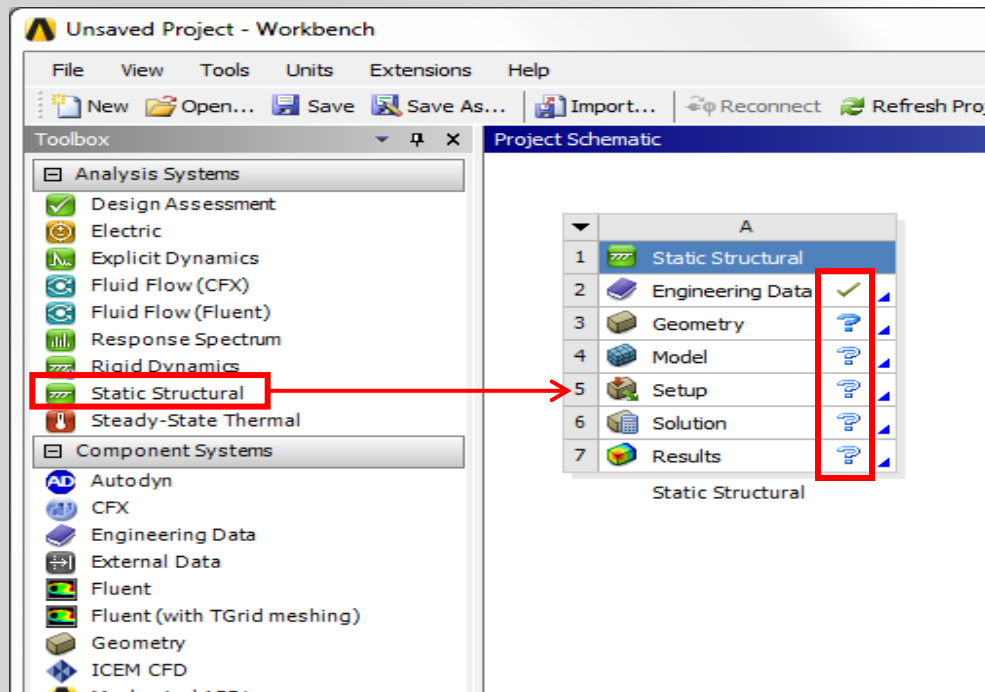
“Design Exploration” provides tools for optimising designs and understanding the parametric response.



“Analysis Systems” are ready-made stencils that include all the individual systems (applications) needed for common analyses (*for example Geometry + Mesh + Solver + Post-Processor*)

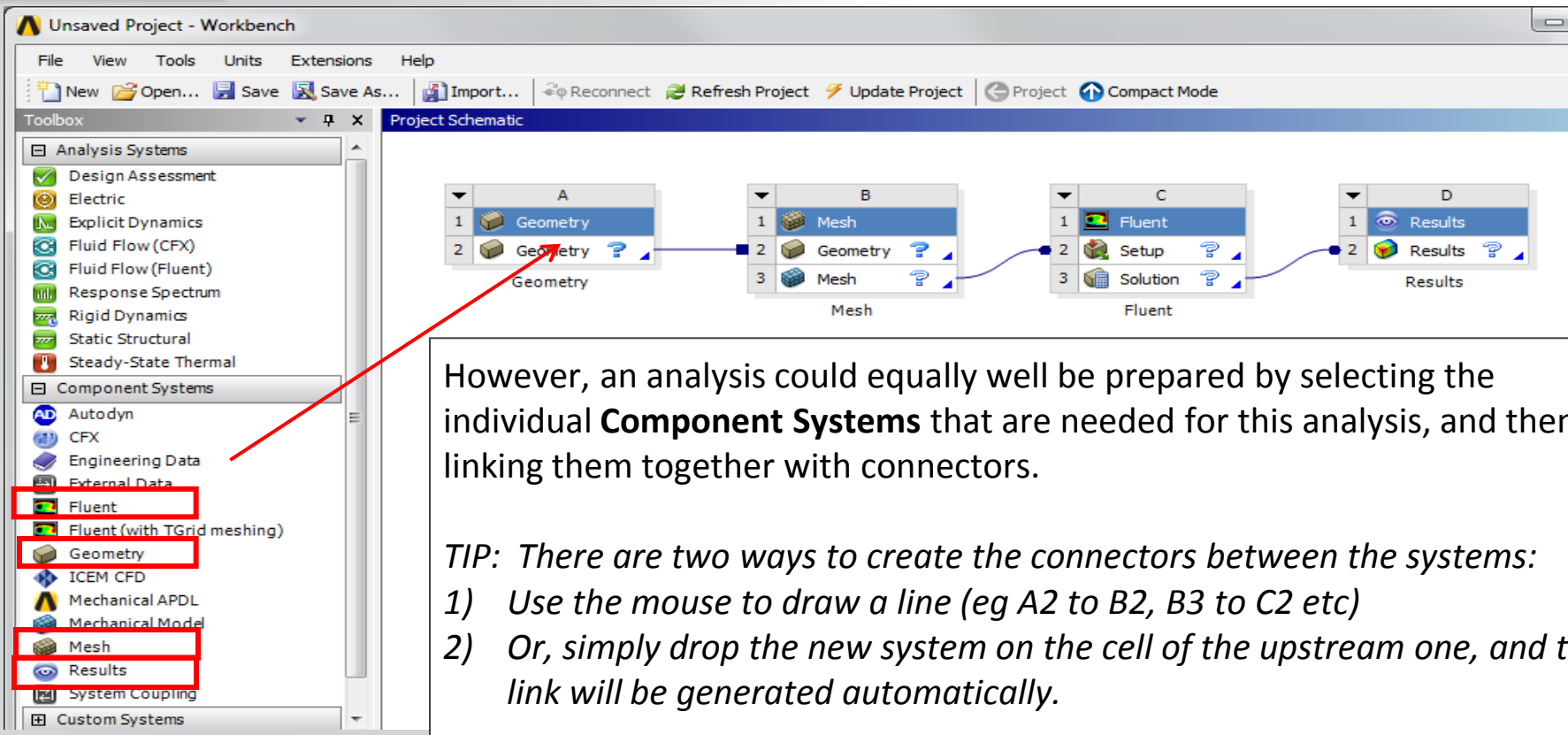
“Component Systems” are the individual building-blocks for each stage of the analysis

Basic Workflow



Dragging an **Analysis System** onto the Project Schematic lays out a workflow, comprising all the steps needed for a typical analysis.

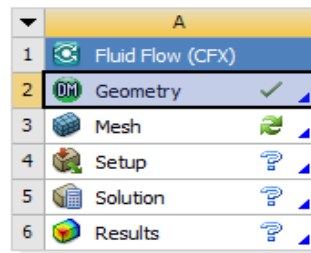
Workflow is from top to bottom. As each stage is complete, the icon at the right-hand side changes



Cell States

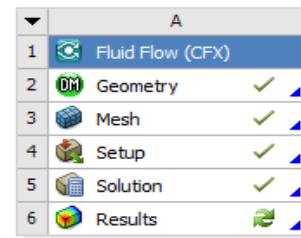
As each stage in the model-build is completed, the state of the cell changes.

Icon Meaning	
	Up to Date
	Refresh required. Upstream data has changed
	Update required. Local data has changed
	Unfulfilled. Upstream data does not exist
	Attention Required
	Solving
	Update Failed
	Update Interrupted
	Changes pending (was up-to-date, but upstream data has changed)



Fluid Flow (CFX)

Status after creating Geometry in A2, not yet opened mesh in A3

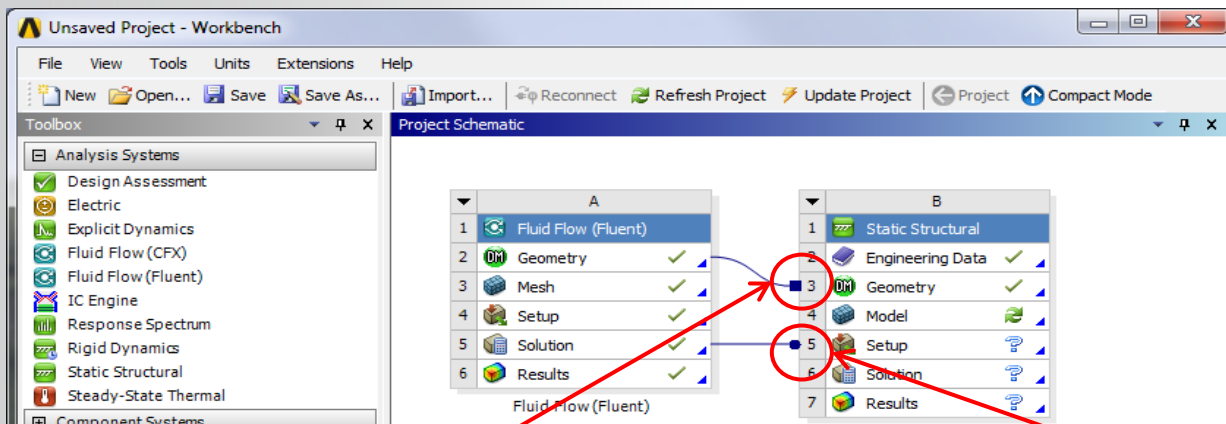


Fluid Flow (CFX)

Status after model has converged, waiting for post-processing

Sharing Data between Different Solvers

Workbench can be used to transfer data between solvers. In this 1-way FSI (fluid-structure-interaction) example, we transfer the loads from a Fluent CFD simulation over to a Mechanical system to perform a stress analysis



The square connector shows that the geometry created in cell A2 (CFD model) is being shared with cell B3 (FEA model).

The round connector shows that the CFD results are being transferred as a Setup (input) condition to be used for FEA stress analysis.

File Location on Disk

Should you need to identify the individual files on your disk for each stage of the project, these can be found by enabling View > Files. The resulting table will cross-reference the directory and filename with the project cells.

The screenshot shows the ANSYS Workbench interface. In the left sidebar, the 'View' menu is open, and the 'Files' option is highlighted with a red circle. A red arrow points from this 'Files' option to the 'Files' table at the bottom of the interface. The 'Files' table is a table with columns: Name, Cell ID, Size, Type, Date Modified, and Location. The table contains 10 rows of data. Two rows are highlighted with red boxes: row 6 and row 7. Row 6 contains the file 'FFF-1.cas.gz' in cell A5, and row 7 contains the file 'FFF-1-00001.dat.gz' in cell A5. The 'Location' column for both rows points to 'E:\WB_Demo\training_files\dp0\FFF\Fluent'. Below the table, there are two labels: 'Filename' and 'Directory'.

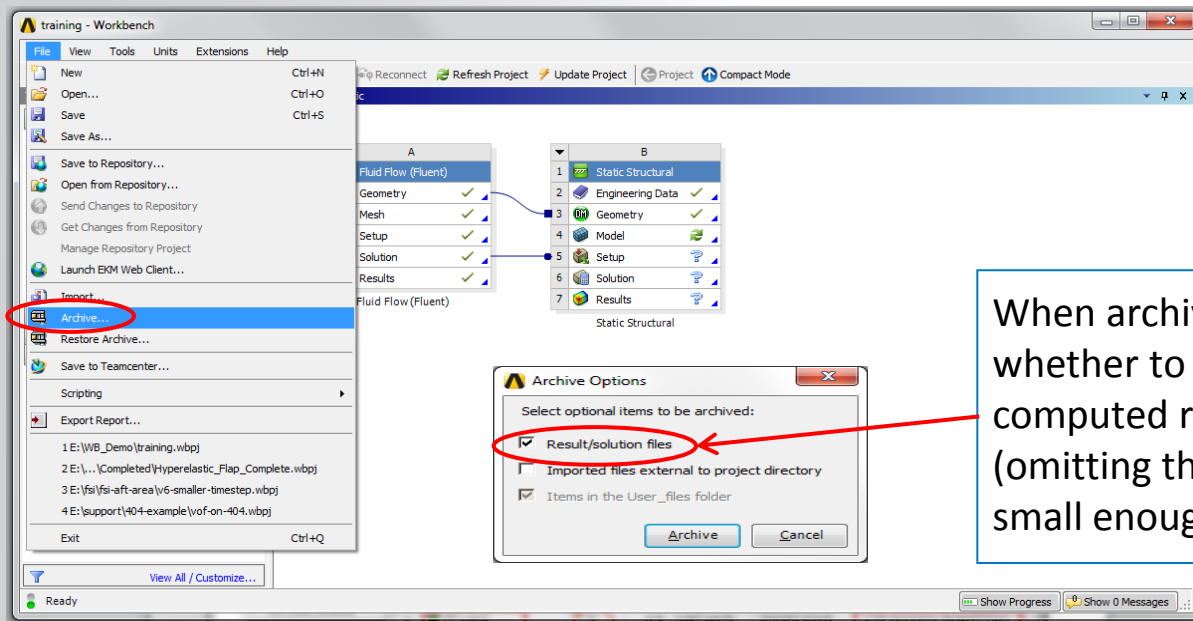
	Name	Cell ID	Size	Type	Date Modified	Location
1						
2	FFF.agdb	A2,B3	2 MB	Geometry File	30/10/2012 15:18:00	E:\WB_Demo\training_files\dp0\FFF\DM
3	FFF.mshdb	A3	56 KB	Mesh Database File	30/10/2012 15:19:25	E:\WB_Demo\training_files\dp0\global\MECH
4	FFF.msh	A3,A4	183 KB	Fluent Mesh File	30/10/2012 15:19:38	E:\WB_Demo\training_files\dp0\FFF\MECH
5	FFF.set	A4	9 KB	FLUENT Model File	30/10/2012 15:20:58	E:\WB_Demo\training_files\dp0\FFF\Fluent
6	FFF-1.cas.gz	A5	92 KB	FLUENT Case File	30/10/2012 15:21:05	E:\WB_Demo\training_files\dp0\FFF\Fluent
7	FFF-1-00001.dat.gz	A5	1 KB	FLUENT Data File	30/10/2012 15:21:05	E:\WB_Demo\training_files\dp0\FFF\Fluent
8	Fluid Flow Fluent.ct	A6	22 KB	CFD-Post State File	30/10/2012 15:21:23	E:\WB_Demo\training_files\dp0\FFF\Post
9			151 KB	ANSYS Project File	30/10/2012 15:29:53	E
10		B2	17 KB	Engineering Data File	30/10/2012 15:29:52	E

Filename

Directory

Use of Archive / Restore

The workbench project comprises many files and directories. If you need to either archive the project, or bundle it to send to us for a Technical Support query, use the **'Archive'** tool. This generates a single zipfile of the entire project.



When archiving, you can choose whether to include the computed result files or not (omitting these may make it small enough to send by email)

Working with Parameters / Refresh and Update

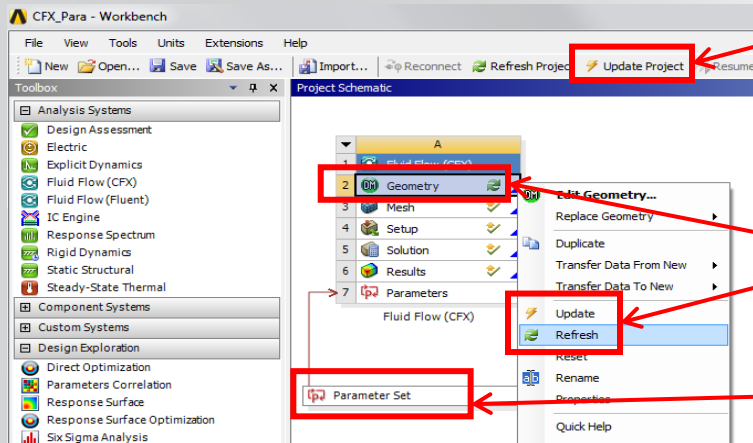
- To make changes, you can manually open up a component cell (e.g. geometry)
- Or: most Workbench applications will let you specify key quantities as a parameter. A new object 'Parameter Set' appears on the Project schematic.
- You then need to update your model. From Workbench you can choose to then update the entire project, or just a single cell.



Refresh: Reads upstream data, but will not do any lengthy operation like solving or meshing.



Update: Performs both a Refresh, AND generates the new output



2] "Update Project" will then work through each component in turn (geometry > mesh > solver) to compute the new design point.

3] However you may want to update an individual component (eg to preview the new geometry before proceeding). Right-click on a individual cell.

1] Clicking here will let you modify the parameters centrally, without having to open the individual application.

Summary

ANSYS Workbench is a convenient way of managing your simulation projects.

Workbench is used to launch the individual software components, and used to transfer data between them.

It is easy to see at-a-glance how a model has been built, and determine which files were used for a particular simulation (pairing geometry files to solver runs)

Workbench also makes it straightforward to perform parametric analyses (without the user needing to manually launch each application in turn), and makes it easy to simulate multi-physics scenarios like fluid-structure interaction.