Introduction to ANSYS Workbench
ANSYS Workbench

• **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.
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!

"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.

"Design Exploration" provides tools for optimising designs and understanding the parametric response.
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.
Alternative Workflow

However, an analysis could equally well be prepared by selecting the individual Component Systems that are needed for this analysis, and then linking them together with connectors.

TIP: There are two ways to create the connectors between the systems:
1) Use the mouse to draw a line (eg A2 to B2, B3 to C2 etc)
2) Or, simply drop the new system on the cell of the upstream one, and the link will be generated automatically.
As each stage in the model-build is completed, the state of the cell changes.

<table>
<thead>
<tr>
<th>Icon Meaning</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Up to Date</td>
<td>Upstream data has changed.</td>
</tr>
<tr>
<td>Refresh required.</td>
<td>Upstream data has changed.</td>
</tr>
<tr>
<td>Update required.</td>
<td>Local data has changed.</td>
</tr>
<tr>
<td>Unfulfilled.</td>
<td>Upstream data does not exist.</td>
</tr>
<tr>
<td>Attention Required</td>
<td></td>
</tr>
<tr>
<td>Solving</td>
<td></td>
</tr>
<tr>
<td>Update Failed</td>
<td></td>
</tr>
<tr>
<td>Update Interrupted</td>
<td></td>
</tr>
<tr>
<td>Changes pending (was up-to-date, but</td>
<td>Upstream data has changed.</td>
</tr>
<tr>
<td>Update Interrupted</td>
<td></td>
</tr>
</tbody>
</table>

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

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 the 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.
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

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

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.

© 2016 ANSYS, Inc. March 12, 2017
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.