ANSYS - Workbench Overview

From zero to results
We are going to work in most advanced

**ANSYS Workbench**
What is Workbench?

- Platform for integration of all ANSYS analysis tools.
- Solid mechanics, fluid dynamics, EM, optimization, etc.
- Entire project contained in common platform:
  - geometry creation, meshing, analysis, and
  - post-processing.
B. ANSYS Workbench Overview

• What is ANSYS Workbench?
  
  – **ANSYS Workbench** provides powerful methods for interacting with the ANSYS family of solvers. This environment provides a unique integration with CAD systems, and your design process.

• **ANSYS Workbench** is comprised of various applications (some examples):
  
  – **Mechanical** for performing structural and thermal analyses using the ANSYS solver
    • Meshing is also included within the Mechanical application
  – **Fluid Flow (CFX)** for performing CFD analyses using CFX
  – **Fluid Flow (FLUENT)** for performing CFD analyses using FLUENT
  – **Geometry (DesignModeler)** for creating and modifying CAD geometry to prepare the solid model for use in Mechanical.
  – **Engineering Data** for defining material properties
  – **Meshing Application** for generating CFD and Explicit Dynamics meshes
  – **Design Exploration** for optimization analyses
  – **Finite Element Modeler (FE Modeler)** for translating a NASTRAN and ABAQUS mesh for use in ANSYS
  – **BladeGen (Blade Geometry)** for creating blade geometry
  – **Explicit Dynamics** for explicit dynamics simulations featuring modeling of nonlinear dynamics
C. Starting Workbench

There are two methods of launching Workbench:

- From the Windows start menu:

- From the CAD system
• For most situations the Workbench GUI is divided into 2 primary sections (there are other optional sections we’ll see in a moment):
The Toolbox

- The toolbox contains 4 subgroups:
  - **Analysis systems**: predefined templates that can be placed in the schematic.
  - **Component systems**: various applications that can be accessed to build, or expand, analysis systems.
  - **Custom Systems**: predefined analysis systems for coupled applications (FSI, thermal-stress, etc.). Users can also create their own predefined systems.
  - **Design Exploration**: Parametric management and optimization tools.
The Project Schematic

• The Workbench project schematic is a graphical representation of the workflow defining a system or group of systems.
• The workflow in the project schematic is always left to right.
• There are currently several applications which are native to Workbench, meaning they run entirely in the Workbench window:
  – Project Schematic, Engineering Data and Design Exploration
• Non-native applications (called data-integrated) run in their own window:
  – Mechanical (formerly Simulation), Mechanical APDL (formerly ANSYS), ANSYS FLUENT, ANSYS CFX, Etc . . .
• Blocks of cells can be deleted by RMB menu selection.
• In this example a Static Structural analysis type is selected for the project schematic.
• From the toolbox the selection can be dragged and dropped onto the schematic or simply double clicked.
The Project Schematic

- By dropping applications and/or systems into various locations in the schematic, an overall analysis project is defined.
- “Connectors” indicate the level of collaboration between systems.
- In the example below a structural system is dragged and dropped onto a thermal system at the Model cell (A4).
- Before completing the operation notice there are a number of optional “drop targets” that will provide various types of linkage between systems (continued next page).
By completing the operation from the previous page, notice the linkage here is only at the Model level and above.

In this case there would be no thermal/structural coupling.

Notice too each system block is given and alphabetic designation (A, B, C, etc.).
By dropping the structural system at the “Solution” level we obtain a structural system that is coupled to the thermal solution.

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Steady-State Thermal (ANSYS)</td>
</tr>
<tr>
<td>2</td>
<td>Engineering Data</td>
</tr>
<tr>
<td>3</td>
<td>Geometry</td>
</tr>
<tr>
<td>4</td>
<td>Model</td>
</tr>
<tr>
<td>5</td>
<td>Setup</td>
</tr>
<tr>
<td>6</td>
<td>Solution</td>
</tr>
<tr>
<td>7</td>
<td>Results</td>
</tr>
</tbody>
</table>

Steady-State Thermal (ANSYS)

Share A2:A1
Transfer A6

Notice, the candidate “drop target” indicates data will be shared from fields A2 to A4, and transferred from A6.
A schematic can also be constructed by RMB and choosing to “Transfer Data To New” or “Transfer Data From New”.

In using this RMB transfer feature all transfer possibilities (upstream and downstream) are displayed.

These selections will vary depending on which cell in a particular system you highlight.
• Identifying cell states:

  • Unfulfilled: missing upstream data.

  • Attention required: may need to correct this or upstream cells.

  • Refresh required: upstream data has changed. Need to refresh cell (update will also refresh the cell).

  • Update required: the data has changed and the output of the cell must be regenerated.

  • Up to date.

  • Input changes pending: cell is locally up to date but may change when the next update is performed due to upstream changes.
To start Workbench - Double click on **ANSYS Workbench 15** icon
• Project built by dropping component systems into schematic.
• Links connect different components, show flow of project/data.
Questions ?