Workbench User's Guide

Table of Contents

List of Figures

List of Tables

Overview

Ansys Workbench is a workflow analysis platform, combining the strength of our core simulation tools with the tools necessary to manage your projects. To build an analysis, you add building blocks called *systems* to the main project workspace. These systems make up a flowchart-like diagram that represent the data flow through your project. Each system is a block of one or more components called *cells*, which represent the sequential steps necessary for the specific type of analysis. Once added, you can link them together to share or transfer data between systems.

From the cells, you can work with various Ansys applications and analysis tasks; some of these open in tabs within the Workbench environment, while others open independently in their own windows.

Ansys applications enable you to define analysis characteristics such as geometry dimensions, material properties, and boundary conditions as *parameters*. You can manage parameters at the project-level in the Workbench environment. To perform your analysis, work through the cells of each system in order—typically from top to bottom—defining inputs, specifying project parameters, running your simulation, and investigating the results.

Workbench enables you to easily investigate design alternatives. You can modify any part of an analysis or vary one or more parameters, and then automatically update the project to see the effect of the change on the simulation result.

Related topics:

The Ansys Product Improvement Program

User Interface Overview

Interacting with Project Objects

Using Help

The Ansys Product Improvement Program

This product is covered by the Ansys Product Improvement Program, which enables Ansys, Inc., to collect and analyze *anonymous* usage data reported by our software without affecting your work or product performance. Analyzing product usage data helps us to understand customer usage trends and patterns, interests, and quality or performance issues. The data enable us to develop or enhance product features that better address your needs.

How to Participate

The program is voluntary. To participate, select **Yes** when the Product Improvement Program dialog appears. Only then will collection of data for this product begin.

How the Program Works

After you agree to participate, the product collects anonymous usage data during each session. When you end the session, the collected data is sent to a secure server accessible only to authorized Ansys employees. After Ansys receives the data, various statistical measures such as distributions, counts, means, medians, modes, etc., are used to understand and analyze the data.

Data We Collect

The data we collect under the Ansys Product Improvement Program are limited. The types and amounts of collected data vary from product to product. Typically, the data fall into the categories listed here:

Hardware: Information about the hardware on which the product is running, such as the:

- brand and type of CPU

- number of processors available

- amount of memory available

- brand and type of graphics card

System: Configuration information about the system the product is running on, such as the:

- operating system and version

- country code

- time zone

- language used

- values of environment variables used by the product

Session: Characteristics of the session, such as the:

- interactive or batch setting

- time duration

- total CPU time used

- product license and license settings being used

- product version and build identifiers

- command line options used

- number of processors used

- amount of memory used

- errors and warnings issued

Session Actions: Counts of certain user actions during a session, such as the number of:

• project saves

• restarts

• meshing, solving, postprocessing, etc., actions

• times the Help system is used

• times wizards are used

• toolbar selections

Model: Statistics of the model used in the simulation, such as the:

• number and types of entities used, such as nodes, elements, cells, surfaces, primitives, etc.

• number of material types, loading types, boundary conditions, species, etc.

• number and types of coordinate systems used

• system of units used

• dimensionality (1-D, 2-D, 3-D)

Analysis: Characteristics of the analysis, such as the:

• physics types used

• linear and nonlinear behaviors

• time and frequency domains (static, steady-state, transient, modal, harmonic, etc.)

• analysis options used

Solution: Characteristics of the solution performed, including:

• the choice of solvers and solver options

• the solution controls used, such as convergence criteria, precision settings, and tuning options

• solver statistics such as the number of equations, number of load steps, number of design points, etc.

Specialty: Special options or features used, such as:

• user-provided plug-ins and routines

• coupling of analyses with other Ansys products

Data We Do Not Collect

The Product Improvement Program does *not* collect any information that can identify you personally, your company, or your intellectual property. This includes, but is not limited to:

- names, addresses, or usernames

- file names, part names, or other user-supplied labels

- geometry- or design-specific inputs, such as coordinate values or locations, thicknesses, or other dimensional values

- actual values of material properties, loadings, or any other real-valued user-supplied data

In addition to collecting only anonymous data, we make no record of where we collect data from. We therefore cannot associate collected data with any specific customer, company, or location.

Opting Out of the Program

You may *stop* your participation in the program any time you wish. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

The Ansys, Inc., Privacy Policy

All Ansys products are covered by the Ansys, Inc., Privacy Policy.

Frequently Asked Questions

1. *Am I required to participate in this program?*

   No, your participation is voluntary. We encourage you to participate, however, as it helps us create products that will better meet your future needs.

2. *Am I automatically enrolled in this program?*

   No. You are not enrolled unless you explicitly agree to participate.

3. *Does participating in this program put my intellectual property at risk of being collected or discovered by Ansys?*

   No. We do not collect any project-specific, company-specific, or model-specific information.

4. *Can I stop participating even after I agree to participate?*

   Yes, you can stop participating at any time. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

5. *Will participation in the program slow the performance of the product?*

   No, the data collection does not affect the product performance in any significant way. The amount of data collected is very small.

6. *How frequently is data collected and sent to Ansys servers?*

   The data is collected during each use session of the product. The collected data is sent to a secure server once per session, when you exit the product.

7. *Is this program available in all Ansys products?*

    Not at this time, although we are adding it to more of our products at each release. The program is available in a product only if this *Ansys Product Improvement Program* description appears in the product documentation, as it does here for this product.

8. *If I enroll in the program for this product, am I automatically enrolled in the program for the other Ansys products I use on the same machine?*

    Yes. Your enrollment choice applies to all Ansys products you use on the same machine. Similarly, if you end your enrollment in the program for one product, you end your enrollment for all Ansys products on that machine.

9. *How is enrollment in the Product Improvement Program determined if I use Ansys products in a cluster?*

    In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

10. *Can I easily opt out of the Product Improvement Program for all clients in my network installation?*

    Yes. Perform the following steps on the file server:

    a. Navigate to the installation directory: `[Drive:]\v222\commonfiles\globalsettings`

    b. Open the file **ANSYSProductImprovementProgram.txt**.

    c. Change the value from "on" to "off" and save the file.

User Interface Overview

Use the following image and topics to understand basic user interface functionality.
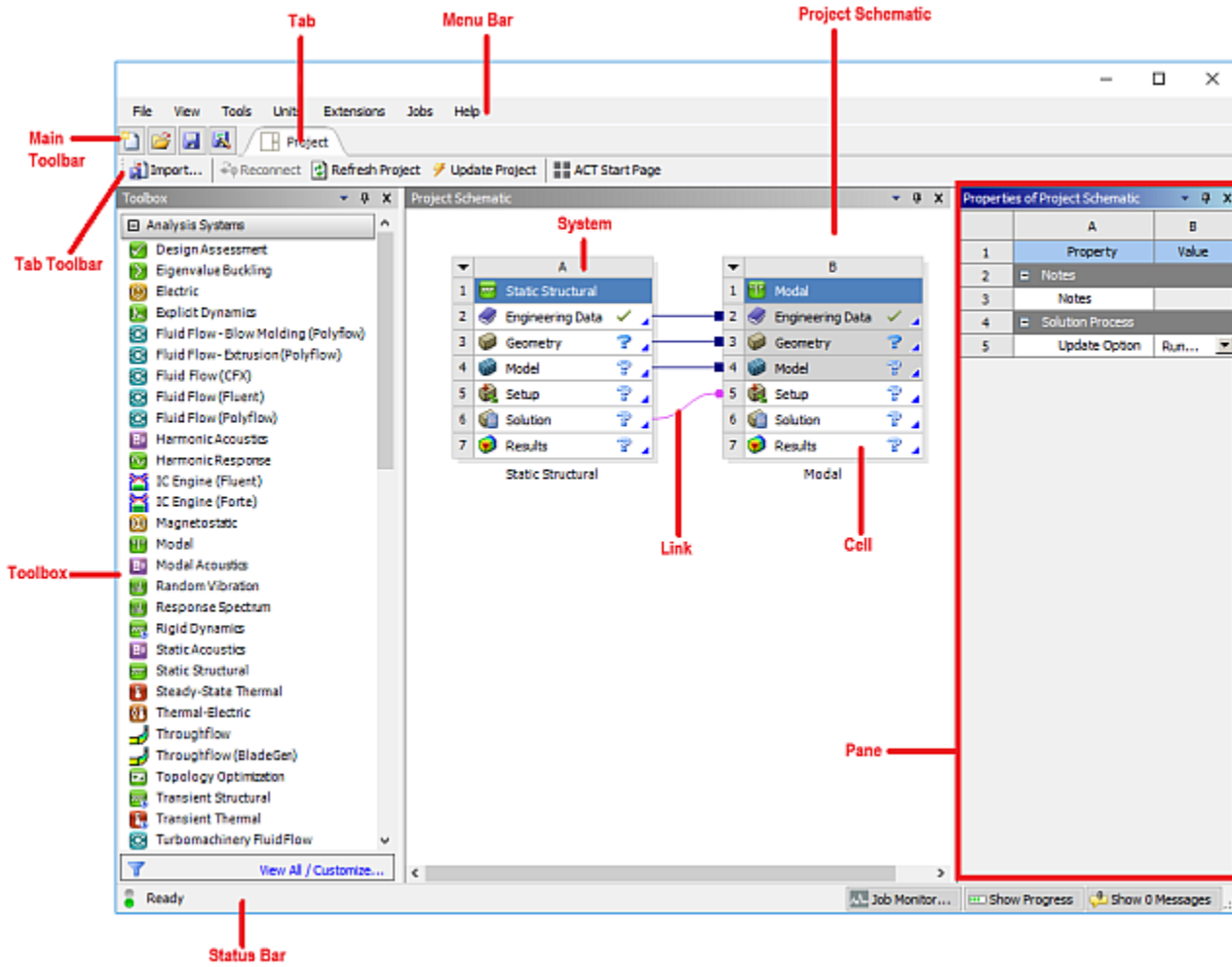
Related topics:

Tabs

Panes

Systems and Cells

Links

Tabs

Tabs are workspaces that allow you to interact with different parts of your project. The **Project** tab is used as the main workspace, but other tabs can be opened; either by double-clicking the associated cell, or by right-clicking it and selecting **Edit** from the context menu.

Each cell that does not launch an external application has a single workspace that can be opened inside Workbench. In some cases, you can open multiple tabs of the same type; for example, if you have three **Engineering Data** cells in three separate systems, you can open three **Engineering Data** tabs at the same time.

You can right-click tabs to **Close Tab**, **Close Other Tabs**, and **Close All**. You can also close individual tabs by clicking the **x** icon. Only the **Project** tab cannot be closed.

For a detailed list of available tabs, see Tabs in Workbench (p. 439).

Panes

Tabs in Workbench are made up of multiple panes that contain information relating to the project shown in the tab.

Some panes are common to multiple tabs, while others are tab-specific. Some panes are only shown in the tab for which they've been enabled, while others, once enabled, are shown across all the tabs either until you either disable them or reset the tab layout.

To configure panes, see Configuring Panes (p. 29). For a detailed list of available panes, see Panes within Tabs (p. 440).

Systems and Cells

A system is a workflow building block and is made up of one or more analysis components called cells.

You can interact with a cell to perform any of the following actions:

- Launch an application that opens independently of Workbench

- Open a tab inside Workbench

- Add connecting systems, either upstream or downstream

- Assign input or reference files

- Assign properties to components of your analysis

Each cell has either an application or a tab associated with it. Some cells are associated with an application that launches in a separate window, such as Fluent or Mechanical; in some cases, multiple cells in a system can be associated with the same application. Other cells, such as the **Parameters** cell or a cell in a **System Coupling** system, are associated with tabs that open inside Workbench.

Icons for each cell indicate the state of that particular cell, such as whether the cell requires attention, is up-to-date, and so on. For more information, see Understanding Cell States (p. 449).

To display a quick help panel for the cell, click the blue triangle in the lower right corner of the cell (where available). The quick help message that displays explains any immediate action that must be taken and can include links to more detailed help.

For more information on working with systems, see Working Through a System (p. 67). For a detailed list of cell types, see Types of Cells (p. 447).

Links

Links represent data sharing or data transfer between the systems added to the **Project Schematic**.

For more information on link types and working with links, see Creating Connected Systems (p. 60).

Interacting with Project Objects

The **Toolbox** and **Project Schematic** pane contain project objects such as systems, cells, and links. You can interact with these objects in a number of different ways:

- **Single-click**: Selects an object. This does not modify data or initiate any action.

- **Double-click**: Initiates the default action from the context menu. This allows users who are familiar with Workbench to quickly move through basic or common operations.

- **Right- click**: Displays a context menu applicable to the current state of the selected object. From the context menu, you can select from multiple actions. The default action is shown in bold.

- **Drag-and-drop**: Previews possible locations for an object in the **Project Schematic**. A drag-and-drop operation can have multiple alternative targets, depending on context and schematic complexity. Holding down the mouse button, hover over any target to see details of how the target location would be implemented (for example, what components would be connected after the operation is completed).

  To cancel a drag-and-drop operation, press the **Esc** key while holding down the mouse button.

Using Help

Workbench offers three levels of help:

- Quick Help (p. 22)

- Sidebar Help (p. 23)

- Online Help (p. 24)

Quick Help

Quick help is available for most cells in a system. Click the blue arrow in the bottom right corner of the cell to see a brief help panel on that cell. Quick help is generally state-sensitive; as the state of a cell changes, the content of the quick help panel updates to match. From quick help, you can also access related help topics in the online help system.
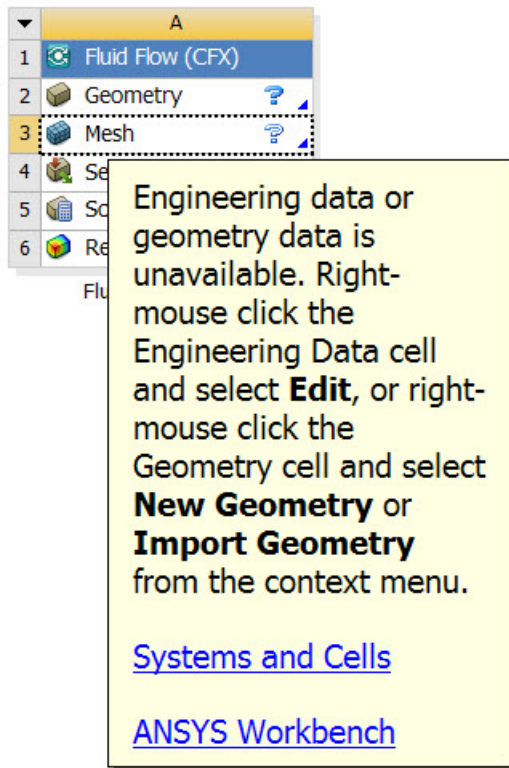
Sidebar Help

The **Sidebar Help** pane is shown on the right side of the screen. The content of this help panel is determined by the portion of the interface that has focus (that is, where the mouse was last clicked).

- If no area has focus, a user interface overview topic is displayed.

- If the **Project Schematic** has focus but no systems are defined, a getting started topic is displayed.

- If the **Project Schematic** has focus and one or more systems have been defined, links to those specific system types, as well as links to general topics are displayed.

To access the **Sidebar Help** pane, press **F1** or, from the menu bar, select **Help** → **Show Context Help**.
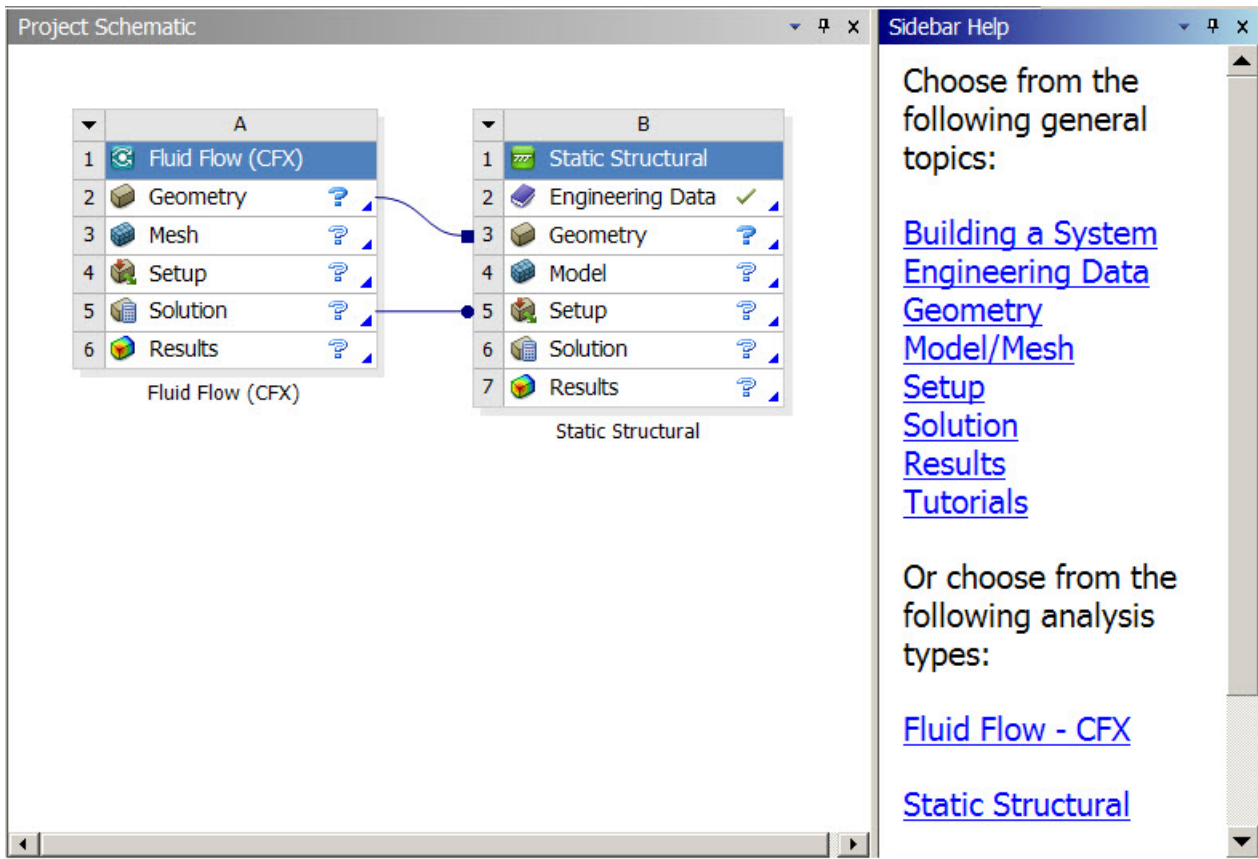
Online Help

Online help is available from the Workbench **Help** menu, or from any of the links in the quick help or the **Sidebar Help** pane. Online help provides a comprehensive overview of all features and capabilities and includes full search capabilities.

Configuring Workbench

Use the following topics to configure Workbench:

Configuring the Toolbox

Configuring Units in Workbench

Configuring Panes

Configuring User Preferences

Configuring External Solvers for Use with Workbench

Customizing Workbench with Ansys ACT

Configuring the Toolbox

When you open Workbench, the **Toolbox** pane on the **Project** tab contains the systems available to you. These are the products you have installed and for which licenses are available.

To configure the Toolbox to show only the systems you use, rather than all available systems:

1. At the bottom of the **Toolbox**, Click **View All / Customize**.

2. In the **Toolbox Customization** pane, deselect systems you want to hide.
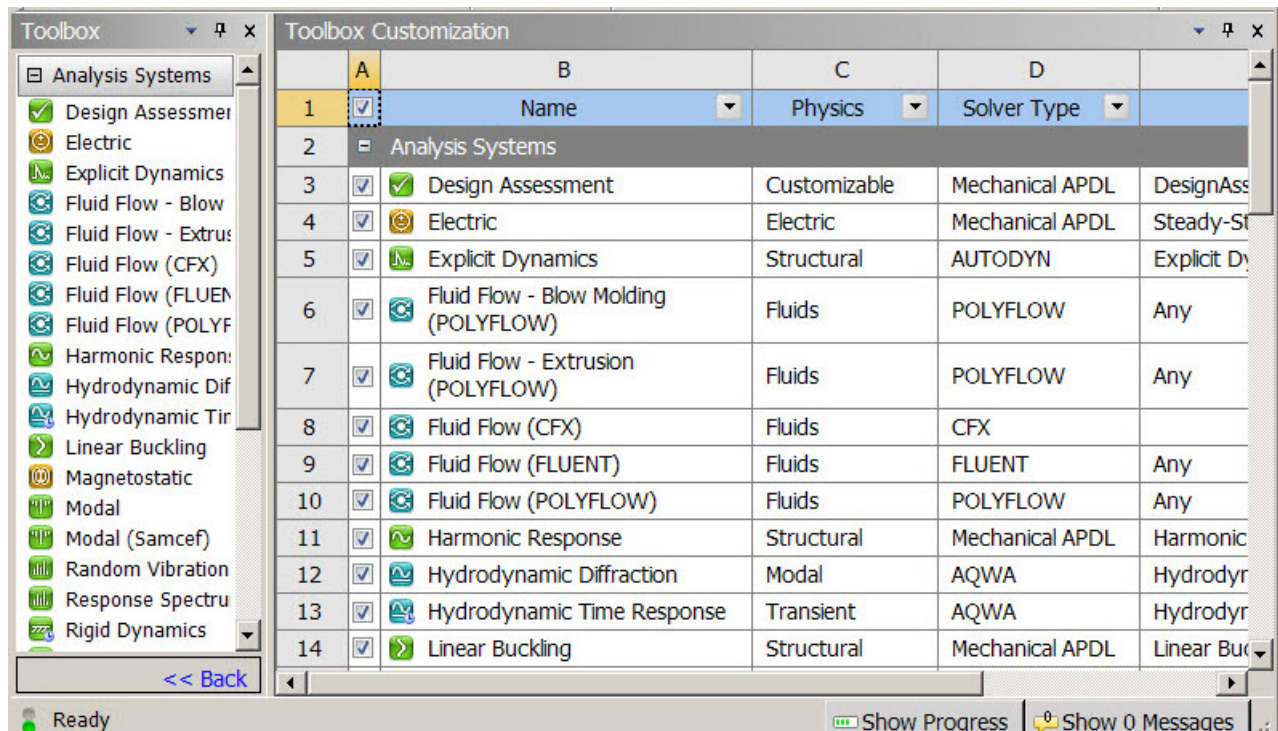
3. To close the **Toolbox Customization** pane, click **<< Back**.

Configuring Units in Workbench

Workbench provides:

- Predefined unit systems that contain the most commonly used sets of units.

- The ability to define custom unit systems (p. 28) based on the predefined unit systems. (You cannot edit or delete predefined unit systems.)

- The ability to display the following project data using the project unit system:
  - Engineering data
  - Parameters
  - Charts.

- The ability to share the unit system between users.

> **Note:**
>
> Unit settings in Workbench are not passed to Fluid Flow analysis systems; to CFX, Fluent, **Results**, or TurboGrid systems; or to FSI: Fluid Flow custom systems.

To open the **Unit Systems** dialog box, from the menu bar select **Units → Unit Systems**.
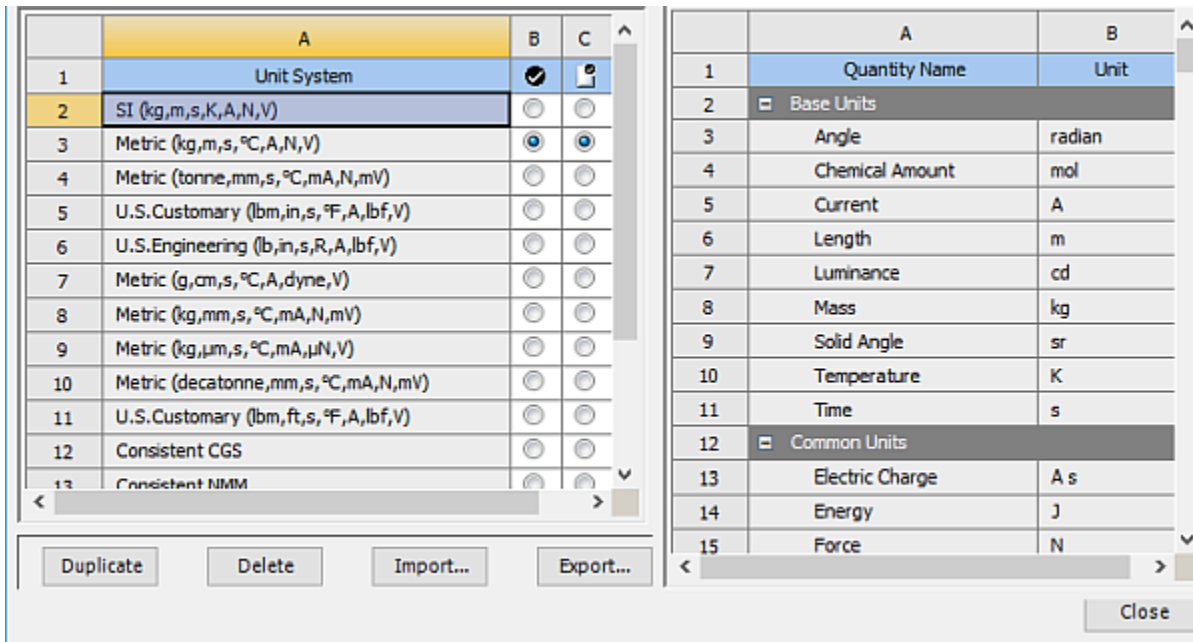
The **Unit Systems** dialog is displayed and contains the following options:

Option	Description
Active Project (icon)	Sets unit system for active project.
Default (icon)	Sets default unit system. This will be default unit system for every project.
Suppress/Unsuppress (icon)	Hides/displays unit system menu item. Only 15 unit systems can be unsuppressed and displayed as in the **Units** menu.
Duplicate	Creates a custom unit system based on selected unit system.
Delete	Deletes unit system. The following unit systems cannot be deleted: • Predefined Unit System • Active Project Unit System • Default Unit System
Import	Imports a units file (`*.xml`).
Export	Exports the active project unit system as a units file (`*.xml`).

Units

A unit system is a collection of the preferred unit for the base, common, and other quantity types.

• Base Units: All other units are derived from these units.
  – Angle
  – Chemical Amount
  – Current
  – Length
  – Luminance
  – Mass
  – Solid Angle
  – Time
  – Temperature

• Common Units: These are units which are derived from the base units and are typically used as base units for other units.
  – Electric Charge
  – Energy
  – Force
  – Power
  – Pressure
  – Voltage

• Other Units: Several other units are derived from base and common units.

For a detailed description on how units are used in expressions, see Expressions, Functions, Quantities, and Units (p. 291).

Predefined Unit Systems

Workbench offers the following predefined unit systems:

• Metric (kg, m, s, °C, A, N, V) (default unit system)
• Metric (tonne, mm, s, °C, mA, N, mV)
• U.S. Customary (lbm, in, s, °F, A, lbf, V)
• SI (kg, m, s, K, A, N, V)
• U.S. Engineering (lbm, in, s, R, A, lbf, V)

You cannot edit or delete predefined unit systems.

Workbench also provides the following additional unit systems. These are suppressed by default.

• Metric (g, cm, s, °C, A, dyne, V)
• Metric (kg, mm, s, °C, mA, N, mV)
• Metric (kg, μm, s, °C, mA, μN, V)
• Metric (decatonne, mm, s, °C, mA, N, mV)
• U.S. Customary (lbm, ft, s, F, A, lbf, V)
• Consistent CGS
• Consistent NMM
• Consistent μMKS
• Consistent BIN
• Consistent BFT

For a detailed description of unit systems for the Mechanical application, see Understanding Solving Units in the *Mechanical User's Guide*.

You can also display values as defined and display values in project units. See Units Menu (p. 454) for more information on using these options.

Custom Unit Systems

To create a custom unit system based on a predefined unit system, select the unit system and click **Duplicate**.

• The default name for the new **Unit** system is **Custom Unit System**. You can change the name.

• You can change the units for any quantity type from the available list.

• The list of units that are available are either consistent with SI or US Customary, depending on the original unit system. This to assure that a consistent unit system can be constructed for solution purposes.

- Changing base units can automatically change **Common** and **Other** derived units if appropriate. For example, if the mass unit is kg, the length unit is m, and the force unit is N, then changing mass to g and length to cm will automatically change force to dyne.

Unit Limitations

Sound Pressure Level and *A Weighted Sound Pressure Level* are dimensionless quantities with units dB and dBA, respectively.

- Math operations between these two units or either of these and a numerical value will result in a value with no unit. For example, 10 dB x 10 = 100. (no unit)

- Math operations between either of these units and a dimensional unit will result in a value with the dimensional unit. For example, 10 dB x 10 m = 100 m.

Configuring Panes

The configuration and persistence of panes across tabs varies to give you the maximum flexibility in customizing the information you see on each tab. The header bar of each pane contains icons and context menus that allow you to control pane attributes such as visibility, size, and floating/docked state. You can access the context menu either by clicking the down arrow or by right-clicking in the header.

Some panes—for example, the **Table**, **Chart**, **Outline**, and **Properties** panes—are defined per tab, so that changes to a pane are specific to that tab. For example, if you resize and float the **Chart** pane in the **Parameter Set** tab and then switch to the **Response Surface** tab in a DesignXplorer system, the **Chart** pane in the new tab is not resized and floated.

For more information on specific panes, see Panes within Tabs (p. 440).

Related topics:

Opening and Closing Panes

Hiding and Resizing Panes

Floating and Docking Panes

Resetting Panes

Opening and Closing Panes

To open a pane, from the menu bar, select **View** → *pane name*. Once the pane is open, it updates its content according to what you've selected; by selecting different objects, you can view or edit the associated properties of those objects.

To close a pane, use one of the following options:

- From the menu bar, select **View** → *pane name* to deselect it.

- Right-click the header bar of the pane and from context menu, select **Close**.

- From the Windowing drop-down menu on the pane header bar, select **Close**.

- On the header bar, click ⊠ .

Hiding and Resizing Panes

To change how the pane displays, from the pane header bar context menu or the **Windowing** drop-down menu, select from the following options:

- **Minimize**: Displays the pane as a tab in the bottom left corner of the Workbench window. To display the pane again, click the tab.

- **Maximize**: Expands the pane to its largest size.

   You can only maximize a floating pane; you cannot maximize a docked pane.

- **Restore**: Returns the pane to its pre-maximized size and location.

To resize a pane, use your mouse to drag its edges to the desired size.

Floating and Docking Panes

By default, most panes are docked, or embedded, in their tab. You can separate a pane from its tab, so it can be moved outside of the Workbench window. When you dock a pane, it returns to its place on its usual tab.

When you float a pane, you separate it from its tab, so it is still available when you switch tabs. This is especially useful when you want to see a pane that is not available as part of a tab. For example, the **Files** pane is not visible in the **Parameter Set** tab, but you can float the **Files** pane so you can see it while working with project parameters.

To float a pane, use one of the following options:

- Right-click the header bar of the pane and from context menu, select **Float**.

- From the Windowing drop-down menu on the pane header bar, select **Float**.

- On the header bar of the pane, click 📌 .

- Tear the pane from its dock by dragging the header bar of the pane to another location.

To dock a pane, use one of the following options:

- Right-click the header bar of the pane and from context menu, select **Dock**.

- From the Windowing drop-down menu on the pane header bar, select **Dock**.

- On the header bar of the pane, click 🔲 .

When you dock a pane, it returns to its usual tab and may not be visible in the current tab.

Resetting Panes

You can reset the panes for one tab or for all the tabs.

- To reset all the panes in the current tab to their default position, select **View** → **Reset Workspace**.

- To reset all the panes in the project, select **View** → **Reset Window Layout**. This resets the panes in all tabs to their default positions and switches to the **Project** tab.

Configuring User Preferences

To set your preferences for Workbench, from the menu bar, select **Tools** → **Options**.

The preferences you set here are local settings, affecting only you. Some changes made in the **Options** dialog take place immediately, while others take place after you restart Workbench.

For detailed descriptions of each preference dialog box, see Workbench User Preferences (p. 417).

Configuring External Solvers for Use with Workbench

This section describes how to set up your environment to use the Samcef and ABAQUS solvers within Workbench.

Configuring Samcef

Configuring ABAQUS

Configuring Samcef

After installing the Samcef program, you must set the following environment variables for compatibility with Workbench:

SAMTECH_LICENSE_FILE

The path to your license server or license file. See the Samcef documentation for more information.

SAM_EXE

The path to the Samcef executable folder. In a standard installation, this is the `Exec` folder in the base directory of the Samcef install. For instance, if Samcef is installed at `C:\Samcef`, the path to the executable folder is `C:\Samcef\Exec`

SAM_WORK

The path to a temporary folder for a local run of Samcef. See the Samcef documentation for more information.

The Samcef Result Storage Configuration File

The exact results stored by the Samcef solver can be controlled using an XML configuration file. This file uses Samcef SAI codes to define which results to write to the results file. Results are classified by result type and category. The configuration file is located at `ANSYS_IN-STALL_DIR\v222\aisol\WBAddins\SamcefAddin\SamcefArchiveSettings.xml`.

The configuration file is read when Mechanical is launched. If changes are made to the file while Mechanical is running, Mechanical must be restarted to reflect the changes.

The XML root element in this file is `<SamcefArchiveSettings>`. The child nodes of this root element represent analysis types. The only valid child node is `<Analysis>`, and this child node has the following attribute:

`name` **(type string)**

> The analysis type:
> - `default`
> - `modal`
> - `non_linear`
> - `harmonic`
> - `thermal`

The `<Analysis>` node can have a child node of `<Output>`, which has the following attribute:

`type` **(type string)**

> The option type, used to order SAI codes in the solver input file:
> - `nodal`
> - `elemental`

The `<Output>` node has child nodes of `<Codes>` which have the following attributes:

`value` **(type string)**

> List of Samcef SAI codes separated by spaces. Usually a list of integers, positive or negative.

`category` **(type string)**

> Optional, used for options enable or disabled by the user.
> - `stress`
> - `strain`
> - `thermal_flux`
> - `contact`

When a solve is executed and the solver input file is created, Mechanical finds the correct `<Analysis>` node in the configuration file to determine the SAI codes to write to the input file. This check is performed by finding the analysis type and whether the analysis is linear or nonlinear. The following table describes the mapping:

Workbench Analysis Type	Option	`<Analysis>` **Node Used**
Static Structural (Samcef)	Linear	default
	Nonlinear	non_linear
Transient Structural (Samcef)	Linear	default
	Nonlinear	non_linear
Modal (Samcef)	Linear	modal
	Nonlinear	non_linear
Harmonic Response (Samcef)	-	harmonic

Workbench Analysis Type	Option	`<Analysis>` Node Used
Eigenvalue Buckling (Samcef)	-	No SAI code written
Steady-State Thermal (Samcef)	-	thermal
Transient Thermal (Samcef)	-	thermal

The Samcef Postprocessing Configuration File

In order to postprocess all results from the Samcef solver in the Workbench interface, an XML file is used to map the results between Samcef and Workbench. The XML file defines all of the Workbench result codes with associated data, and lists the corresponding Samcef codes. The configuration file is located at $ANSYS\_INSTALL\_DIR$\v222\aisol\WBAddins\SamcefAddin\SamcefResultCodes.xml.

The configuration file is read when Mechanical starts postprocessing Samcef data. If changes are made to the file while Mechanical is running, Mechanical must be restarted to reflect the changes.

The following is an example result as expressed in the XML configuration:

```
<Result result_name="U" result_id="101" data_type="nodal" out_unit="length" style="vector"
   num_comp="5" comp_labels="XYZLABELS" description="Deformations(XYZs)">
      <Codes>
            <Code name="Code 163" />
      </Codes>
</Result>
```

The listing provides the following information:

- The Workbench code "U" identified by Mechanical by the ID 101 is a nodal result and is interpreted as a length.

- The result is represented in the vector style, has 5 components and is part of the XYZLABELS group.

- The result corresponds to Samcef result code, "Code 163".

The XML **Result** object can have the following attributes:

result_name (type string)

> The Workbench name of the result. Standard Mechanical names are reserved.

result_id (type integer)

> The Workbench ID of the result. Standard Mechanical IDs are reserved; you should start numbering solver-specific results at 100000. You should not choose a number larger than 1000000 because those numbers are internally reserved.

data_type (type string enumerate)

> The result type:
> - **nodal**
> - **elem_nodal**

- **elemental**

out_unit (type string enumerate)

The unit type of the result. If this value is **defined_per_component**, the subtag **<Units-PerComponent>** must be defined.

The **<UnitsPerComponent>** node contains individual **<Component>** nodes with the attributes **name** and **out_unit**. The **name** attribute corresponds to the component name, and the **out_unit** attribute has the same requirements as this attribute.

```
<UnitsPerComponent>
     <Component name="XXX" out_unit="XXX" />
     <Component out_unit="XXX" />
</UnitsPerComponent>
```

- **no_units**
- **acceleration**
- **angle**
- **angular_velocity**
- **area**
- **capacitance**
- **charge**
- **charge_density**
- **conductivity**
- **current**
- **current_density**
- **density**
- **displacement**
- **electric_conductivity**
- **electric_field**
- **electric_flux_density**
- **electric_resistivity**
- **energy**
- **film_coeff**
- **force**
- **force_intensity**
- **frequency**
- **heat_flux**
- **heat_generation**
- **heat_rate**
- **inductance**
- **inverse_stress**
- **length**
- **magnetic_field_intensity**
- **magnetic_flux**
- **magnetic_flux_density**

- **mass**
- **moment**
- **moment_inertia**
- **permeability**
- **permittivity**
- **poisson**
- **power**
- **pressure**
- **relative_permeability**
- **relative_permittivity**
- **section_modulus**
- **specific_heat**
- **specific_weight**
- **shear_strain**
- **stiffness**
- **strain**
- **stress**
- **strength**
- **thermal_expansion**
- **temperature**
- **time**
- **velocity**
- **voltage**
- **volume**
- **gasket_stiffness**
- **moment_inertia_mass**
- **psd_acceleration**
- **psd_acceleration_grav**
- **psd_displacement**
- **psd_velocity**
- **rotational_damping**
- **rotational_stiffness**
- **translational_damping**
- **angular_acceleration**
- **seedbeck_coefficient**
- **decay_constant**
- **fracture_energy**
- **shock_velocity**
- **energy_density_mass**
- **electric_conductance_per_unit_area**
- **psd_stress**
- **psd_strain**
- **psd_force**

- **psd_moment**
- **psd_pressure**
- **force_per_angular_unit**
- **impulse**
- **impulse_per_angular_unit**
- **temperature_difference**
- **material_impedance**
- **rs_acceleration**
- **rs_acceleration_grav**
- **rs_displacement**
- **rs_velocity**
- **warping_factor**
- **thermal_conductance**
- **inverse_length**
- **inverse_angle**
- **thermal_capacitance**
- **normalized_value**
- **mass_flow_rate**
- **unitless**
- **stress_intensity_factor**
- **sqrt_length**
- **energy_per_volume**
- **thermal_gradient**

`style` **(type string enumerate)**

The style of the value:
- **scalar**
- **vector**
- **tensor**
- **tensor_strain**
- **euler_angles**
- **coordinate**
- **shear_moment_diagram**

`num_comp` **(type integer)**

Number of components, between 1 and the length of **comp_labels**.

`comp_labels` **(type string enumerate)**

The component labels:
- **XYZLABELS**
- **STRESSLABELS**
- **CONTACTLABELS**
- **BEAMLABELS**

- **GASKETLABELS**
- **NLLABELS**
- **MISCLABELS**
- **SENGLABELS**
- **SPRINGLABELS**
- **BEAM188LABELS**
- **RADIOSITYLABELS**
- **EULERLABELS**
- **BEAMRXLABELS**
- **THERMALMASSLABELS**
- **SHELLMBPLABELS**
- **SLASHPNUMLABELS**
- **CINTLABELS**
- **PRIN_S_LABELS**
- **PDMG_LABELS**
- **PFC_LABELS**
- **CDM_LABELS**
- **CONTDEBONDLABELS**
- **BEARINGLABELS**
- **BLANKLABELS**

description (type string)

> Free text.

Once the **<Result>** node is defined, the **<Codes>** node with the individual **<Code>** child nodes are defined. The **<Codes>** node can have the following attributes:

hasShell (type string)

> If present and value = true, the **layer** attribute should be defined.

layer (type integer)

> Only necessary if **hasShell** is set to true.
> - 1 - bottom
> - 2 - middle
> - 3 - top

Note:

It is possible to have a **<Codes>** node without any child **<Code>** nodes. It can be used to define a result with layers but no associated codes for those layers, for example TEMP_UPSHELL.

The **<Code>** child nodes can have the following attributes:

`name` (type string)

Value of the Samcef solver code, generally of the form "Code XXX" or "Sdb XXX".

`module` (type string)

Optional, if set, defines the code only for a specific Samcef module. Possible values are:

- ba - bacon
- me - mechano
- as - asef
- st - stabi
- dy - dynam
- mt - mechano/thermal
- re - repdyn

Limitations

- If the XML of the configuration file is missing or not well-formed, no results are available.

- If a result has invalid syntax (missing XML components, invalid attributes), it is not available.

- Mechanical must be restarted if any changes are made to the result configuration file during operation.

Configuring ABAQUS

ABAQUS version 6.14 is supported.

After installing ABAQUS, you must configure the *ANSYS_INSTALL_DIR*`\v222\aisol\WBAddins\AbaqusAddin\config.xml` file in order to postprocess results. Edit the `config.xml` file with a text editor, and specify the version of ABAQUS you have installed:

`<AbaqusVersion version="6.14-3">`

If more than one version of ABAQUS is installed on your machine, the paths to the version configured for use with Workbench must appear first in your **PATH** environment variable.

Assuming ABAQUS is installed at `C:\SIMULIA\Abaqus`, for ABAQUS 6.14-3, the **PATH** environment variable must contain:

- `C:\SIMULIA\Abaqus\Commands`

- `C:\SIMULIA\Abaqus\6.14-3\code\bin`

- `C:\SIMULIA\Abaqus\6.14-3\tools\SMApy\python2.7\Lib`

If the ABAQUS version installed does not match the version specified in the `config.xml` file, Mechanical returns an error when trying to postprocess the ODB result file.

The ABAQUS Result Storage Configuration File

The exact results stored by the ABAQUS solver can be controlled using an XML configuration file. This file uses ABAQUS Program Controlled codes to define which results to write to the results file. Results are classified by result type and category. The configuration file is located at `ANSYS_IN-STALL_DIR\v222\aisol\WBAddins\AbaqusAddin\AbaqusArchiveSettings.xml`.

The configuration file is read when Mechanical is launched. If changes are made to the file while Mechanical is running, Mechanical must be restarted to reflect the changes.

The XML root element in this file is **`<AbaqusArchiveSettings version="1">`**. The child nodes of this root element represent analysis types. The only valid child node is **`<Analysis>`**, and this child node has the following attribute:

`name` (type string)

> The analysis type:
> * **`struct`**
> * **`eigen`**
> * **`thermal`**

The **`<Analysis>`** node can have a child node of **`<Output>`**, which has the following attribute:

`type` (type string)

> The option type, used to order codes in the solver input file:
> * **`nodal`**
> * **`elemental`**
> * **`contact`**
> * **`energy`**
> * **`radiation`**

The **`<Output>`** node has child nodes of **`<Codes>`** which have the following attributes:

`value` (type string)

> List of ABAQUS codes separated by spaces. Usually a list of integers, positive or negative.

`position` (type string)

> For elemental results, if a result exists with a matching position, it is used. Otherwise results without position are used.

`category` (type string)

> Optional, used for options enable or disabled by the user.
> * **`stress`**
> * **`strain`**
> * **`thermal`**
> * **`contact`**

When a solve is executed and the solver input file is created, Mechanical finds the correct **`<Analysis>`** node in the configuration file to determine the SAI codes to write to the input file. This check is performed by finding the analysis type and whether the analysis is linear or nonlinear. The following table describes the mapping:

Workbench Analysis Type	`<Analysis>` Node Used
Static Structural (ABAQUS)	struct
Transient Structural (ABAQUS)	struct
Modal (ABAQUS)	eigen
Steady-State Thermal (ABAQUS)	thermal
Transient Thermal (ABAQUS)	thermal

The ABAQUS Postprocessing Configuration File

In order to postprocess all results from the ABAQUS solver in the Workbench interface, an XML file is used to map the results between ABAQUS and Workbench. The XML file defines all of the Workbench result codes with associated data, and lists the corresponding Samcef codes. The configuration file is located at `ANSYS_INSTALL_DIR\v222\aisol\WBAddins\AbaqusAddin\VkiAbaqus-ResultCodes.xml`.

The configuration file is read when Mechanical starts postprocessing ABAQUS data. If changes are made to the file while Mechanical is running, Mechanical must be restarted to reflect the changes.

The following is an example result as expressed in the XML configuration:

```
<Result result_name="U" result_id="101" data_type="nodal" out_unit="length" style="vector"
    num_comp="5" comp_labels="XYZLABELS" description="Deformations(XYZs)">
        <Solver name="Abaqus">
            <Code name="U" />
        </Solver>
        <vki vki_root_name="D" />
</Result>
```

The listing provides the following information:

- The Workbench code "U" identified by Mechanical by the ID 101 is a nodal result and is interpreted as a length.

- The result is represented in the vector style, has 5 components and is part of the XYZLABELS group.

- The result corresponds to vki result "D", which comes from ABAQUS code "U".

The XML **`Result`** object can have the following attributes:

`result_name` (type string)

The Workbench name of the result. Standard Mechanical names are reserved.

`result_id` (type integer)

The Workbench ID of the result. Standard Mechanical IDs are reserved; you should start numbering solver-specific results at 100000. You should not choose a number larger than 1000000 because those numbers are internally reserved.

`data_type` (type string enumerate)

The result type:
- **`nodal`**
- **`elem_nodal`**
- **`elemental`**

`out_unit` (type string enumerate)

The unit type of the result:
- **`no_units`**
- **`acceleration`**
- **`angle`**
- **`angular_velocity`**
- **`area`**
- **`capacitance`**
- **`charge`**
- **`charge_density`**
- **`conductivity`**
- **`current`**
- **`current_density`**
- **`density`**
- **`displacement`**
- **`electric_conductivity`**
- **`electric_field`**
- **`electric_flux_density`**
- **`electric_resistivity`**
- **`energy`**
- **`film_coeff`**
- **`force`**
- **`force_intensity`**
- **`frequency`**
- **`heat_flux`**
- **`heat_generation`**
- **`heat_rate`**
- **`inductance`**
- **`inverse_stress`**
- **`length`**
- **`magnetic_field_intensity`**
- **`magnetic_flux`**

- **magnetic_flux_density**
- **mass**
- **moment**
- **moment_inertia**
- **permeability**
- **permittivity**
- **poisson**
- **power**
- **pressure**
- **relative_permeability**
- **relative_permittivity**
- **section_modulus**
- **specific_heat**
- **specific_weight**
- **shear_strain**
- **stiffness**
- **strain**
- **stress**
- **strength**
- **thermal_expansion**
- **temperature**
- **time**
- **velocity**
- **voltage**
- **volume**
- **gasket_stiffness**
- **moment_inertia_mass**
- **psd_acceleration**
- **psd_acceleration_grav**
- **psd_displacement**
- **psd_velocity**
- **rotational_damping**
- **rotational_stiffness**
- **translational_damping**
- **angular_acceleration**
- **seedbeck_coefficient**
- **decay_constant**
- **fracture_energy**
- **shock_velocity**
- **energy_density_mass**
- **electric_conductance_per_unit_area**
- **psd_stress**
- **psd_strain**

- **psd_force**
- **psd_moment**
- **psd_pressure**
- **force_per_angular_unit**
- **impulse**
- **impulse_per_angular_unit**
- **temperature_difference**
- **material_impedance**
- **rs_acceleration**
- **rs_acceleration_grav**
- **rs_displacement**
- **rs_velocity**
- **warping_factor**
- **thermal_conductance**
- **inverse_length**
- **inverse_angle**
- **thermal_capacitance**
- **normalized_value**
- **mass_flow_rate**
- **unitless**
- **stress_intensity_factor**
- **sqrt_length**
- **energy_per_volume**
- **thermal_gradient**

style (type string enumerate)

The style of the value:
- **scalar**
- **vector**
- **tensor**
- **tensor_strain**
- **euler_angles**
- **coordinate**
- **shear_moment_diagram**

num_comp (type integer)

Number of components, between 1 and the length of **comp_labels**.

comp_labels (type string enumerate)

The component labels:
- **XYZLABELS**
- **STRESSLABELS**
- **CONTACTLABELS**

- **`BEAMLABELS`**
- **`GASKETLABELS`**
- **`NLLABELS`**
- **`MISCLABELS`**
- **`SENGLABELS`**
- **`SPRINGLABELS`**
- **`BEAM188LABELS`**
- **`RADIOSITYLABELS`**
- **`EULERLABELS`**
- **`BEAMRXLABELS`**
- **`THERMALMASSLABELS`**
- **`SHELLMBPLABELS`**
- **`SLASHPNUMLABELS`**
- **`CINTLABELS`**
- **`PRIN_S_LABELS`**
- **`PDMG_LABELS`**
- **`PFC_LABELS`**
- **`CDM_LABELS`**
- **`CONTDEBONDLABELS`**
- **`BEARINGLABELS`**
- **`BLANKLABELS`**

`is_membrane` (type boolean)

If this attribute is present and marked true, the result is only added if the model has a membrane.

`is_six_dof` (type boolean)

Optional, applies for only "R" and "M" type Mechanical results.

`description` (type string)

Free text.

The **`<Solver>`** node references the solver used by the attribute **`name`**. All child **`<Code>`** nodes are the codes associated to the Workbench result for this solver. A code is an alphanumerical string referred by attribute **`name`**. It is possible to define several codes for the same solver.

The **`<vki>`** node is another way to match the solver result. This node has the following attributes:

`vki_root_name` (type string)

The vki root name as described in the documentation/API.

`vki_aux_name` (type string)

Only required if **`vki_root_name`** is set to UNKNOWN. Matches the vki result in case of unknown vki type.

> **Note:**
>
> If solver-specific data and vki data are fulfilled for a specific result, Mechanical will first try to match the vki code. If it is not found, it will try to match the solver-specific code. For example, when reading a result file, the vki dataset E_ELAST is found, with solver-specific data EE. Mechanical looks for a Workbench result having vki root name E_ELAST. If not found, it looks for a Workbench result having Abaqus code EE.

Limitations

- If the XML of the configuration file is missing or not well-formed, no results are available.

- If a result has invalid syntax (missing XML components, invalid attributes), it is not available.

- Mechanical must be restarted if any changes are made to the result configuration file during operation.

The following results don't appear in the XML file, as they have behavior which is not compatible:

- SERR - not associated to VKI or ABAQUS, is available if S and SVOLU are also available.

- NDIR, EDIR - not associated to VKI or ABAQUS, is available if the model contains shells.

Customizing Workbench with Ansys ACT

With Ansys ACT, you can create *extensions* to customize supported Ansys products, including Workbench.

ACT provides three levels of Workbench customization:

- **Feature Creation**

  Feature creation is the direct, API-driven customization of an Ansys product. In addition to leveraging the functionality already available in Workbench, ACT enables you to add functionality and operations of your own. Examples of feature creation include the creation of custom loads and geometries, the addition of custom preprocessing or postprocessing features, and the integration of third-party solvers, sampling methods, and optimization algorithms.

  For general feature creation information, see Feature Creation in the *ACT Developer's Guide*. For Workbench-specific feature creation information, see Workbench Feature Creation in the *ACT Customization Guide for Workbench*.

- **Simulation Workflow Integration**

  Simulation workflow integration is the incorporation of external knowledge such as apps, processes, and scripts into the ANSYS ecosystem. Using ACT, you can customize simulation workflows in the Workbench **Project** tab.

For example, you can create custom *task groups* (systems) and custom *tasks* (components) for insertion in the **Project Schematic**, constructing consistent and cohesive simulation workflows that allow your business-specific elements to coexist and interface with pre-built Ansys solutions.

For components that support updates in Remote Solve Manager (RSM), you can use ACT to configure job submission options for remote and background updates.

For more information, see Simulation Workflow Integration and Simulation Workflow Integration Examples in the ACT Customization Guide for Workbench.

Process Compression

Process compression is the encapsulation and automation of existing processes available in an Ansys product. The result is a *wizard* that provides guidance within the product workflow, walking the end user step-by-step through a simulation. You can create single wizards to run in the Workbench **Project** tab or mixed wizards to be run across several Ansys products.

Wizards allow you to leverage both the existing functionality of Workbench and the scripting capabilities of the Workbench framework API. With a wizard, you can manipulate existing features and simulation components, organizing them as needed to produce a custom automated process. You can also customize the user interface of a wizard as a whole and define layouts for individual interface components.

For general wizard information, see Simulation Wizards in the *ACT Developer's Guide*. For Workbench-specific wizard information, see Workbench Wizards in the *ACT Customization Guide for Workbench*.

Creating, Opening, and Importing Projects and Files

When you start Workbench, or open a project in Workbench, it opens to the **Project** tab. The **Project** tab is the main workspace where you build your analysis.

Like all tabs, the **Project** tab is made up of different panes that can be reconfigured according the information you want to see. By default, the **Project** tab has the **Toolbox** pane and the **Project Schematic** pane.
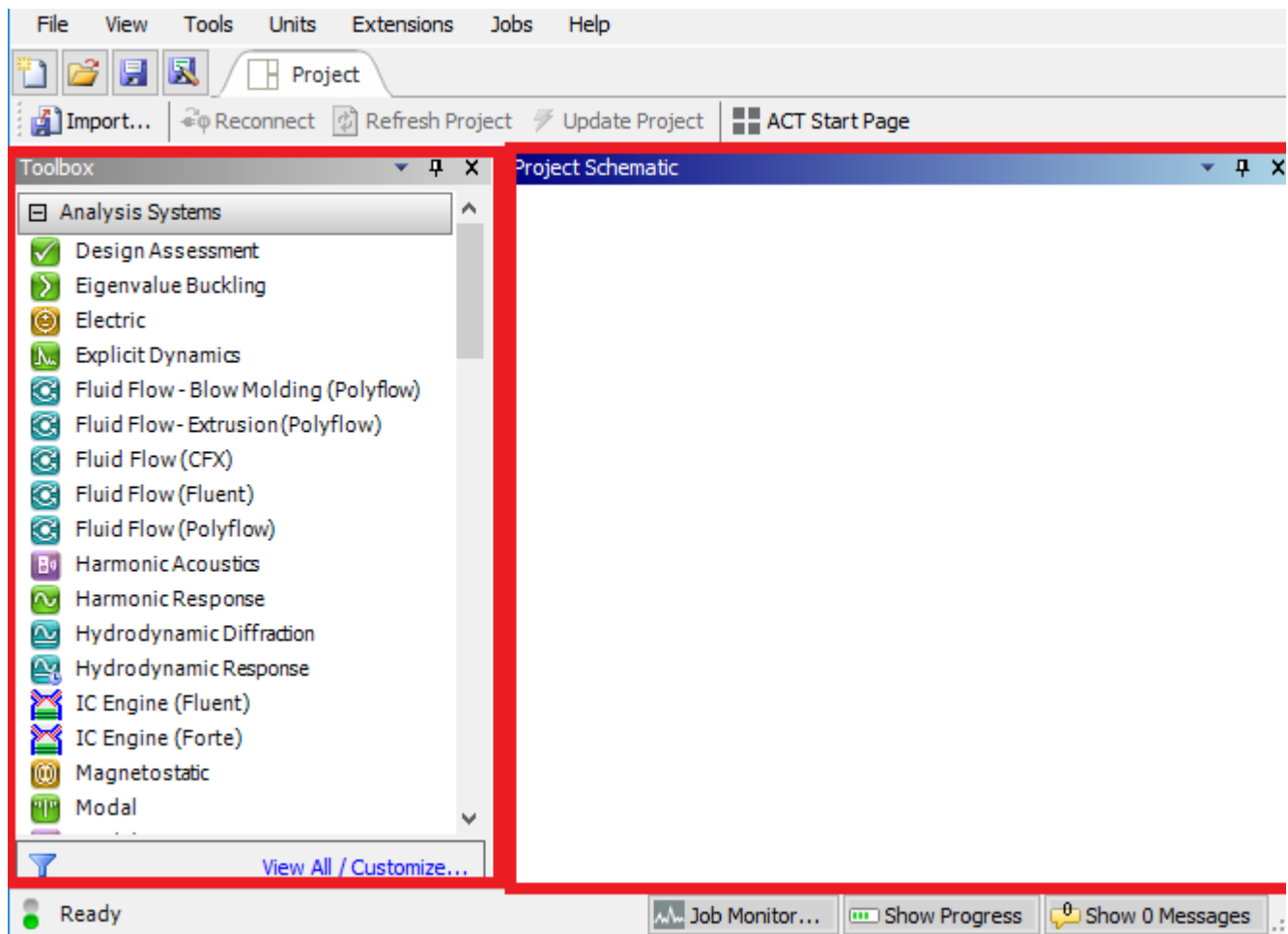
You can start adding systems to the **Project Schematic** immediately or you can create, open, or import a project.

Related topics:

Creating a New Project

Opening an Existing Project or File

Importing Files

Creating a New Project

When Workbench opens, it displays a blank **Project Schematic** that you can start adding systems to. If you need to open a new blank **Project Schematic**, you can create a new project.

To create a new project:

1. From the menu bar, select **File** → **New**.

   If you have added a system to the **Project Schematic**, the following message is displayed:

   ```
   The project has been modified. Do you want to save it?
   ```

2. To save the modified project before opening the new one:

   a. From the dialog box, select **Yes**.

   b. Type a name for the project.

   c. Browse to the location you want to save the project in.

   d. Click **Save**.

   To delete the project, click **No**.

Opening an Existing Project or File

You can open a previously saved project or file to continue working with it.

On Windows systems, you can also double-click a `.wbpz` archive file from the directory it is saved in to open the archive.

If you have deleted any project files through the file system and not through the Workbench interface, you may see a warning message when you attempt to open the project. The error message lists the missing files. The corrective action depends on the type of files that are missing:

- For files that are programmatically integrated with Workbench (such as database files, geometry files, engineering data files, application-generated files, and so on), open the **Files** pane and use the context menu to repair a file or permanently remove it (p. 276) from the project's **Files** list if it is no longer needed. Repairing these types of files should be done carefully; it is possible to use files or file types that are not similar to the original. Missing or erroneous files can cause unintended consequences for project stability and usability.

- For files that are not programmatically integrated with Workbench (such as user-supplied input files, `dpn` files, and so on), you can replace them using your file system and then open the project again. These files may not be necessary for the project; in this situation, you can safely ignore the warning message.

To open an existing project:

1. From the menu bar, select **File** → **Open**.

2. Optionally, select a file type from the file selection filter list.

    - **Workbench Files (*.wbpj, *.wbpz, *.wbjn)**

    - **Project Files (*.wbpj)**

    - **Archive Files (*.wbpz)**

    - **Journal Files (*.wbjn)**

    - **Python Script Files (*.py)**

    - **All Files (*.*)**

3. Browse to the file location.

4. Select the file to open.

5. Click **Open**.

6. If you are opening an archive file (`*.wbpz`), browse to the location where you would like to extract the archive to and click **Save**.

Importing Files

When working on a Workbench project, you may need to import files such as input files, existing mesh files, and geometries.

When you edit an imported file, Workbench saves a copy of the file to the project directory, rather than overwriting the original file. This process ensures that your original files are never compromised.

Importing an archived project adds the data contained in the project to the currently-loaded project. Systems in the imported project are added to the schematic, along with all the data they contain. The connections between imported systems are kept, but no connections are made to the systems in the existing project by the import process.

Older archived projects containing Ansoft systems cannot be imported. To import these projects, re-archive the project from the current version first.

To import files:

1. Select **File** > **Import**. There is also a **Import** option on the **Project** tab.

2. Optionally, specify a desired file type from the file selection filter list. File types include:

    - **Workbench Project Archive Files (*.wbpz)**

    - **Workbench R10/11 Project (*.wbdb)**

    - **Importable Mechanical File (*.dsdb, *.mechdat)**

    - **NASTRAN File (*.nas)**

    - **NASTRAN Data File (*.d**

    - **ABAQUS Input (*.inp)**

- **Fluent Mesh File (*.msh)**

- **Importable Mesh File (*.cmdb, *.meshdat)**

- **Engineering Data File (*.xml, *.engd)**

- **Geometry File (*.sat, *.sab, *.ant, *pmdb, *.agdb, *.model, *.exp, *.session, *.div, *CATPart, *CATProduct, *.3dxml, *.ipt, *.iam, *.jt)**

- **FEModeler Database (*.fedb)**

- **Mechanical APDL CDB (*.cdb)**

- **ICEM CFD Mesh File (*.uns)**

- **Fluent Case File (*.cas)**

- **NASTRAN Bulk Data File (*.bdf)**

- **LS-DYNA Input File (*.k**

- **Ansys Amco File (*.acm**

- **ICEM CFD Project (*.prj**

- **BladeGen File (*.bgd)**

- **AQWAWB Database (*.a**

  AUTODYN Save File (*.a

- **CFX-Pre-Case File (*.cfx**

- **CFX-Solver Input File (***

- **CFX-Solver Results File**

3. Browse to the file location.

4. Select the file to import.

5. Click **Open**.

All data for the current design point is imported; systems that were up-to-date in the archive are up-to-date after import and contain any generated results. User files in the imported project are added to the current project and are renamed if needed to avoid conflicts with existing files.

Parameters from the imported project are added to the parameter set. If the imported project contains design points, those design points are added to the design point table, using the current parameter values from the existing project. Retained design point files from the imported project are not added to the new project. However, if the archived project to be imported contains DX systems, the DX systems are not imported, although the other systems in the project are imported.

Importing Legacy Databases

Workbench offers several methods to import databases from earlier releases:

- **Context Menu Import**: Create the appropriate system and cell in the **Project Schematic**, right-click the cell, and import the file into the cell from the context menu.

- **File Import**: From the menu bar, select **File** → **Import**, then select the file to import. The proper systems and cells are created and populated with the data from the imported file. You can also drag-and-drop one or multiple files from Windows Explorer onto the **Project Schematic**. These files are treated as if they were imported.

- **Project Import**: From the menu bar, select **File** → **Import**, then select the project to import. All necessary systems, cells, and links are established and populated with data from the various files that made up the earlier release's project.

In each of these cases, you must launch the associated application to use the imported file, which is typically the same application that was used to edit the file in the earlier release. The imported files are not undergoing any transformation in this release of Workbench; rather, access to the files is being co-ordinated through the **Project Schematic** interface.

Not all products/databases use all of these methods.

DesignXplorer Release 11 (`.dxdb`) file import is not supported; however, basic parameter import is supported.

The following table shows which applications can use the three methods described above.

Application	Context Menu Import	File Import	Project Import
Ansys Workbench, Release 10 or 11 (`.wbdb`)			X
Mechanical (`.dsdb`)		X	X
Engineering Data (`.eddb`)			X (except convections and load histories)
Engineering Data (`.xml`, `.engd`)		X	
Ansys Aqwa, Release 12 only (`.aqdb`)		X	X
Autodyn (`.ad`)		X	X
Mechanical APDL (`.inp`, `.dat`, `.cdb`, `.mac`, `.anf`)	X		
DesignModeler (`.agdb`)	X	X	X
Meshing (`.cmdb`)		X	X
Fluent (`.msh`, `.cas`, `.dat`)	X		
CFX (`.cfx`, `.def`, `.res`)	X	X	
BladeGen (`.bdg`)	X	X	

Additional Mechanical Application Import Details

- Any `.dsdb` files and `.wbdb` project files created in Release 11 or earlier containing items that convert into Mesh or Mechanical systems cannot be imported. To import these files, import them first into a version between Release 12 and 19.2, save as file in that version, and then import into Release 2022 R2.

- Legacy `.dsdb` files that contain multiple models are split into multiple files.

- `.dsdb` files become `.mechdb` files internally.

- Separate systems are created to correspond to each of the Release 11 environments. Environments that use the same geometry and model are represented with a link between the **Geometry** and **Model** cells of the systems. Physics environments are typically represented as a link between a **Solution** cell of the originating system and the **Setup** cell of the receiving system. For example, a

thermal condition is represented by a connection between the **Solution** cell of a thermal system and the **Setup** cell of a structural system.

- Information that is transferred from the Release 11 system to the current system includes:

  – Model name

  – Model state

  – Physics type

  – Internal IDs for model, mesh, environment, and answer set

  – Engineering Data file path

  – Material property suppression path

  – Solver files directory path

  – Solver type

  – Geometry preferences

  – Parameters

- Legacy `.dsdb` files that do not contain an environment are imported as systems.

- When you import a database that does not include a result file (such as .mechdat files), certain Mechanical result-based features may be lost. For example, if no result file is available:

  – Results do not deform or animate properly.

  – A Harmonic Acoustics environment has multiple RPM values selected. The **Tabular Data** for a result lists all frequencies in the analysis, not just the frequencies in the selected **RPM Set Number**.

Additional Aqwa Import Details

- Only Ansys Aqwa Release 12 databases can be imported. More recent versions are not imported correctly and may cause functional issues.

- An Aqwa Release 12 database is imported as a standalone Hydrodynamic Diffraction system.

- If the Release 12 database contains multiple analyses, they are converted into multiple Hydrodynamic Diffraction systems.

- Geometry associated with the legacy database is associated with the **Geometry** cell of the Hydrodynamic Diffraction system and is editable.

Additional Autodyn Import Details

Links are not created when Autodyn files are imported from Release 11.0.

Additional CFX Import Details

- If CFX files are included in the `.wbdb` project, they appear in the **Files** pane, but system or links associated with these files are not created. To create a CFX system and import the selected file, select **Import to Schematic** from the **Files** pane. This creates a CFX system and imports the selected file.

- You can import CFX-Solver `.bak` files or full `.trn` files into the **Project Schematic** so that you can postprocess results for debugging when a run fails. You may find it useful to use full `.trn` files as a backup mechanism because all timesteps are retained, instead of just the most recent ones, and it is easier to postprocess multiple timesteps.

Additional DesignModeler Import Details

- DesignModeler files (`.agdb`) can be imported through context menu import.

- `.agdb` and `.modeldv` files can be imported by selecting **File → Import**.

- Links with other cells are automatically generated when `.agdb` files that are present in the Release 11.0 `.wbdb` project are imported into the appropriate system (Mechanical Model or other appropriate system).

- Parameters contained in the `.agdb` file are not immediately published to the **Project Schematic** interface. They are published when the DesignModeler application is opened.

- CAD files that were imported into the `.agdb` are not immediately registered in the **Files** pane of the **Project Schematic**. Registration of these files occurs when the DesignModeler application is opened.

Additional Meshing Import Details

- `.cmdb` files from Release 10 that contain CFX-Mesh data are not supported. To import these files, import them first into Release 11, save as a Release 11 file, and then import into Release 2022 R2.

- Links with other cells are automatically generated when `.cmdb` files that are present in the Release 11.0 `.wbdb` project are imported into the appropriate system (Mechanical model or other appropriate system).

- CFX-Mesh files (`.gtm`, `.cfx`) can be imported via context menu import from the **Mesh** cell of a Mesh system.

Adding Systems to the Project Schematic

In the **Project** tab, you take systems from the **Toolbox** and add them to the **Project Schematic**. Projects can vary in complexity, from a single system representing all the necessary steps for a desired analysis, to a complex set of connected (linked) systems representing coupled analyses or variations in modeling approaches.

> **Note:**
>
> We recommend that a project contain only systems that are relevant to a specific analysis or coupled analysis with a well-defined focus. Adding systems for multiple unrelated analyses to the same project can have an adverse effect on performance and cause corruption with portions of the project.

Most analysis systems are defined by three primary attributes: physics type, analysis type, and solver type. Workbench uses these attributes to determine valid data transfer and system replacement possibilities. For more information on the types of systems, see Working Through a System (p. 67).

Related topics:

System Placement

Adding a System by Double-Clicking

Adding a System using Drag-and-Drop

Adding a System using the Context Menu

Adding Multiple Independent Systems

Creating Connected Systems

System Placement

Systems are added from left-to-right, and from top-to-bottom. All data transfer occurs from left (upstream) to right (downstream); you cannot transfer data from right to left.

When placing or moving systems, it is important that you place receiving systems to the right of sending systems. All processing of data (such as updates) also occurs in the same direction, from left-to-right and top-to-bottom.
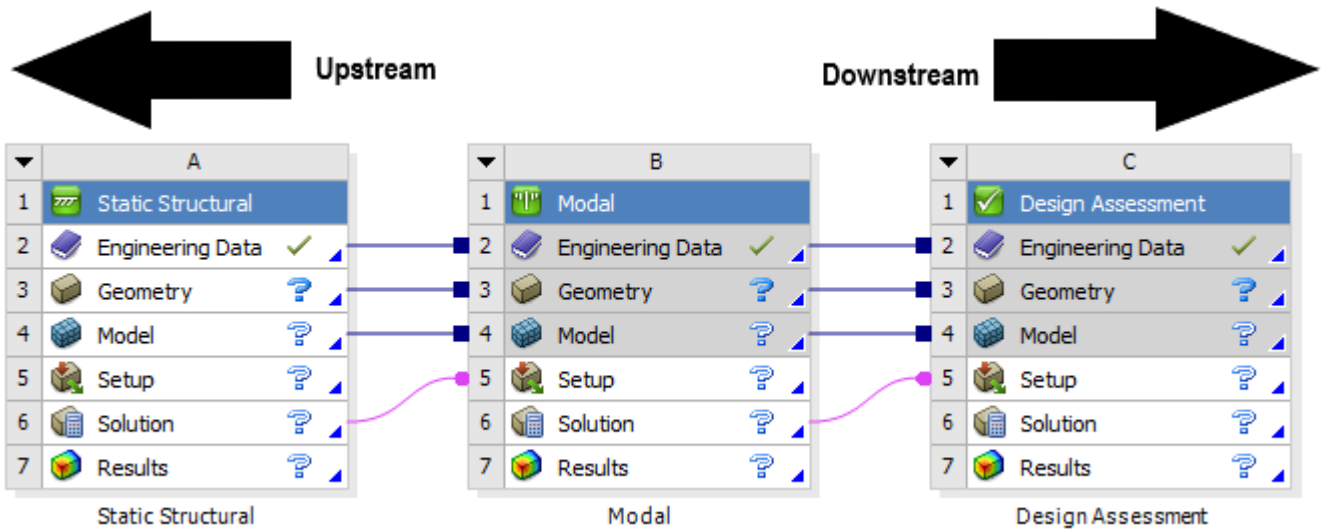
Adding a System by Double-Clicking

The simplest way to add a new system to the **Project Schematic** is to double-click the system in the Toolbox.

The system is placed in a new row in the **Project Schematic**, below any existing systems.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
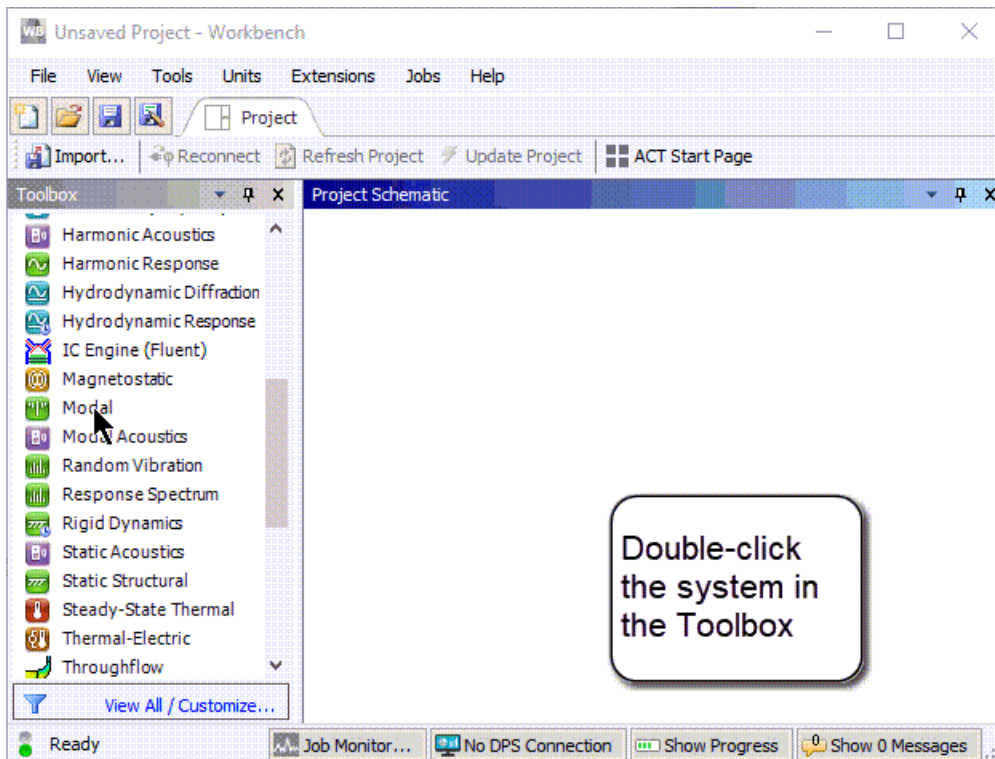
Adding a System using Drag-and-Drop

You can add a system to the **Project Schematic** by dragging it from the **Toolbox** and dropping it in the desired location. This method provides a preview of possible target locations, allowing you to choose the best location for that system.

To add a system using drag-and-drop:

1. In the **Toolbox**, click the required system.

2. Holding down the mouse button, drag the system over to the **Project Schematic**.

3. Hover the mouse over the green drop targets to see the result of dropping the system in that location.

    The drop target turns red when the mouse is over the target. In cases where the new system can be linked to one or more existing systems, drop targets are also shown on the eligible cells of the existing systems.

4. Release the mouse button to place the system onto the drop target.

    Note:

    When you attempt to add an invalid system (for example, if the mesh type is not compatible with the system you are attempting to add), the drop target preview will be visible, but the system is not added when you release the mouse. Details are written to the Messages pane (p. 446).

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
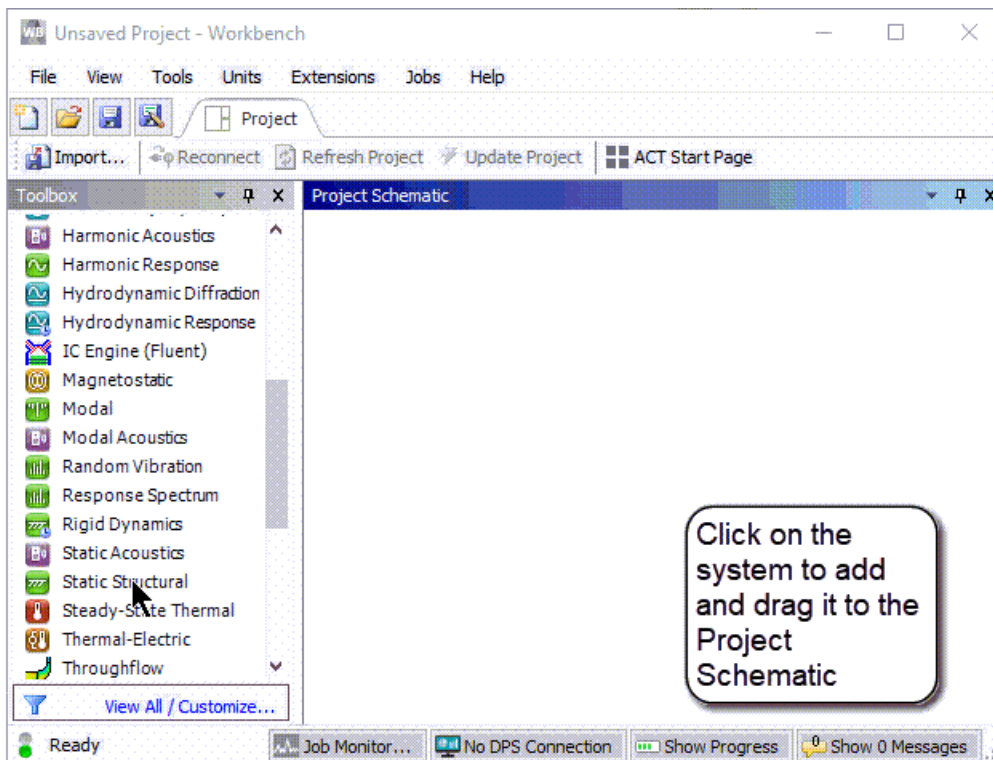
Adding a System using the Context Menu

The context menu includes a list of available systems, divided into categories. From this list, you can browse and select a system to add to the **Project Schematic**.

The available systems in this list reflect your **Toolbox** configuration (p. 25).

To add a system using the context menu:

1. Right-click the white space in the **Project Schematic**.

2. From the context menu, select **New** *category* **Systems** → *system name*.
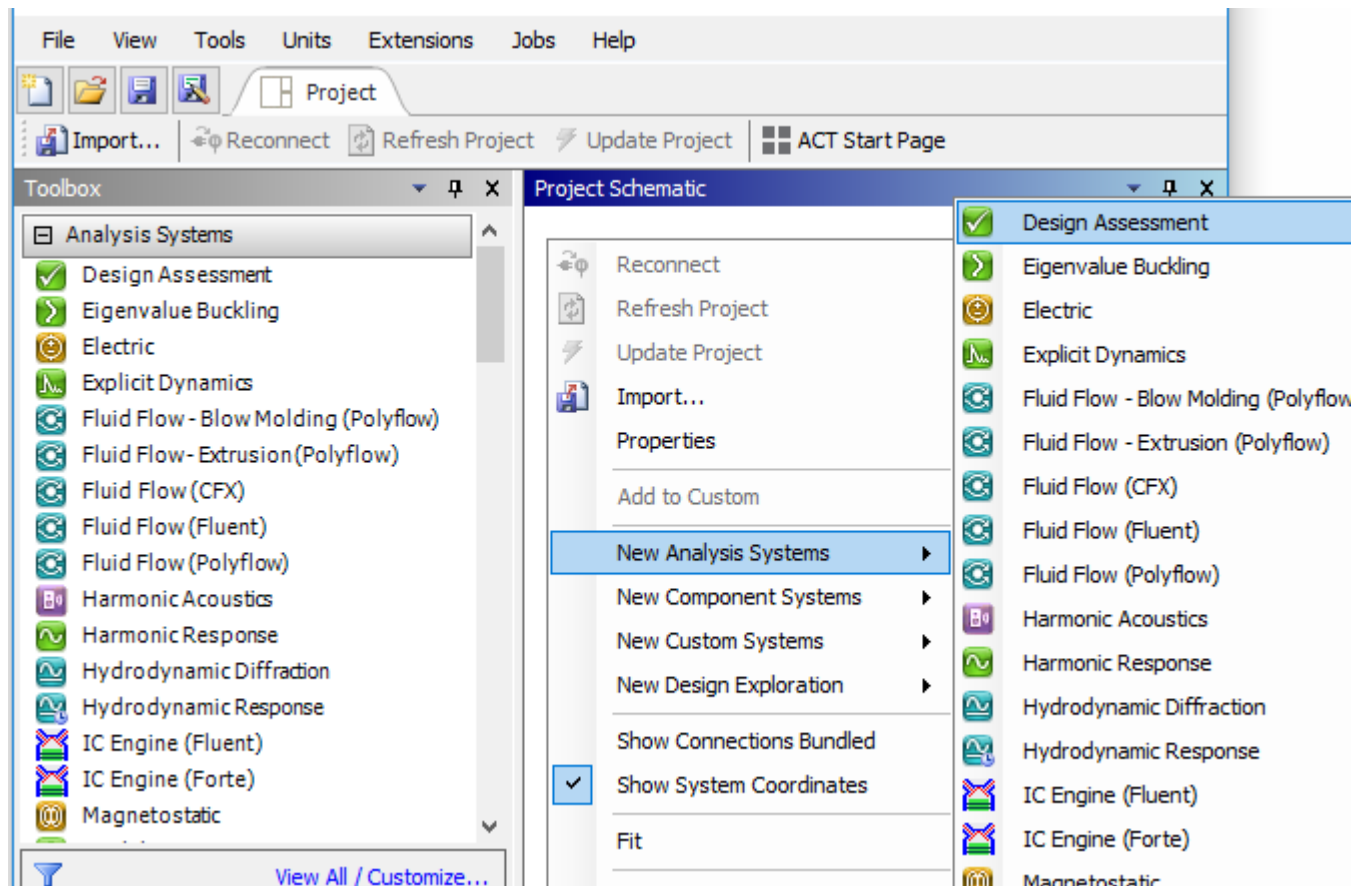
Adding Multiple Independent Systems

Independent systems are systems on the **Project Schematic** that are not connected to other systems by data links. Multiple independent systems can be used to keep the analyses for related but separate components within the same project (for example, two parts of the same vehicle). You may also want to create independent systems if you are analyzing the same model but using different solvers to compare the results, or are using different editors on independent yet related data (for example, you are using Mechanical APDL on an independent input file, and using Mechanical on a geometry file that has related origins to the input file of the Mechanical APDL system).

You can create a new independent system using either of the following methods:

- Double-click the desired system (p. 56).

  The new system is created on the **Project Schematic** below an existing system and is not linked to any existing system.

- Drag-and-drop the desired system (p. 57).
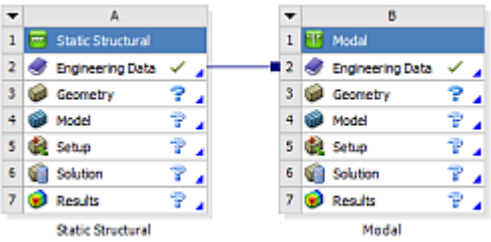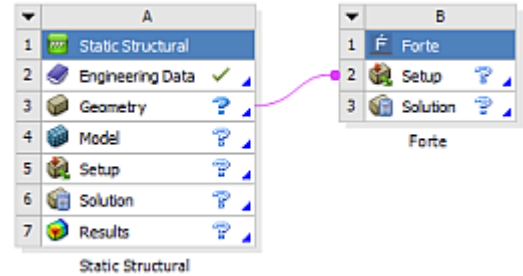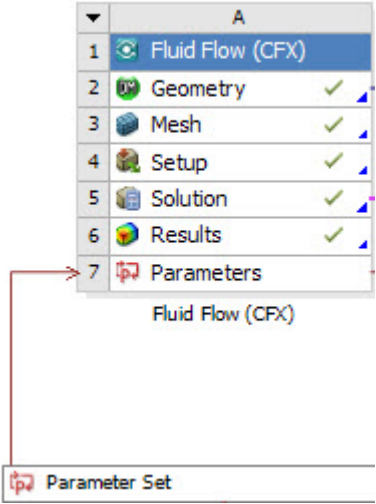
  When you drag the system from the **Toolbox** and move it over the **Project Schematic**, a preview of all possible drop targets is shown. Drop the system on a target that displays the message `Create standalone system`.
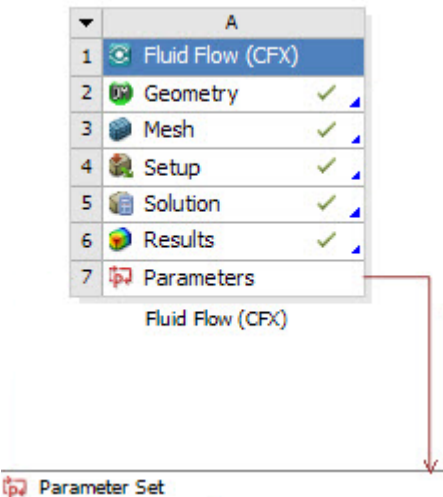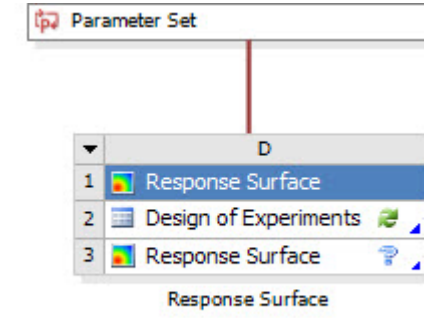
Creating Connected Systems

Connected systems are systems on the **Project Schematic** that are connected together by data links so data can be shared and/or transferred between the two systems. You can use connected systems for sequential physics coupling (for example, thermal-stress) or for sequential simulation steps, such as a pre-stress modal (stress analysis followed by modal), mode-superposition (modal followed by harmonic), and so on. You can either create connections between existing systems or create a new system that is connected to an existing system.

Link Types

Links connecting systems represent data sharing or data transfer between the systems. The following table provides descriptions of links used in the **Project Schematic**.

Link Name	Image	Description
Shared data		Data is shared between systems.
Data transfer		Data is transferred from an upstream system to a downstream system.
Input parameters		A system is consuming input parameters. These links connect systems to the **Parameter Set** bar.

Link Name	Image	Description
Output parameters		A system is providing output parameters. These links connect systems to the **Parameter Set** bar.
Design Exploration system		A Design Exploration system is connected to project parameters. These links connect Design Exploration systems to the **Parameter Set** bar.
Design point data	N/A	Design point data is transferred from a Design Exploration component to a Design Exploration Direct Optimization system. For more information, see Transferring Design Point Data for Direct Optimization in the *DesignXplorer User's Guide*.

System Highlighting

To trace how data is shared and/or transferred between systems, click a system cell. The shared data and data transfer links connected from this cell to upstream and downstream systems are highlighted, allowing you to easily see how the data in this cell is connected to other systems in the project.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
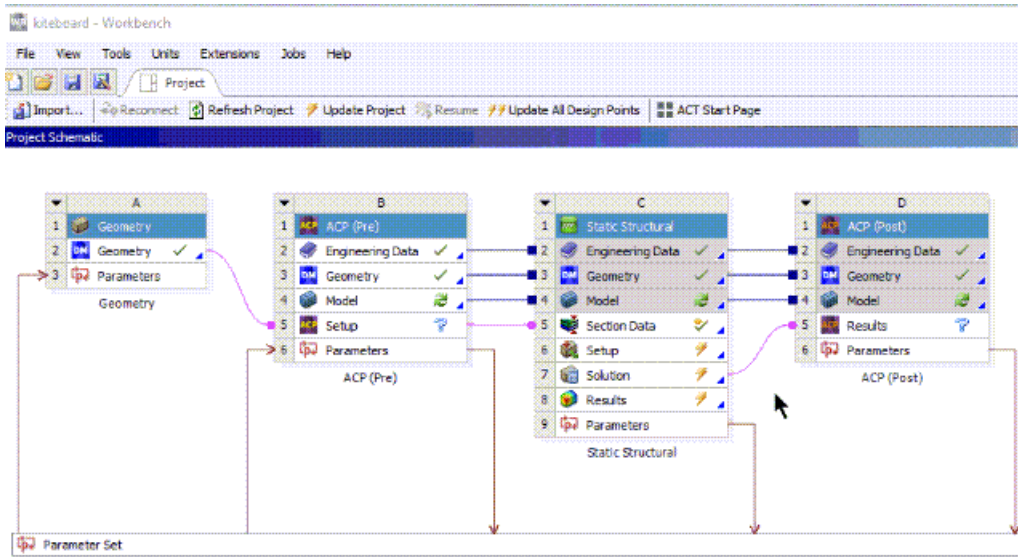
Connecting Two Existing Systems

You can connect systems on the **Project Schematic** by linking one or more component cells (for example, a **Geometry** cell and/or an **Engineering Data** cell) from one system to eligible cells in another system.

To connect two existing systems:

1. To preview possible drop targets, drag a cell from the source system and hold it over cells in the target system.

2. Drop the system on the target best suited to your engineering goals.

Data transfer links are displayed between the connected cells.

Creating a New Connecting System

When you create a new system that is connected to an existing system, Workbench generates data shares and/or data transfers between the systems to achieve a compound analysis. In this case, the two systems are then interdependent.

The new system can be created either upstream (the new system provides inputs to the existing system) or downstream (the new system receives inputs from the new system) of the existing system. The system receiving input is also called the dependent system. To create a new connected system, you can either a drag-and-drop the new system or use the **Transfer Data** context-menu option.

Connecting a New System Using Drag-and-Drop

You can use drag-and-drop functionality to create a new dependent system.

To drag-and-drop a new connected system:

1.   Drag a system from the **Toolbox** to the **Project Schematic**.

2.   Move the new system over cells in the existing system to preview available drop targets.

3.   Drop the system on the desired target.

The new system is created downstream on the **Project Schematic**.

The following animation demonstrates the various drop targets and their results. For example, selecting the **Geometry** cell as the drop target results in a second system that shares the Engineering Data and Geometry data. Selecting the **Model** cell as the drop target results in a second system that shares the Engineering Data, Geometry and Model data. The preview for each drop target indicates how the new system will be connected to the existing one.

If you select the **Solution** cell as the drop target, the preview displays the text `Share A2:A4 Transfer A6`, meaning that the data for cells A2, A3, and A4 (**Engineering Data**, **Geometry**, and **Model** in the following animation) would be shared, and data from cell A6 (in this case, **Solution**) would be transferred as input to the new system. It is important to review each potential drop target carefully to ensure that you select the target that best suits your needs. Note that in addition to the red, linked drop targets, you can also preview the green independent drop targets.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
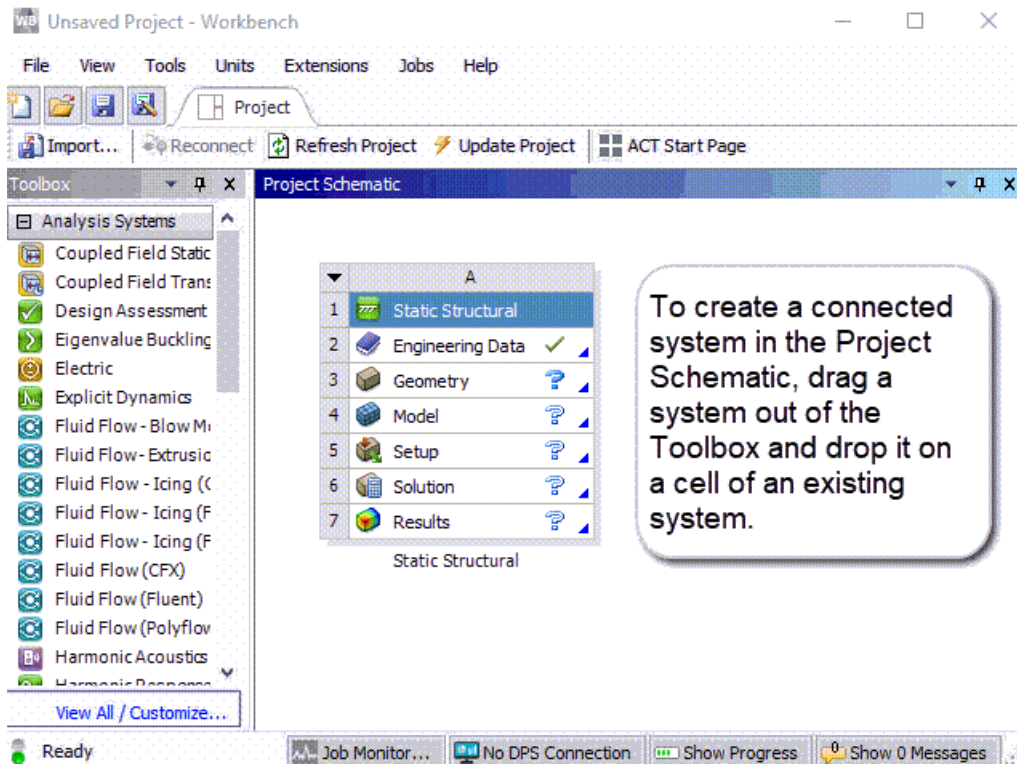
Connecting a New System Using Transfer Data

The **Transfer Data** options available in the context menu allow you to select a cell in an existing system and create a new system either upstream or downstream of that system.

To transfer data to a new connected system:

1. Right-click the target cell in the existing system.

2. Select from the following options:

    *   To create the new system upstream of the existing system (so the existing system is dependent), select **Transfer Data from New** from the context menu. Select your new system from a list of all possible analysis types that can *provide data* to the existing cell.

    *   To create the new system downstream of the existing system (so the new system is dependent), select **Transfer Data to New** from the context menu. Select your new system from a list of all possible analysis types that can *accept data* from the existing cell.

When using either of the **Transfer Data** options, all possible cells are shared, up to the position of the selected cell.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.

You can add a connected system upstream of an existing system by using Transfer Data From New.
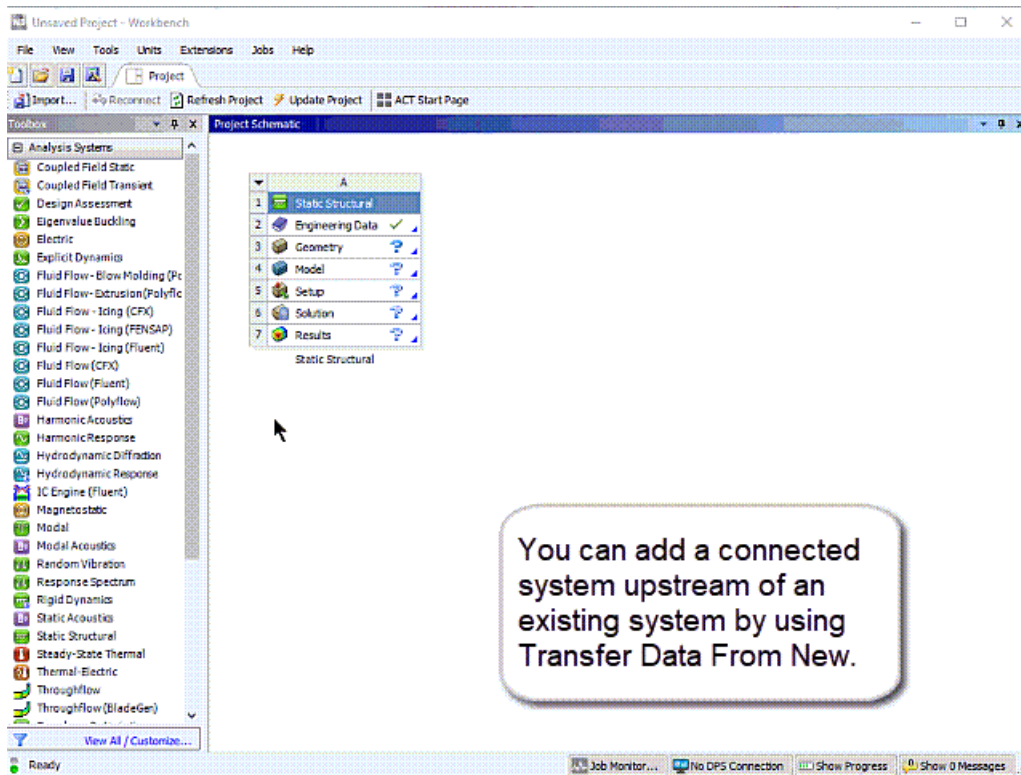
Deleting Links

In most cases, you can delete links by right-clicking the link and selecting **Delete** from the context menu. The data associated with the cell in the upstream system is copied to the downstream system so the cells can be edited independently.

In some cases (for example, links between two mechanical model cells), you cannot delete shared data links. In these cases, the linked cells in the downstream system have a gray background (as shown in the following image).
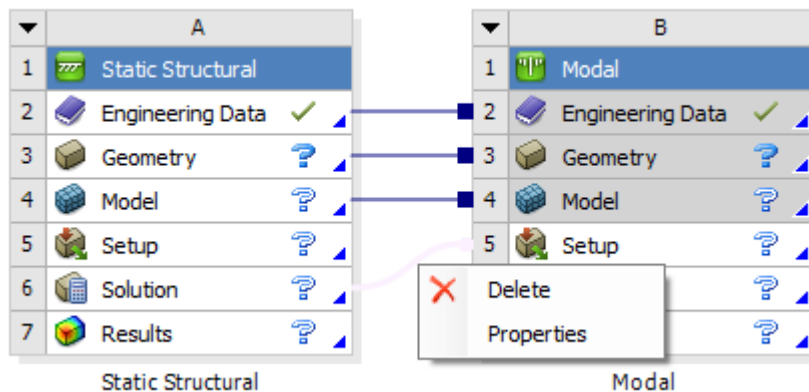
Working Through a System

Workbench provides you with a straightforward workflow for creating and working through a system. First, you select a system from the Toolbox and add it to the **Project Schematic** (see Adding Systems to the Project Schematic (p. 55)). Then you work through the cells in the system, generally from top-to-bottom, until you have completed all the required steps for your analysis.

In most cases, data flows from top-to-bottom through the system, as well. For example, in a Mechanical system, the geometry must be defined before you can define the model; the **Model** cell uses the geometry defined in the **Geometry** cell as its input.

Because the workflows for different types of analyses differ to some degree, we have included two typical examples of working with analysis systems: one for a mechanical analysis and one for a fluid flow analysis.

Related topics:

Understanding Data Flow

Defining your Simulation Geometry

Basic Mechanical Analysis Workflow

Basic Fluid Flow Analysis Workflow

Analysis Systems

Component Systems

Custom Systems

Design Exploration

Submitting Solutions

Opening Mechanical without a License (Read-Only Mode)

Understanding Data Flow

To complete your analysis, you work downward through each cell in order. In general, data flows downstream (from top-to-bottom within systems and from left-to-right across systems). Output data from upstream component cells is provided as the input data for downstream cells. Output data from certain types of cells in one system can also be transferred and/or shared with cells in another system.

The following example shows two systems in the **Project Schematic**, a **Fluid Flow (Fluent)** system (system A) and a **Static Structural** system (system B).
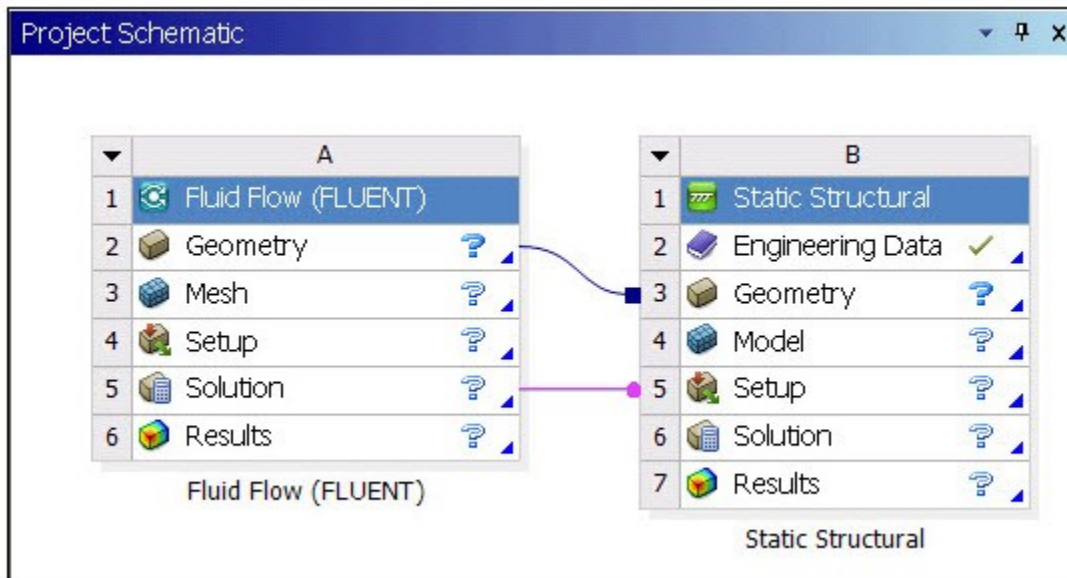
The geometry from the **Geometry** cell in system A becomes the input for the **Mesh** cell of system A. The mesh generated in the **Mesh** cell of system A, in turn, becomes input to the downstream **Setup** cell of system A, and so on.

Data also flows between system A and system B. These two systems share the same geometry, as indicated by the blue link with the square terminator between the **Geometry** cell in system A and the **Geometry** cell in system B.

Solution data from System A is provided to the **Setup** cell of system B, as indicated by the fuchsia connector with the round terminator.

At a glance, you can see the data relationship between the two types of analysis systems.

Defining your Simulation Geometry

All analysis systems and several component systems begin with a geometry-definition step. You can define the geometry differently depending on the type of simulation you are running. In most cases, you use the **Geometry** cell. With the **Geometry** cell, you can:

• Create a geometry in DesignModeler or SpaceClaim.

• Import an existing geometry:

  – From neutral formats such as IGES, STEP, Parasolid, and ACIS.

  – From CAD files.

  – From a CAD session running on the same machine.

  – From ECAD files.

For Fluid Flow simulations, you can also start with an imported mesh or case file; see Basic Fluid Flow Analysis, Starting from an Imported Mesh (p. 73) for details.

To specify the geometry using the context menu:

1. Right-click the **Geometry** cell.

2. Select **New DesignModeler Geometry** or **New SpaceClaim Geometry** to launch those respective programs and create a new model or select **Import Geometry** and browse to an existing CAD model.

Alternatively, you can also launch Workbench directly from some CAD systems. When doing so, Workbench starts with a Geometry system in place and the CAD file already attached.

After you have attached or imported your geometry, the state appears as Up to Date (p. 449), and the icon indicates the type of file imported. Geometry types include:

ACIS (`.sat`)

ANSYS Neutral File (`.anf`)

Autodesk Inventor (`.ipt`, `.iam`)

BladeGen (`.bgd`)

CATIA v4 (`.model`, `.dlv`)

CATIA v5 (`.CATPart`, `.CATProduct`)

Creo Elements/Direct Modeling (`.pkg`, `.bdl`, `.ses`, `.sda`, `.sdp`, `.sdac`, `.sdpc`)

Creo Parametric (formerly Pro/ENGINEER) (`.prt`, `.asm`)

DesignModeler (`.agdb`)

IGES (`.iges`, `.igs`)

JT Open (`.jt`)

Monte Carlo N-Particle (`.mcnp`)

NX (`.prt`)

Parasolid (`.x_t`, `.xmt_txt`, `.x_b`, `.xmt_bin`)

Solid Edge (`.par`, `.asm`, `.psm`, `.pwd`)

SOLIDWORKS (`.SLDPRT`, `.SLDASM`)

SpaceClaim (`.scdoc`)

STEP (`.stp`, `.step`)

Stereolithography (`.stl`).

No file imported.

If you do not need to make any additional changes to your geometry, you can continue working through the analysis as described in the next sections.

If your geometry must be modified before continuing with your analysis, you can edit the geometry in DesignModeler or SpaceClaim. After modifying the geometry, the icon in the **Geometry** cell changes to the icon of the application you used to make those modifications. For a file imported and then modified in DesignModeler, you can open the file in DesignModeler, and the DesignModeler model tree indicates the original source of the geometry.

After the geometry is defined, you can share it with other systems. See Data Sharing and Data Transfer for more information on sharing geometry systems.

For detailed CAD-related information specific to the Ansys DesignModeler application and Workbench, see CAD Integration.

Basic Mechanical Analysis Workflow

After introducing a new Mechanical analysis system to the **Project Schematic** and assigning an appropriate name, the focus is typically directed to the **Geometry** cell, because this is usually the first cell in the system that requires user input. You can edit or define material models in the **Engineering Data** cell, but this example assumes that the default materials are sufficient.

	A		
1	▦ Static Structural		
2	Engineering Data	✓	◢
3	Geometry	❓	◢
4	Model	❓	◢
5	Setup	❓	◢
6	Solution	❓	◢
7	Results	❓	◢
	My Static Analysis		
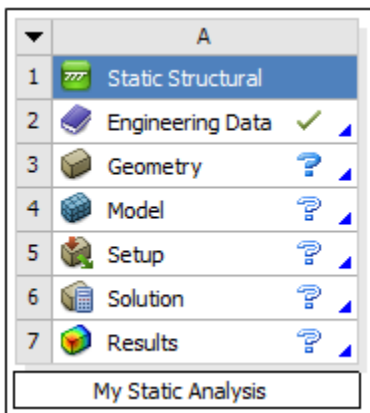
Important:

You are not required to define a geometry. You can open Mechanical without an attached geometry. You can simply skip the step of specifying a geometry and open Mechanical. This can be done to create a template for use with multiple geometries. Moreover, you are not required to define an analysis system. You can begin your simulation with a **Mechanical**

Model system and build the simulation as desired until you wish to add a geometry as well as specify an analysis type. This flexibility can be beneficial.

You typically work through the system from top to bottom. Use the context menus for each cell to view and select operations that can be performed for that cell.

1. Import or create a geometry:

    •   To import an existing geometry, right-click the **Geometry** cell and select **Import Geometry**. From the context menu, select a previously used geometry, an active CAD model, or browse to a desired folder location.

    •   To create a new geometry, select **New DesignModeler Geometry** or **New SpaceClaim Geometry**, as appropriate.

2. To define all loads and boundary conditions, right-click the **Setup** cell and select **Edit**.

    Mechanical opens. Set up your analysis using the application's tools and features.

3. Typically, you solve your analysis from within Mechanical, however, you can right-click the **Solution** cell and select **Update**. The results appear automatically.

For more information on setting up and running specific Mechanical analyses, see Analysis Systems (p. 74).

Basic Fluid Flow Analysis Workflow

After introducing a new Fluid Flow analysis system to the project schematic and assigning it an appropriate name, the focus is typically directed to the **Geometry** cell, because this is usually the first cell in the system that requires user input. An example of both a CFX and a Fluent fluid flow system is shown below.
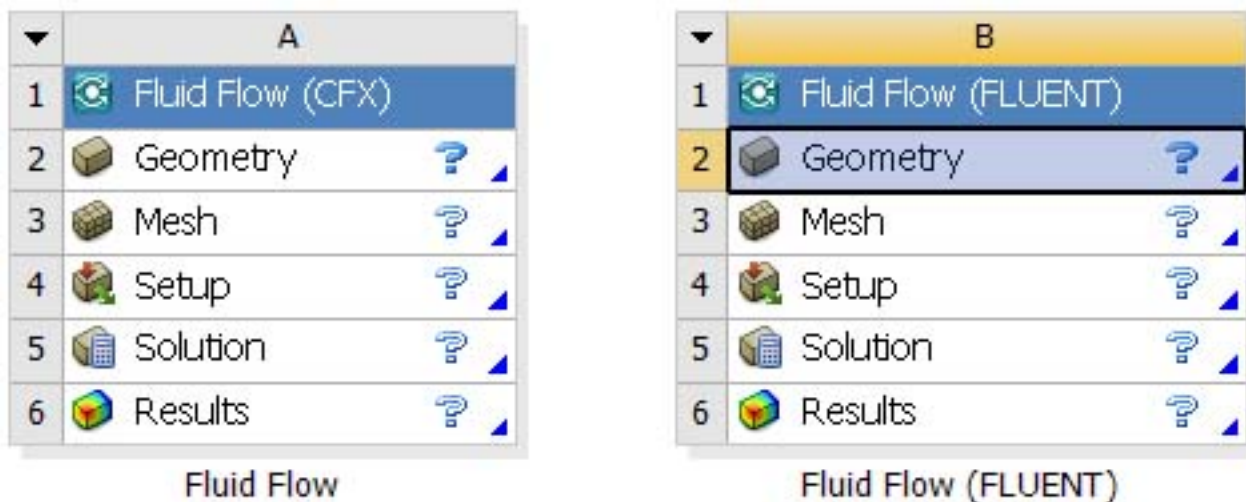
Fluid Flow

Fluid Flow (FLUENT)

You typically work through the system from top to bottom. Use the context menus for each cell to view and select operations that can be performed for that cell. For fluid flow systems, the process is somewhat

flexible; you can start from geometry, from an existing mesh, or from an existing case file; each is described in the following sections.

Basic Fluid Flow Analysis, Starting from Geometry

Typically, you start working through the system at the **Geometry** cell.

1. To launch a geometry program and create a new model, right-click the **Geometry** cell and select **New DesignModeler Geometry** or **New SpaceClaim Geometry** from the context menu or select **Import Geometry** and browse to an existing CAD model.

    For details, see Defining your Simulation Geometry (p. 68).

2. After your geometry is defined and the **Geometry** cell shows the green check mark indicating that the cell is up to date, you can proceed to the **Mesh** cell. To launch the meshing application, double-click the **Mesh** cell or right-click the cell and select **Edit** from the context menu.

    To generate a default mesh, right-click the **Mesh** cell and selected **Update**. This generates the mesh in the background without launching the meshing application.

    When this step has successfully completed, the **Mesh** cell displays a green check mark, indicating an up-to-date state.

3. To load the physics pre-processor, double-click the **Setup** cell or right-click the cell and select **Edit** from the context menu.

    If you are working a Fluid Flow (Polyflow) system, the editor is the Fluent application. If you are working with a Fluid Flow (Ansys CFX) system, the editor is CFX-Pre.

    When you have successfully defined your physics, the **Setup** cell displays a green check mark, indicating an up-to-date state.

4. To open the associated **Solution** cell editor, double-click the **Solution** cell or right-click the cell and select **Edit** from the context menu.

    to run the solution in the background, right-click the **Solution** cell and select **Update** from the context menu.

    If you are working a Fluid Flow (Polyflow) system, the associated editor is the Fluent application. If you are working with a Fluid Flow (Ansys CFX) system, the associated editor is the CFX-Solver Manager.

    When the solution is complete, the **Solution** cell displays a green check mark, indicating an up-to-date state.

5. You can now post-process the resulting solution. To open the resulting solution in Ansys CFD-Post, double-click the **Results** cell or right-click the cell and select **Edit** from the context menu.

Once the process has been completed, all cells in the system should display a green check mark, indicating an up-to-date state.

Basic Fluid Flow Analysis, Starting from an Imported Mesh

You can bypass the **Geometry** cell and start the simulation process by importing a mesh.

1. To import your mesh file, right-click the **Mesh** cell and select **Import Mesh File** from the context menu.

2. In the **Open** dialog, browse to find your mesh file and click **Open**.

   The process for CFX and Fluent differs only in the types of files that are supported (as shown by the options shown in the **Files of Type** drop-down list).

   After selecting **Open**, the **Mesh** cell changes to **Imported Mesh** and shows a green check mark, to indicate an up-to-date state. The **Geometry** cell (which must be unused for the **Import Mesh File** option to be shown in the **Mesh** cell context menu) is deleted from the system (examples shown below for both a CFX and a Fluent-based Fluid Flow system).
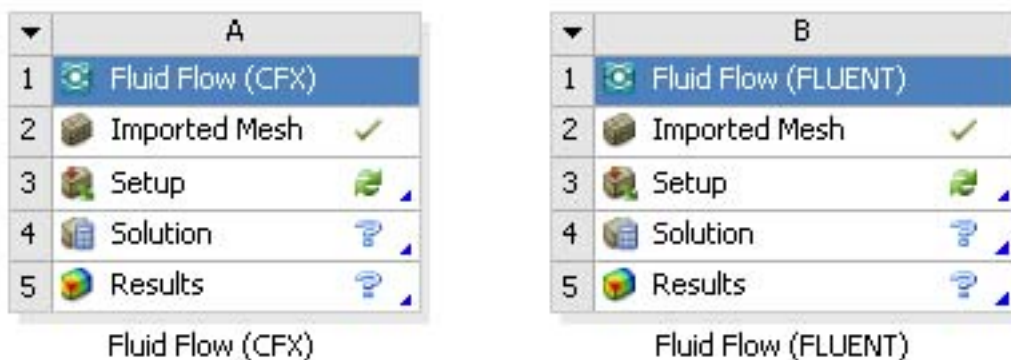
3. To load the physics pre-processor, double-click the **Setup** cell or right-click the cell and select **Edit** from the context menu.

4. From this point, follow the procedure described in Basic Fluid Flow Analysis, Starting from Geometry (p. 72).

Basic Fluid Flow Analysis, Starting from an Imported Case File: Fluid Flow (Ansys CFX) System

You can bypass the **Geometry** cell and the **Mesh** cell and begin the simulation process by importing a case file. The process differs slightly for CFX and Fluent-based systems.

1. Right-click the **Setup** cell and select **Import Case** → **Browse**.

2. In the **Open** dialog, browse to find your case file and click **Open**.

3. If the **Geometry** and **Mesh** cells are unused (empty), then the unused cells are automatically deleted.

If either cell has an incoming or outgoing connection, then it is considered used, and is not deleted.

Because CFX supports multiple meshes imported into a single setup cell, if either of the **Geometry** or **Mesh** cells contains data, neither cell is deleted. As a result, both the Mesh generated in the **Mesh** cell and the mesh imported from the case file is combined in CFX-Pre.

4. Once the import is complete, to start CFX-Pre, double-click the **Setup** cell or right-click the cell and select **Edit** from the context menu.

5. From this point, follow the procedure described in Basic Fluid Flow Analysis, Starting from Geometry (p. 72).

Basic Fluid Flow Analysis, Starting from an Imported Case File: Fluid Flow (Polyflow) System

You can bypass the **Geometry** cell and the **Mesh** cell and begin the simulation process by importing a case file. The process differs slightly for CFX and Fluent-based systems.

1. To import a case file, right-click the **Setup** cell and select **Import Polyflow Case** from the context menu.

2. Select a case file either from the list of recently used case files or select **Browse…**.

3. A warning dialog informs you that completing this action deletes the **Geometry** and **Mesh** cells and any associated data. Click **OK**.

The **Geometry** and **Mesh** cells are deleted from the system. Any existing connections to the **Geometry** and **Mesh** cells are also deleted.

4. From this point, follow the procedure described in Basic Fluid Flow Analysis, Starting from Geometry (p. 72).

Analysis Systems

Analysis systems include all the necessary component cells already defined and ready to be populated. For example, a static structural analysis system includes all the cells needed for the analysis, **Engineering Data** through **Results**.

Some analysis types offer different solvers, noted in parentheses. The features available can differ from one solver to another.

The following analysis systems are available:

Ansys Motion

Coupled Field Harmonic

Coupled Field Modal

Coupled Field Static

Coupled Field Transient

Eigenvalue Buckling

Electric

Explicit Dynamics

Fluid Flow - Icing (CFX)

Fluid Flow - Icing (FENSAP)

Fluid Flow - Icing (Fluent)

Fluid Flow (CFX)

Fluid Flow (Fluent)

Fluid Flow (Fluent with Fluent Meshing)

Fluid Flow (Polyflow)

Harmonic Acoustics

Harmonic Response

Hydrodynamic Diffraction

Hydrodynamic Response

LS-DYNA

LS-DYNA Restart

Magnetostatic

Modal Acoustics

Modal

Random Vibration

Response Spectrum

Rigid Dynamics

Speos

Static Acoustics

Static Structural

Steady-State Thermal

Structural Optimization

Substructure Generation

Thermal-Electric

Throughflow and Throughflow (BladeGen)

Transient Structural

Transient Thermal

Turbomachinery Fluid Flow

Ansys Motion

The **Ansys Motion** system performs Multibody Dynamic (MBD) Analysis, a means of analyzing the dynamic behavior of a system of interconnected bodies.

This system is configured in Ansys Mechanical, using the Ansys Motion solver to compute the solution.

The system is an Ansys ACT app and is not automatically visible in the toolbox until it is installed. For installation instructions, see Adding Motion Analysis.

To work through a **Ansys Motion** system:

1. To add a **Ansys Motion** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

   Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   For information on standard Mechanical features, see Using Standard Mechanical Features. For Ansys Motion specific features, see Define Motion Specific Features.

5. To connect the **Ansys Motion** system to a **Harmonic Acoustics** system:

   a. Drag the **Harmonic Acoustics** system from the **Toolbox** to the **Project Schematic**.

   b. Drag the **Solution** cell of the **Ansys Motion** system onto the **Setup** cell of the **Harmonic Acoustics** system.

Coupled Field Harmonic

The **Coupled Field Harmonic** system determines the steady-state response of a structure and the surrounding fluid medium to loads and excitations that vary sinusoidally (harmonically) with time. The system supports 2D coupled, structural acoustic, and structural electric physics.

This system is configured in Ansys Mechanical, using the Mechanical APDL solver to compute the solution.

To work through a **Coupled Field Harmonic** system:

1. To add a **Coupled Field Harmonic** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

   Once the analysis type is selected, the following prompt displays.
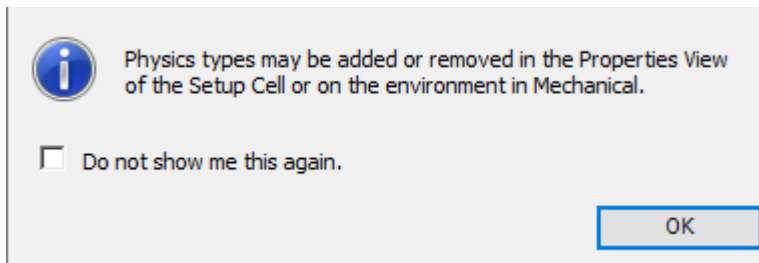
   This prompt informs you that you can automatically create a **Physics Region** (object) in the downstream Mechanical system by selecting physics types in the **Setup** cell properties. Both the **Structural** and **Electric** properties, as illustrated below, are read-only properties that are always active. The **Acoustics** property is also read-only and is off by default.

2. To ensure the prompt is not displayed again, select the **Do not show me this again** check box.

3. To close the prompt, click **OK**.

4. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

5. To import the model, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

6. In the Mechanical application window, complete your analysis using the application's tools and features.

7. To link one Coupled Field Harmonic analysis with another one:

   a. Drag a second **Coupled Field Harmonic** system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

b. Drag the **Solution** cell of the first system to the **Setup** cell of the second system.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
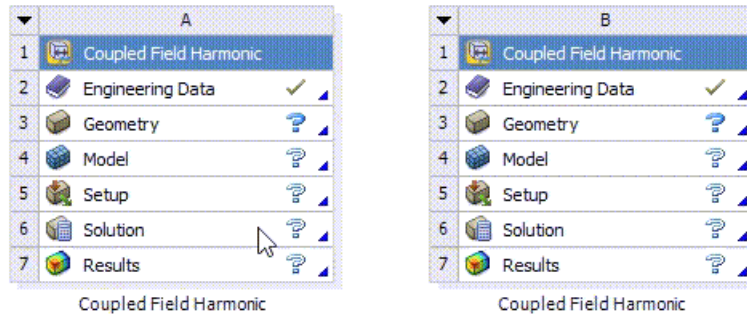
Coupled Field Modal

The **Coupled Field Modal** system models a structure and the surrounding the fluid medium to determine frequencies and standing wave patterns within a structure. The system supports 2D coupled, structural acoustics, and structural electric physics.

This system is configured in Ansys Mechanical, using the Mechanical APDL solver to compute the solution.

To work through a **Coupled Field Modal** system:

1. To add a **Coupled Field Modal** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

    Once the analysis type is selected, the following prompt displays.
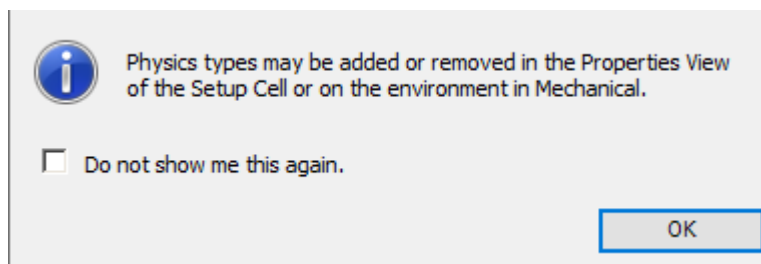
This prompt informs you that you can automatically create a **Physics Region** (object) in the downstream Mechanical system by selecting physics types in the **Setup** cell properties. Both the **Structural**, **Acoustics**, and **Electric** properties, as illustrated below, are read-only properties that are always active. The **Acoustics** property is also read-only and is off by default.

Coupled Field Modal

2. To ensure the prompt is not displayed again, select the **Do not show me this again** check box.

3. To close the prompt, click **OK**.

4. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

5. To import the model, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

6. In the Mechanical application window, complete your analysis using the application's tools and features.

Coupled Field Static

The **Coupled Field Static** system determines the displacements, stresses, strains, and forces caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed. The loads and the responses are assumed to vary slowly with respect to time. The system supports 2D coupled, structural thermal, structural acoustics, and structural electric physics.

This system is configured in Ansys Mechanical, using the Mechanical APDL solver to compute the solution.

To work through a **Coupled Field Static** system:

1. To add a **Coupled Field Static** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

    Once the analysis type is selected, the following prompt displays.

Physics types may be added or removed in the Properties View of the Setup Cell or on the environment in Mechanical.

☐ Do not show me this again.

OK
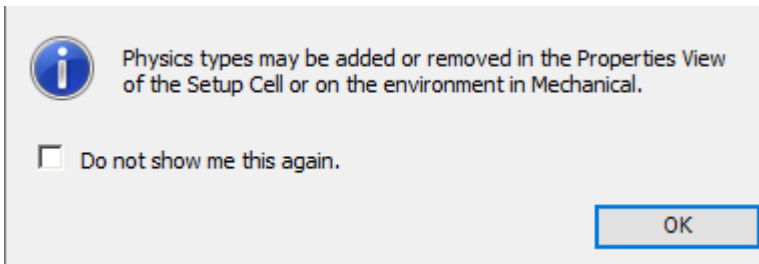
This prompt informs you that you can automatically create a **Physics Region** (object) in the downstream Mechanical system by selecting physics types in the **Setup** cell properties. Both the **Structural** and **Thermal** properties, as illustrated below, are read-only properties that are always active. The **Acoustics** and **Electric** properties are also read-only and are off by default.
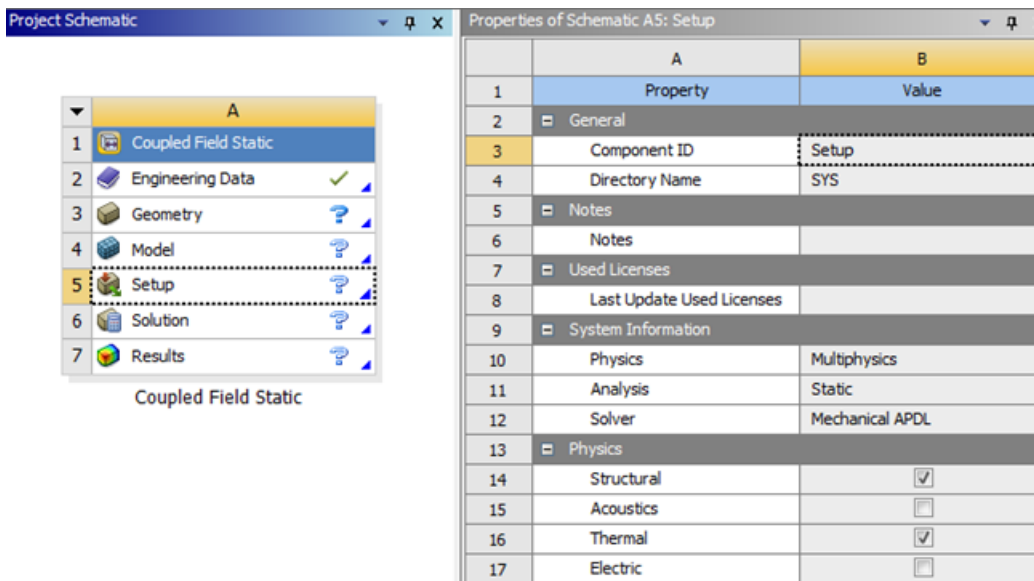
Coupled Field Static

2. To ensure the prompt is not displayed again, select the **Do not show me this again** check box.

3. To close the prompt, click **OK**.

4. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

5. To import the model, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

6. In the Mechanical application window, complete your analysis using the application's tools and features.

Coupled Field Transient

The **Coupled Field Transient** system determines the time-varying displacements, strains, stresses, and forces as they respond to any transient loads. The system supports 2D coupled, structural acoustics, structural thermal, and structural electric physics.

80

This system is configured in Ansys Mechanical, using the Mechanical APDL solver to compute the solution.

To work through a **Coupled Field Transient** system:

1. To add a **Coupled Field Transient** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

   Once the analysis type is selected, the following prompt displays.
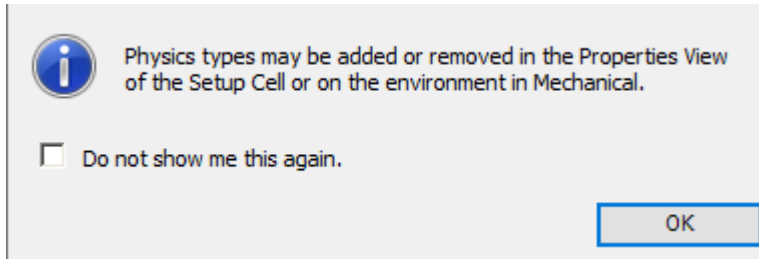
   This prompt informs you that you can automatically create a **Physics Region** (object) in the downstream Mechanical system by selecting physics types in the **Setup** cell properties. Both the **Structural** and **Thermal** properties, as illustrated below, are read-only properties that are always active. The **Acoustics** and **Electric** properties are also read-only and off by default.
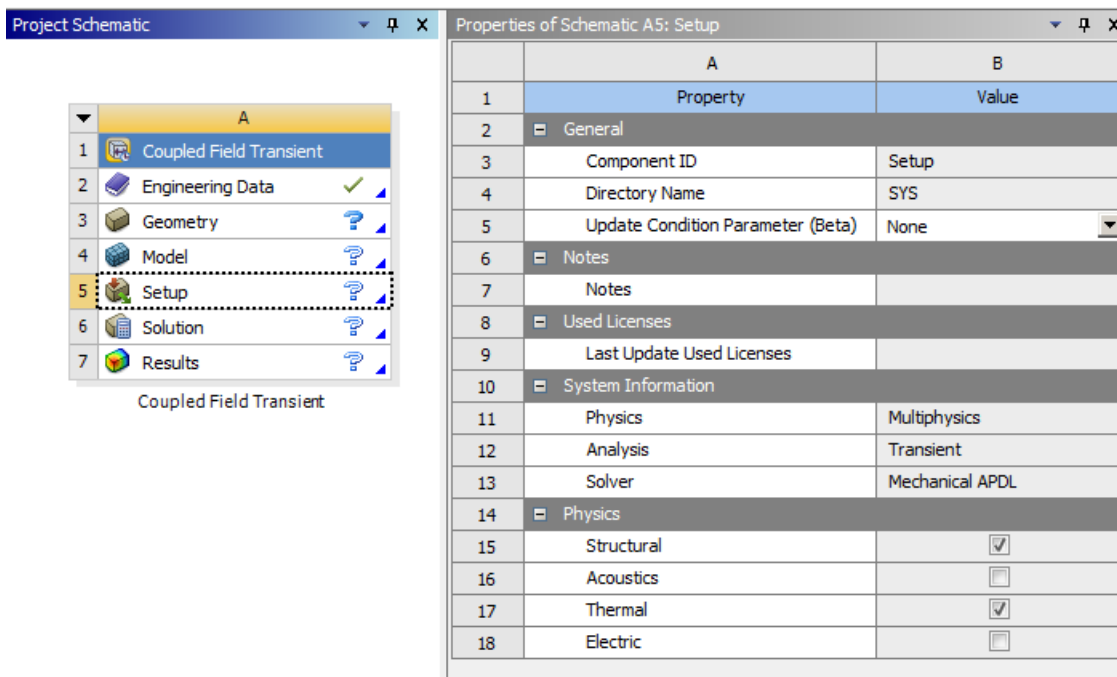
2. To ensure the prompt is not displayed again, select the **Do not show me this again** check box.

3. To close the prompt, click **OK**.

4. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

5. To import the model, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

6. In the Mechanical application window, complete your analysis using the application's tools and features.

Eigenvalue Buckling

The **Eigenvalue Buckling** system predicts the theoretical buckling strength of an ideal elastic structure. This method corresponds to the textbook approach of elastic buckling analysis; for instance, an Eigenvalue Buckling analysis of a column matches the classical Euler solution. However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. The Eigenvalue Buckling analysis often yields quick but non-conservative results.

This analysis is configured in Ansys Mechanical, using the Ansys or the Samcef solver to compute the solution.

An Eigenvalue Buckling analysis must follow a prestressed static structural analysis. Follow the instructions in Static Structural (p. 98) to build a prestressed Static Structural system, and then complete the following instructions to build and link an **Eigenvalue Buckling** system.

To work through an **Eigenvalue Buckling** system:

1. From the Static Structural system, right-click the **Solution** cell and select **Transfer Data to New → Eigenvalue Buckling**.

   A new Eigenvalue Buckling system is created, with the **Engineering Data**, **Geometry**, **Model**, and **Setup** cells linked from the static structural system.

2. To open the Mechanical application, from the **Eigenvalue Buckling** system, right-click the **Setup** cell and select **Edit** from the context menu or double-click the **Setup** cell.

3. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Eigenvalue Buckling Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

4. On the **Project** tab toolbar, click **Update Project**.

Electric

The **Electric** system supports steady-state electric conduction. Primarily, this analysis type is used to determine the electric potential in a conducting body created by the external application of voltage or current loads. From the solution, other results items are computed such as conduction currents, electric field, and joule heating.

This analysis is configured in Ansys Mechanical, using the Ansys solver to compute the solution.

To work through an **Electric** system:

1. To add an **Electric** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To open the Mechanical application, right-click the **Setup** cell and select **Edit** from the context menu or double-click the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Electric Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Explicit Dynamics

The **Explicit Dynamics** system can perform a variety of engineering simulations, including the modeling of nonlinear dynamic behavior of solids, fluids, gases and their interaction.

This analysis is configured in Ansys Mechanical.

To work through an **Explicit Dynamics** system:

1. To add an **Explicit Dynamics** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To open the Mechanical application, right-click the **Setup** cell and select **Edit** from the context menu or double-click the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   See the Explicit Dynamics Analysis Guide for more information on conducting this analysis.

Fluid Flow - Icing (CFX)

The **Fluid Flow - Icing (CFX)** system performs a complete Icing analysis. It contains cells to connect to a Ansys CFX component system and run airflow with the CFX solver, set up and run droplets with the DROP3D solver, set up and run icing with the ICE3D solver, displace the grid due to ice accretion, and visualize the results in CFD-Post or Viewmerical.

To display this system in the **Toolbox**, install the `FENSAPICE-WB` extension located in the `[Install-Directory]\ANSYS Inc\v222\fensapice\workbench` folder, then load it using the Extension Manager.

For more details, see Creating CFX/Fluent-based Icing Systems in the *Ansys FENSAP-ICE in Ansys Workbench User's Guide*.

Fluid Flow - Icing (FENSAP)

The **Fluid Flow - Icing (FENSAP)** system performs a complete Icing analysis. It contains cells to set up and run airflow with the FENSAP solver, set up and run droplets with DROP3D solver, set up and run icing with ICE3D solver, displace the grid due to ice accretion, and visualize the results in CFD-Post or Viewmerical.

To display this system in the **Toolbox**, install the `FENSAPICE-WB` extension located in the `[Install-Directory]\ANSYS Inc\v222\fensapice\workbench` folder, then load it using the Extension Manager.

For more details, see Creating FENSAP-ICE Based Systems in the *Ansys FENSAP-ICE in Ansys Workbench User's Guide*.

Fluid Flow - Icing (Fluent)

The **Fluid Flow - Icing (Fluent)** system performs a complete Icing analysis. It contains cells to connect to a Ansys Fluent component system and run airflow with the Fluent solver, set up and run droplets with the DROP3D solver, set up and run icing with the ICE3D solver, displace the grid due to ice accretion, and visualize the results in CFD-Post or Viewmerical.

To display this system in the **Toolbox**, install the `FENSAPICE-WB` extension located in the `[Install-Directory]\ANSYS Inc\v222\fensapice\workbench` folder, then load it using the Extension Manager.

For more details, see Creating CFX/Fluent-based Icing Systems in the *Ansys FENSAP-ICE in Ansys Workbench User's Guide*.

Fluid Flow (CFX)

The **Fluid Flow (CFX)** system performs fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries.

This system is also used as part of the FSI: Fluid Flow (Ansys CFX) > Static Structural (p. 225) custom system.

This analysis is configured in Ansys CFX.

For detailed information on working with Ansys CFX, see Ansys CFX in Ansys Workbench.

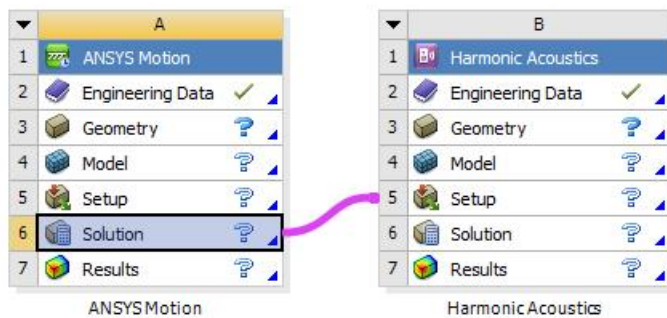To work through a **Fluid Flow (CFX)** system:

1. To add a **Fluid Flow (CFX)** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select one of the following options from the context menu:

    •   **New SpaceClaim Geometry**

    •   **New DesignModeler Geometry**

    •   **Import Geometry**

3. To create a mesh, right-click the **Mesh** cell and select **Edit** from the context menu.

4. Right-click the **Setup** cell and select one of the following options from the context menu:

   - To specify the Ansys CFX physics definitions in CFX-Pre, select **Edit**.

   - To import a previously saved case file, select **Import Case** → **Browse**.

5. Right-click the **Solutions** cell and select one of the following options from the context menu:

   - To start the solver immediately, select **Update**.

   - To set the solver execution controls in CFX-Solver Manager select **Edit**.

   - To import an existing CFX-Solver Results file, select **Import Solution** → **Browse**.

6. To analyze the results of the calculations in CFD-Post, right-click the **Results** cell and select **Edit** from the context menu.

Note:

If you import a CFX-Solver Input File into a CFX **Setup** cell and the reload file format is not a CFX-Solver Input File (for example, a `DEF` file) and the Reload Options were not defined or did not contain a `replacetype` option, you may need to modify the definition of the Mesh Reload Options.

If a definition file contains a mesh from a GTM file and is imported into a **Setup** cell, the original mesh type is automatically changed to *CFX-Solver Input file*. If the remeshing process is not explicitly told to import the GTM file as a GTM file, the remeshing process assumes that the mesh is also a CFX-Solver Input file, rather than a GTM file.

This fails when the remeshing process runs. This is a limitation and must be modified by hand. To do this, add a line similar to the following to the `DEF` file:

```
Mesh Reload Options = "replacetype=GTM,notransform=false"
```

See Remeshing Tab in the *CFX-Pre User's Guide* for details.

Resuming a Failed Run

If your CFX run fails and a backup file is available, you can resume a run from the latest standard or essential backup, or equivalent transient results file. This capability is especially useful for cases with very long runtimes to avoid restarting from initial conditions or, if the solver failed due to an error, reviewing backed up results to diagnose and correct the issue.

To take advantage of this capability, follow these steps when starting your run:

1. Import a case file or begin with an existing CFX setup definition.

2. Open **Setup** and define backups:

      a.   From the menu bar, select **Insert** → **Solver** → **Output Control**.

      b.   Switch to the **Backup** tab.

          For more information on output controls and the **Backup** tab in CFX-Pre, see Output Control.

      c.   Define your backup settings.

          Note that only **Essential** or **Standard** backup files provide sufficient data for a clean restart. Additional backup files using the **Smallest** or **Selected Variables** option can be included, but are not available for restarting a solution.

      d.   Close CFX-Pre.

3. In the Workbench **Project Schematic**, update the system (p. 244) to start the solution.

4. If the solver fails to write a results file, right-click the **Solution** cell and select **Resume Solution from Backup** from the context menu to resume the solution or **Copy Backup to New Solution** to review the results in CFD-Post or keep the backup to use as initial conditions.

- If you selected **Resume Solution from Backup**, the update continues from the last completed backup.

- If you selected **Copy Backup to New Solution**, a new CFX system with only a **Solution** cell and a new **Results** cell is created in the **Project Schematic**. You can review the results from this system to determine the status of the solution at the last completed backup point. From there, you can choose to resume the original interrupted run or you can choose to revise your original case as necessary.

> **Note:**
>
> The solution is restarted directly from the backup file and includes any modifications made through the Command Editor. Also, if the setup is modified, **Resume Solution from Backup** is not available. To modify the setup and restart from a backup, first copy the backup to a new solution using **Copy Backup to New Solution** and link this backup solution to your solution cell to use it as an initial guess.

Fluid Flow (Fluent)

The **Fluid Flow (Fluent)** system performs fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries.

This analysis is configured in Ansys Fluent.

You can use a Fluent fluid flow analysis system to apply a computational mesh to a geometry within Workbench, then use Fluent to define pertinent mathematical models (for example, low-speed, high-speed, laminar, turbulent, and so on), select materials, define boundary conditions, and specify solution controls that best represent the problem to be solved. Fluent solves the mathematical equations, and

the results of the simulation can be displayed in CFD-Post for further analysis (for example, contours, vectors, and so on).

For detailed information on working with Fluent, see the Fluent User's Guide, the Fluent in Workbench User's Guide, and the other online documentation available under the Help menu within Fluent.

To work through a **Fluid Flow (Fluent)** system:

1. To add a **Fluid Flow (Fluent)** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select one of the following options from the context menu:

   • **New SpaceClaim Geometry**

   • **New DesignModeler Geometry**

   • **Import Geometry**

3. To create a mesh, right-click the **Mesh** cell and select **Edit** from the context menu.

4. Right-click the **Setup** cell and select one of the following options from the context menu:

   • To specify the Fluent settings, select **Edit**.

   • To import a previously saved Fluent case file, select **Import Fluent Case**.

   • To import previously saved Fluent case and data files, select **Import Fluent Case And Data**.

5. To analyze the results of the calculations in CFD-Post, right-click the **Results** cell and select **Edit** from the context menu.

Fluid Flow (Fluent with Fluent Meshing)

The **Fluid Flow (Fluent with Fluent Meshing)** system performs fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries.

This analysis is configured in Ansys Fluent.

You can use a Fluent fluid flow analysis system to apply a computational mesh to a geometry using Fluent's Watertight Meshing workflow, then use Fluent to define pertinent mathematical models (for example, low-speed, high-speed, laminar, turbulent, and so on), select materials, define boundary conditions, and specify solution controls that best represent the problem to be solved. Fluent solves the mathematical equations, and the results of the simulation can be displayed in CFD-Post for further analysis (for example, contours, vectors, and so on).

For detailed information on working with Fluent, see the Fluent User's Guide, the Fluent in Workbench User's Guide, and the other online documentation available under the Help menu within Fluent.

To work through a **Fluid Flow (Fluent with Fluent Meshing)** system:

1. To add a **Fluid Flow (Fluent with Fluent Meshing)** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select one of the following options from the context menu:

    - **New SpaceClaim Geometry**

    - **New DesignModeler Geometry**

    - **Import Geometry**

3. To create a mesh, right-click the **Mesh** cell and select **Edit** from the context menu.

4. Right-click the **Setup** cell and select one of the following options from the context menu:

    - To specify the Fluent settings, select **Edit**.

    - To import a previously saved Fluent case file, select **Import Fluent Case**.

    - To import previously saved Fluent case and data files, select **Import Fluent Case And Data**.

5. To analyze the results of the calculations in CFD-Post, right-click the **Results** cell and select **Edit** from the context menu.

Fluid Flow (Polyflow)

The **Fluid Flow (Polyflow)** systems analyze fluid flows with free surfaces, complex rheology (including non-Newtonian behavior with viscoelasticity), heat transfer, and chemical reactions.

This analysis is configured in Ansys Polyflow.

There are three Polyflow fluid flow analysis systems available in Workbench:

- The **Fluid Flow (Polyflow)** system provides the full simulation capabilities of Polyflow.

- The **Fluid Flow - Blow Molding (Polyflow)** system provides only the application-specific capabilities of Polyflow that are suited to blow-molding simulations.

- The **Fluid Flow - Extrusion (Polyflow)** system provides only the application-specific capabilities of Polyflow that are suited to extrusion simulations.

Use a Polyflow, Blow Molding (Polyflow), or Extrusion (Polyflow) fluid flow analysis system to apply a computational mesh to a geometry within Workbench, then use Polydata to define pertinent mathematical models (for example, Generalized Newtonian, Viscoelastic, and so on), select materials, define boundary conditions, and specify solution controls that best represent the problem to be solved. Polyflow solves the mathematical equations, and the results of the simulation can be displayed in CFD-Post for further analysis (for example, contours, vectors, and so on).

For detailed information on working with Polyflow, see the online help in Polyflow, the Polyflow User's Guide, and the Polyflow in Workbench User's Guide.

SpaceClaim Meshing

SpaceClaim mesh transfer to the Mesh cell of the Fluid Flow systems (Fluent, CFX, and Polyflow) now uses a `.uns` file to handle the import.

The recommended method of transferring SpaceClaim meshing is to connect a Geometry component system to the Setup cell of either a Fluent, CFX, or Polyflow component system, instead of using the Fluid Flow system.
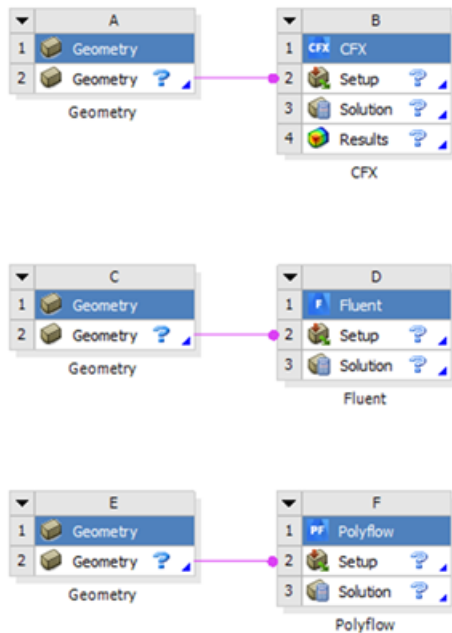
If you choose to use the Fluid Flow system, enable meshing in SpaceClaim then display the Properties pane for the Fluid Flow system Mesh cell. Control the import using the mesh conversion options.

Harmonic Acoustics

The **Harmonic Acoustics** system determines the steady-state response of a structure and the surrounding fluid medium to loads and excitations that vary sinusoidally (harmonically) with time.

This system is configured in Ansys Mechanical, using the Mechanical APDL solver to compute the solution.

To work through a **Harmonic Acoustics** system:

1. To add a **Harmonic Acoustics** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

   Once the analysis type is selected, the following prompt displays.

   This prompt informs you that you can automatically create a **Physics Region** (object) in the downstream Mechanical system by selecting physics types in the **Setup** cell properties. The **Acoustics** property, as illustrated below, is a read-only property that is always active.

2. To ensure the prompt is not displayed again, select the **Do not show me this again** check box.

3. To close the prompt, click **OK**.

4. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

5. To import the model, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

6. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Harmonic Acoustics Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Harmonic Response

The **Harmonic Response** system determines the steady-state response of a linear structure to loads and excitations that vary sinusoidally (harmonically) with time.

Use this system to verify whether or not your designs successfully overcome resonance, fatigue, and other harmful effects of forced vibrations. This analysis technique calculates only the steady-state, forced vibrations of a structure, typically at a number of discrete points within a range of frequencies. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic response analysis.

This system is configured in Ansys Mechanical, using the Ansys solver to compute the solution.

To work through a **Harmonic Response** system:

1. To add a **Harmonic Response** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

   Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   A mode-superposition harmonic analysis automatically runs the modal portion of the solution and cannot transfer data from a separate modal system in the **Project Schematic**.

   See Harmonic Response Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Hydrodynamic Diffraction

The **Hydrodynamic Diffraction** system calculates wave forces and structure motions in regular or irregular waves.

Use this system to apply a computational mesh to a geometry within the Aqwa application and produce a solution.

This analysis is configured in DesignModeler and solved in Ansys Aqwa.

Optionally, you can connect a **Hydrodynamic Diffraction** system **Solution** cell to a **Static Structural system (p. 98)** **Setup** cell and transfer the data to the **Static Structural** system.

You can also import a previously saved Aqwa `.aqdb` file (p. 50) and navigate to the database file.

For detailed information on working with Aqwa in Workbench, see the Aqwa User's Manual.

1. To add a **Hydrodynamic Diffraction** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu or select **New Geometry** from the context menu to create it in DesignModeler.

3. To specify the Aqwa settings, right-click the **Setup** cell and select **Edit** from the context menu.

4. In the Aqwa application window, complete your analysis using the application's tools and features.

5. To start the solution, in the Aqwa application select **Solve**, or right-click the **Solution** cell and select **Update** from the context menu.

Hydrodynamic Response

The **Hydrodynamic Response** system calculates wave forces and structure motions in regular or irregular waves.

Use this system to apply ocean environment forces (wind, wave, current) to a structure.

This analysis is configured in DesignModeler and solved in Ansys Aqwa.

You can also import a previously saved Aqwa `.aqdb` file (p. 50) and navigate to the database file.

For detailed information on working with Aqwa in Workbench, see the Aqwa User's Manual.

1. To add a **Hydrodynamic Diffraction** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu or select **New Geometry** from the context menu to create it in DesignModeler.

3. To add a **Hydrodynamic Response** analysis system, drag the system from the **Toolbox** onto the system added in step 1, sharing the **Geometry**, **Model**, and **Solution** cells.

4. To specify the Aqwa settings, right-click the **Hydrodynamic Response** system **Setup** cell and select **Edit** from the context menu.

5. In the Aqwa application window, complete your analysis using the application's tools and features.

6. To start the solution, in the Aqwa application select **Solve**, or right-click the **Solution** cell and select **Update** from the context menu.

LS-DYNA

LS-DYNA is a general-purpose finite element program capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. The LS-DYNA solver is optimized for shared and distributed memory Unix, Linux, and Windows based platforms. The code's origins lie in highly nonlinear, transient dynamic finite

element analysis using explicit time integration. The **LS-DYNA** system takes the power of the LS-DYNA solver and wraps it up into the familiar and easy to use environment of Ansys Workbench and Ansys Mechanical.

See the LS-DYNA User's Guide for more information on conducting this analysis.

LS-DYNA Restart

The **LS-DYNA Restart** system allows you to restart an LS-DYNA analysis. Restarting means performing an analysis which continues from a previous analysis. A restart can begin from either the conclusion of or the middle of a prior analysis.

For detailed steps on using this system, see Restarting an LS-DYNA Analysis in the *LS-DYNA User's Guide*.

Magnetostatic

The **Magnetostatic** system performs 3D static magnetic field analysis. You can model various physical regions including iron, air, permanent magnets, and conductors.

This system is configured in Ansys Mechanical, using the Ansys solver to compute the solution.

To work through a **Magnetostatic** system:

1. To add an **Magnetostatic** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu, or double-click the **Model** cell.

   Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Magnetostatic Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Modal Acoustics

The **Modal Acoustics** system models a structure and the surrounding the fluid medium to determine frequencies and standing wave patterns within a structure.

This system is configured in Ansys Mechanical, using the Mechanical APDL solver to compute the solution.

To work through a **Modal Acoustics** system:

1. To add a **Modal Acoustics** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

   Once the analysis type is selected, the following prompt displays.

   This prompt informs you that you can automatically create a **Physics Region** (object) in the downstream Mechanical system by selecting physics types in the **Setup** cell properties. The **Acoustics** property, as illustrated below, is a read-only property that is always active.

2. To ensure the prompt is not displayed again, select the **Do not show me this again** check box.

3. To close the prompt, click **OK**.

4. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

5. To open the Mechanical application, right-click the **Setup** cell and select **Edit** from the context menu or double-click the **Setup** cell.

6. In the Mechanical application window, complete your analysis using the application's tools and features.

See Modal Acoustics Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Modal

The **Modal** system determines the vibration characteristics (natural frequencies and corresponding mode shapes) of a structure or a machine component. It can serve as a starting point for other types of analyses by detecting unconstrained bodies in a contact analysis or by indicating the necessary time-step size for a transient analysis, for example. In addition, the modal-analysis results may be used in a downstream dynamic simulation employing mode-superposition methods, such as a harmonic response analysis, a random vibration analysis, or a spectrum analysis. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions.

This analysis is configured in the Mechanical application, which uses either the Ansys, ABAQUS, or Samcef solver to compute the solution, depending on which system you selected.

To work through a **Modal** system:

1. To add a **Modal** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

    Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

    See Modal Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Random Vibration

The **Random Vibration** system determines the response of structures to vibration loads that are random in nature. The loads on a structural system may not always be known or quantifiable with certainty. Sensitive electronic equipment mounted in an automobile, for example, may experience slightly-different conditions each day, due to changes in engine vibration or pavement roughness, even if the same road is traveled. Since the input loads are described using statistical quantities, the outputs are said to be non-deterministic, meaning that the results can be known only within a certain probability.

This analysis is configured in the Mechanical application, which uses either the Ansys solver to compute the solution.

A Random Vibration analysis must follow a Modal analysis that extracts the natural frequencies and mode shape. Follow the instructions in Modal (p. 95) to build a Modal analysis system and then complete the following instructions to build and link a **Random Vibration** system.

Alternatively, you can select the Random Vibration (p. 226) custom system. This option creates a Random Vibration system template that includes both the Modal analysis and Random Vibration analysis system templates with the links pre-defined.

To work through a **Random Vibration** system:

1. From the **Modal** analysis system, right-click the **Solution** cell and select **Transfer Data to New** → **Random Vibration**.

    Workbench creates a new **Random Vibration** system. The **Engineering Data**, **Geometry**, and **Model** cells are linked, and the **Modal** system **Solution** cell is linked to the **Random Vibration** system **Setup** cell.

2. To open the Mechanical application, right-click the **Setup** cell in the **Random Vibration** system and select **Edit**, or double-click the **Setup** cell.

3. In the Mechanical application window, complete your analysis using the application's tools and features.

    See Random Vibration Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

4. On the **Project** tab toolbar, click **Update Project**.

Response Spectrum

The **Response Spectrum** system calculates the maximum response for a given excitation based upon the input response spectrum and the method used to combine the modal responses. The combination methods available are the Square Root of the Sum of the Squares (SRSS), the Complete Quadratic Combination (CQC) and the Rosenblueth's Double Sum Combination (ROSE). A response spectrum analysis has similarities to a random vibration analysis. However, unlike a random vibration analysis, responses from a response spectrum analysis are deterministic maxima.

This analysis is configured in the Mechanical application, which uses either the Ansys solver to compute the solution.

A Response Spectrum analysis must follow a Modal analysis that extracts the natural frequencies and mode shape. Follow the instructions in Modal (p. 95) to build a Modal analysis system and then complete the following instructions to build and link a **Response Spectrum** system.

To work through a **Response Spectrum** system:

1. From the **Modal** analysis system, right-click the **Solution** cell and select **Transfer Data to New** → **Response Spectrum**.

    Workbench creates a new **Response Spectrum** system. The **Engineering Data**, **Geometry**, and **Model** cells are linked, and the **Modal** system **Solution** cell is linked to the **Response Spectrum** system **Setup** cell.

2. To open the Mechanical application, right-click the **Setup** cell in the **Response Spectrum** system and select **Edit** or double-click the **Setup** cell.

3. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Response Spectrum Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

4. On the **Project** tab toolbar, click **Update Project**.

Rigid Dynamics

The **Rigid Dynamics** system determines the dynamic response of an assembly of rigid bodies linked by joints and springs. You can use this type of analysis to study the kinematics of a robot arm or a crankshaft system, for example.

This analysis is configured in the Mechanical application, which uses the Ansys Rigid Dynamics solver to compute the solution.

To work through a **Rigid Dynamics** system:

1. To add a **Rigid Dynamics** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

   Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Rigid Dynamics Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Speos

The **Speos** system lets you design and optimize light and systems.

For more detailed information, see Creating a Speos system in Ansys Workbench in the *Speos User's Guide*.

Static Acoustics

The **Static Acoustics** system determines displacements, stresses, strains, and forces in your structure or structural components that are caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed. The loads and the structure's response are assumed to vary slowly with respect to time.

it is used as a pre-stress environment for future downstream Modal Acoustics or Harmonic Acoustics analyses.

This analysis is configured in the Mechanical application, which uses the Mechanical APDL solver to compute the solution.

To work through a **Static Acoustics** system:

1. To add a **Static Acoustics** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

    Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

    See Static Acoustics Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Static Structural

The **Static Structural** system determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed. The loads and the structure's response are assumed to vary slowly with respect to time.

This analysis is configured in the Mechanical application, which uses the Ansys, ABAQUS, or Samcef solver to compute the solution, depending on which system you have selected.

To work through a **Static Structural** system:

1. To add a **Static Structural** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

    Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

    See Static Structural Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Steady-State Thermal

The **Steady-State Thermal** system determines temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineers often perform a steady-state analysis before performing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished.

> **Important:**
>
> By default, the application does not write thermal gradient results to the result file. To have these results written to the results file, use a Command object and insert the command **OUTRES**,**ERASE**.

This analysis is configured in the Mechanical application, which uses the Ansys, ABAQUS, or Samcef solver to compute the solution, depending on which system you have selected.

To work through a **Steady-State Thermal** system:

1. To add a **Steady-State Thermal** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

    Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

    See Steady-State Thermal Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

    See the *Thermal Analysis Guide* for more information on thermal analyses using the Mechanical APDL application.

Structural Optimization

See the Structural Optimization Analysis section of the Mechanical User's Guide for the steps to conduct this type of analysis.

Substructure Generation

The **Substructure Generation** system supports the preliminary computation (Generation Pass) for top-down substructuring of condensed geometry. It also supports load vectors and constraints for super element Generation Pass.

For a more detailed overview of substructures, see Substructure Analysis in the *Mechanical User's Guide*.

This analysis is configured in the Mechanical application, which uses the Mechanical APDL solver to compute the solution.

To work through a **Substructure Generation** system:

1. To add a **Substructure Generation** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

    Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

    See Condensed Part in the *Mechanical Object Reference* for more information on conducting this analysis.

Thermal-Electric

The **Thermal-Electric** system allows for a simultaneous solution of thermal and electric fields. This coupled-field capability models joule heating for resistive materials as well as Seebeck, Peltier, and Thomson effects for thermoelectricity.

This analysis is configured in the Mechanical application, which uses the Ansys solver to compute the solution.
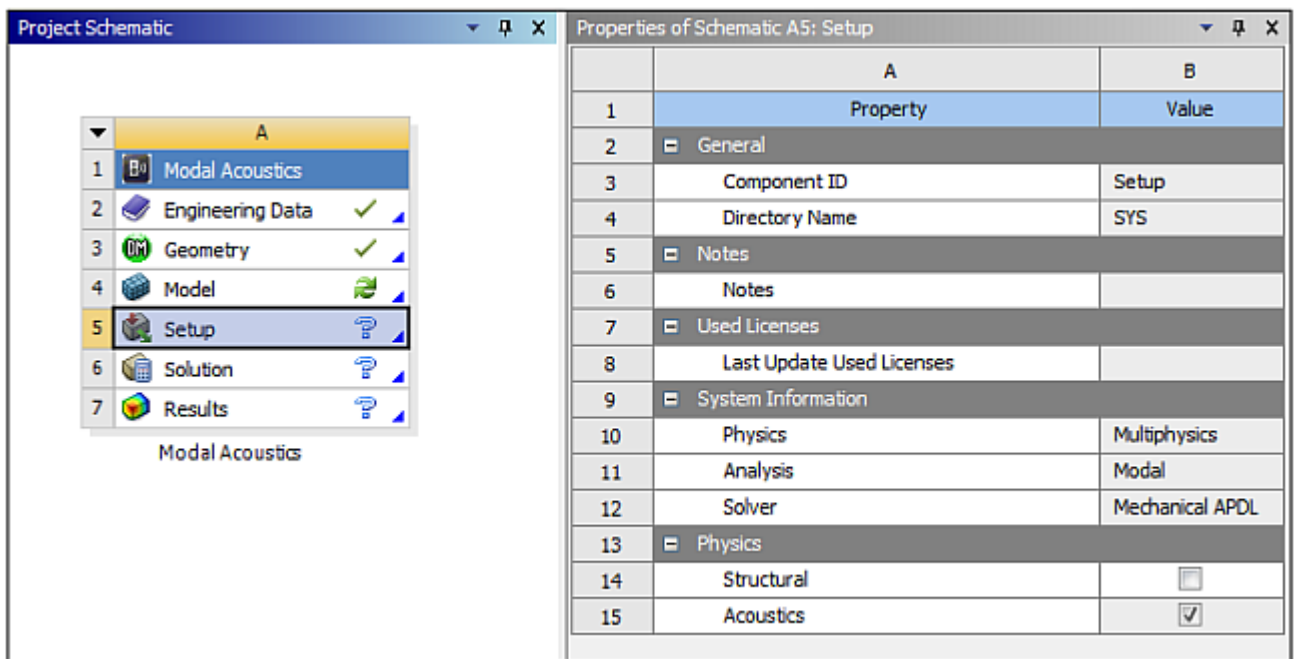
To work through a **Thermal-Electric** system:

1. To add a **Thermal-Electric** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

    Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

    See Thermal-Electric Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Throughflow and Throughflow (BladeGen)

The **Throughflow** system contains cells for conducting a study with Vista TF starting with the geometry and ending with a report. **Throughflow** is a Vista TF system with an added **Geometry** cell and **Throughflow (BladeGen)** is a Vista TF system with an added **Blade Design** cell.

For details on the Vista TF system, see Vista TF (p. 222).

Transient Structural

The **Transient Structural** system determines the dynamic response of a structure under the action of any general time-dependent loads. You can use it to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to any transient loads. The time scale of the loading is such that the inertia or damping effects are considered to be important. If the inertia and damping effects are not important, you might be able to use a static analysis instead.

This analysis is configured in the Mechanical application, which uses the Ansys, ABAQUS, or Samcef solver to compute the solution, depending on which system you have selected.

To work through a **Transient Structural** system:

1. To add a **Transient Structural** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

   Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Transient Structural Analysis in the *Mechanical User's Guide* for more information on conducting this analysis.

Transient Thermal

The **Transient Thermal** system determines temperatures and other thermal quantities that vary over time. The variation of temperature distribution over time is of interest in many applications such as with cooling of electronic packages or a quenching analysis for heat treatment. Also of interest are the temperature distribution results in thermal stresses that can cause failure. In such cases the temperatures from a transient thermal analysis are used as inputs to a static structural analysis for thermal stress evaluations. Many heat transfer applications (such as heat treatment problems, electronic package design, nozzles, engine blocks, pressure vessels, and so on) involve transient thermal analyses.

This analysis is configured in the Mechanical application, which uses the Ansys, ABAQUS, or Samcef solver to compute the solution, depending on which system you have selected.

To work through a **Transient Thermal** system:

1. To add a **Transient Thermal** analysis system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To load the geometry, right-click the **Geometry** cell and select **Import Geometry** from the context menu.

3. To view the geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

   Alternatively, you can complete this step using the **Setup** cell.

4. In the Mechanical application window, complete your analysis using the application's tools and features.

   See Transient Thermal Analysis for more information on conducting this analysis.

   See the *Thermal Analysis Guide* for more information on thermal analyses using the Mechanical APDL application. For more information on time stepping, see the discussion on General Options.

Turbomachinery Fluid Flow

The **Turbomachinery Fluid Flow** system contains cells for conducting a study with Ansys CFX starting with the mesh and ending with a report. **Turbomachinery Fluid Flow** is a CFX system with an added **Turbo Mesh** cell.

For details on the CFX system, see CFX (p. 106).

Component Systems

Component systems enable you to launch and use familiar stand-alone editors to build a project, but represent only a subset of a complete analysis. They usually do not include all components and steps necessary to achieve an analysis system; however, if you are familiar with a particular application, you can use your product knowledge to complete an analysis.

For example, you can use a geometry component system to define your geometry and then connect the component system to several downstream systems, so component system can then be connected to several downstream systems, so that the downstream systems share the same geometry source.

Component systems also include applications that open outside of Workbench (rather than as a tab), allowing you to use Workbench to manage your analysis data and files. This can be useful for products such as Mechanical APDL, which uses numerous files during an analysis.

The following component systems are available:

ACP-Pre and ACP-Post

Autodyn

BladeGen

CFX

Chemkin

Discovery

Engineering Data

EnSight (Forte)

External Data

External Model

Fluent

Fluent (with Fluent Meshing)

Forte

Geometry

GRANTA MI

GRANTA Selector

ICEM CFD

Icepak

Injection Molding Data

Material Designer

Mechanical APDL

Mechanical Model

Mesh

Microsoft Office Excel

Performance Map

Polyflow

Results

Sherlock Pre and Post

System Coupling

Turbo Setup

TurboGrid

Vista AFD

Vista CCD and Vista CCD with CCM

Vista CPD

Vista RTD

Vista TF

ACP-Pre and ACP-Post

The **ACP-Pre** and **ACP-Post** systems launch Ansys Composite PrepPost (ACP) for the modeling and postprocessing of composite models.

For detailed usage instructions and example workflows, see Implementation in Workbench in the *ACP User's Guide*.

Autodyn

The **Autodyn** system launches Ansys Autodyn, an explicit analysis tool for modeling nonlinear dynamics of solids, fluids, gas, and their interaction. This system supports the full range of Autodyn capabilities including the Explicit Eulerian solvers, mesh free SPH solvers, and explicit solver coupling (FSI).

> **Note:**
>
> The Explicit Dynamics (p. 83) analysis system supports the FE components of the Autodyn solver.

The system takes the binary database (`.ad`) file as input and produces the following output files:

- Results file (`.adres`) for postprocessing

- Save file (`admodel_cycle.adres`) for postprocessing and database modification during a run

- History data files (`.his`, `.sum`) that record time dependent data at gauge locations and summary data for materials/parts

- Print file (`.prt`) recording a text summary of the model definition and results

- Log file (`.log`) listing solution information and warnings or errors encountered

For detailed information on using Autodyn, see Autodyn User's Manual.

To work through an **Autodyn** system:

1. To add an **Autodyn** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To connect other systems, right-click the **Setup** cell, select **Transfer Data From New** from the context menu, then select one of the following options:

   - **Mesh**: Inserts a **Mesh** component system and generates a link between its **Mesh** cell and the **Autodyn** system **Setup** cell. This option enables the transfer of a mesh from the Ansys Meshing system into the **Autodyn** system.

   - **Explicit Dynamics**: Inserts an **Explicit Dynamics** analysis system and generates a link between its **Setup** cell and the Autodyn system **Setup** cell. This option enables the transfer of the initial model defined in the **Explicit Dynamics** system or the **Autodyn** system. The initial model includes materials, mesh, connections, coordinate systems, initial conditions, loads, constraints, and analysis settings.

3. Right-click the **Mesh** cell of the **Mesh** component system, or the **Setup** cell of the **Explicit Dynamics** analysis system, and select **Update** from the context menu.

A `CAERep.xml` file is produced, and then consumed by the **Autodyn Setup** cell.

Note:

Any modifications made in Autodyn to a model that originated from an **Explicit Dynamics** system are likely to be overwritten during the update process. Items defined in the **Explicit Dynamics** or **Mesh** system represent the main version of the data.

The files generated during the previous transfer or solve are not deleted during the update. As good practice, you can right-click the Analysis cell and select **Clear Generated Data** to clear the previously generated files.

---

4. To launch Autodyn interactively, right-click the **Autodyn Setup** cell and select one of the following options from the context menu:

    - **New Model**: Opens the Autodyn editor where you can set up a new 2D or 3D Autodyn model, solve, or postprocess results.

    - **Edit Model**: Opens the Autodyn editor and loads in the database currently associated with system. You can then further edit the model, solve, or postprocess results.

5. To launch Autodyn with input and reference files specified:

    a. Right-click the **Autodyn Setup** cell and select **Import Model** from the context menu.

        This action imports an existing Autodyn database (`.ad` file) into the system. Any existing database associated with the system is replaced. If the import detects that there are other files in the source directory for the model that is being imported, you are presented with the option to import all the associated files.

    b. Right-click the **Autodyn Setup** cell and select **Edit Model** from the context menu.

        Autodyn launches in interactive mode, and the input files specified are loaded.

6. To select a user-customized executable, right-click the **Autodyn Setup** cell and select **Select User Executable** from the context menu.

    This action selects the Autodyn executable file (`autodyn.exe`) you want to associate with the system and use for subsequent preprocessing, solving, and postprocessing.

BladeGen

The **BladeGen** system launches Ansys BladeGen, a component of Ansys BladeModeler. The BladeModeler software is a specialized, easy-to-use tool for the rapid 3D design of rotating machinery components.

With BladeGen, you can re-design existing blades to achieve new design goals or create completely new blade design. BladeGen imports blade geometry interactively or through files you supply. BladeGen allows sculpted or ruled element blades with linear or compound lean leading or trailing

edges. Over/Under-Filing can be applied, and leading and trailing edge shapes are easily specified as a full radius, an ellipse ratio, or a simple cutoff.

For more information on Ansys BladeGen, see BladeGen User's Guide.

To work through a **BladeGen** system:

1. To add a **BladeGen** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To launch BladeGen interactively, right-click the **Blade Design** cell and select **Edit** from the context menu.

3. To launch BladeGen with existing files:

   a. Right-click the **Blade Design** cell and select **Import Existing Case** from the context menu.

   b. Right-click the **Blade Design** cell and select **Edit** from the context menu.

CFX

The **CFX** system launches Ansys CFX, a general-purpose Computational Fluid Dynamics (CFD) software suite that performs fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You import meshes, specify the materials, boundary conditions, and solution parameters, solve the calculations, view the results, then create reports using built-in tools.

For detailed information on working with Ansys CFX, see Ansys CFX in Ansys Workbench.

For information on submitting Ansys CFX jobs to Remote Solve Manager, see Submitting CFX Jobs to Remote Solve Manager (p. 337).

To work through a **CFX** system:

1. To add an **CFX** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To Import a previously saved case file, right-click the **Setup** cell and select **Import Case** → **Browse**.

3. To read a mesh and specify the Ansys CFX physics definitions in CFX-Pre, Right-click the **Setup** cell and select **Edit** from the context menu.

4. To start the solver:

   • Right-click the **Solution** cell and select **Update** from the context menu.

   • Right-click the **Solution** cell and select **Edit** from the context menu, set the solver execution controls in CFX-Solver Manager, and start the solver.

   • To import an existing CFX-Solver Results file, right-click the **Solution** cell and select **Import Solution** → **Browse**.

5. To analyze the results of the calculations in CFD-Post, right-click the **Results** cell and select **Edit** from the context menu.

Chemkin

The **Chemkin** system launches Ansys Chemkin-Pro, allowing you to employ both an array of reactor models that include 0-D, 1-D, and 2-D models and complex detailed chemical kinetics mechanisms to investigate concepts in combustion, emissions, and materials synthesis.

For detailed information on working with Chemkin-Pro, see the Chemkin-Pro Getting Started Guide, as well as the other online documentation available in the **Help** menu within Chemkin-Pro.

To work through a **Chemkin** system:

1. To add a **Chemkin** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Right-click the **Setup** cell and select one of the following options from the context menu:

   - To specify the Chemkin-Pro settings, select **Edit**.

   - To import a previously saved Chemkin-Pro project, select **Import Chemkin Project**.

Discovery

The **Discovery** system launches Ansys Discovery, allowing you to transfer the geometry model bidirectionally, with parametric updates. It is also used when you start Workbench from Discovery using the Geometry Transfer tools.

For more details, see Geometry Transfer.

Engineering Data

The **Engineering Data** system launches the **Engineering Data** tab, allowing you to define or access material models for use in an analysis. You can also use the Engineering Data (p. 447) cell with Mechanical systems.

For more information, see Overview.

To work through an **Engineering Data** system:

1. To add an **Engineering Data** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To add or modify material data, double-click the **Engineering Data** cell.

EnSight (Forte)

The **Ensight (Forte)** system launches EnSight, a full-featured postprocessing and analysis tool that supports visualization and quantitative analysis of simulation results. You can run EnSight as a component from within Workbench by activating the EnSight Forte extension. You must have already

installed EnSight as part of the unified installation in order to access the extension. This system uses Forte solvers for analysis and reads the results into EnSight.

To work through an **EnSight (Forte)** system:

1. If **EnSight (Forte)** does not appear in the **Toolbox**:

    a. From the menu bar, select **Extensions** → **Manage Extensions**.

    b. In the **Extensions Manager** dialog, select the **EnSight Forte** extension name.

    c. Right-click and select **Load** from the context menu to run once or **Load as Default** to load the extension every time you start Workbench.

        Once you've loaded the extension, **EnSight (Forte)** is listed in the **Toolbox** as a component system.

2. To add a **Forte** component template, drag the template from the **Toolbox** to the **Project Schematic** or double-click the template in the **Toolbox**.

3. To add an **EnSight (Forte)** component template, drag the template from the **Toolbox** to the **Project Schematic**, dropping it onto the **Solution** cell of the **Forte** system.

4. If required, to connect the **EnSight (Forte)** system to a structural system, drop the structural system on the **Results** cell of the **EnSight (Forte)** system.

5. Right-click the **EnSight (Forte)** system header and select **Update** from the context menu.

6. To launch EnSight with the results loaded, double-click the **EnSight (Forte)** system **Results** cell or right-click the **Results** cell and select **Edit**.

External Data

The **External Data** system enables you to import data from text files into a Mechanical system or a **System Coupling** component system.

You can link an **External Data** system with the following Mechanical analysis types:

*   **Coupled Field Static**.

*   **Coupled Field Transient**. Supported loads include Imported Thermal Condition and Imported Temperature.

*   **Explicit Dynamics**. Note that, at this time, the Explicit Dynamics system and the LS-DYNA ACT extension provide only limited support for External Data Import.

*   **Modal**. Supported loads include Imported Body Temperature.

*   **Static Structural** and **Transient Structural**. These analysis types do not support the use of the Rigid Dynamics solver for External Data import.

- **Steady-State Thermal**, **Transient Thermal**, and **Thermal-Electric**.

> **Note:**
>
> There are certain limitations and/or requirements for some of the analysis types. See External Data Import in the *Mechanical User's Guide* for additional information about defining and processing your source data once you have imported it into Mechanical.

You must specify the data format in order to process the files in the **External Data** tab. This information can then be transferred to a downstream Mechanical system where the data can be applied as loads in an analysis.

> **Note:**
>
> If the project has an **External Data** system that consumes files stored within the design point path and multiple design points, updating the multiple design points can cause the project to fail or produce inaccurate data. When a different design point is run, the **External Data** system does switch to the new design point's file path for the files it requires, and cannot be made to do so within an update of multiple design points.

For more details, see the following topics:

Importing External Data

Importing a Trace File

Importing a CDB File as Input

Importing an Ansys External Data File as Input

Importing Multiple Data Sets

Applying Source Point Analytical Transformations

Performing System Coupling Simulations Using External Data

Importing External Data

To import external data using an **External Data** system:

1. To add an **External Data** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open the **External Data** tab, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

3. In the **Outline of Schematic** pane, in the **Location** column, click [...].

4. Browse to the location of the file to add.

> **Note:**
>
> Importing files from design point folders within the same project directory (with the exception of the Current design point) is not supported.

5.   Select the file and click **Open**.

     The selected file name, location, and identifier are automatically displayed in the **Data Source** column. The identifier is a string that can be used to identify the file in the downstream Mechanical application.

6.   If required, click the cell in the **Description** column and type a description for the file.

7.   Optionally, to duplicate an existing file, right-click the file in the **Data Source** column and select **Duplicate** from the context menu.

8.   To assign a main file, select the check box next to the required file.

     Set a main file so that the system displays and uses the **Properties** for this file only. The nodal coordinate data for the selected main file is processed by the mapping utility. Any nodal co-ordinate data contained in non-main files is skipped.

     You can select only one file to be the main file. If you multi-select a main file and non-main files, the **Dimension**, **Length Unit** and **Transformation** properties in the **Definition** pane are hidden. If you duplicate the main file, only one instance is designated as the main file.

9.   In the **Properties of File** pane, input the information required to process the file and apply the data in Mechanical.

     If you select multiple files in the **Data Source** column, the **Properties of File** pane displays:

     •   A value when that value is the same for all selected files

     •   A blank field when values differ between selected files

     •   A yellow field when a value is required, but not currently specified for at least one of the files

     If you edit any field in the **Properties of File** pane when multiple files are selected, your change is applied to all files.

     > **Note:**
     >
     > Although you can multi-select files in the **Data Source** pane, when you click away from that pane the highlighting applied to those files disappears. However, the files remain selected and operations subsequently performed will be applied to all of the selected files.

Property	Description
Dimension	Imports data from 2D or 3D models. If the 2D option is selected, you can import data only at the X and Y coordinates. The Z coordinate is not supported for the 2D option.
Start Import at Line	Sets the line number at which you want the data import to start. Line numbers start at 1.

Property	Description
Format Type	Sets the format specification used for the file. If you select **Delimited**, specify the character that delimits data elements in a line in the **Delimiter Character** field. The options are Comma, Semicolon, Space, Tab, or User-Defined. The latter choice enables you to specify any character as the data delimiter. If you select **User-Defined**, specify the format using the following allowed format specifiers: • Decimal floating point: F or f • Scientific notation (exponent) using E/e character: E or e • Signed decimal integer: D or d • Signed integer: I or i • Character: A or a • Ignore flag: X or x
Dummy Net Data	For **External Data** files that include trace mapping, selecting this option enables you to import trace data belonging to the dummy net of the file.
Length Unit	Specifies the unit system in which source point locations are defined.
Coordinate System Type	Specifies the nature of the source point locations. Options include Cartesian (X, Y, Z) or Cylindrical (R, theta, Z). The default value is Cartesian.
Average Data at Midside Nodes	If set to *Yes*, the data at Midside nodes, if not specified, is calculated as average of data specified at corner nodes. **Note:** This property is only available when **Format Type** is **Delimited** or **User Defined** and **.cdb** file is chosen as the main file.
Material Field Data	When selected, maps values from an external user-defined file to the nodes or elements of your mode. Once active, Field Variables become available in the **Data Type** drop-down menus in the **Table of File** pane.
X/Y/Z Coordinate	Enables you to apply scaling factors or functions to the corresponding column **Data Type** selected in the **Table of File** pane. For example, to scale all values for the X Coordinate column by 90%, change the X Coordinate text entry to x*0.9. When the Coordinate System Type is cylindrical, the X Coordinate will refer to the radius and the Y Coordinate refers to the angle. By default, each X, Y, and Z Coordinate is set to x, y, and z.

Property	Description
	For a complete list of supported functions, see Working with Parameters (p. 289). For an example, see Applying Source Point Analytical Transformations (p. 123). **Note:** When the **Dimension** type is 2D, the **Z Coordinate** is not shown. Rotations, resulting from specified analytical transformations, do not get applied to mapped data (pressure, displacement, force) in a downstream Mechanical system.

In the example below, the line has one integer followed by four exponential data types.

```
385 8.333333333E-003 9.375000000E-003 3.375000000E-001 0.000000000E+000
```

In the example below, the line has one integer followed by four exponential data types.

```
385 8.333333333E-003 9.375000000E-003 3.375000000E-001 0.000000000E+000
```

The corresponding format is

```
1i3,4e17.9
```

Where:

- The first entry is the number of occurrences

- The second entry is the format specifier

- The third entry is the number of characters of data in the definition (including numeric values (0-9), the letter 'e' (for instances of scientific notation), and any white space and + or - signs)

- The fourth entry is the number of digits after the decimal point.

Tip:

By default, Excel puts a minus sign at the beginning of a negative number but it does not put a plus sign at the beginning of a positive number, which changes the number of characters in the negative version of a number. However, if you select a column in Excel and apply a custom format, you can have a plus sign added to positive numbers. In Excel's **Format Cells** dialog box, set the Type with an example of the format for a positive number and a negative number, separating the two types with a semicolon. For example:

```
+00.00E+00;-00.00E+00
```

10. Based on the format details you specified in the **Properties** pane, you must define the data types and their corresponding units in the **Table of File** pane. The allowed **Data Type** values include:

- Body Force Density
- Displacement
- Element ID
- Element Orientations
- Force
- Heat Flux
- Heat Rate

- Heat Generation
- Heat Transfer Coefficient
- Node ID
- Not Used
- Pressure
- Strain
- Stress

- Temperature
- Thickness
- Velocity
- X Coordinate
- Y Coordinate
- Z Coordinate

Note:

- The **Node ID** field is useful when the nodal locations and values on nodes are specified in different files, as is the case when one of the files is specified as the main file. They are not meant to apply the values directly on the nodes of the target mesh. If a file is set up without specifying the nodal locations (X/Y/Z), all the points from the file default to (0,0,0). Similarly, the **Element ID** field is useful only when used in conjunction with a CDB file.

- **Thickness** uses the `Length` **Data Unit** and the **Data Identifier** should start with `Thickness`.

Tip:

You can multi-select rows (from the left-most column in the **Table of File** pane), then right-click to set all highlighted Data Type values or Data Unit values (where applicable) at the same time.

Column data is ignored if the **Data Type** is set to **Not Used**.

When **Coordinate System Type** is set to **Cartesian** in the **Properties** pane, the **Data Unit** cell for the X, Y, and Z Coordinates is read-only in the **Table of File** pane. When **Coordinate System Type** is set to **Cylindrical**, the **Data Unit** cell for the Y has a combo box that can specify its **Data Unit** string (either as **Degrees** or **Radian**).

You can change the data identifier from the default string for allowed data types. The data identifiers are appended to the file identifier, specified in the **Properties** pane, so that you can pick the correct source data in the downstream Mechanical application.

A preview of the file is shown in the **Preview** pane. The first ten imported lines are shown.

When you multi-select files, the **Preview** pane is disabled and the **Table of File** pane displays data in columns that you can sort and filter from the down arrow beside each column heading.

11. To delete files that you have selected (or multi-selected) right-click one of the files in the **Outline of Schematic** pane and select **Delete** from the context menu.

12. To modify any file in the **Outline of Schematic** pane, in the **Location** column, click [···] and select a new file.

13. Switch to the **Project** tab.

14. Display the properties for the **Setup** cell of the **External Data** system by right-clicking on the cell and selecting **Properties**. Specify the **Files Import Type** property as needed. Options include **By Copy** and **By Reference** (default). The **By Reference** option instructs the application to directly reference the External Data file instead of transferring the file to the project folder. For large size files, this action can drastically reduce disk storage requirements and improve performance.

> **Note:**
>
> When you specify the **By Reference** option and wish to archive your project using the option **Imported files external to project directory** of the **Archive Options** dialog, the application automatically copies the External Data files to the archived project.

> **Important:**
>
> The **By Reference** option cannot be used with the System Coupling feature.

15. If required, link (p. 60) the **Setup** cell of the **External Data** to a **Model** cell or to a **Setup** cell of a Mechanical system (except for Rigid Dynamics Systems).

You can modify any file in the **Outline** pane by browsing to a new file using the browse option provided in the **Location** column

> **Note:**
>
> If you modify an **External Data** system's data file outside of Workbench, you must re-read the data file. To do so:
>
> 1. Right-click the **Setup** cell
>
> 2. Select **Re-read Input Files** from the context menu.
>
> The **Re-read Input Files** operation causes Workbench to regard the file as having changed, whether the file has changed or not, and the status of the **Setup** cell is updated appropriately.

To add a downstream Mechanical system, either drag a valid analysis system from the Toolbox and drop it on the appropriate **Setup** cell of the **External Data** system, or right-click the **Setup** cell and choose the **Transfer Data To New** context menu option.

For additional information for using the data in a downstream Mechanical application, refer to External Data Import.

Importing a Trace File

You can import a circuit board's trace layout file (ODB++ `TGZ`, Ansoft `ANF`, `BOOL + INFO`, or Icepak `COND + INFO`) into the **Model** cell of aMechanical system.

To import a trace file:

1. To add an **External Data** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open the **External Data** tab, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

3. In the **Outline of Schematic** pane, in the **Location** column, click [...].

4. Browse to the location of the trace file.

5. Select the file and click **Open**.

The **External Data** component enables you to perform rigid transformations to the trace file. When any required transformations are complete, drag the **External Data** Setup cell to a Mechanical system's **Model** cell.

Note:

- If you import a project from Release 16 that has an **External Data** system, you will not immediately be able to load in trace files. In order to import a trace file, you must delete the connection between the **External Data Setup** cell and the downstream **Model** cell, and then recreate the connection. The new connection will support the transfer of trace files.

- For trace-file imports, the **Table** and **Preview** panes are disabled.

Importing a CDB File as Input

You can import a Mechanical APDL-generated CDB file as a source file.

To import a CDB file:

1. To add an **External Data** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open the **External Data** tab, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

3. In the **Outline of Schematic** pane, in the **Location** column, click [...].

4. Browse to the location of the CDB file.

5. Select the file and click **Open**.

   The **Format Type** property is automatically set to `.cdb`.

CDB files have certain restrictions:

- The **Start Import at Line** setting is disabled as the entire file is read as data. **Delimiter Type**, **Delimiter Character**, and **Format String** are also disabled.

- The data is always in the Cartesian global frame-of-reference.

- The **Table** and **Preview** panes are disabled, so you cannot specify data types or preview data.

Mechanical APDL CDB files can be added as a parent mesh in the **External Data** system. The file must be generated using 'blocked' formatting (see **CDWRITE** in the *Command Reference*). Files generated in unblocked format are not supported. Only NBLOCK and EBLOCK data will be read from the file. Elements defined in the EBLOCK command should also have a corresponding element type and number (ET command) defined in the CDB file. No load/data transfer information is read.

The following element types are ignored during reading of the file:

- Solid168

- Targe169,170

- Conta171,172,173,174,175,176,177,178

- SURF152,153,154,156

Data transfer information must be defined in separate files, which also must be added to the same **External Data** system. These files must have a column providing node identifiers that match the node IDs defined in the Mechanical APDL CDB file. If the data transfer file contains element nodal values, then both node and element identifier columns need to be defined. Element values are not supported.

> **Note:**
>
> - An element defined in the CDB file is ignored by the common mapping utility if any of the below conditions exist. The ignored elements are not used when data is mapped in the downstream application.
>
>   – One or more of the nodes forming the element are not defined in the NBLOCK command.
>
>   – The element does not have a corresponding element type and number (ET command) defined in the CDB file.
>
> - CDB files must contain only solid elements or shell elements, but not both.

- For nodal data transfers, the number of nodes in the Mechanical APDL CDB file must match the number of data transfer items in all child files.

- If data is not available at midside nodes, then the **Average Data at Midside Nodes** property can be used to specify the data at midside nodes as an average of data at corner nodes.

Importing an Ansys External Data File as Input

You can import an Ansys external data file (`.axdt`) as a source file.

To import an Ansys external data file:

1. To add an **External Data** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open the **External Data** tab, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

3. In the **Outline of Schematic** pane, in the **Location** column, click .

4. Browse to the location of the Ansys external data file.

5. Select the file and click **Open**.

   The **Format Type** property is automatically set to **Ansys External Data File**.

The **Length Unit** property is not available because this information is specified in the file. All length unit information in the `.axdt` file must be the same (all meters, for example).

The **Start Import at Line** setting is disabled as the entire file is read as data. **Delimiter Type**, **Delimiter Character**, and **Format String** are also disabled.

Ansys External Data File Format

CFD-Post and the Mechanical application can export external data files (`.axdt`) that can be used by the **External Data** system.

> **Note:**
>
> The Mechanical application does not export temperature data in units of K.

The following is an example of an `.axdt` file that has been exported from CFD-Post:

```
[Name]
Plane 1

[Data]
X [m] (X coordinate), Y [m ] (Y coordinate), Z [ m ] (Z coordinate),
Wall Heat Transfer Coefficient [W m^-2 K^-1] (Heat Transfer Coefficient),
Wall Adjacent Temperature [K] (Temperature)
-1.77312009e-02, -5.38203605e-02, 6.00000024e-02, 7.12153496e-06, ...
-1.77312009e-02, -5.79627529e-02, 5.99999949e-02, 5.06326614e-06, ...
```

```
.
.
.
[Faces]
369, 370, 376, 367
350, 374, 367, 368
.
.
.
```

This file contains three blocks, each with one of the labels: **[Name]**, **[Data]** and **[Faces]**. The **[Name]** block contains the name of the region contained/defined in the file.

The **[Data]** block contains node coordinates and values. The first line following the **[Data]** label is a header that contains a comma separated list of unique labels, units and quantity type for coordinates and values at each node. Units are contained in square brackets and quantity types are contained in parentheses. Subsequent lines, one per node, contain a comma separated list of data defined in the header. The **[Data]** block ends in the line before the **[Faces]** block label.
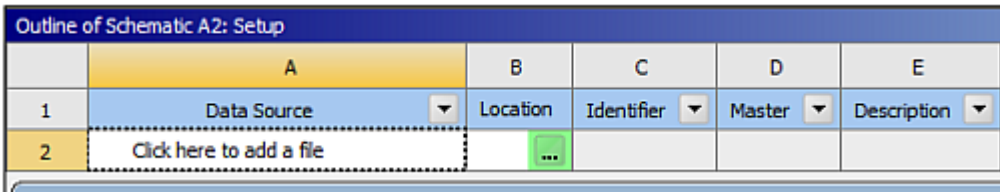
The **[Faces]** block contains definitions for topologically two-dimensional faces (small surfaces), each by 3 (triangle) or 4 (quadrilateral) points. The points must be ordered to trace a path going around the face. For proper rendering, the faces should have consistent point ordering, either clockwise or counterclockwise. Each face is automatically closed by connecting the last point to the first point. Face connectivity data is listed in the **[Faces]** block and references the points in the **[Data]** block, where the latter are implicitly numbered, starting with 0.
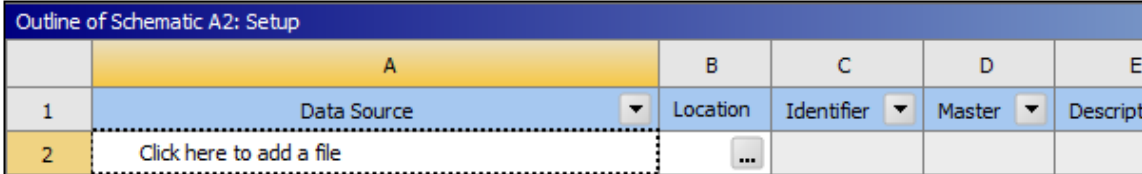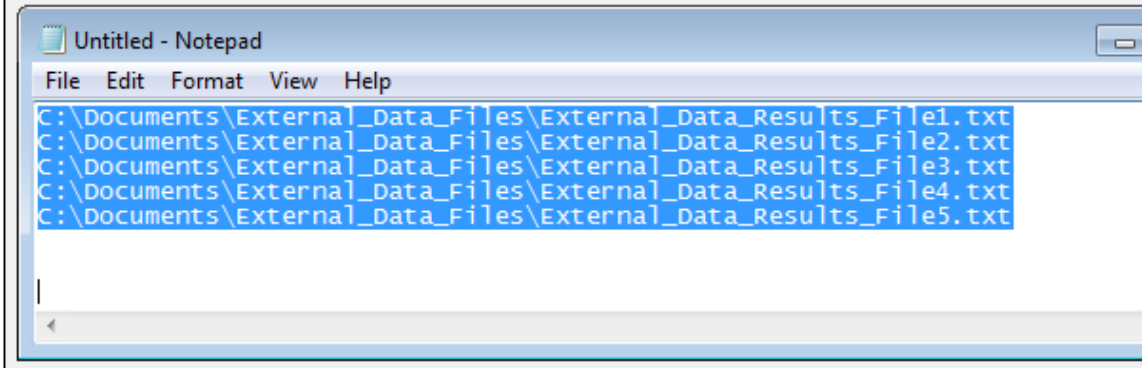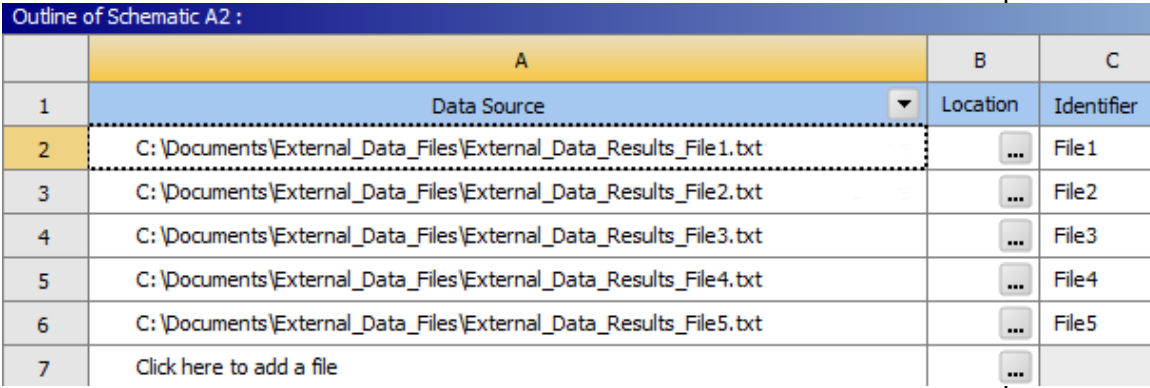
Importing Multiple Data Sets

External data can be configured to efficiently import multiple data sets (for example, from a transient analysis). It can handle multiple sets through a single file or from multiple files.

To import multiple data sets:

1. To add an **External Data** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open the **External Data** tab, double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu.

3. In the **Outline** pane, select the data files using one of the following methods:

Method	Steps
Add one or more files to the **Outline** pane.	1. In the **Outline of Schematic** pane, in the **Location** column, click […]. 2. Browse to the location of the files. 3. Select one or more files and click **Open**.

Method	Steps
Copy a list of fully-qualified paths from a text file or an Excel file and paste it into the **Outline** pane's **Data Source** field.	To paste from a flat-text editor (for example, Notepad): 1. List the paths to the files in the editor. Ensure that there are no trailing spaces in the lines. 2. Select all the files and copy them (**Ctrl**+**A**, then **Ctrl**+**C**). 3. In the **Outline** pane, click the row number The line becomes highlighted, and the text **Click here to add a file** *remains visible.* 4. Press **Ctrl**+**V** to add the files.

4. In the **Properties** pane, set the properties of the files.

If the files are of the same or similar format, you can multi-select the files to set common properties.

When you multi-select the files in the **Outline** pane, you see:

* A value when that value is the same for all selected files.

* A blank field when values differ between selected files.

* A yellow field when a value is required, but is not currently specified for any of the files.

If you edit any field in the **Properties** pane when multiple files are selected, your change is applied to all files.

If the X, Y, Z locations of the source points is common between all the files, you can make use of the main file designation. By designating a main file, all other files use that file's values for the X, Y, Z locations. This leads to faster user interface set up, as well as much faster mapping

times as the mapping weight calculations need to be done only once and then are shared for all child files.

5. Use the **Table** pane (which is populated from the **Format String** field in the **Properties** pane) to specify the **Column** data in the file.

    You can multi-select the files to fully populate the **Table** pane. You can span data from all se-lected files and use various right-mouse button actions to efficiently define the column data.

    a. In the **Outline** pane, multi-select the desired files.

    b. In the **Table** pane, sort the table by **Column** to order the file data. For example, if you have four data fields and the first three specify the X, Y, Z locations, sorting by column places the remaining data-field entries together at the bottom of the column.

    c. Select the first row to be changed by clicking on the *row number* (which is in the table boundary).

    d. Press and hold **Shift** key.

    e. Select the last row to be changed by clicking on the row number.

    f. Right-click anywhere over the selected cells, select **Set Data Type To** from the context menu, and set the desired data type.

        Repeat as required to set the data units using the **Set Unit To** option.

        Tip:

        If **Set Data Type To** or **Set Unit To** are not available, ensure that your mouse cursor is over the body of the table, not on the table boundary.

        ---

6. Select and copy the cell entries in the **Combined Identifier** column that correspond to the multiple data sets (using **Ctrl+C** or right-click and select **Copy**). These data-identifier strings are used to specify which data set will be imported at each load step inside Mechanical.

7. Link the **External Data** system into the desired Mechanical system/cell.

8. Update the **External Data** system.

9. Edit (or, if editor is already open, refresh) the Mechanical system.

10. Set the required **Number Of Steps** in the Mechanical **Analysis Settings** object. Set the step end times as required (you can copy and paste).

11. As needed, create the desired **Imported Load/Thickness** in Mechanical.

12. Select the **Imported Object**, then paste the data identifier text into the appropriate cells inside the **Data** pane on the imported object.

13. As required, copy and paste the desired step end times on the **Analysis Settings Object** and the **Imported Data Object** (in the **Analysis Time(s)** column).

14. Right-click **Import Load** to invoke the mapping calculations.

15. After mapping has completed, you can review the various mappings by adjusting the **Active Row** entry. The graphics render a contour plot of the imported data at the specified row. Additionally, this data can be exported out of Mechanical by right-clicking the tree object and selecting **Export**.

Applying Source Point Analytical Transformations

Analytical transformation options can be applied to the source nodal locations. You can enter constant or functional values that are applied to the x, y, and z (or r, theta, z for cylindrical) values read from the input file from within the External Data User Interface. For a complete list of supported functions, go to Expressions, Functions, Quantities, and Units (p. 291). The order of operations for conversion of the original node locations into a format the common mapping utility uses must be taken into consideration when setting up analytical transformation functions, rotation, and translation information.

The nodal data, as well as any unit system information, is read into the common mapping utility. If any nodal analytical transformation values or functions are provided, they are applied directly to the nodal coordinates as they are read in from the file. If the data is provided in a cylindrical system, it is converted into Cartesian coordinates. Once the nodes are in Cartesian, all nodal data is converted into MKS, so that the mapping utility stores all data in the same unit system. If any rotational information is provided, this is applied next, followed by any translations.
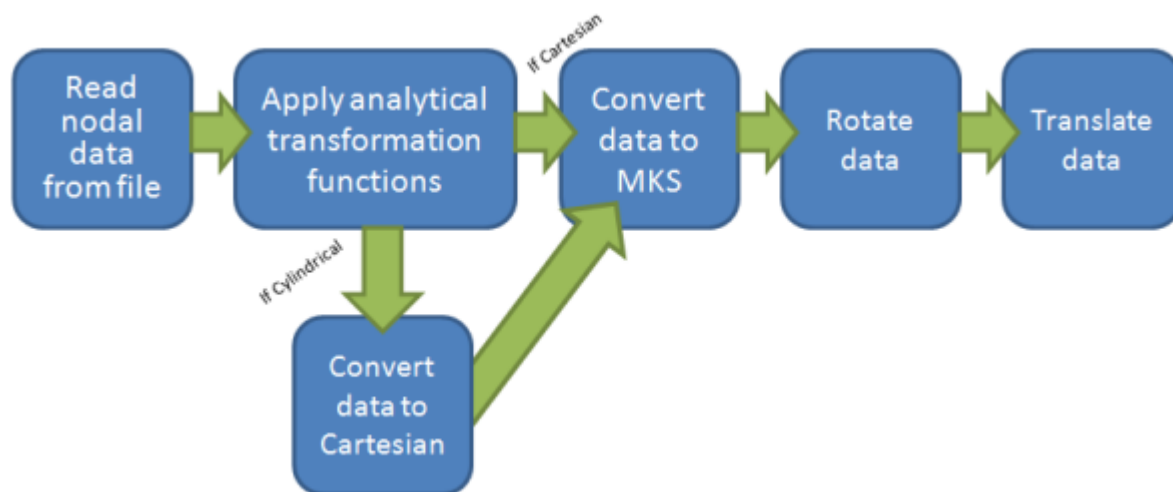
The following example takes nodal data, written in a cylindrical system, from a ring with an inner radius of 8 mm and an outer radius of 10 mm and allows an analytical transformation value to be applied to the radius of the source data, such that it aligns with a smaller ring with an inner radius of 7.2 mm and an outer radius of 9 mm.

Thermal results on an expanded or stretched ring (inner radius is 8 mm and outer radius is 10 mm)

A: Steady-State Thermal
Temperature
Type: Temperature
Unit: °C
Time: 1
5/17/2011 1:26 PM

100 Max
99.997
99.993
99.99
99.986
99.983
99.98
99.976
99.973
99.97 Min

0.000   5.000   10.000 (mm)

2.500   7.500
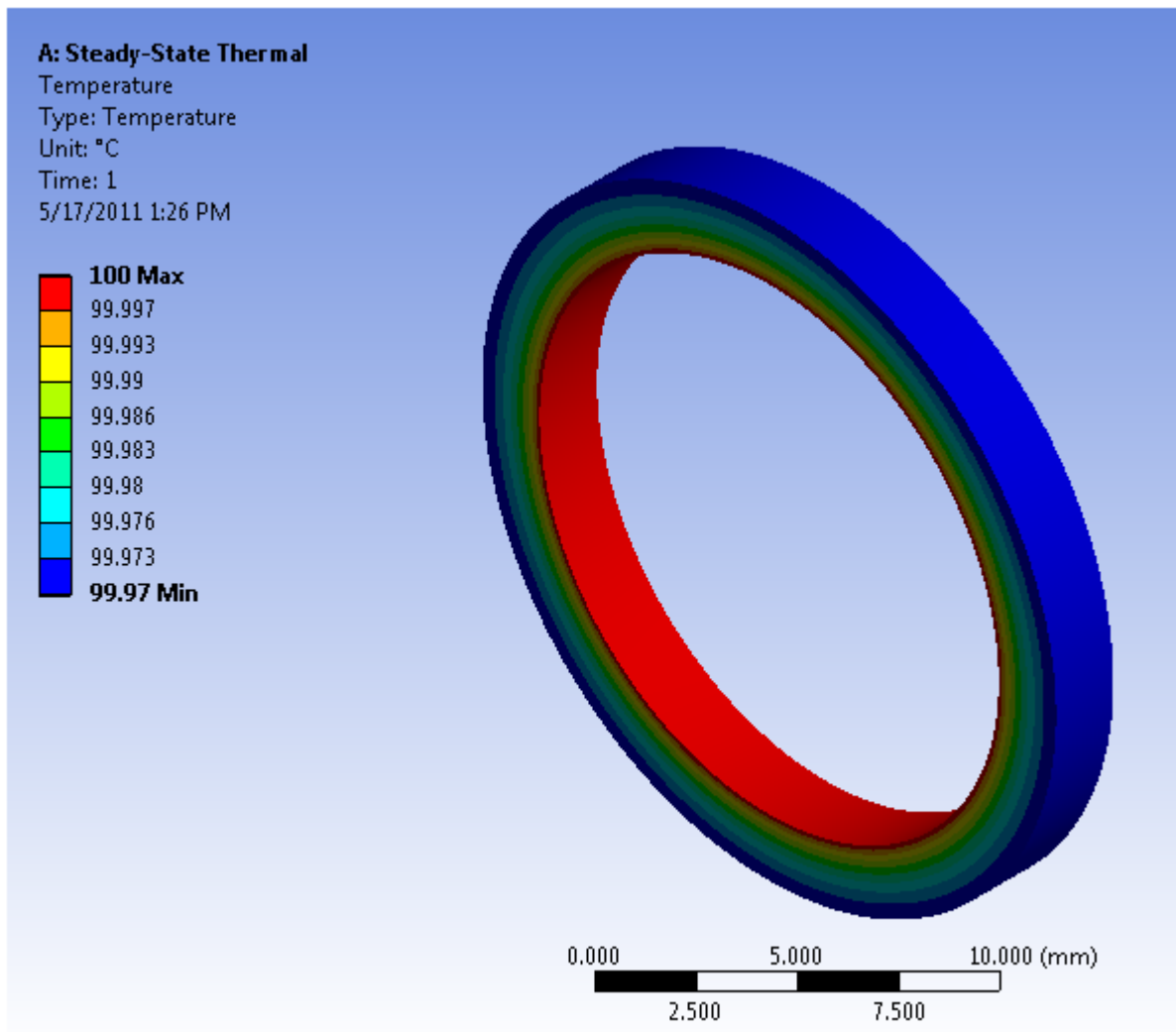
Sample cylindrical nodal data:

Node Number	Radius (mm)	Theta (radians)	Z Location (mm)	Temperature (−C)
1	8.914523065	−0.247535105	1.6	99.983
2	8.914523065	−0.247535105	1.2	99.984
3	8.914523065	−0.247535105	0.8	99.984
4	8.914523065	−0.247535105	0.4	99.983
5	8.917873803	−0.192647608	1.6	99.983
6	8.917873803	−0.192647608	1.2	99.984
7	8.917873803	−0.192647608	0.8	99.984
8	8.917873803	−0.192647608	0.4	99.983
9	8.927166575	−0.137916029	1.6	99.983

If we simply import the nodal data, you can see how the source nodes are not contained within the target volume.

Imported temperature load with unmodified source points displayed:
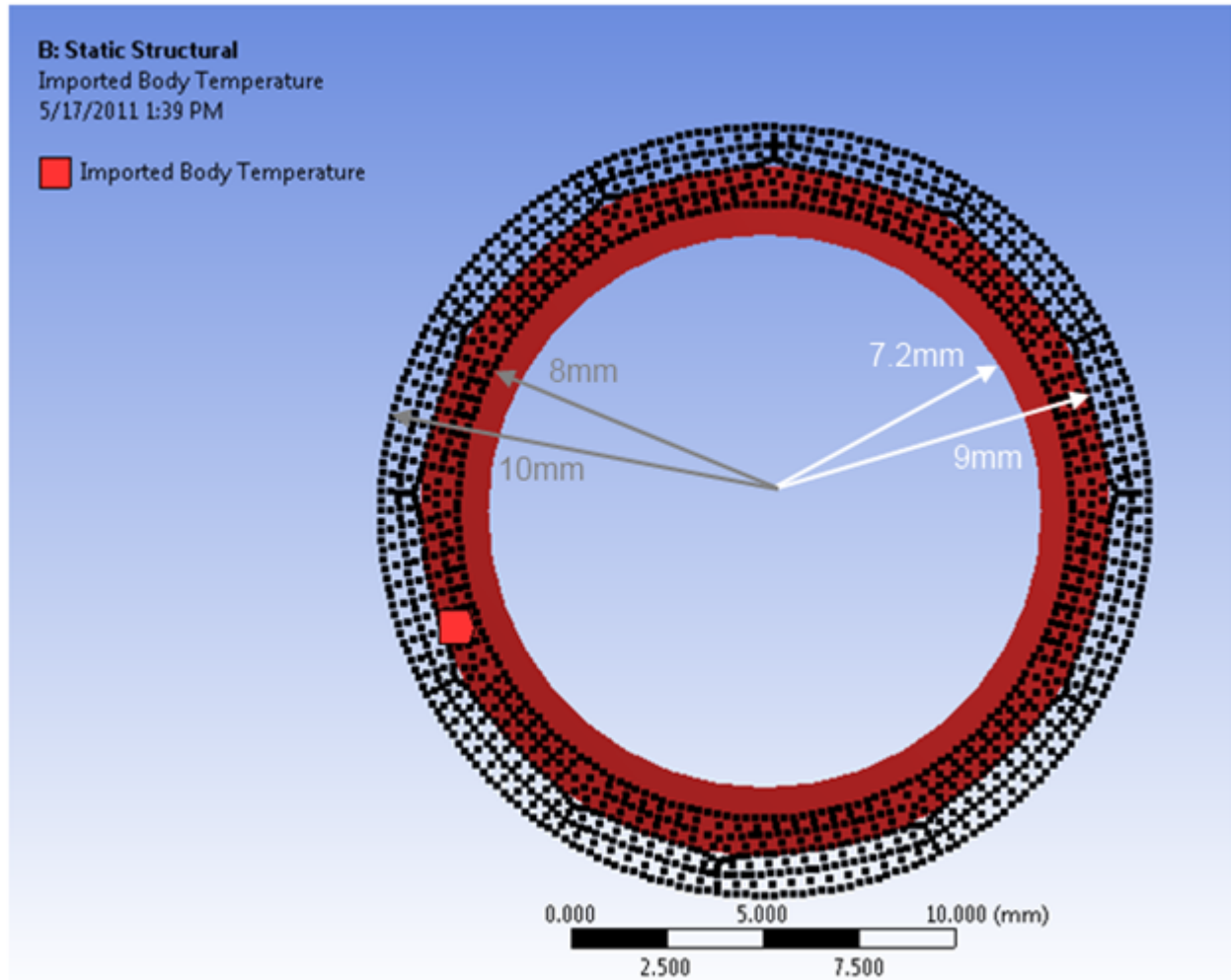
Within the External Data System user interface, you can enter analytical transformation values as either constants, or as functions of x, y, or z. For our example, we want to scale the source radius (that is, X Coordinate) by 90%.

Applying a constant analytical transformation scale factor to the radius (that is, X Coordinate) of the cylindrical source nodal locations:

		A	B	C
	1	Property	Value	Unit
	2	⊟ Definition		
	3	Dimension	3D	
	4	Start Import At Line	2	
	5	Format Type	Delimited	
	6	Delimiter Type	Tab	
	7	Delimiter Character	Tab	
	8	Length Unit	mm	
	9	Coordinate System Type	Cylindrical	
	10	⊟ Analytical Transformation		
	11	X Coordinate	x*.9	
	12	Y Coordinate	y	
	13	Z Coordinate	z	
	14	⊟ Rigid Transformation		
	15	Origin X	0	m
	16	Origin Y	2	mm
	17	Origin Z	0	m
	18	Theta XY	0	radian
	19	Theta YZ	90	degree
	20	Theta ZX	0	radian

Updating the schematic and looking at the imported load we see the analytical transformation value has been applied to the nodal locations and now coincides with the target model.

Imported temperature load with "transformed" source points displayed:
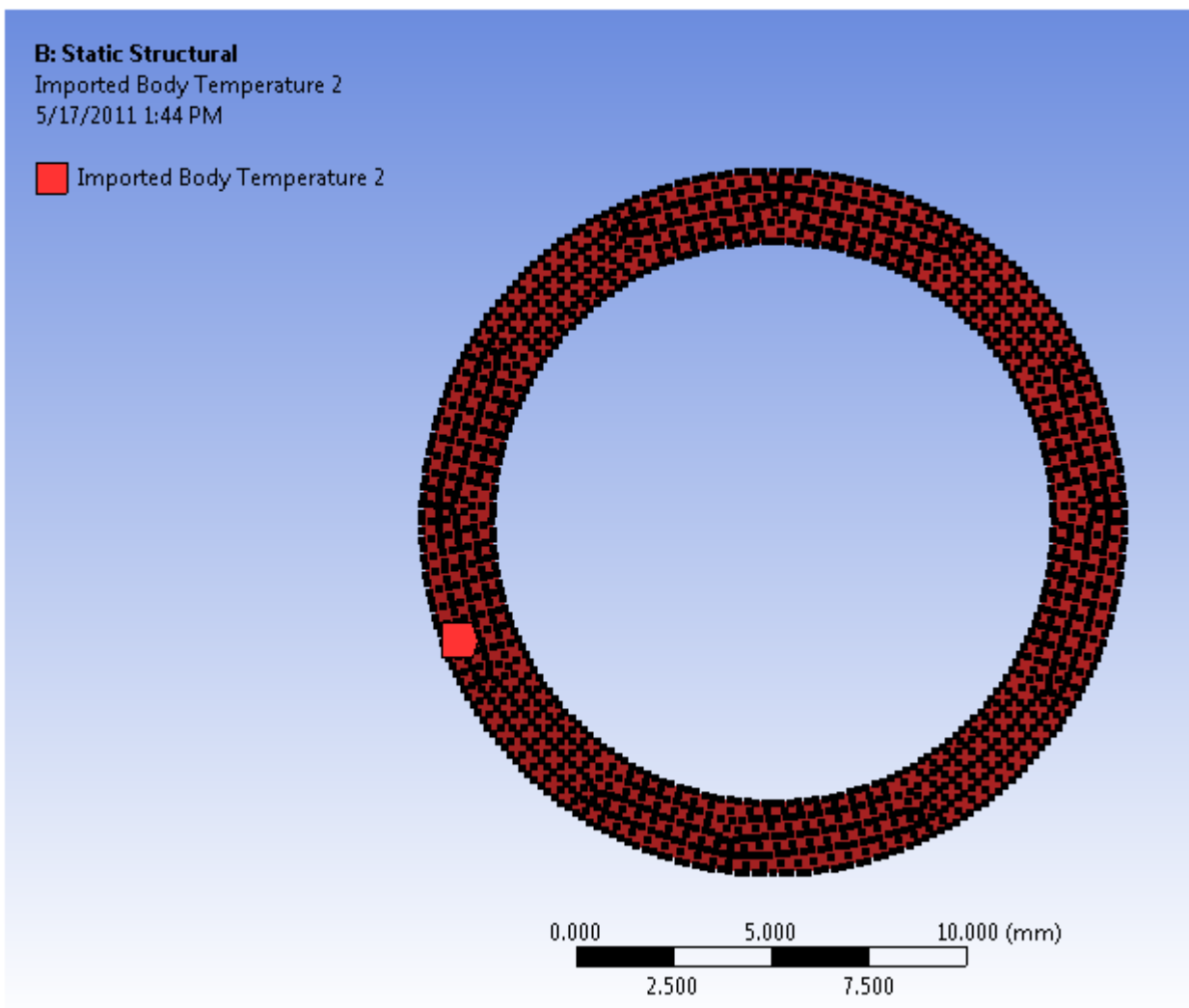
Importing the load generates the following imported load:
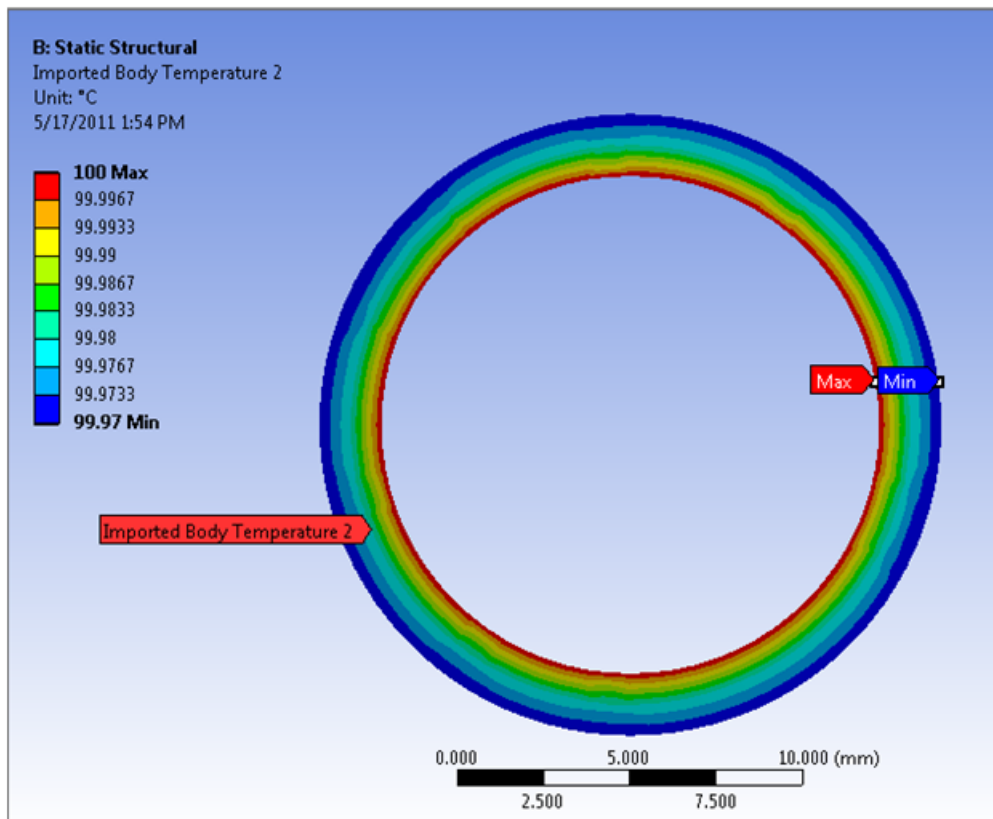
Performing System Coupling Simulations Using External Data

You can use Workbench to perform coupled simulations using multiple analysis or component systems. The **External Data** system may be used as a source of static (that is, unchanging) data for other co-simulation participants such as Ansys Forte or Ansys Mechanical, as described in the System Coupling User 's Guide

External Data System and System Coupling Configuration

Each **External Data** system that is connected to the System Coupling system is represented as a coupling participant in the System Coupling setup. The name of this participant is the name of the **External Data** system specified in the Workbench **Project Schematic**. Since the **External Data** system acts as a source of static data, it can only be defined as the source in System Coupling's Data Transfer definition.

Each input file added to an instance of the **External Data** system is represented as a coupling region for the associated coupling participant in the System Coupling setup.

Supported Input File Formats

To use the **External Data** system as a coupling participant, the Ansys external data file format should be used. For information regarding use of this file format, see Importing an Ansys External Data File as Input (p. 117).

Supported Data Types

Data types provided by external data that are currently consumable by the System Coupling system include temperature and heat rate.

In the XML file transferred from the **External Data** system to System Coupling, if the units of the coordinates in are not all consistent, an error is reported.

External Model

The **External Model** component system enables you to import finite element files created outside of Workbench into Workbench. Supported file formats include:

• Mechanical APDL common database (.cdb)

• Workbench mesh data file (.acmo)

• Abaqus Input (.inp)

• NASTRAN Bulk Data (.bdf, .dat, .nas)

• Fluent Input (.msh, .cas)

• ICEM CFD Input (.uns)

• LS-DYNA Input (.k and .key)

Important:

The **External Model** system only supports .cdb files archived from the Mechanical APDL application using the Blocked format. As needed, you can import your model into the Mechanical APDL application and properly archive the model. Use the PREP7 command **CDWRITE,DB** to create a compatible file. Or, there is a python script available: `<install_dir>\Addins\ExternalLoad\Scripts` that will automatically convert files to the Blocked CDB format. Execute the script using the **File** menu (p. 453) option **Scripting** (p. 251).

Requirements and Restrictions

Review the following processing requirements and restrictions:

• Only data included in the selected **External Model** file is imported. Furthermore:

  – Model assembly is not supported for Ansys data. That is, the **/INPUT** and **CDREAD** commands are not supported for Mechanical APDL files.

  – The *include* command for NASTRAN, Abaqus, and LS-Dyna files are supported.

  – For Abaqus files, if you are using the *Instance* command, only the data from the very first command contained in the file is read. All other data from any additional *Instance* command is ignored. The element and node sets, as well as the materials, that are as-

sociated with the first *Instance* command are processed. Any data that follows the first *End Part* of *End Instance* commands is ignored.

- Scripting commands used to loop over data or generate additional items are not supported.

- The Abaqus scripting command *NGEN* is not supported. The commands *NSET*, *ELSET*, and *NODAL THICKNESS* are the only commands that support the GENERATE parameter.

- Only the first load step of the Abaqus file is read by the **External Model** system.

- The following capabilities are not supported on the Linux operating systems:

  – Data transfer from the Engineering Data workspace.

  – Import of LS-DYNA input (`.k` and `.key`) files is not supported on Linux.

Geometry

For the listed file types, you can import solid, shell, and line body finite element meshes. The geometry is automatically synthesized and made available inside Mechanical.

See the Importing Mesh-Based Geometry section in the Mechanical User's Guide for additional information about how to process and work with imported geometry data.

Finite Element Data

For Mechanical APDL common database (.cdb), Abaqus Input (.inp), and Nastran Bulk Data (.bdf, .dat, .nas) files, you can import the following finite element data for use within your simulation.

- Bolt Pretensions and Premeshed Bolt Pretensions

- Boundary Conditions

- Composite Plies

- Constraint Equations and Couplings

- Contact

- Coordinate Systems

- Cross Sections

- Element Orientations

- Flexible Remote Connections

- Named Selections

- Nodal Orientations

- Point Masses

- Rigid Remote Connections

- Shell Thicknesses

- Spring Connectors

See Importing Mesh-Based Databases in the *Mechanical User's Guide* for additional information about the data types and how to use the data in Mechanical.

Working with the External Model System

Select a link below to jump to steps for working with finite element data from the **External Model** system:

Creating and Configuring an External Model System

Transferring Data to Mechanical

Transferring Data to Engineering Data

Note:

If you open Mechanical in such a way that the simulation does not yet include a mesh, you can use the **Open** option of the File tab to import an **External Model** file.

Creating and Configuring an External Model System

To specify an **External Model** system:

1. To add an **External Model** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

   Note:

   You can drag and drop your mesh file directly onto the Workbench **Project Schematic**. The application automatically creates an **External Model** system linked to a **Mechanical Model** system. Any further specification is still required.

   ---

2. Double-click the **Setup** cell or right-click the **Setup** cell and select **Edit** from the context menu. This opens the **External Model** tab in order to select a desired file or files.

3. In the **Outline of Schematic** pane, in the **Location** column, click the icon to add a data source file ( ).

4. Browse to the location of your source files.

5. Select one or more files and click **Open**. The selected file names and locations are automatically displayed in the **Data Source** column and an entry appears in the **Identifier** column.

   Note:

   The application does not copy input files specified in **External Model** into the project folders of downstream systems. The application references these files by absolute path only. Be sure you don't move or rename these files.

   ---

6. If required, update the name in the **Identifier** column. By default this is **File**n.

7. If required, enter descriptions for the files in the **Description** column.

8. For the ABAQUS (.inp), NASTRAN Bulk Data (.bdf, .dat, .nas), and LS-DYNA Input (.k and .key) file, a row labeled **Click here to add a supporting file** is added to the schematic that enables you to select and attach additional files to the parent file, such as node and element files. These attached files are included in the same import group as the parent file and Mechanical treats them as a package. There is no limit on the number of files you can add and/or attach.

    If your parent file contains an "Include" command, property formatted, that references the desired supporting files, the application software can automatically complete the supporting files list for you if you update the Setup cell (RMB > Update) of the External Model system. The supporting file list is auto-filled only if it is empty. In addition, if the parent file references a file that is not found in the target directory, a warning message is issued. Note that you can also delete files from the supporting files list. Any supporting file that appears in the parent file (as "included") but not in the supporting file list will be ignored.

9. Optionally, you can right-click a file (or files) in the **Outline** pane and use the context menu to duplicate them.

    All files (whether imported or duplicated) can be sorted or filtered.

10. Once you have opened your files, use the **Properties** pane to modify the unit system, components, and/or coordinate system transformation properties. These properties transform the mesh coordinate systems of the sub-assemblies for proper alignment in Mechanical.

11. If you select multiple files in the **Data Source** column, the **Properties of File** pane displays:

    • A value when that value is the same for all selected files

    • A blank field when values differ between selected files

    • A yellow field when a value is required, but not currently specified for at least one of the files.

If you edit any field in the **Properties** pane when multiple files are selected, your change is applied to all files.

> **Caution:**
>
> Although you can multi-select files in the **Data Source** pane, when you click away from that pane the highlighting applied to those files disappears. However, the files remain active and any subsequent operations that are applied affect the files.

Table 1: Properties Pane: Description Section

Property	Description
Application Source	This property provides a drop-down list of available file types.

Table 2: Properties Pane: Definition Section

Property	Description
Unit System	The unit system in which the file is defined. Source points are interpreted in these units. > **Important:** > > When you import a project from a previous release that contains an **External Model** system, the Length Unit from that project is converted to a compatible Unit System. You must confirm that the Unit System chosen is appropriate; if it is not, choose the correct system from the dropdown list of consistent unit systems. > > The **External Model** system must have a consistent unit system before you can perform an update.
Process Nodal Components	Enables the **External Model** system to import node-based components defined in the mesh files. The application transfers the data to downstream Mechanical systems as node-based Named Selections. The application renames the node-based Named Selection objects in Mechanical based on the selection made in the **Object Renaming** property.
Nodal Component Key	This entry field enables you to filter and import only those node-based components that start with a specified name/string value in the mesh files. For example, you want to import only node-based components that start with the prefix string "`nodal_*`." Enter that string into this field and the application filters through all component names and returns only the components that begin with this key string value.
Process Element Components	Enables the **External Model** system to import any element-based components defined in the mesh files. The application transfers data to downstream Mechanical systems as elemental-based Named Selections. The application renames the element-based Named Selection objects in Mechanical based on the selection made in the **Object Renaming** property.

Property	Description
Element Component Key	This entry field enables you to filter and import only those element-based components that start with a specified name/string value in the mesh files. For example, you want to import only element-based components that start with the prefix string "**elemental_***." Enter that string into this field and the application filters through all component names and returns only the components that begin with this string value.
Process Face Components	Enables the **External Model** system to import any face components defined in the mesh files. The application transfers data to downstream Mechanical systems as face-based Named Selections. The application renames the face-based Named Selection objects in Mechanical based on the selection made in the **Object Renaming** property.
Face Component Key	This entry field enables you to filter and import only those face components that start with a specified name/string value in the mesh files. For example, you want to import only face components that start with the prefix string "**face_***." Enter that string into this field and the application filters through all component names and returns only the components that begin with this string value.
Process Model Data	When you uncheck/deactivate this option, the application will only process data directly needed for the creation of the mesh and geometry of the external file. All other data from the external file will be skipped. This includes data such as Connections and engineering abstractions such as plies, bolt pretensions, boundary conditions, etc. By default, this the option is active/checked and the application processes all data.
Process Mesh200 Elements	This option supports .cdb files only. When selected, Mesh200 elements present in your mesh file are included with your geometry in Mechanical.
Check Valid Blocked CBD File	This property displays when the **Application Source** property is set to **MAPDL** (default). This property instructs the application to verify that the specified file is a valid Blocked `.cdb` file. **External Model** only supports Blocked format. If the property is checked and the file is not a valid blocked file, an error message displays. As needed, you can import your model into the Mechanical APDL application and properly archive the model. Use the PREP7 command **CDWRITE,DB** to create a compatible file. You can also automatically convert files to the Blocked CDB format using the script file contained in this directory: `<install_dir>\Addins\ExternalLoad\Scripts`. Execute the script using the **File** menu (p. 453) option **Scripting** (p. 251).note **Important:** If you unselect this property and your file is not a valid Blocked cdb file, the application may not properly import your model.
Node and Element Renumbering Method	When you connect the **Setup** cell of an **External Model** to a Mechanical system, this property controls whether mesh nodes and elements are automatically renumbered to prevent conflicts. The property's options include **Automatic** (default) and **Offset**. The application does not renumber nodes and elements if you specify **Offset**.

Property	Description
	In this case, and in order to avoid conflicts, use the **Node Offset** and **Element Offset** fields to prepend your node and element IDs with the positive number of your choice. This property requires that the **Number of Copies** property is set to **0**. If you enter a value in the **Number of Copies** property, other than zero, the application requires automatic node and element ID renumbering.
Transformation Type	Use this property to apply a transformation to your upstream model. The transformation options for this property include **Rotation and Translation** (default) and **Mirroring**. The **Rotation and Translation** option enables you to translate the origin of the upstream model along the X, Y, or Z axis or to rotate the model about its origin in the XY, YZ, or ZX plane. Using the **Mirroring** option enables you to import your upstream model as well as a mirror copy of the model about a specified plane.

Table 3: Properties Pane: Rigid Transformation Section

Property	Description
Number Of Copies	When set to zero (default), only the source mesh is transformed. If you specify a number of copies greater than zero, these will be in addition to the source mesh. For example, if you import a `.cdb` file with a single part and set **Number Of Copies** to 2, you will get 3 parts in Mechanical.
Transform Original	This property is available only when **Number Of Copies** is set to 1 or greater. Select the check box if you want to apply the specified transformation to the source mesh as well as any copies.
Origin X/Y/Z	These properties allow you to translate the origin of the model along the X, Y, or Z axis. If you specify any copies, the translation will be applied relative to the previous copy (or source mesh in the case of the first copy).
Theta XY/YZ/ZX	These properties allow you to rotate the model about its origin in the XY, YZ, or ZX plane. If you specify any copies, the rotation will be applied relative to the previous copy (or source mesh in the case of the first copy).

Note:

For the **Rotation and Translation** option, transformations are applied in the following order:

1. Rotation about the Y Axis

2. Rotation about the X Axis

3. Rotation about the Z Axis

4. Translations

Note:

The application does not apply Rigid Transform values if they expand to the "identity transformation." The identity transformation is when the values for Origin X, Origin Y, Origin Z are all zero (within a tolerance) and Theta XY, Theta YZ, Theta ZX are all multiples of 360 degrees (within a tolerance). The application uses an absolute and relative tolerance equal to the 2D Tolerance specified in the Geometry category of the Options preferences. You can change the specified default value (1e-5) using the preference dialog.

Table 4: Properties Pane: Mirror Transformation Section

Property	Description
Plane Point X/Y/Z	These properties enable you to define the coordinates of a point on the mirroring plane.
Plane Normal X/Y/Z	These properties enable you to define a vector that is normal (orthogonal) to the mirroring plane.

Note:

The **Plane Point Z** and **Plane Normal Z** properties are hidden for 2D models.

12. To modify any file in the **Outline of Schematic** pane, in the **Location** column, click [...] and select a new file.

13. To delete files that you have selected (or multi-selected) right-click one of the files in the **Outline of Schematic** pane and select **Delete** from the context menu.

14. Return to the **Project** tab.

15. Update the project (p. 243).

16. If required, link (p. 60) the **Setup** cell of the **External Model** to the following cells of a Mechanical Model or a Mechanical analysis system:

   - **Model**

   - **Engineering Data** (**Windows Platform Only**)

Transferring Data to Mechanical

Once the **External Model** system is created, open your mesh files in Mechanical.

To add a downstream Mechanical system:

1. Drag a valid analysis system from the **Toolbox** onto the **Project Schematic**.

2. Link (p. 60) the **External Model** system **Setup** cell to the Mechanical system **Model** cell to complete the connection.

   This action deletes the **Geometry** cell. Multiple model cells in the **Project Schematic** can link to one analysis system. See Assembling External Models and Mechanical Models in the *Mechanical User's Guide* for more details.

3. Modify the **Mesh Conversion Options** associated with the Mechanical **Model** cell as required.

   See the Importing Mesh-Based Geometry section in the Mechanical User's Guide for more details.

4. Launch Mechanical.

Associativity between External Model and Mechanical

Geometry from **External Model** (`.cdb`) files is partially associative. When you have geometry from multiple **External Model** system assembled, and you refresh upstream model data into the downstream system, any geometry scoping that you have performed on an object in the downstream analysis is lost for the modified **External Model** system only. That is, only **External Model** systems that you change lose scoping. For example, if you have two **External Model** systems assembled, System 1 and System 2, and you have objects scoped to geometry in the assembled system. If you modify System 1 and then refresh the upstream system, geometry scoping on objects is lost only for System 1. System 2 experiences no scoping losses. A more robust way to maintain scoping is to properly define imported named selections or criterion-based named selections. These scoping features automatically update when the upstream model updating is complete.

Re-reading Modified External Mesh Files

If you change your mesh file, such as making a manual change or as a result of an automated tool, these types of changes are not automatically updated in Workbench. You must reread your external data file and update the system to re-import the changed data.

To re-read modified external mesh files:

1. Right-click the **Setup** cell

2. Select **Re-read Input Files** from the context menu.

3. Right-click the **Setup** cell

4. Select **Update** from the context menu.

The **Re-read Input Files** operation causes Workbench to regard the file as having changed whether the file has changed or not, and the status of the **Setup** cell changes appropriately.

Transferring Data to Engineering Data

As required, you can incorporate Engineering Data.

To add a downstream Engineering Data system:

1. Drag a valid analysis system, or an Engineering Data system, from the **Toolbox** onto the **Project Schematic**.

2. Link (p. 60) the **External Model** system **Setup** cell to the Engineering Data system.

    You can link the **Setup** cell to multiple Engineering Data cells.

3. Launch Engineering Data.

    > **Note:**
    >
    > - Review the supported MAPDL, NASTRAN, and ABAQUS material commands listed in the following subsections.
    >
    > - The application creates material names in the Engineering Data workspace based on the material identifier in the imported file (.cdb, ABAQUS Input, or NASTRAN Bulk Data). The names include the corresponding (linked) Workbench cell and number as well as the value in the **Identifier** column in the **External Model** setup interface.

MAPDL Material Commands

The following MAPDL material commands are supported when importing material data into the Engineering Data workspace.

Command	Description
MPTEMP MPDATA	The following temperature-dependent material property labels are supported for these commands: ALPX, ALPY, ALPZ, C, DENS, EX, EY, EZ, GXY, GYZ, GXZ, KXX, KYY, KZZ, NUXY, NUYZ, NUXZ, PRXY, PRYZ, PRXZ, REFT, MU, MURX, MURY, MURZ, RSVX, RSVY, RSVZ, MGXX
TB TBDATA	Bilinear isotropic hardening (BISO) is the only non-linear material property and label supported.

NASTRAN Supported Material Specifications

The following NASTRAN material properties are supported when importing material data into the Engineering Data workspace.

Card	Properties
MAT1 (Material 1)	• Young's Modulus • Shear Modulus • Poisson's Ratio • Mass Density • Thermal Expansion Coefficient

Card	Properties
	• Reference Temperature
MAT2 (Material 2)	• 3 X 3 symmetric material property matrix • Mass density is supported • Thermal expansion coefficient vector • Reference temperature for thermal expansion, if thermal expansion is defined
MAT3 (Material 3)	No data is supported. Only the material ID is maintained.
MAT4 (Material 4)	• Thermal Conductivity • Specific Heat
MAT5 (Material 5)	No data is supported. Only the material ID is maintained.
MAT8 (Material 8)	• Moduli of elasticity – Modulus in the Z direction uses the value in the Y direction. • Poisson's ratio – Ratio in the YZ directions uses the value in the XY direction. – Ratio in the XZ direction uses the ½ the value in the XY direction. • Shear moduli • Mass density • Thermal expansion coefficients • Reference temperature for thermal expansion, if thermal expansion is defined
MAT9 (Material 9)	• 6 X 6 symmetric material property matrix • Mass density is supported **Note:** The stiffness terms must be positive (which requires that all determinants to be positive). Otherwise, the properties are not imported.
MAT10 (Material 10)	No data is supported. Only the material ID is maintained.

ABAQUS Supported Materials Keywords

The following ABAQUS Materials Keywords are supported when importing material data into the Engineering Data workspace.

Keyword	Description
*MATERIAL	The NAME parameter is supported.

*ELASTIC	• Supported for TYPE = ISOTROPIC, ENGINEERING CONSTANTS, and LAMINA.
	
	 • The DEPENDENCIES parameter is *not* supported.
	
	 Material property definition is *not* processed.
	
	 • For TYPE = ISOTROPIC:
	
	   – Young's Modulus and Poisson's Ratio are supported.
	
	   – If Poisson's Ratio is not specified, **0.3** is the value that is used.
	
	   – Temperature is dependency supported.
	
	 • For TYPE = ENGINEERING CONSTANTS:
	
	 Young's Moduli, Poisson's Ratios, and the Shear Moduli in the principal directions is supported.
	
	 • For TYPE = LAMINA:
	
	   – Young's Moduli, Poisson's Ratios, and the Shear Moduli in the principal directions is supported.
	
	       Modulus in the Z direction uses the value in the Y direction.
	
	       Ratio in the YZ directions uses the value in the XY direction.
	
	       Ratio in the XZ direction uses the ½ the value in the XY direction.
	
	   – Temperature dependency is **not** supported. The data for the first temperature is used.

*HYPERELASTIC	The following mutually exclusive parameters (material models) are supported: • MOONEY-RIVLIN • NEO HOOKE: This parameter is equivalent to *HYPERELASTIC, REDUCED POLYNOMIAL and as a result, is imported as Yeoh 1st Order. • OGDEN: Supports parameter N = 1 (default), 2, or 3. • POLYNOMIAL (default): Supports parameter N = 1 (default), 2, or 3. • REDUCED POLYNOMIAL: Supports parameter N = 1 (default), 2, or 3.

Keyword	Description
	• YEOH **Note:** Temperature dependence is not supported.
*DENSITY	• The DEPENDENCIES parameter is *not* supported. Material property definition is *not* processed. • Temperature dependency is supported.
*EXPANSION	• TYPE = ISOTROPIC only • The DEPENDENCIES parameter is *not* supported. Material property definition is *not* processed. • Temperature dependency is supported.
*PLASTIC	• HARDENING = ISOTROPIC only. • The RATE parameter is not supported. These stress-strain curves are ignored. • Curve must have a positive slope.
*CONDUCTIVITY	• Supported for TYPE = ISOTROPIC and ORTHO. • The DEPENDENCIES parameter is *not* supported. Material property definition is *not* processed. • For TYPE = ISOTROPIC, temperature dependency is supported. • For TYPE = ORTHO, temperature dependency is *not* supported; the data for the first temperature is used.
*SPECIFIC HEAT	• The DEPENDENCIES parameter is *not* supported. Material property definition is *not* processed. • Temperature dependency is supported.

Fluent

The **Fluent** system launches Ansys Fluent, allowing you to complete fluid flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries.

Within Fluent, a computational mesh is applied to a geometry, pertinent mathematical models are applied (for example, low-speed, high-speed, laminar, turbulent, and so on), materials are chosen, boundary conditions are defined, and solution controls are specified that best represent the problem to be solved. Fluent solves the mathematical equations, and results of the simulation can be displayed in Fluent for further analysis (for example contours, vectors, and so on).

To work through a **Fluent** system:

1. To add a **Fluent** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Right-click the **Setup** cell and select one of the following options from the context menu:

    - To specify the Fluent settings, select **Edit**.

    - To import a previously saved Fluent case file, select **Import Fluent Case**.

    - To import previously saved Fluent case and data files, select **Import Fluent Case And Data**.

Fluent (with Fluent Meshing)

The **Fluent (with Fluent Meshing)** system (also known as Fluent Meshing or Fluent in meshing mode) launches Ansys Fluent, allowing you to complete fluid flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries.

Within Fluent in meshing mode, a computational mesh is imported and manipulated. Switch to Fluent in solution mode where pertinent mathematical models are applied (for example, low-speed, high-speed, laminar, turbulent, and so on), materials are chosen, boundary conditions are defined, and solution controls are specified that best represent the problem to be solved. Fluent solves the mathematical equations, and results of the simulation can be displayed in Fluent for further analysis (for example, contours, vectors, and so on).

For detailed information on working with Fluent Meshing, see the Fluent User's Guide as well as the other on-line documentation available under the Help menu within Fluent Meshing. In addition, see the Fluent in Workbench User's Guide.

For information on submitting Fluent jobs to Remote Solve Manager, see Fluent (p. 141).

To work through a **Fluent (with Fluent Meshing)** system:

1. To add a **Fluent (with Fluent Meshing)** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open Fluent in meshing mode and import a computational mesh, double-click the **Mesh** cell.

3. Right-click the **Setup** cell and select one of the following options from the context menu:

    - To specify the Fluent settings, select **Edit**.

    - To import a previously saved Fluent case file, select **Import Fluent Case**.

    - To import previously saved Fluent case and data files, select **Import Fluent Case And Data**.

Forte

The **Forte** system launches ANSYS Forte, allowing you to complete fluid flow analysis of engines and other detailed reacting systems, employing heat transfer and advanced spray models in complex geometries and using automated mesh generation.

Import a mesh or allow Forte to mesh on-the-fly with automated mesh generation. Within Forte, a computational mesh is applied to a geometry, pertinent mathematical models are applied (for example, low-speed, high-speed, laminar, turbulent, and so on), chemistry and reaction condition are selected, boundary conditions are defined, and solution controls are specified that best represent the problem to be solved. Forte solves the mathematical equations, and the results of the simulation can be displayed in Forte for further analysis (for example contours, vectors, and so on).

For detailed information on working with Forte see the Ansys Forte User's Guide as well as the other on-line documentation available under the Help menu within Forte.

To work through a **Forte** system:

1. To add a **Forte** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Right-click the **Setup** cell and select one of the following options from the context menu:

   • To specify the Forte settings, select **Edit**.

   • To import a previously saved Forte project, select **Import Forte Project**.

   • To import previously saved Forte replay file, select **Replay Journal**.

Geometry
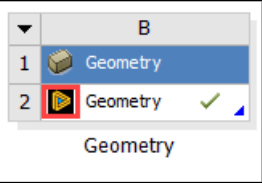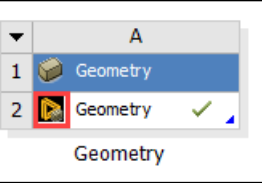
The **Geometry** system launches either Ansys Discovery, SpaceClaim, or DesignModeler, allowing you to create and import a model. It also allows you to import an existing geometry file.

> **Note:**
>
> When creating or importing geometry from SpaceClaim, there are user interface and functional differences between SpaceClaim and SpaceClaim Meshing.
>
> The Geometry cell has a different icon.

SpaceClaim	SpaceClaim Meshing

Downstream systems like Mechanical recognize the different types.
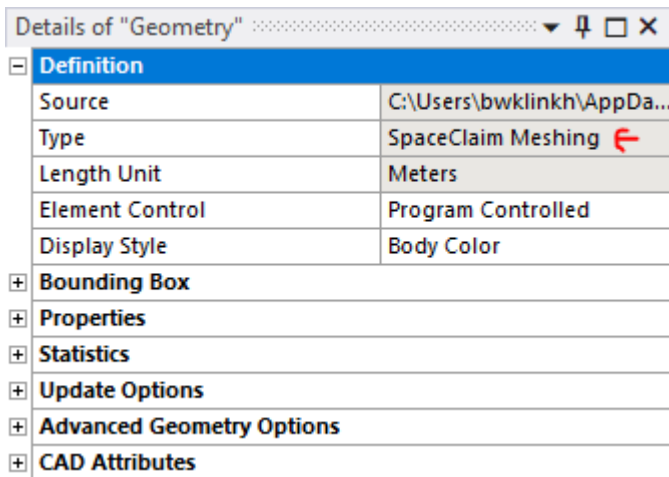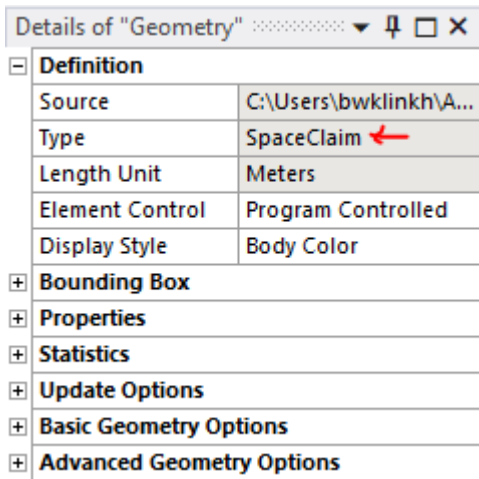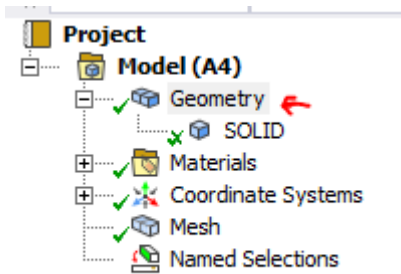
There are also different transfer types. For more details, see Transferring a Mesh to Downstream Systems in in the *Discovery SpaceClaim* documentation.

You can also import BladeGen model files into DesignModeler using the BladeEditor feature. BladeEditor provides a seamless path to both structural and fluid analysis, which enables you to efficiently transition from preliminary blade design, to full 3D viscous flow analysis, and finally to your native CAD system. BladeEditor features are available in DesignModeler depending on your license preferences. For more information on Ansys BladeEditor, see Ansys BladeEditor help.

You can set your user preferences to select your preferred editor as default. For more information, see Geometry Import (p. 433).

For more information on the geometry capabilities in Workbench, see DesignModeler Project Schematic Operations in the *Ansys DesignModeler User's Guide*

To work through a **Geometry** system:

1. To add a **Geometry** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

    You can also drag a geometry file to the **Project Schematic**. A **Geometry** component system is created with the geometry file attached.

2. Right-click the **Geometry** cell and select one of the following options from the context menu:

    -   To open SpaceClaim, select **New SpaceClaim Geometry**.

        You can then build a model using its features. When you save the geometry, the file becomes an `.scdoc` file.

    -   To open DesignModeler, select **New DesignModeler Geometry**.

        You can then build a model using its features. When you save the geometry, the file becomes an `.agdb` file.

    -   To import an existing geometry file, select **Import Geometry**.

GRANTA MI

The **GRANTA MI** system launches GRANTA MI:Materials Gateway, an application that can be embedded in a variety of CAD and CAE environments, allowing you to directly access your company's GRANTA MI database. You can then search and browse the GRANTA MI database, find materials, view their datasheets, choose applicable CAE materials models, and then import these models directly to Ansys projects, complete with full traceability information.

For more information, see GRANTA MI Support.

To work through a **GRANTA MI** system:

1. 1. To add a **GRANTA MI** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Right-click the **MI Materials Gateway** cell and select **Edit** from the context menu.

GRANTA Selector

The **GRANTA Selector** system launches GRANTA Selector, a suite of software tools and materials property data that supports systematic materials selection.

For more information, see GRANTA Selector Support.

To work through a **GRANTA Selector** system:

1.   1. To add a **GRANTA Selector** component system, drag the system from the **Toolbox** to the
      Project Schematic or double-click the system in the **Toolbox**.

2.   Right-click the **GRANTA Selector** cell and select **Edit** from the context menu.

ICEM CFD

The **ICEM CFD** system launches Ansys ICEM CFD. This application extends Ansys meshing capabilities
with robust and varied geometry import, the ability to efficiently mesh large or complex models with
extended meshing controls, advanced interactive blocking tools for structured or unstructured mesh
generation, extended mesh diagnostics, advanced interactive mesh editing, and output to a wide
variety of solver formats including CFD, FEA, and neutral formats.

Ansys ICEM CFD can input geometry in almost any format, whether a commercial CAD design package,
third-party universal database, scan data, point data, or even combinations of CAD, facets, and mesh.
It includes a variety of "patch independent" meshing methods which are able to work with dirty cad
and does not require that surfaces be formed into solids or that flow volumes be extracted. It does
include a wide range of interactive geometry, blocking and mesh editing tools that can be used to
generate advanced or high quality meshes for any application.

The data-integrated ICEM CFD component system, or "Ansys ICEM CFD Add-in", enables you to launch
ICEM CFD from Ansys Workbench and use it to build a project, with the option of adding upstream
data from Geometry, Mesh, Mechanical Model, or combined Geometry and Mesh system components.
You can also use ICEM CFD to provide data to downstream component systems, such as Ansys Forte,
Ansys CFX, Ansys Polyflow, FENSAP-ICE, and Mechanical APDL.
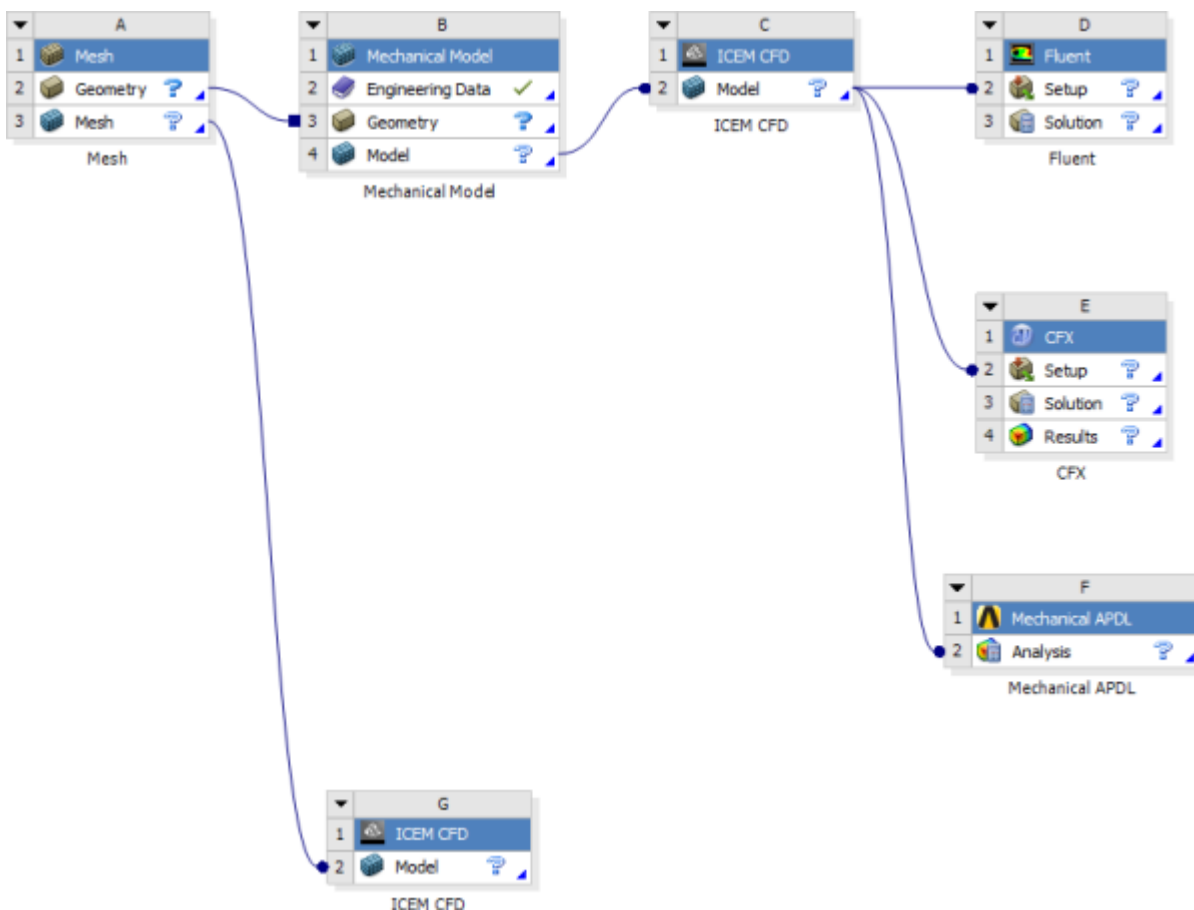
Elements of the ICEM CFD Component
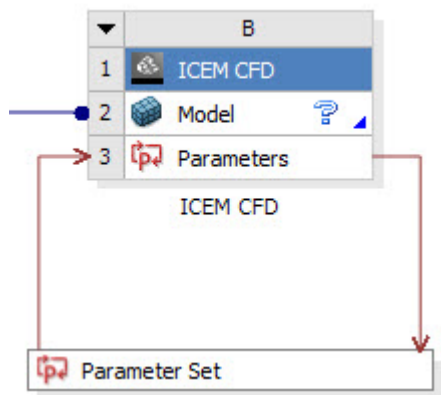
The ICEM CFD component system contains the following cells.

Cell	Description
System header	Identifies the component type and provides access to standard Workbench context menu options (p. 457).
Model	Modifies some aspects of the project and opens the project in ICEM CFD. The **Model** cell context menu items include the following: • **Edit**: Opens the ICEM CFD application and loads an existing Geometry/ICEM CFD file. • **Duplicate**: Copies the entire geometry and mesh data, enabling you to edit the **Model** cell in the duplicate system to investigate an alternative modeling approach. • **Transfer Data From New**: Enables the transfer of data from upstream **Geometry**, **Mesh**, Mechanical **Model**, or combined **Geometry** and **Mesh** components. **Note:** If the **Geometry** cell option **Use Associativity** is **On** (its default setting), Part Reference IDs are stored in a Workbench database and remain persistent in ICEM CFD even if the topology of the geometry is changed. • **Transfer Data to New**: Enables the transfer of data from an ICEM CFD project to downstream data-integrated system projects, such as: – Forte – CFX – Polyflow – FENSAP-ICE

Cell	Description
	– Mechanical APDL
	• **Update**, **Refresh**, **Reset**, **Rename**, **Properties**, and **Add/Edit Note**. These standard actions are described in System Header Context Menu Options (p. 457).
	• **Create Subset(s) from Named Selection**: Transfers overlapping named selections to multiple subsets instead of to exclusive parts if this option is set. This creates geometry subsets instead of parts, allowing you determine the part where the geometry entity (point/curve/surface) should be associated. When this option is disabled, each geometry is assigned to only one part, losing association with other named selections (which are mapped to parts).
Parameters (optional)	Displays and allows you to edit input and output parameters for ICEM CFD.

Note:

Scripts written in ICEM CFD may not be parametric with upstream or downstream projects. Care should be taken to write scripts whose functions do not exceed the capabilities of the upstream or downstream component systems.

Creating an ICEM CFD Component

You can create an ICEM CFD component system in Workbench using any of the following methods:

• Double-click the ICEM CFD system template in the **Toolbox**.

• Drag-and-drop the ICEM CFD system template onto the **Project Schematic**.

• Right-click a **Geometry** or **Mesh** system and select **Transfer Data to New** → **ICEM CFD**.

• Import (p. 49) a standalone ICEM CFD project into Workbench.

Updating ICEM CFD Projects

Updating a project in Workbench brings the entire ICEM CFD system up to the most current status, including upstream and downstream data.

Changes that require you to update your ICEM CFD project include changes to upstream data and changes you make in the ICEM CFD editor. When changes are made, the cell in the **Project Schematic** window indicates that an update is required. Note, however, that changes made in the ICEM CFD editor will not cause the system to go out of date until the project, tetin file (geometry), blocking file, and/or Replay file are saved.

The actions taken by Workbench depend on whether the following conditions are met:

• Blocking exists.

• A Replay file exists.

- Blocking parameters are set.

- Other input parameters are set.

The following table describes the actions performed by ICEM CFD according to these conditions:

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
No	No	No	No	1. Runs tetra default meshing. 2. Saves the unstructured mesh. 3. Saves the project
Yes	No	No	No	1. Runs hexa default meshing. 2. Saves the unstructured mesh. 3. Saves the project.
No	No	No	Yes	1. Sets all input parameters. 2. Runs tetra meshing. Runs prism meshing if any `Part PART_NAME:  Prism` input parameters exist. (See Setting Parameters for Prism Meshing (p. 157).) 3. Saves the unstructured mesh. 4. Saves the project.
No	Yes	No	Yes	1. Sets all input parameters. 2. Runs the Replay file. 3. Saves the unstructured mesh. 4. Saves the project.

No	Yes	Yes	Yes	1. Sets all input parameters except blocking parameters.
				
				2. Runs the Replay file.
				
				3. If blocking now exists:
				
				   a. Sets blocking input parameters.
				
				   b. Runs hexa meshing.
				
				   c. Converts pre-mesh to unstructured.
				
				   d. Saves the unstructured mesh

149

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
				4. Saves the project.
Yes	No	Yes	Yes	1. Sets all input parameters. 2. Sets blocking input parameters. 3. Runs hexa meshing. 4. Converts pre-mesh to unstructured. 5. Saves the unstructured mesh. 6. Saves the project.

Yes	Yes	Yes	Yes	1. Sets all the input parameters except blocking.
				
				 2. Runs the Replay file.
				
				 3. If blocking still exists:
				
				    a. Sets blocking input parameters.
				
				    b. Runs hexa meshing.
				
				    c. Converts pre-mesh to unstructured.
				
				    d. Saves the unstructured mesh.
				
				 4. Saves the project.

ICEM CFD saves the unstructured mesh and project only if the ICEM CFD project is closed and you update from Workbench. If the ICEM CFD project is open, you must manually save the unstructured mesh and project.

Meshing input parameters are set before the **Replay** file is run; blocking Input parameters are set after a replay file is run, but only if blocking exists after the replay file is run.

If no **Replay** file exists, the default mesher is determined by the presence or absence of blocking: if no blocking exists in the project, **tetra** is the default. Conversely, if blocking exists in the project, **hexa** is the default mesher.

Updating a Project

To update a project and refresh upstream and downstream data, right-click the **Model** cell and select **Update**.

> **Note:**
>
> If available, **Update** uses the **ICEM CFD Replay** file to update the ICEM CFD project.

Interface Differences in the Data-Integrated ICEM CFD

The data-integrated ICEM CFD interface has been modified to provide additional functionality that enhances the integration of ICEM CFD and Workbench.

- **One-click menus** that enable you to control basic project and **Replay Recording** functions.

  > **Note:**
  >
  > If you create Workbench input parameters and want to step through the script line-by-line or using a line range, you must open the **Workbench Replay Control** dialog from this menu.

- The **Workbench Replay Control** dialog enables you to test and edit Replay scripts created within Workbench.

You can ensure that you are working within the data integrated environment by checking the **Message** window within ICEM CFD. The first line identifies that the application is integrated in the Workbench framework.

One-Click Menus

The following one-click menu options are available in the **Toolbar** when you open ICEM CFD from Workbench:

- **Save Project**: Saves the entire project, including Workbench data.

- **Refresh Project**: Refreshes the upstream data in the ICEM CFD project.

- **Update Project**: Brings the entire ICEM CFD system up to the most current status, including upstream and downstream data.

- **Start Replay Recording**: Begins recording the commands needed to generate a custom meshing process. All of the steps in the mesh development process are recorded, including blocking, mesh size, edge meshing, boundary condition definition, and final mesh generation. See Replay Scripts in the *Ansys ICEM CFD Help Manual*. After you click the **Start Replay Recording** icon, the icon changes to the **Stop Replay Recording** icon. You can click this icon to stop recording.

  You can also click the arrow to choose **Pause Replay Recording**, **Run Replay File**, **Delete Replay File**, and **Replay Control**, which opens the **Workbench Replay Control** dialog.
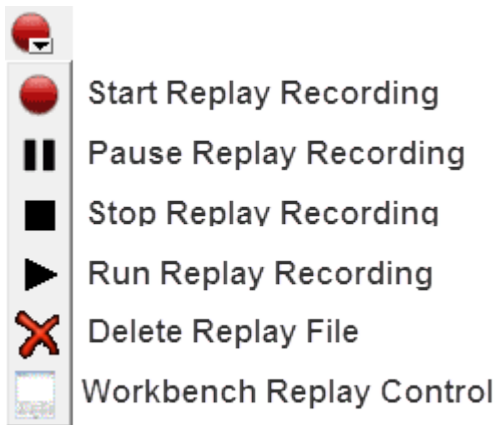
  Start Replay Recording
  Pause Replay Recording
  Stop Replay Recording
  Run Replay Recording
  Delete Replay File
  Workbench Replay Control

  ### Note:

  If you create Workbench input parameters and want to step line by line or using a line range through a replay file, you must use the **Replay Control** item from this menu to start the **Workbench Replay Control** dialog.

- **Output Mesh**: You can choose to save the ICEM CFD mesh output to one of several formats including Forte, CFX, Polyflow, Ansys Meshing, or FENSAP-ICE projects.

Workbench Replay Control Dialog

The **Replay Control** dialog helps you create, test, and edit script files by performing operations in Ansys ICEM CFD and recording the equivalent Tcl/Tk commands in a Replay file. You can then use the dialog to step through and edit the script.

The **Workbench Replay Control** dialog works exactly the same as the standalone **Replay Control** dialog, with two key exceptions:

- The **Workbench Replay Control dialog** supports Workbench input parameters and allows you to step through them.

- The dialog automatically loads the current Replay script file (`ICM.rpl`) instead of opening a file browser.
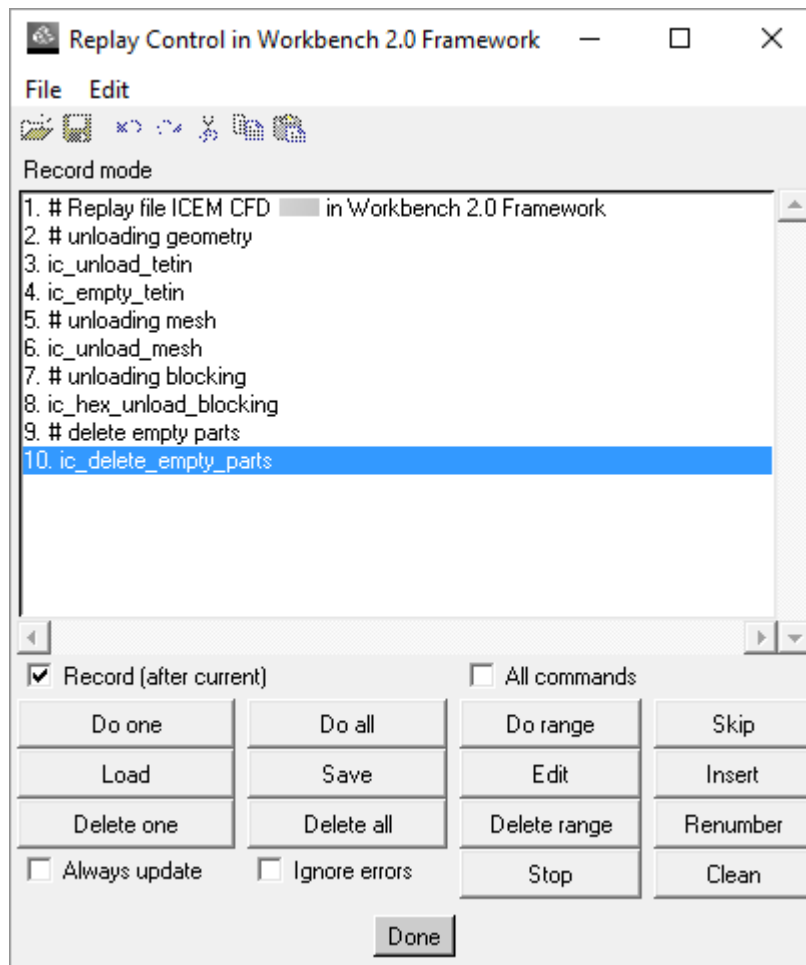
For more information, see the Replay Scripts in the *Ansys ICEM CFD Help Manual*.

Setting Parameters

Using input parameters in Workbench enables you to pass data to ICEM CFD, while output parameters allow you to receive data back from ICEM CFD. The interaction of parameters between applications provides you with greater flexibility and capabilities to run optimization and what-if scenarios. For

more information about using parameters in Workbench, see Working with Parameters and Design Points (p. 283).

Parameters may be set globally or individually, with individual parameters taking precedence over global values.

You can perform the following operations involving parameters:

- Setting Input Parameters (p. 154)

- Setting Workbench Mesh Parameters for Parts (p. 156)

- Setting Parameters for Prism Meshing (p. 157)

- Setting Output Parameters (p. 158)

- Setting User-Defined Parameters (p. 160)

- Deleting Parameters (p. 161)

Setting Input Parameters

Clicking the box to the right of certain **Meshing Input** parameters enables you to select whether the parameter is controlled from within ICEM CFD or from within Workbench. A "**P**" in the check box indicates that it has been selected as a Workbench input parameter. If the check box is empty, you can control the input from within ICEM CFD.
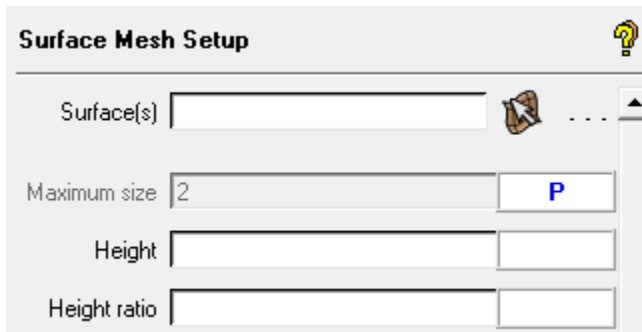
You can set the following input parameters in Workbench:

- **Global Mesh Size** (See Global Mesh Size in the *Ansys ICEM CFD Help Manual*).

- **Shell Meshing** (See Shell Meshing Parameters in the *Ansys ICEM CFD Help Manual*).

- **Volume Meshing** (See Volume Meshing Parameters in the *Ansys ICEM CFD Help Manual*).

- **Prism Meshing** (See Prism Meshing Parameters in the *Ansys ICEM CFD Help Manual*).

- **Surface Mesh Setup** (See Surface Mesh Setup in the *Ansys ICEM CFD Help Manual*).

- **Curve Mesh Setup** (See Curve Mesh Setup in the *Ansys ICEM CFD Help Manual*).

- **Edge Params** (See Edge Params in the *Ansys ICEM CFD Help Manual*).

- **Vertex Location** (See Set Location in the *Ansys ICEM CFD Help Manual*).

> **Tip:**
>
> - If you create Workbench input parameters and want to step line by line or using a line range through a replay file, you must use the **Workbench Replay Control** item from the **One-Click** menu to start the **Workbench Replay Control** dialog.
>
> - For **Surface Mesh Setup**, **Curve Mesh Setup**, and **Edge Params**, you can set parameters for all existing surfaces, curves, or edges at once; in addition to setting a parameter for a single surface, curve, or edge.
>
> - Parameterized edges and vertices (location) can be easily located in the graphics display. Use **Edges** → **Show Parameterized Edges**, under **Blocking** in the **Display Tree** to color all parameterized edges red. Similarly, **Vertices** → **Show Parameterized Vertices** will color the parameterized vertex number red.

Setting General Input Parameters

To set input parameters in Workbench:

1. In ICEM CFD, select an input parameter.

2. Select the check box next to the parameter.

3. To confirm the selection, click **Yes**.

   A **P** in the check box indicates that the parameter has been created for Workbench.

   > **Note:**
   >
   > You will not be able to edit the parameter within ICEM CFD unless you click the check box again and deselect the parameter.

4. In **Workbench**, double-click the project's **Parameters** cell.

5. Edit the parameter values in the **Outline of Schematic: Parameters** pane.
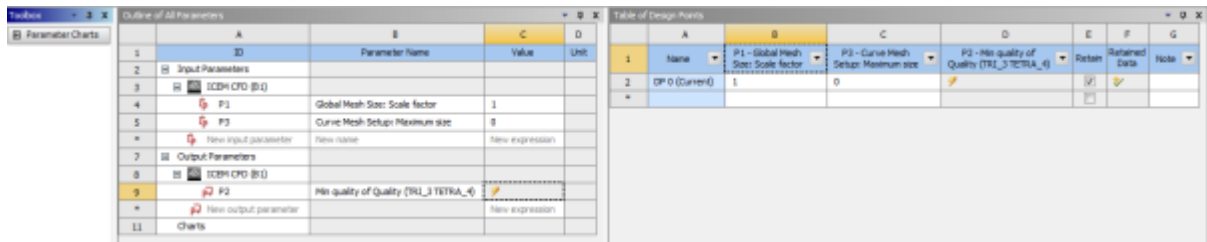
6. Switch to the **Project** tab.

You can now update the project using the new parameter settings.

Setting the parameters for a single curve, surface, or edge

To set single curve, surface, or edge parameters in Workbench:

1. In ICEM CFD, open the **Surface Mesh Setup**, **Curve Mesh Setup**, or **Edge Params** parameters from the **Tab** menu.

2. At the top of the **Parameters** window, click **Select**.

3. To select the curve, surface, or edge for which you want to set parameters, click the **Left Mouse** button

4. Click the **Middle Mouse** button to complete the selection.

   The surface, curve, or edge you selected are listed in the selection entry.
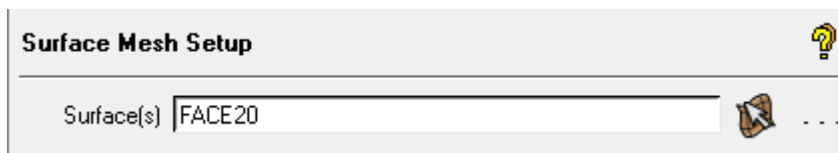
5. Select the check box next to the parameter you want as the input parameter.

6. To confirm the selection, click **Yes**.

   A **P** in the check box indicates that the parameter has been created for Workbench.

Setting the Parameters for All Existing Curves, Surfaces, or Edges

To set single curve, surface, or edge parameters in Workbench:

1. In ICEM CFD, open the **Surface Mesh Setup**, **Curve Mesh Setup**, or **Edge Params** parameters from the **Tab** menu.

2. Leave the surface, curve or edge selection field empty (do not select any surface, curve or edge).

3. Select the check box next to the parameter you want as the input parameter.

4. To confirm the selection, click **Yes**.

   A **P** in the check box indicates that the parameter has been created for Workbench.

Setting Workbench Mesh Parameters for Parts

The **Part Mesh Setup** parameters allow you to specify the mesh parameters for different parts of a geometry.

For more information, see Part Mesh Setup in the *Ansys ICEM CFD Help Manual*.

To create a Workbench mesh parameter for a part:

1. Switch to the **Mesh** tab.

156

2.   Click the **Part Mesh Setup** icon.

3.   In the **Part Mesh Setup** dialog, select a part and assign non-zero values to one or more of its parameters.

4.   Click the **Parameter** column for the part.

5.   A dialog asks you to confirm that you want to create a Workbench parameter for each value you have changed.

> ### Note:
>
> If any Workbench parameters are already set for the part, a dialog will ask if you want to delete that Workbench parameter.

Parameters assigned as Workbench parameters are highlighted in blue.
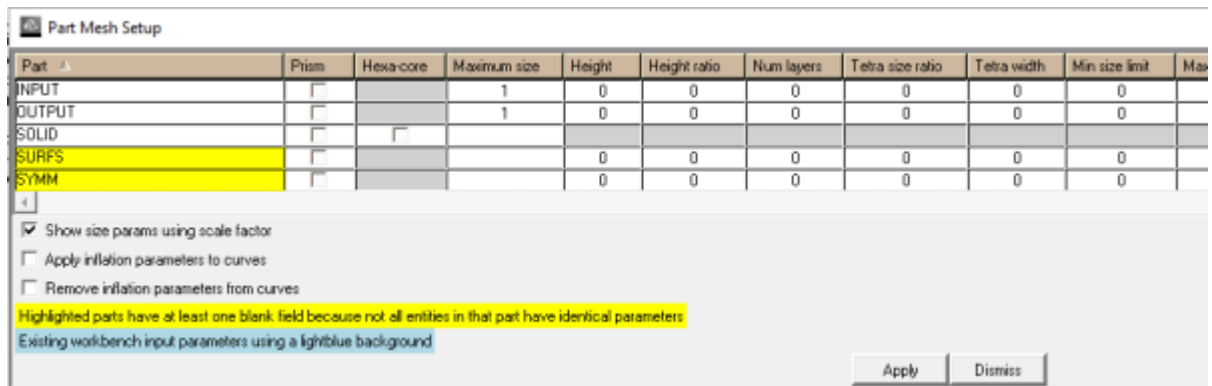
To delete a parameter, click in the **Parameter** column. A dialog asks you to confirm that you want to delete each individual Workbench parameter in the row.

As with ICEM CFD, the **Apply inflation parameters to curves** and **Remove inflation parameters from curves** options affect Workbench parameter behavior as well. The current value of **Apply inflation parameters to curves** is saved to the `project file/.aienv_options` file, so it is always available in **GUI** or **Batch** mode.

Setting Parameters for Prism Meshing

You can add prism meshing parameters using the **Part Mesh Setup** dialog box.

To set parameters for prism meshing:

1.   Switch to the **Mesh** tab.

2.   Click the **Part Mesh Setup** icon.

3.   In the **Part Mesh Setup** dialog box, select a part and select the check box in the **Prism** column.

4.   Click the **Parameter** column for the part.

5. A dialog asks you to confirm that you want to create a Workbench parameter for **Part PARTNAME: Prism**.

> **Note:**
>
> If any Workbench parameters are already set for the part, a dialog asks if you want to delete that Workbench parameter.

Setting Output Parameters

You can set Workbench output parameters to:

- List unstruct mesh and pre-mesh blocking quality metrics

- List the number of mesh errors and possible problems

- List the number of element and block types created.

You set the output parameters within the ICEM CFD application, then view them in either Workbench or ICEM CFD.
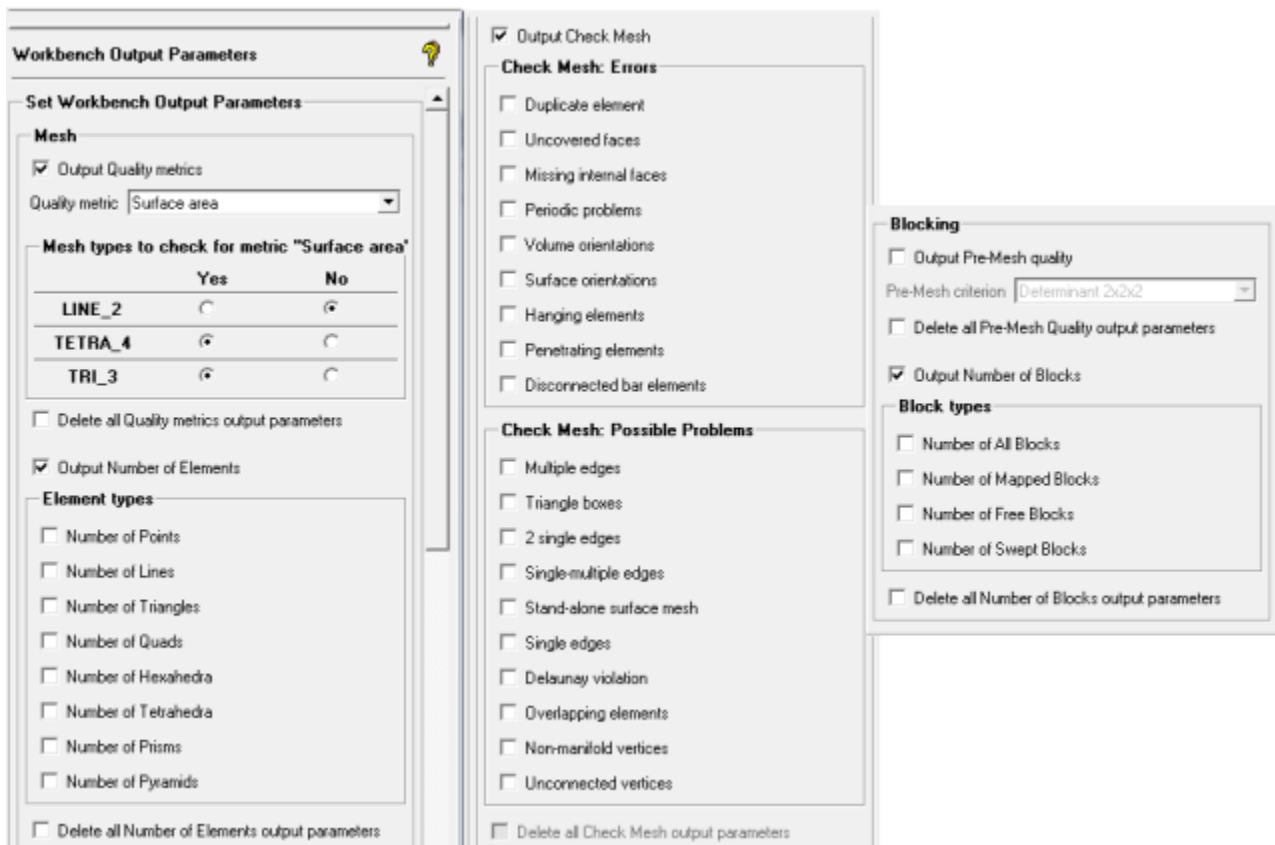
Setting Output Parameters

To set the output parameters:

1. In ICEM CFD, select **Settings → Workbench Parameters → Workbench Output Parameters**.

2. Optionally, select the **Output quality metrics** check box to select quality metrics.

    a. Use the **Quality Metrics** drop-down menu to select the metric you want to set.

    b. Click the radio buttons to select the mesh types to check for the metric.

3. Optionally, select the **Output number of elements** check box and select the check boxes next to the element types where you want an output.

4. Optionally, select the **Output Check Mesh** check box and select the check boxes next to the errors you want to have tallied in the output results.

    ICEM CFD performs an analysis as part of the meshing process and list the results in the **Output** window and in the **Workbench Parameters** window.

5. To set the blocking output parameters, select the **Output Pre-Mesh Quality** and **Output Number of Blocks** check boxes.

    For more information about pre-mesh quality, see Determining the Pre-Mesh Quality in the Ansys ICEM CFD User's Manual.

    For more information about Hexa Block types, see Hexa Block Types in the Ansys ICEM CFD User's Manual.

Deleting Output Parameters

To set the output parameters:

1. In ICEM CFD, select **Settings** → **Workbench Parameters** → **Workbench Output Parameters**.

2. In the **Workbench Output Parameters** window, select one or more of the following options, as required:

    • **Delete all quality metrics output parameters**

    • **Delete all number of elements output parameters**

    • **Delete all Pre-Mesh Quality output parameters**

    • **Delete all Number of Blocks output parameters**

3. Click **OK**.

Setting User-Defined Parameters

Setting user-defined parameters provides flexibility and control over the meshing operation. For example, parameters that cannot be applied as single input parameters can be individually set with user-defined parameters.

Note:

See the *Ansys ICEM CFD Programmer's Guide* for information about using user-defined parameters with replay scripting.

Setting User-Defined Input Parameters

To set user-defined input parameters:

1. In ICEM CFD, select **Settings** → **Workbench Parameters** → **Workbench User Defined Parameters**.

2. In the **Workbench User Defined Input/Output Parameters** window, select the **Create Input Parameter** check box.

3. Enter a value for **Parameter name**.

4. Enter a value for **Parameter**. This field must not be empty.

5. Click **Apply** or **OK**.

You can edit this value in the **Outline of Schematic: Parameters** pane.

Setting User Defined Output Parameters

To set user-defined output parameters:

1. In ICEM CFD, select **Settings** → **Workbench Parameters** → **Workbench User Defined Parameters**.

---

2. In the **Workbench User Defined Input/Output Parameters** window, select the **Create Output Parameter** check box.

3. Enter a value for **Parameter name**.

4. Enter a value for **IC Function**.

    The IC Function must start with `ic_geo_` or `ic_uns_` or `ic_hex_` and must return an integer or a float value.

Note:

See the *Ansys ICEM CFD Programmer's Guide* for information about IC functions.

---

5. Click **Apply** or **OK**.

Deleting Parameters

Use this procedure to delete any input or output parameter, whether user-defined or otherwise.
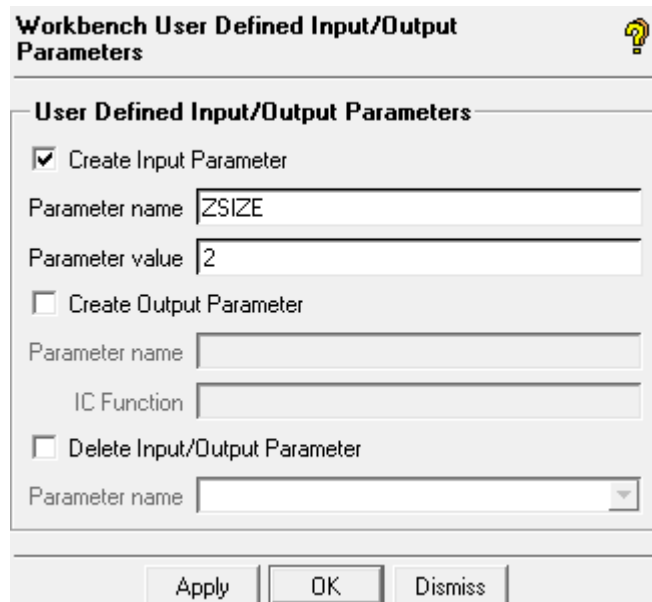
To delete parameters:

1. In ICEM CFD, select **Settings** → **Workbench Parameters** → **Workbench User Defined Parameters**.

2. In the **Workbench User Defined Input/Output Parameters** window, select the **Delete Input/Output Parameter** check box.

3. Using the drop-down menu, select the name of the parameter you want to delete.

4. Click **Apply** or **OK**.

User-Defined Parameters Example

This example illustrates how you can use user-defined parameters to test different meshing scenarios for a simple box:

1. To add a data-integrated ICEM CFD component system to the Workbench **Project Schematic**, in the **Toolbox**, double-click **ICEM CFD**.

2. To open ICEM CFD, double-click the **Model** cell.

3. First, create an input parameter, **ZSIZE**, that you can manipulate from Workbench.

    a. Select **Settings → Workbench Parameters → Workbench User Defined Parameters**.

    b. Click **Create Input Parameter**.

    c. Name the parameter `ZSIZE` and set **Parameter Value** to **2**.

    d. Click **OK**.

4. Now create a box model to perform a meshing operation on. Record the process so the operation can be performed again when you update it from Workbench.

    a. Click **Start Replay**.

    b. Switch to the **Geometry** tab.

    c. Select **Create/Modify Surface**.

    d. In the **Create/Modify Surface** window, select **Standard Shape**.

    e. Select **Box** and click **Apply**.

    f. Switch to the **Mesh** tab.

g.   Select **Compute Mesh**.

h.   Select **Volume Mesh** and click **Compute**.

i.   To use autosizing for the mesh size, click **Yes**.

j.   Click **Stop Replay Recording**.

k.   To save the project, select **File → Save Project**.

5.   Next, edit the script to use the input parameter.

a.   From the **One-Click Replay Recording** menu, select **Workbench Replay Control**.

b.   Click **Edit**.

c.   Change **Line 16** (or whichever line is appropriate) from `ic_vid_objectbox8 GEOM 00 {0 0 0} 1 1 1` to `ic_vid_object box8 GEOM 00 {0 0 0} 1 1 [ic_wb2_get_parameter user_defined ZSIZE]`

d.   Select **File → Save**.

e.   Close the **Edit** window.

f.   Click **Save** save the file as `ICM.rpl`.

g.   Click **Done**.

Do not close ICEM CFD.
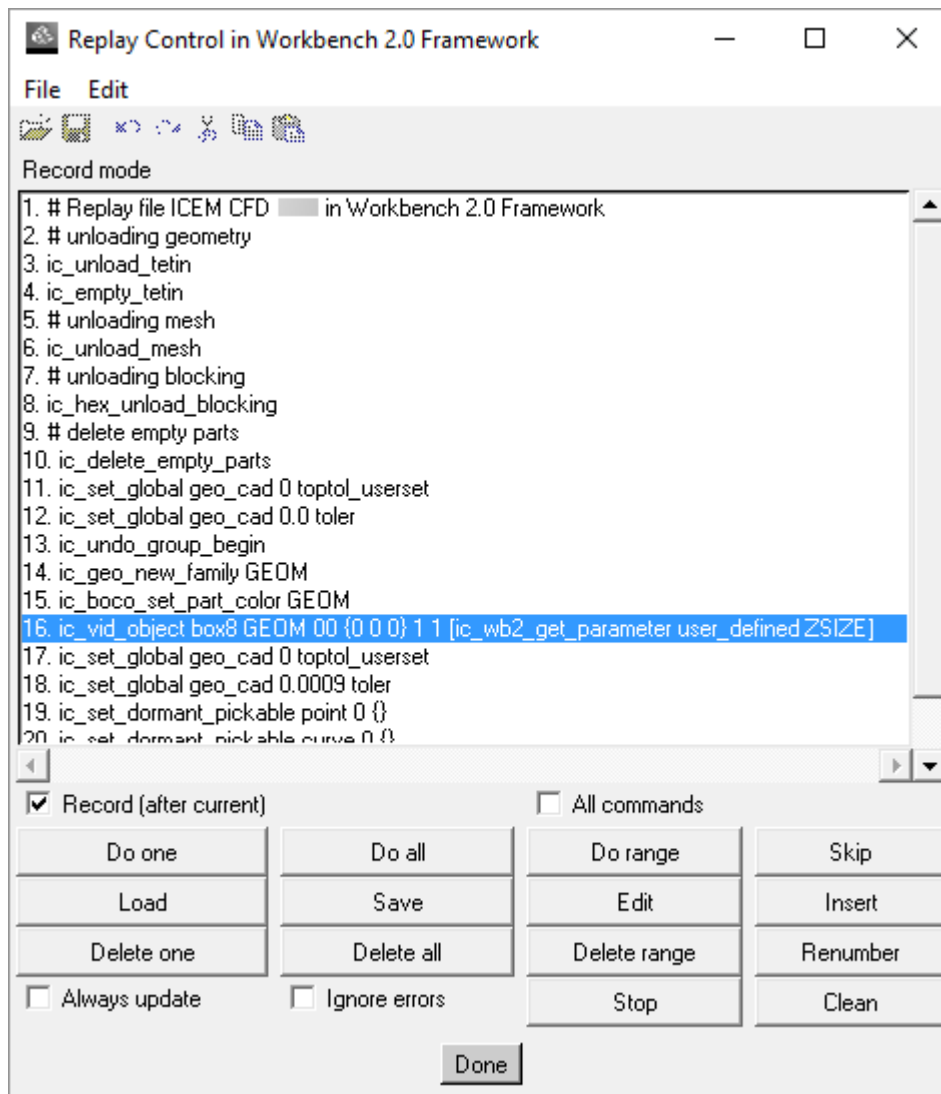
6. Now you can change the input parameter through Workbench.

   a. In the Workbench **Project Schematic**, double-click the **Parameters** cell of the ICEM CFD component.

   b. Change the value of **ZSIZE** to **5**.

   c. Return to the **Project** tab.

   d. Right-click the **Model** cell and select **Update**.

You can watch the Replay script run using the new parameter in the ICEM CFD interface. Each time you change the **ZSIZE** parameter in Workbench, a new part will be generated and meshed with the new **ZSIZE** parameter value.

Transferring an ICEM CFD Project to Static Structural

When you create a project that transfers ICEM CFD data downstream to a Static Structural system, data that is applied to the geometry, such as load data, is not transferred to the structural system. To work around this limitation, you can use an External Model system as a bridge between the two systems and then manually set the connections between them.

To transfer an ICEM CFD project to Static Structural:

1. To start with an **ICEM CFD** component, do one of the following:

    - To select an existing **ICEM CFD** project, select **File → Import**.

    - Drag an **ICEM CFD** component system onto the **Project Schematic** and then click the model to open the ICEM CFD Add-In and create your model.

2. Drag an **External Model** system from the **Toolbox** and drop it on the **ICEM CFD** system.

    Note that they are connected through the **Model** cell.

3. Drag a **Static Structural** system from the **Toolbox** and drop it on the **Project Schematic** as a standalone system.

    > **Tip:**
    >
    > If you drag the **Static Structural** system and drop it on the **External Model** system, the two cells link **Model** to **Engineering Data**, and model information is not passed downstream.

4. Drag the **Model** cell from the **External Model** system and drop it on the **Model** cell of the **Static Structural** system.

5. Right-click the **Model** cell of the **ICEM CFD** system and select **Update** from the context menu.

6. Right-click the **Model** cell of the **External Model** system and select **Update** from the context menu.

The model is now ready to use within Static Structural.

Icepak

The **Icepak** system launches Ansys Icepak. This application allows engineers to model electronic designs and perform heat transfer and fluid flow simulations. You can construct your model geometry or import model data from other CAD and CAE packages. Icepak then creates a mesh for your model geometry and passes the mesh and model definition to the solver for computational fluid dynamics simulation. The resulting data can then be postprocessed using Icepak and/or CFD-Post.

Workbench has the capability of coupling Ansoft steady-state to Icepak steady-state cases and Ansoft transient to Icepak steady-state cases.

To work through an **Icepak** system:

1. To add an **Icepak** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Right-click the **Setup** cell and select **Edit** or double-click the **Setup** cell. This launches Icepak.

3. Create a geometry within the Icepak application.

    After creating a geometry, the system displays the cell state as Unfulfilled (p. 449).
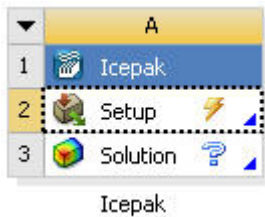
    The Update Required icon ( ) indicates the system has not been solved.

4. In Icepak, set up the problem and complete your analysis using Icepak's tools and features.

    After solving, the Icepak system displays an up to date cell state as shown below.
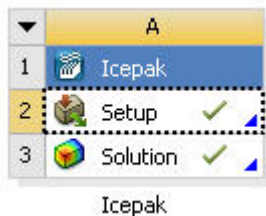
    The Up to Date icon ( ) indicates that all necessary files are loaded and up to date.

5. Load or import an existing Icepak model by selecting one of the following options:

    • To load an existing Icepak model, right-click the **Setup** cell and select **Import Icepak Project**.

    • To import an existing Icepak model:

        1. Right-click the **Setup** cell and select **Import Icepak Project From .tzr**.

        2. Click **Browse**

        3. In the dialog box, browse the directory structure and select the model to load.

            The system displays an incomplete cell state.
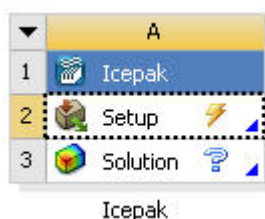
The Update Required icon (⚡) indicates the system has not been solved.

Icepak project files are located in the system directory under *IPK*. All Icepak projects saved in Workbench follow this naming convention which is described in Project Directories (p. 278). All other Icepak projects can be saved anywhere. Note, there are no special icons for Icepak projects to differentiate them from other files.

> **Note:**
>
> You can only import one Icepak project per system.

6. In Icepak, set up the problem and complete your analysis using Icepak's tools and features.

   After solving, the Icepak system displays an up to date cell state as shown below.
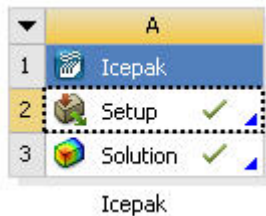
   The Up to Date icon (✓) indicates that all necessary files are loaded and up to date.

   > **Note:**
   >
   > If you load an Icepak project that already has a solution, the **Solution** cell state will show an up to date cell state after the model is loaded.

7. You can also import geometry into Icepak from DesignModeler. See the DesignModeler to Icepak (p. 170) section for details on importing geometry from DesignModeler into Icepak or loading a geometry.

Workbench Menu Options Overview for Icepak Projects

Many Icepak options remain the same when running Icepak from the Workbench framework. However, there are a few changes that should be noted.

Menu Item	Description
Save project	Saves all changes to your project. The **Save** button in Workbench works the same way as **Save project** in Icepak, you can use either one. All Icepak projects saved in Workbench will follow the directory structure described in Project Directories (p. 278). See Saving a Project File (p. 269) for a description of **Save** and **Save as** options in Workbench.
Archive	Generate a single archive file that contains all project files. This archive includes the project file and all files in the *project_name_files* directory, not just Icepak files. The archive is saved

Menu Item	Description
	as a zip file. Icepak users familiar with the Pack option in the standalone Icepak application will find this option works in a similar fashion except that all Workbench files are archived. See Archiving a Project (p. 270) for a complete description of the **Archive** option in Workbench. > **Note:** > > Previously imported external files from a restored archive directory are treated as internal files if archived again.

Icepak Context Menu Options

When the Icepak system is active in the **Project Schematic**, use the right mouse button to initiate the following **Setup** cell actions.

Menu Item	Description
Edit	Launches Icepak. This is the default operation.
Import Icepak Project From .tzr	Opens a **Browse** dialog box to add an input file, then launches Icepak and reads the input files. This is the default action. This option is available only when a project is empty.
Import Icepak Project	Opens a **Browse For Folder** dialog box to add an input file, then launches Icepak and reads the input files. This is the default action. This option is available only when a project is empty.
Duplicate	Creates a duplicate of the Icepak system and any upstream data. If the duplicate operation is performed on a system containing a solution, the solution is transferred and you must re-solve. The system cell state indicates the need to re-solve.
Transfer Data From New	Creates a second, dependent (connected) system with the existing system.
Update	Updates the system if data is changed or modified. You have the option to incorporate the changes in the **Refresh Input Data** pane and initiating the solver inside Icepak. See DesignModeler to Icepak (p. 170) for a further description of the update option and how the **Refresh Input Data** panel is used. > **Note:** > > You must open the Icepak editor before doing an update.

Menu Item	Description
Refresh	Refreshes the system if data is changed or modified. You have the option to incorporate the changes in the **Refresh Input Data** panel. See DesignModeler to Icepak (p. 170) for a further description of the refresh option and how the **Refresh Input Data** panel is used. **Note:** You must open the Icepak editor before doing a refresh.
Rename	Renames the system or cell.
Properties	Displays applicable cell properties in the **Properties** pane.
Enable Update	Allows you to run the solution if data is unchanged. This changes the **Setup** cell icon from a green check mark to the Update lightning icon.
Quick Help	Displays a quick help panel for the cell. Quick help provides a brief description of how to use the cell in its current state. You can also click the blue triangle in the lower right corner of a cell to view quick help.

The **Solution** cell actions are similar to those of the **Setup** cell; however, there are additional actions and functions. These are described in the list below. Use the right mouse button to initiate the following **Solution** cell actions.

Menu Item	Description
Set Case File...	Opens an **Open** dialog box where you can specify an Icepak solution file to load. This option is used primarily to view multiple solutions for a project. If the you re-solve in Icepak, the solution case file is overwritten with the latest solution.
Transfer Data to New	Creates a downstream system that can accept data from the selected cell. Only those systems that can provide valid data to the selected cell are shown. When you select a system from the options shown here, that system is displayed to the right of the currently-selected system, with all appropriate connections drawn.
Update	Performs an update if solution data is changed or modified. You can bring in the latest solution case file into the solution cell for consumption.

System Names

You can enter an Icepak system name for your project as described in Naming Systems (p. 233). In addition, you can use the **Rename** option to change the name of a system or cell. In the Icepak application, the system coordinate, system name, and application name are displayed in the top left corner.
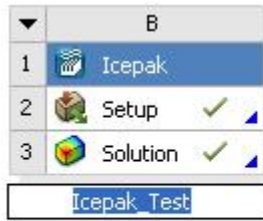
Icepak Properties

Select or enable **Use Workbench Color Scheme** to keep the Workbench background graphics colors and display defaults in Icepak.

To enable this option:

1. Right-click the **Setup** cell and select **Properties** from the context menu.

2. In the **Value** column, select the **Use Workbench Color Scheme** check box.

DesignModeler to Icepak

CAD models created or edited in DesignModeler can be imported into Icepak. Icepak instructs DesignModeler to export the current DesignModeler geometry into Step file format so it is readable by Icepak.

To transfer geometry upstream to Icepak:

*   To add a **Geometry** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

The Attention Required icon ( ) indicates that an input file is required.

1. To launch the DesignModeler application, double-click the **Geometry** cell or right-click the **Geometry** cell and select **New Geometry** from the context menu.

    You can import any geometry or load an existing DesignModeler geometry by right-clicking the **Geometry** cell and selecting **Import Geometry** from the context menu.
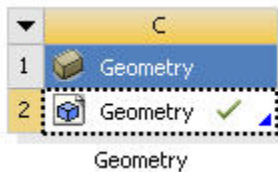
The Up to Date icon ( ) indicates that all necessary files are loaded and up to date.

2. Establish the connection to Icepak using one of the following options:

   • Drag and drop an **Icepak** system on top of the **Geometry** cell.

   • In the **Toolbox**, double-click **Icepak** to add the **Icepak** system to the **Project Schematic**, then click the **Geometry** cell and drag it to the Icepak **Setup** cell.

   • Right-click the **Geometry** cell and select **Transfer Data From New** from the context menu, then select a system that uses DesignModeler.

   • Right-click the **Geometry** cell and select **Transfer Data to New** from the context menu, then select a connection to Icepak.
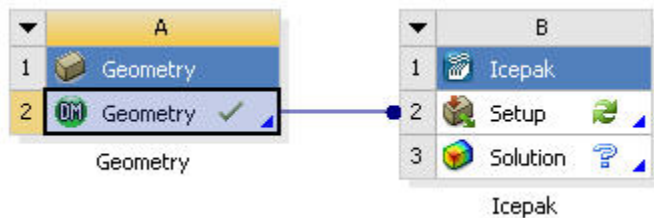
3. To launch the Icepak application, double-click the **Setup** cell.

   If the geometry is different from DesignModeler's native format (`.agdb`), you must edit the geometry first before exporting into Icepak.

   A new project is created and DesignModeler geometry is imported into Icepak as STEP geometry.
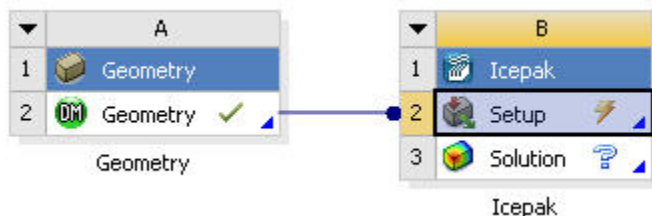
4. If the geometry has been modified, refresh the Icepak **Setup** cell to bring in the new geometry.
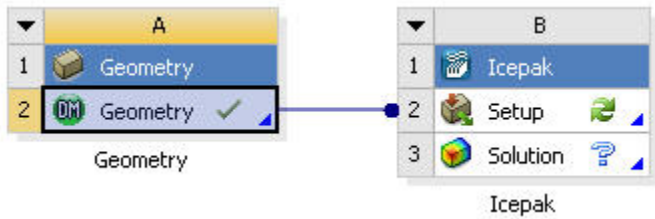
After you refresh the data, you must either replace the entire model in Icepak or update individual geometries that have changed. You are prompted with the following dialog box.
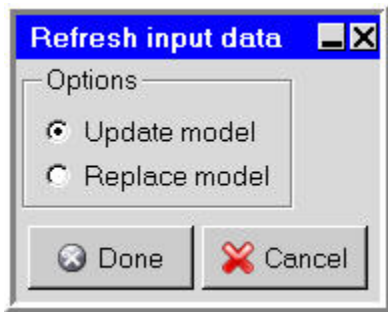
If you select **Update model**, the Icepak model is updated to match the existing model in DesignModeler which may result in creating objects that are not currently present in the Icepak model, the geometries of modified objects will be updated, and the thermal and material properties of the updated objects are retained. Objects created separately in Icepak will not be updated.

If you select **Replace model**, the entire Icepak model will be replaced with the latest input CAD data from DesignModeler.

See the sample session in Chapter 2 of the Icepak documentation for an example on how to use Icepak. See Understanding Cell States (p. 449) for a complete list of all cell states.

5. Perform an analysis on the model if you have not yet done so.
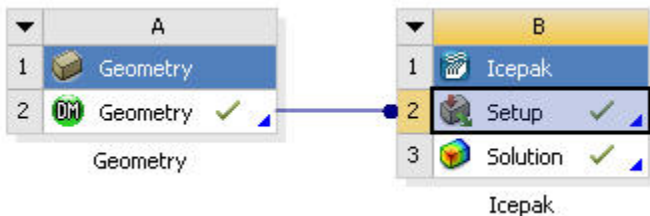
6. If the geometry has changed after you have solved, update the Solution cell (p. 244) to bring in the new changes.
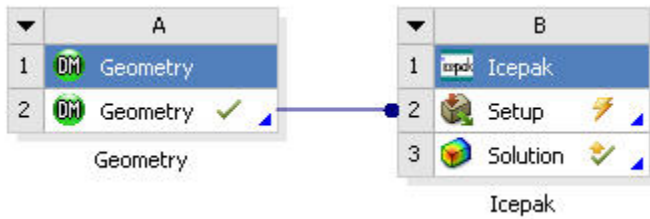
You can choose to not overwrite the solution using the dialog box shown below, and the update is cancelled ( ).
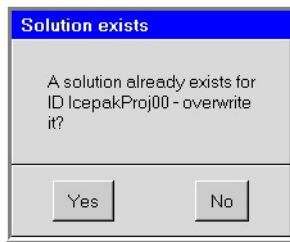
Note:

See Workbench Menu Options Overview for Icepak Projects (p. 167) for information on how to save your project.

Postprocessing Icepak Results

Icepak provides two methods for examining the results of your simulation; you can postprocess results inside of Icepak or by using CFD-Post.

To postprocess results in Icepak, add an Icepak system to the **Project Schematic** and perform an analysis on the model. You can then create graphical displays and examine your results in Icepak. For details, see Examining the Results in the *Icepak User's Guide*

To postprocess Icepak results in CFD-Post:

1. Connect the **Icepak** system to a CFD-Post **Results** system using one of the following options:

   • Drag and drop a **Results** system on top of the Icepak **Solution** cell.

   • In the **Toolbox**, double-click **Results** to add the **Results** system to the **Project Schematic**, then click the Icepak **Solution** cell and drag it to the **Results** cell.

   • Right-click the **Results** cell and select **Transfer Data From New** from the context menu, then select the **Icepak** system.

   • Right-click the **Geometry** cell and select **Transfer Data to New** from the context menu, then select the **Results** system.
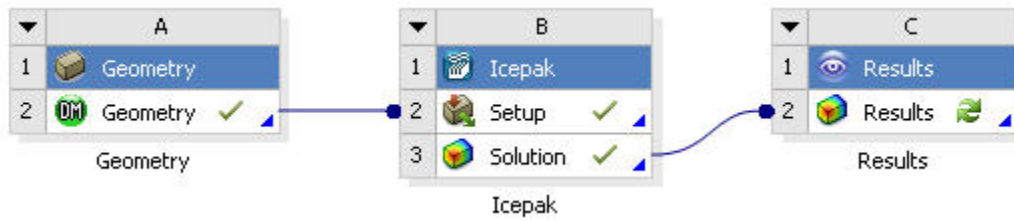
2. Icepak results can be postprocessed in CFD Post by double-clicking the **Results** cell. If you have more than one solution or your solution has changed, you need to update the **Results** cell. If solution data is not transferred, then Results cannot be launched. See CFD-Post help for more details.
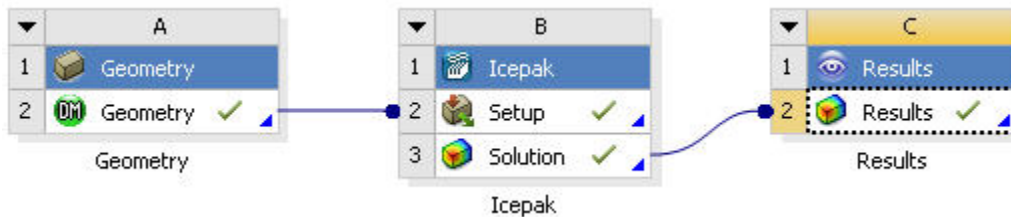
Thermal Results from Icepak to Mechanical

Icepak results can be transferred to the following analysis types within Workbench: Static Structural, Steady-State Thermal, Transient Structural, or Transient Thermal.

To add an analysis type:

1. Add an Icepak system to the **Project Schematic** and perform an analysis on the model inside Icepak.

2. Connect the **Icepak** system to a Mechanical system using one of the following options.

   • Drag and drop an **Analysis** cell on top of the Icepak **Solution** cell.

   • In the **Toolbox**, double-click a Mechanical analysis system to add it to the **Project Schematic**, then click the Icepak **Solution** cell and drag it to the analysis **Setup** cell.

   • Right-click the analysis **Setup** cell and select **Transfer Data From New** from the context menu, then select the **Icepak** system.

   • Right-click the Icepak **Solution** cell and select **Transfer Data to New** from the context menu, then select a Mechanical analysis system.

174
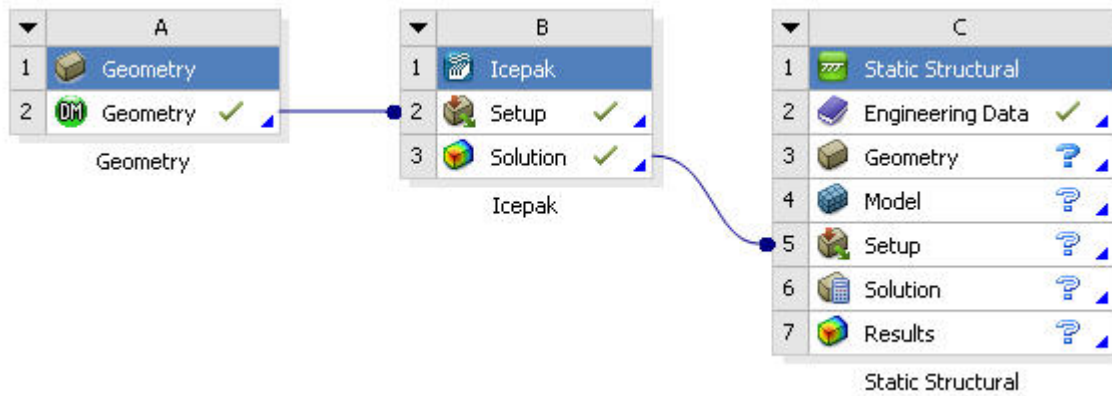
3.   Load geometry to satisfy the system.
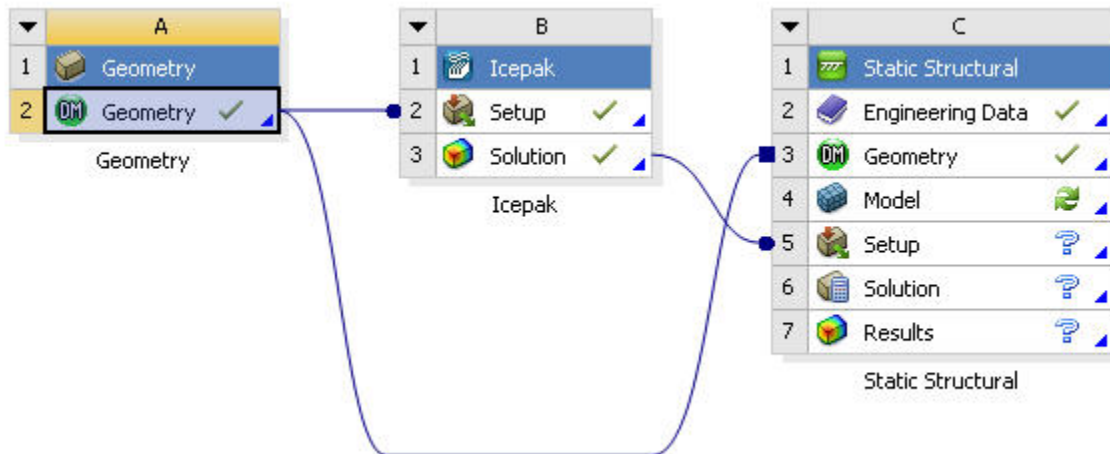
4.   Double-click the **Setup** cell to launch Mechanical.

5.   While in Mechanical insert the appropriate loads based on the type of analysis. See Icepak to Mechanical Data Transfer for a detailed description of how to import an external load.

Design Explorer to Icepak Coupling in Workbench

The exploration of a given design can be performed by using optimization algorithms in Ansys DesignXplorer. Parameters exposed from Icepak provide a method for solving an optimization and/or parameterization problem.

To publish Icepak variables:

1. In Icepak, define input and output parameters. See Overview of Parameterization for a description on how to define parameters.

> **Note:**
>
> The **Design variables** tab of the **Parameters and optimization** panel displays all the parameters names that are currently defined, along with their associated values.

2. In the **Setup** tab of the **Parameters and optimization** panel, select **Single trial (current values)** for **Trial type** if not already selected.

3. At the bottom of the **Parameters and optimization** panel, click **Publish to WB** to display the **Publish to WB** panel.
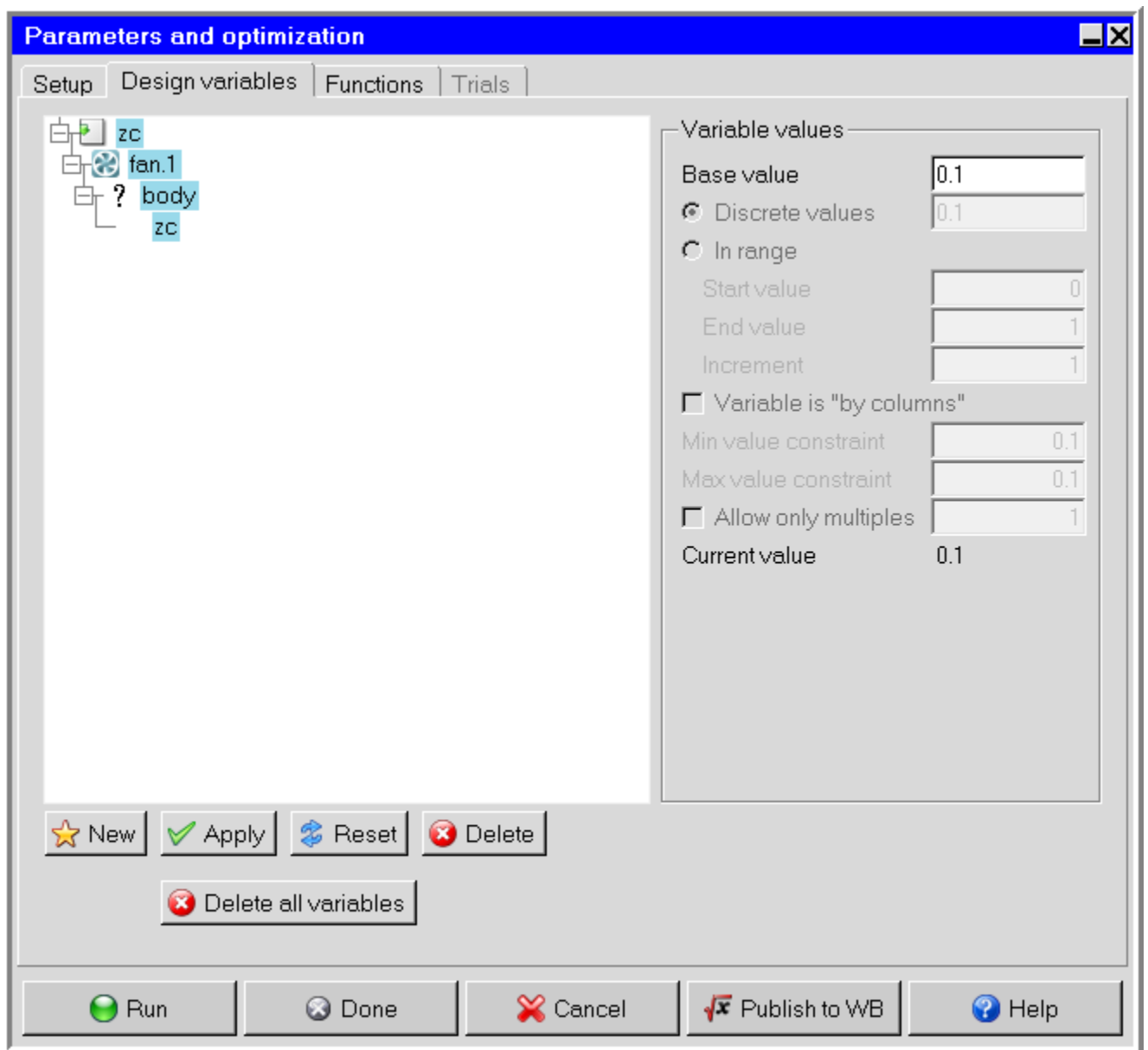
4. In the **Publish to WB** panel, select the input and output variables to publish to Workbench.

Note:

Variables can be selected independently. In the case of multiple variables, click the green check mark to toggle all variables on and off.

---
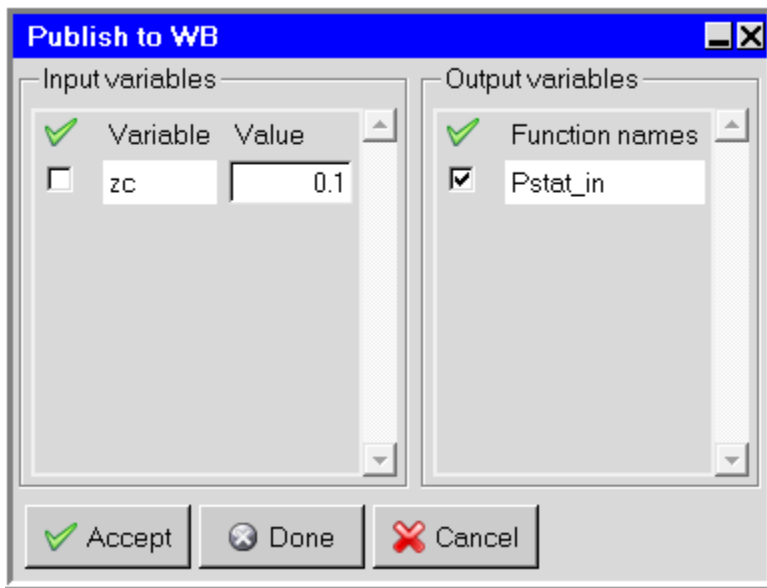
5. Click **Accept** to publish variables to Workbench, click **Done** to publish variables and close the panel, or click **Cancel** to withdraw the request.

Workbench recognizes the parameters defined and exposes them in a parameter set bar that can be shared by multiple systems.

6. To access the **Parameters** tab, double-click the parameter bar or right-click and select **Edit** from the context menu.

For information on defining parameters, refer to Working with Parameters and Design Points (p. 283).
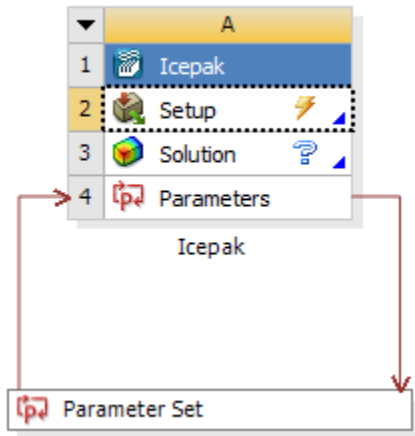
> **Important:**
>
> You cannot create, edit, delete, or rename parameters in Icepak if any iterations (or time-steps) have been performed. If you want to create, edit, delete, or rename parameters in Icepak for a case with an existing solution, you must first initialize the solution.

7. To create a new design point, either duplicate an existing design point (p. 305) or add a new one by entering values (p. 305).

8. Update the design points (p. 306).

> **Important:**
>
> Only the data from the design point in the row labeled **Current** is saved with the project. If you want to postprocess the results from a different design point in either Ansys Icepak or Ansys CFD-Post, click the box in the **Exported** column for that design point before you update that design point. Otherwise, the data for that design point is automatically deleted after the output parameters for that design point are updated. If you choose to export a design point, the data associated with that design point is exported to a new project. The new project is located in the same directory as the original project. The name of the project is the same as the name of the original project, except that it is appended with _dpn, where n is the row number that corresponds to the design point in the original project's **Table of Design Points**.

9. Optimization of an Icepak system can be performed in Ansys DesignXplorer. Ansys DesignXplorer provides various optimization methods with parameters as its fundamental components. These parameters can come from any supported analysis system, such as Icepak, DesignModeler, and various CAD systems. Responses can be studied, quantified and graphed. For information on how to set up an Ansys DesignXplorer analysis, see the DesignXplorer User's Guide.

Ansoft to Icepak Coupling in Workbench

Icepak can be coupled with Ansoft applications within Workbench in order to perform a one-way or two-way electromagnetic-thermal interaction problem. Coupling between Ansoft and Icepak applications within Workbench can be used for simulating fluid flow around or inside electromechanical (EM) devices when the temperature of the device is influenced by electromagnetic losses.

The coupling involves solving an electromagnetic problem in the Ansoft application, and mapping the resulting volumetric heat loss and/or surface loss information into Icepak. Volumetric loss is mapped onto the solid cell zones as a heat source (load) at the cell centroids that is then added to the energy equation. Surface loss is applied to the adjacent cells of the solid zones at the surface that contribute to the source terms of these cells.

> **Note:**
>
> Surface loss is highly concentrated near the surface of the solid zone, so you should have a fine layer of good quality hexahedral or prism mesh elements located where surface loss occurs.
>
> You can analyze the results of volumetric or surface losses using the heat flow postprocessing variable under **Summary report**.
>
> When surface losses are enabled, the double precision solver is recommended.

One-way Coupling between Ansoft and Icepak Within Workbench

To work through an Icepak- HFSS/Maxwell/Q3D Extractor analysis:

1. Connect Ansoft to Icepak using one of the following options:

   • Add a HFSS/Maxwell/Q3D Extractor system to the **Project Schematic**.

     1. To add a HFSS/Maxwell/Q3D Extractor system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

     2. Import/create the geometry in the Ansoft application.

     3. Set up the problem and solve to obtain a solution for transfer into an Icepak system.

     4. Connect the HFSS/Maxwell/Q3D Extractor **Solution** cell to the Icepak **Setup** cell.
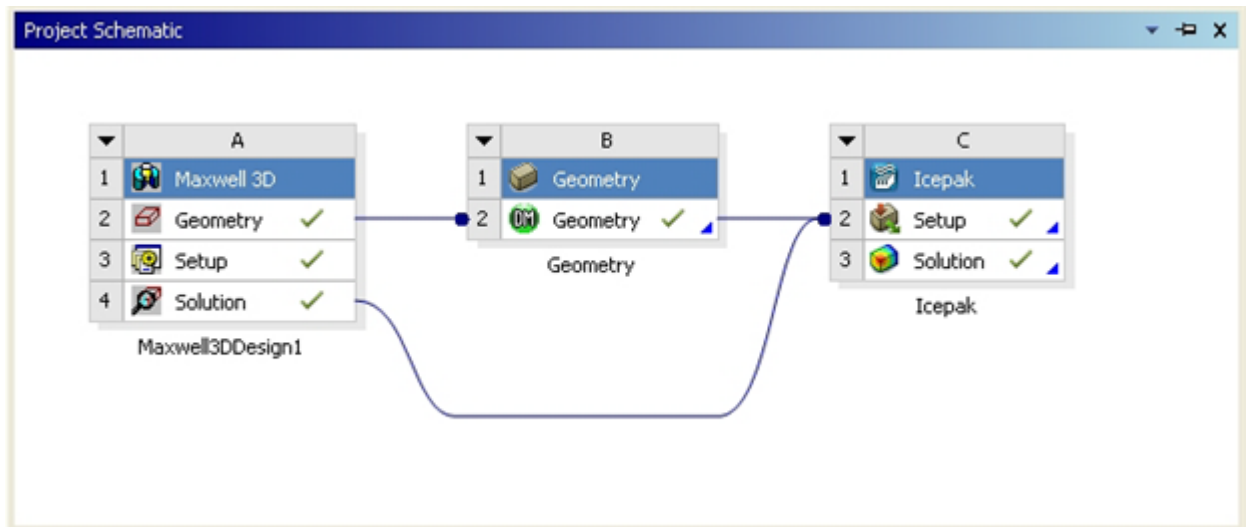
- Transfer data from the **Icepak** system:

    1. On the **Icepak** system, right-click the **Solution** cell and select **Transfer Data From New** from the context menu.

    2. Select a connection to a HFSS/Maxwell/Q3D Extractor system.

- Transfer data to an **Icepak** system:

    1. On the **Geometry** system, right-click the **Geometry** cell and select **Transfer Data to New** from the context menu.

    2. Select the **Icepak** system.
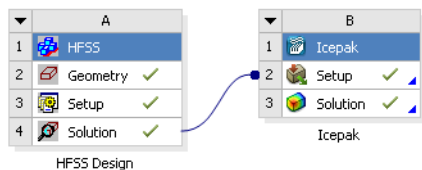
2. To launch the Icepak application, double-click the Icepak cell.

    A new project is created in the name of the project cell. If a DesignModeler system is used the geometry is imported into Icepak automatically. Otherwise, the geometry must be created in Icepak using Icepak primitive objects.

    > **Note:**
    >
    > The geometry can be exported from the HFSS/Maxwell/Q3D Extractor cell to a **Geometry** cell and imported to the Icepak cell using DesignModeler Electronics.

3. In Icepak, go to the **File** menu and select **EM Mapping** and **Volumetric heat losses** or **Surface heat losses**.

    The **Volumetric heat losses** panel contains the following inputs:

    - A list of solid objects onto which the loss information can be mapped. For these objects, Icepak requests the heat source (loss) terms from the Ansoft application. Click the **Objects** button to toggle your current object selections. Right-click **Objects** and select **All** or **None** to select all or none of the objects.

    - **Solution ID** contains available solution sets. Since the HFSS/Maxwell/Q3D Extractor application may have multiple solutions, Icepak requests the generated heat source data for the selected solution.

    - **Frequency** (steady-state only) contains available frequencies. Icepak will request that the HFSS/Maxwell/Q3D Extractor application provide the heat source data for the selected frequency.

    - **Start time(s)** and **End time(s)** (transient only) contains the available time steps. Icepak will request that the HFSS/Maxwell/Q3D Extractor application provide an averaged heat loss for the selected time steps.

    > **Note:**
    >
    > The **Start time(s)** and **End time(s)** specified are applied to both volumetric and surface heat losses.

    - **Temperature feedback** option is used in two-way coupling problems. This option is explained below.

Figure 1: Volumetric heat losses - Steady-state
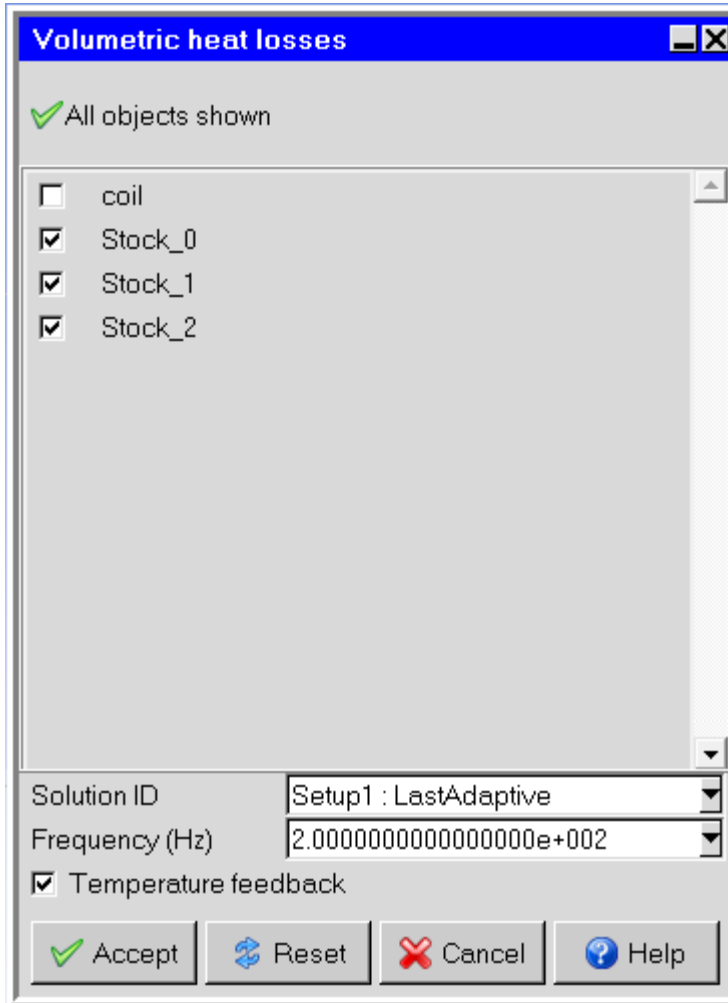
Figure 2: Volumetric heat losses - Transient
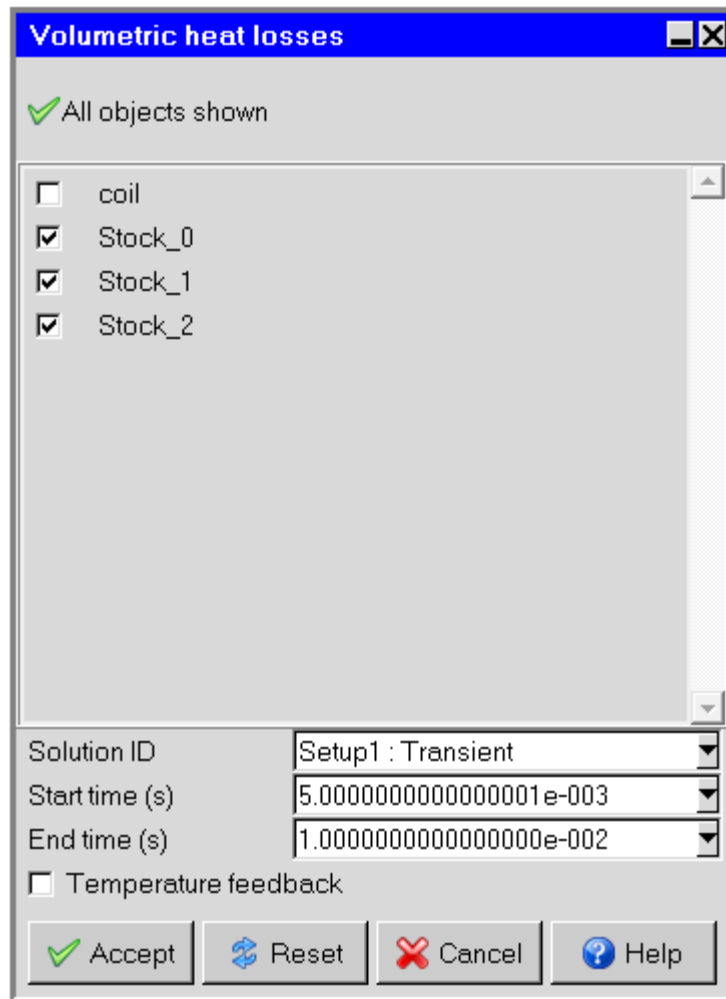
4.   Click **Accept** to close the panel.

5.   In Icepak, click **Start solution** to solve the project.

      During solving, HFSS/Maxwell/Q3D Extractor is launched in the background and the volumetric
      losses calculated and mapped onto the selected Icepak solid objects. This loss mapping from
      HFSS to Icepak is conservative. This is especially important for accurate temperature calculation.

Two-Way Coupling Between Ansoft and Icepak Within Workbench

Two-way coupling enables thermal feedback to be provided between the systems so that you can
exchange temperature data from Icepak to Ansoft applications within Workbench.

To enable two-way coupling between Icepak and Ansoft:

1.   Open the Maxwell (or HFSS/Q3D Extractor) project and define temperature–dependent mater-
      ial properties (enabling the Thermal Modifier field in the **View/Edit Material** dialog box and
      editing the material's thermal property definition).

This makes sure that the Ansoft and Icepak applications generate the temperature–dependent data.
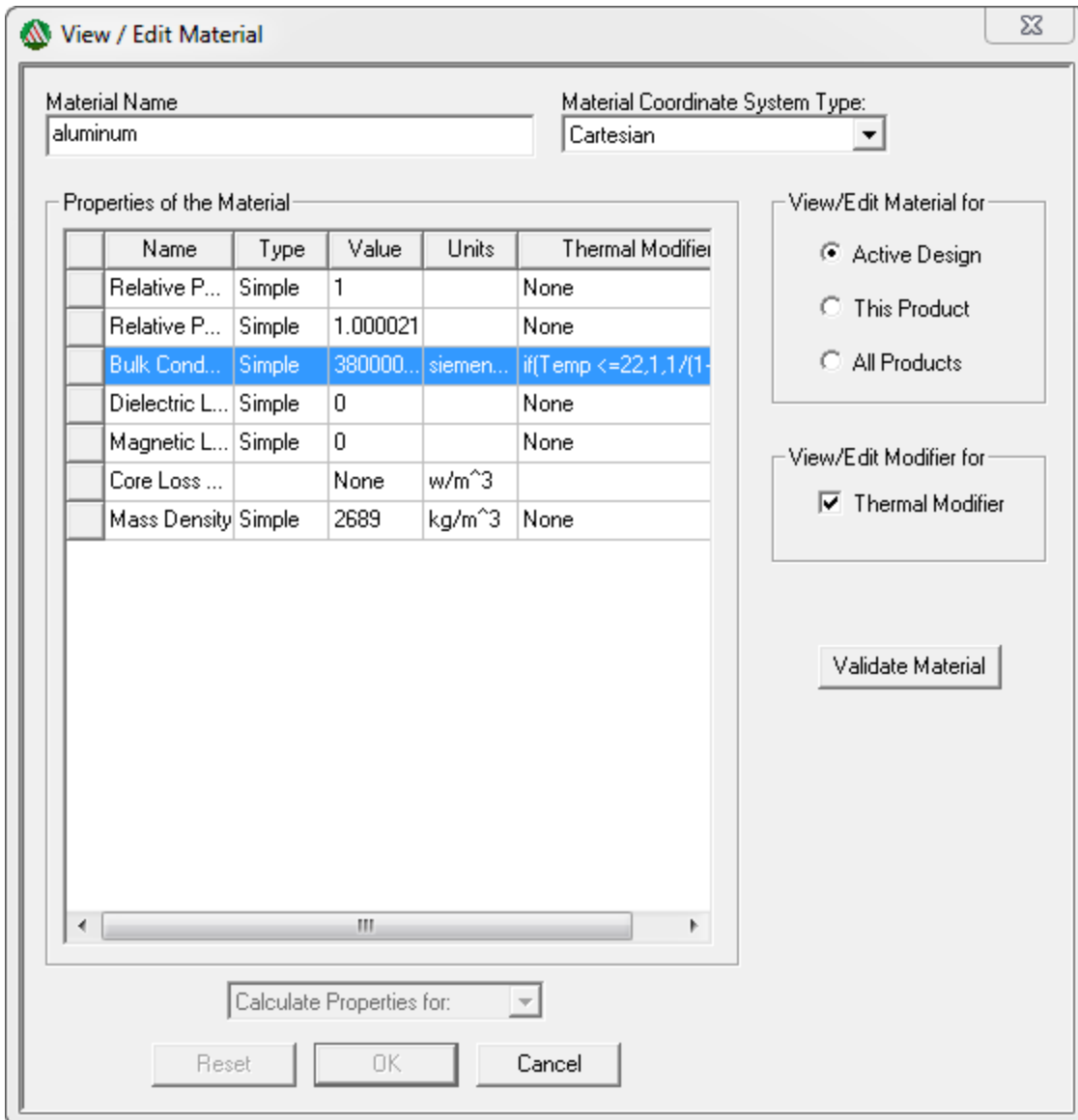
2. In the **Temperature of Objects** dialog box, select **Enable Feedback**.

   The **Temperature of Objects** dialog box is displayed when selecting **Set Object Temperature** in the **Maxwell 3D** /**HFSS**/**Q3D Extractor** menu.
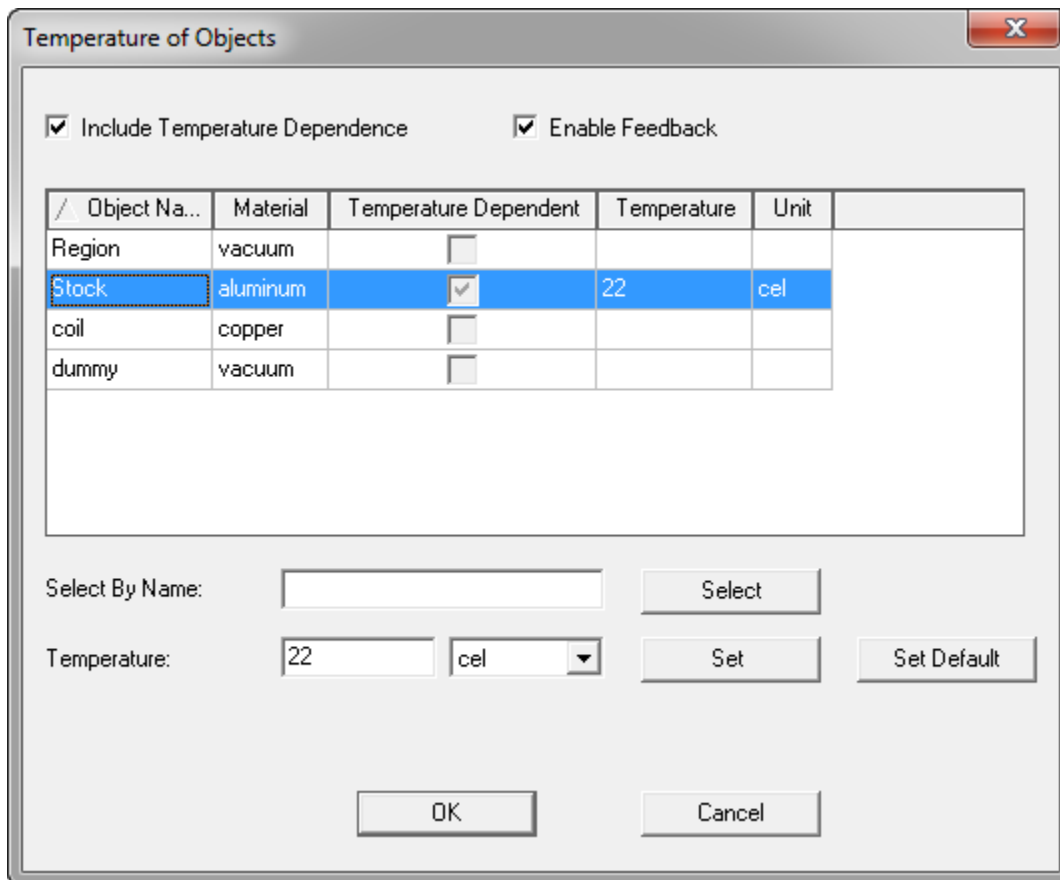
Note:

In Maxwell/HFSS/Q3D Extractor, you can have different losses (Ohmic loss, Core loss, Hysteresis loss) depending on the type of simulation. Maxwell/HFSS/Q3D Extractor can also compute the "total loss" which is the sum of all the losses when appropriate. All the different losses mentioned above including the "total loss" are a function of space and they are also a function of time in the Maxwell/HFSS/Q3D Extractor transient solver. When doing the coupling simulation, the "total loss" is mapped to Icepak for the temperature calculation. For transient simulations, the total loss is time averaged between two times that you specify before it is mapped to Icepak.

3. Connect Ansoft to Icepak using one of the following options:

    • Transfer data from the **Icepak** system:

       1. On the **Icepak** system, right-click the **Solution** cell and select **Transfer Data From New** from the context menu.

       2. Select a connection to a HFSS/Maxwell/Q3D Extractor system.

    • Transfer data to an **Icepak** system:

       1. On the **Geometry** system, right-click the **Geometry** cell and select **Transfer Data to New** from the context menu.

2. Select the **Icepak** system.
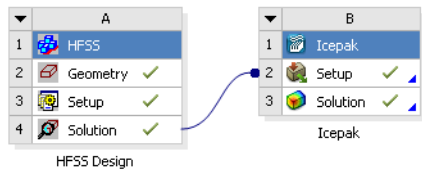
4. To launch the Icepak application, double-click the Icepak cell.

    A new project is created in the name of the project cell. If a DesignModeler system is used the geometry is imported into Icepak automatically. Otherwise, the geometry must be created in Icepak using Icepak primitive objects.

    > **Note:**
    >
    > The geometry can be exported from the HFSS/Maxwell/Q3D Extractor cell to a **Geometry** cell and imported to the Icepak cell using DesignModeler Electronics.

5. In Icepak, go to the **File** menu and select **EM Mapping** and **Volumetric heat losses** or **Surface heat losses**.

    The **Volumetric heat losses** panel contains the following inputs:

    -   A list of solid objects onto which the loss information can be mapped. For these objects, Icepak requests the heat source (loss) terms from the Ansoft application.

    -   **Solution ID** contains available solution sets. Since the HFSS/Maxwell/Q3D Extractor application may have multiple solutions, Icepak will request the generated heat source data for the selected solution.

    -   **Frequency** contains available frequencies. Icepak will request that the HFSS/Maxwell/Q3D Extractor application provide the heat source data for the selected frequency.

    -   **Temperature feedback** enables the writing of temperature data by the Icepak solver to be sent to the Ansoft application to perform two-way coupling. This option must be enabled for two-way coupling.

    > **Note:**
    >
    > When volumetric heat losses are not available, two-way coupling can still be performed for surface heat losses by enabling the **Temperature feedback** option in the **Volumetric heat losses** panel.

6. Click **Accept** to close the panel.

7. After solving in Icepak, to send temperature data back to the Ansoft application to solve again, right-click the Ansoft application's **Solution** cell and select **Enable update** from the context menu.

8. Solve Icepak again using this newly generated data. This process can go on indefinitely until you are satisfied with the results.

Handling Coupling Iterations Between Ansoft and Icepak

You can also perform manual cyclic updates of individual system components until the solution stops changing within a desired level of tolerance. For example:

*   Coupling Iteration 1

    –   Update Maxwell/HFSS/Q3D Extractor Solution cell

    –   Perform EM Mapping on solid zones in Icepak

    –   Update Icepak Solution cell

    –   Perform **Enable update** on Maxwell/HFSS/Q3D Extractor Solution cell

    –   Update Maxwell/HFSS/Q3D Extractor Solution cell

*   Coupling Iteration 2

    –   Update Icepak Setup cell

    –   Update Icepak Solution cell

    –   Perform **Enable update** on Maxwell/HFSS/Q3D Extractor Solution cell

    –   Update Maxwell/HFSS/Q3D Extractor Solution cell

*   Coupling Iteration 3

    –   Update Icepak Setup cell

    –   Update Icepak Solution cell

    –   Perform **Enable update** on Maxwell/HFSS/Q3D Extractor Solution cell

    –   Update Maxwell/HFSS/Q3D Extractor Solution cell

*   Coupling Iteration 4

    –   Update Icepak Setup cell

    –   Update Icepak Solution cell

    –   Perform **Enable update** on Maxwell/HFSS/Q3D Extractor Solution cell

    –   Update Maxwell/HFSS/Q3D Extractor Solution cell

*   and so on.

187

You can perform automatic system updates (coupling iterations) using the Ansoft Feedback Iterator. Refer to Ansoft help documentation for more information on Feedback Iterator.

Icepak and Workbench Integration Tutorial

To access the Ansys Icepak - Ansys Workbench integration tutorial, go to http://support.ansys.com/training and download the Ansys Icepak Tutorial Guide.

Injection Molding Data

The **Injection Molding Data** system allows you to take the data from result-files of injection molding simulations and import it into a Mechanical system. Supported analysis types include **Static** and **Transient Structural**; **Steady-State** and **Transient Thermal**; **Modal** and **Harmonic Response**. File formats of the most popular injection molding software tools are supported. The following quantities can be imported:

- Residual stresses (before ejection)

- Weld lines

- Fiber orientation tensor (for reinforced materials)

- Fiber volume fraction (for reinforced materials)

The input data is parsed, processed, and then transferred to a downstream Mechanical system where it can be applied in the form of element orientations, material fields, and initial stresses.

When simulating the thermo-mechanical behavior of short fiber reinforced composites, the Injection Molding Data system allows you to import injection molding simulation results from other software tools. It's part of a streamlined workflow in Workbench which also employs Material Designer and Mechanical. You will find detailed instructions and a sample workflow in the Short Fiber Composites Guide.

Supported Formats

In the table below, you will find a list of software tools, their file-types, and whether they are supported.

	Mesh File Format	Fiber Orientation Tensor File Format	Initial Stress file Format	Fiber Volume Fraction File Format	Weld Lines File Format
Autodesk Moldflow[a]	Patran Neutral file (.pat) or CDB (generated when exporting initial stresses)	For 3D Meshes: • Patran node-based (.nod) and element-based (.ele) formats.	.ist (only for 3D meshes)[b]	Not supported	.xml, .nod

		• Node and element-based XML (.xml) formats. For Midplane meshes: • Element-based XML (.xml) format.			
Moldex3D	.cdb, .ans	.o2d	.ist	.fcd	.nwd
SigmaSoft[c]	.ans, .sac	.ans, .sac	.sac	.sac	Not supported
Cadmould[c]	.cof	.cof	Not supported	Not supported	Not Supported

[a] Only the 3D and Midplane mesh options are supported. Dual Domain meshes are not supported.

[b] Importing an .ist file in combination with the Patran mesh format is only valid for data in SI units.

[c] To import results from SigmaSoft and Cadmould, you must specify the same file as the **Mesh** and **Fiber Orientation Tensor File**.

Procedure

To import injection molding simulation results, do the following:

1. Drag and drop an **Injection Molding Data** system onto the project schematic. The **Injection Molding Data System** is available under the **Component** systems group of the Workbench application.

2. Right-click the **Setup** cell and select **Edit** (see A, below) to display the **Injection Molding Data** properties (B, below). Alternatively, right-click the cell and select **Properties**.
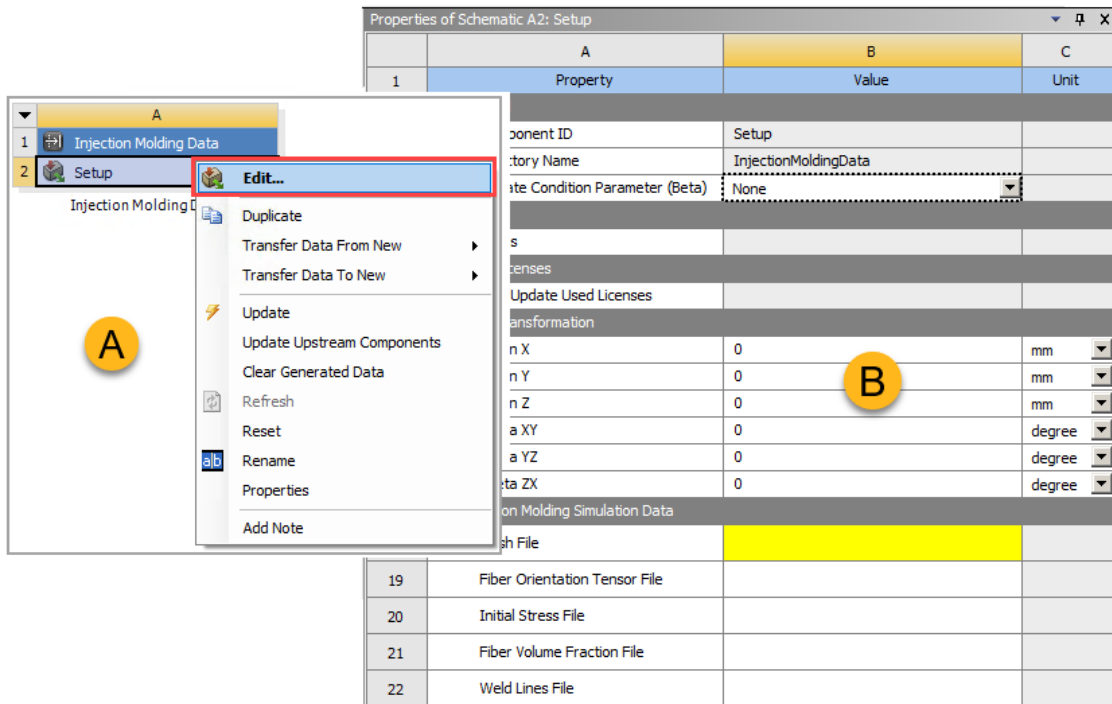
3. Browse to and select the location of the **Mesh File**.

4. Optional: Browse to and add the **Fiber Orientation Tensor File**.

5. Optional: Specify the location of the **Initial Stress File**.

6. Optional: Specify the location of the **Fiber Volume Fraction File**.

7. Optional: Specify the location of the **Weld Lines File**.

8. If needed, apply a **Rigid Transformation** to the input data by specifying the translation and/or rotation components. Transformations are applied in the following order:

   • Rotation about the Y Axis (Theta ZX)

   • Rotation about the X Axis (Theta YZ)

   • Rotation about the Z Axis (Theta XY)

   • Translations

   Because of the order of rotations, the angles Theta XY/YZ/ZX can be considered as intrinsic ZXY Euler angles.

9. Connect to Mechanical:

- To transfer fiber orientation tensor, volume fraction, and weld lines data to Mechanical, drag the **Setup** cell of the **Injection Molding Data** system and drop it onto the **Model** cell of a **Mechanical** system.
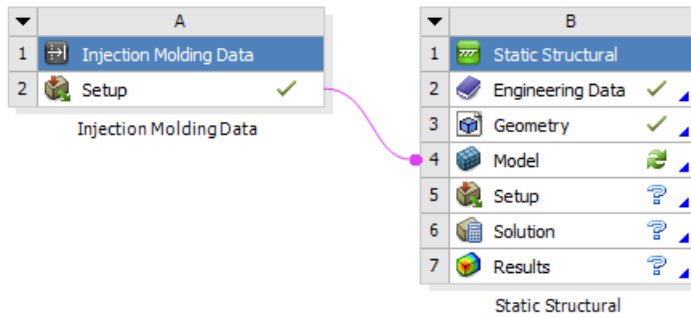
- To transfer initial stress data to Mechanical, drag the **Setup** cell of the **Injection Molding Data** system and drop it onto the **Setup** cell of a **Mechanical System**.
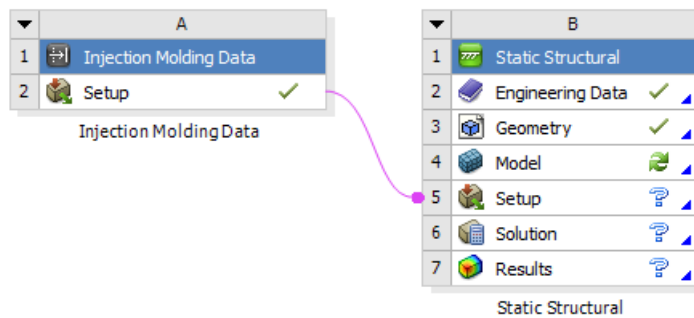
You can combine the two transfers by right-clicking the **Setup** cell of the **Injection Molding Data** cell and choosing **Transfer Data to New** in the context menu.
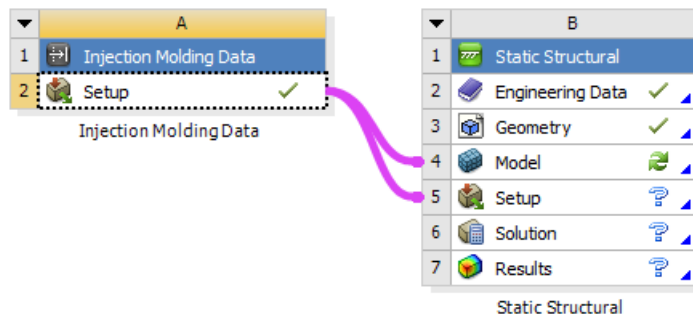
Material Designer

The **Material Designer** system launches Ansys Material Designer, allowing you perform material homogenization. You can simulate the micro-structure of composite materials and lattice structures as well as user-defined materials. Material Designer generates a representative volume element (RVE) and analyzes it in order to compute homogenized (averaged) material properties. The computed materials can then be used in subsequent analyses as any predefined or user-defined materials would be.

Example workflows can be found in Workbench Workflow Examples. For more general information about Material Designer, see the Material Designer User's Guide.

Mechanical APDL

The **Mechanical APDL** system launches Mechanical APDL (formerly known as the Ansys software), allowing you to manage the various files often used and created by Mechanical APDL, especially when working with a linked analysis (for example, thermal-stress, substructuring, submodeling, and so on). It is important that you understand the types of files that the Mechanical APDL application uses and generates, because the actions you take in Workbench act on these files. These files fall into three broad categories: input, reference, and output.

Input Files

Files that are consumed directly by the Mechanical APDL application. Examples include:

- Files consisting of Mechanical APDL commands, generated manually or by Mechanical APDL (log files) or by the Mechanical or Meshing applications.

- Coded input files, such as `.cdb` files, generated by Mechanical APDL and third-party preprocessors

- Mechanical APDL geometry files (`.anf`), generated by Mechanical APDL or DesignModeler

> **Note:**
>
> In some cases, the Mechanical APDL solver overwrites one of its input files with its generated output (for example, the `.rst` file from a Modal system in a Modal to Response Spectrum analysis linked to a Mechanical APDL component system). If this occurs, subsequent updates of the Mechanical APDL component system will fail. To copy the correct input from an upstream system, perform a reset operation on the Mechanical APDL component system.

Referenced Files

Files that are referenced by the execution of an input file. Examples include:

- Database files

- Results files

- Command macro files

- Superelement files

- Solver files

- CAD geometry files

Output Files

Files that are produced by all Mechanical APDL application runs. Primary output files include:

- Results file (`.rst`, `.rth`, etc.)

- Output file (`.out`) of the command echoes, solution information, and requested data listings

- Log file (`.log`) of the commands issued to the Mechanical APDL application

- Error file (`.err`) listing any warnings or errors encountered

Creating a Mechanical APDL Analysis

Use the following procedure to create a Mechanical APDL analysis.

To work through a **Mechanical APDL** system:

1. To add a **Mechanical APDL** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. If required, connect other systems to a Mechanical APDL system by using the **Transfer Data From New** or **Transfer Data To New** context menu options. You can also drag systems from the toolbox, or manually create links between systems.

    When transferring data to a Mechanical APDL system from another system, you can transfer data from the following cell sources:

    - **Geometry**: transfers just the geometry in the form of an `.anf` file. This option is only supported for geometry that is represented as DesignModeler geometry.

    - **Model** (**Mesh** cell if a meshing system): transfers an input file containing only the mesh, contact, coordinate system, and named selections data.

    - **Setup** (Mechanical Systems): transfers an input file containing all data necessary to solve the analysis, including geometry, model, loads, materials, etc. Any supporting files needed to execute the input file will be transferred as well. Examples include pre-stress modal or random vibration.

    - **Setup** (Finite Element Modeler): Transfers input file containing any finite data recognized inside Finite Element Modeler, such as mesh, materials, components, constraints, etc.

    - **Solution** (Mechanical Systems): transfers the database file (`.db`) if it exists and result file only.

    > **Note:**
    >
    > For **Model**, **Setup** (Mechanical Systems), and **Solution** transfer cells, if you solve within Mechanical, you must run an update on the appropriate cell in the Mechanical system to obtain the correct state on the schematic.
    >
    > In most cases, **Model** and **Setup** components from the same Mechanical system should not be linked to one Mechanical APDL system. Doing so will cause the Mechanical system to provide two different (and possibly conflicting) input files to the Mechanical APDL system.

Named selections and coordinate systems that are added to a solved Mechanical system are not immediately reflected in downstream Mechanical APDL systems. They will be available in future solution attempts.

Important:

The Mechanical APDL system consumes all input data without unit system knowledge. You must assure that all input data being used by the Mechanical APDL system is in a consistent unit system. See Solving Units for more information on unit system.

When transferring data from a Mechanical APDL system to another Mechanical APDL system, you can transfer four types of data:

- Results: transfers all results files (including `.rst`, `.rth`, etc.)

- Database: transfers all database files (`.db`)

- Solver: transfers all files in the system folder

- CDB: transfers `.cdb` files

You can also transfer data to a new Finite Element Model system, which uses the `.cdb` file(s).

These files are simply copied to the new system if they exist; Workbench does not generate the files. Before transferring data to a new system, be sure that you have an input file that generates the necessary files from the existing Mechanical APDL system.

When you transfer data to or from another system, right-click the link connecting the systems and select **Properties** from the context menu. The **Properties** pane opens, detailing the nature of the transfer (such as Transfer CDB File).

3. To open the Components workspace to select Mechanical APDL parameters, double-click the **Analysis** cell or right-click the cell and select **Edit** from the context menu.

From the Components workspace, you can select Mechanical APDL parameters or specify setup properties (p. 425) (such as command line options, memory settings, number of processors, etc.).

When you add an input file using the context menu, Workbench automatically searches the file for potential parameters (for example, ***SET**, ***GET**, = assignments). Those parameters are then displayed in the **Properties** pane when that input file is selected in the **Outline** pane. To use one of those parameters, select that parameter's check box and indicate whether it should be used as an input or an output parameter. Input parameters are sent to Mechanical APDL with the value specified when updating. After the update, Workbench retrieves the output values from Mechanical APDL and sets those values in Workbench.

Note:

The presence of a **/EXIT** command in the input file causes state and parameters to malfunction. Make sure you remove this command before adding the input file.

---

4. Launch the Mechanical APDL application using one of the following options:

    - To launch the Mechanical APDL application interactively, right-click the **Analysis** cell and select **Edit in Mechanical APDL** or **Open in Mechanical APDL** from the context menu.

    - To launch the Mechanical APDL application with input and reference files specified, right-click the **Analysis** cell and select **Add Input File** or **Add Reference File** from the context menu. Then select **Edit in Mechanical APDL**

      The Mechanical APDL application launches in interactive mode, and the input file(s) specified will be sent to the Mechanical APDL application and processed in the order listed. After all of these files are processed, the Mechanical APDL application remains active and you can continue your analysis using the standard Mechanical APDL application interface. Any action you take in the Mechanical APDL application will not be reflected in Workbench state indicators or parameters.

Be aware that any time you launch the Mechanical APDL application, Workbench does not log or record the actions that occur in the Mechanical APDL application. If you make changes in the Mechanical APDL application, be sure that the changes are reflected appropriately in the input files. To maintain connectivity (such as to read output parameters), complete an update, either at the project level or at the appropriate system/cell level.

To save Mechanical APDL changes from an open session, you must include a **SAVE** command in one of your input files. The Workbench save capability does not invoke the Mechanical APDL **SAVE** command.

When you add an input file, the files are processed in the order shown. You can change the order in which the files are processed by dragging the files into the proper order. To delete files, right-click the file to be deleted and select **Delete** from the context menu.

To stop a Mechanical APDL batch run, you can .

Mechanical APDL Context Menu Options

When the Mechanical APDL system is active in the schematic, use the right mouse button to initiate the following **Analysis** cell actions.

Menu Item	Description
Edit	Opens the Components workspace, where you can specify Mechanical APDL parameters and setup properties. This is the default action.
Edit in Mechanical APDL	Launches the Mechanical APDL application interactively and reads the input files. If the state is currently up to date, Workbench sets the state to Update Required at this time, even if you do not make any changes in the Mechanical APDL application.
Open in Mechanical APDL	Launches the Mechanical APDL application interactively without reading any input files. Any action you take in the Mechanical APDL application are not be reflected in Workbench state indicators.
Add Input File	Opens a dialog box where you can browse to an input file. When you add an input file, the file is immediately copied into the project directory. To make changes to this file, change the file in the project directory, not the original file. If you have a large input file

Menu Item	Description
	and have disk space concerns, keep the file in the directory of your choice and use a separate input file to reference it (using the **/INPUT** command).
Add Referenced File	Opens a dialog box where you can browse to a referenced file. When you add a referenced file, the file is immediately copied into the project directory. To make changes to this file, change the file in the project directory, not the original file. If you have a large reference file and have disk space concerns, keep the file in the directory of your choice and reference it manually.
Track Solution	Launches the Results Tracking tool during an update, allowing you to monitor diagnostics results of interest in real time during the solution. For more information, see the **NLHIST** command.
Update	Runs the Mechanical APDL application in batch mode, processing all input files in the order listed. If you make changes in the Mechanical APDL application, be sure that the changes are reflected appropriately in the input files before running an update. If you do not, an update could potentially overwrite the work you have done in the Mechanical APDL application. **Note:** An update launches the Mechanical APDL application in batch mode, using all input and referenced files in the order shown in the Outline pane. After all files are processed, the Mechanical APDL application exits. Updating captures any output parameters generated in the Mechanical APDL application and allows you to continue working in Workbench.
Refresh	Copies the latest transfer files into the project directory. Input and referenced files are not re-copied from their original locations. If you change an upstream system after you make changes to the Mechanical APDL application, a refresh could potentially overwrite your Mechanical APDL application changes. Be sure that any changes you do in the Mechanical APDL application are reflected appropriately in the input files before running a refresh. Only changes that occur within the schematic are captured with a Refresh Required state (p. 449); Workbench does not indicate Refresh Required for changes made directly to a file (such as manually editing an input file).
Clear Generated Data	Deletes all files on disk in the system directory except input or reference files. It will not affect any input or reference files.
Rename, Reset	Standard actions as described in Common Context Menu Options (p. 456).
Properties	Launches the **Properties** pane, where you can define graphics settings, command line options, database and tab memory, and other settings. Be aware when selecting graphics settings that some options are potentially platform-specific and must be changed when switching platforms before running the project with Mechanical APDL in interactive mode.

Solver Input File Generated from the Setup cell of a Mechanical APDL System

The solver input file transferred from the **Setup** cell contains all the commands needed to execute a complete run, including any **SOLVE** commands that are necessary. However, the input also contains

a conditional **/EOF** statement to halt reading of the file and therefore not execute the solve. This conditional statement will be executed when the Mechanical APDL application is invoked from the **Edit in Mechanical APDL** context menu option, therefore running the analysis to the point just prior to the **SOLVE** command. If a different behavior is desired, you can edit the input file in the Mechanical APDL system folder to obtain a different behavior or to add an additional input file containing the **SOLVE** command.

Mechanical Model

The **Mechanical Model** system launches the Mechanical application, allowing you to create an analysis using a single model and multiple system analysis branches. You can also create other **Mechanical Model** systems that share data at any cell level (**Engineering Data**, **Geometry**, or **Model**). **Mechanical Model** Model systems are unfiltered (physics and solver).

The system consists of **Engineering Data**, **Geometry**, and **Model** cells. In the Mechanical application, this system corresponds to that of a Model-only system.

This system is created when you resume a legacy database that does not have an analysis environment already defined. It can also be used as a system replacement for a **Mesh** system.

Working Through a Mechanical Model system

To work through a **Mechanical Model** system:

1. To add a **Mechanical Model** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Create or attach a geometry using the **Geometry** cell context menu.

3. If required, right-click the **Model** cell and select **Edit** to edit the model.

4. To add a connected system, either drag a valid analysis system from the **Toolbox** and drop it on the appropriate target location, or right-click the **Geometry** or **Model** cell and select **Transfer Data To New** from the context menu.

5. To create multiple system analysis branches, repeat step 4 with other analysis systems.

Resuming an Existing Legacy Database

To resume an existing legacy database that contains no physics environment:

1. From the menu bar, select **File** → **Import**.

2. Browse to the legacy database, select it, and click **Open**.

   A **Mechanical Model** system appears in the **Project Schematic**, with the legacy database loaded.

3. To open the Mechanical application, double-click the **Model** cell, or right-click the cell and select **Edit** from the cell context menu.

4. In the Mechanical application, complete any necessary model updates.

5. To add a connected system, either drag a valid analysis system from the **Toolbox** and drop it on the appropriate target location, or right-click the **Geometry** or **Model** cell and select **Transfer Data To New** from the context menu.

6. Continue with the analysis in the analysis system as you normally would.

Replacing a Mesh System with a Mechanical Model System

To replace an existing Mesh system with a Mechanical Model system (or vice-versa), right-click the system header and select **Replace with** → **Mechanical Model** or **Replace with** → **Mesh** (respectively) from the context menu. When a Mesh system is replaced with a Mechanical Model system, the Mechanical Model system can then be shared with any analysis system as described above.

The units setting specified in an existing system is not maintained in the replacement system. In the replacement system, you must select the units setting that you want to use.

Model-to-Model Connections

Multiple Mechanical Model component systems can be merged together by creating a connection between the **Model** cells of each component system. This allows you to build up more complicated models from smaller, simpler models. This behaves in the same way as Mesh-to-Mesh Connections (p. 203). More detailed information can be found in Assembling External Models and Mechanical Models in the *Mechanical User's Guide*.

Solution-to-Model Connections

You can transfer a deformed geometry from a Static/Transient/Modal/Buckling/Explicit Dynamics system to a Mechanicall Model or to a Mechanical analysis system. You might want to do this to introduce imperfections to an otherwise perfect geometry to overcome convergence issues when running a nonlinear simulation.

To use the deformed mesh from a solved analysis as the initial geometry/mesh for a follow-on analysis:

1. Drag a connection from the Mechanical **Solution** cell to a downstream **Model** cell in a Mechanical system.

   You can create connections to multiple downstream **Model** cells.

2. To share Engineering Data between the systems, link the **Engineering Data** cells as required.

   Note:

   If you are incorporating a deformed geometry into Model Assembly systems, you need to share the Engineering Data cell of the deformed geometry system with one or more of the Model Assembly systems in order to have the materials automatically transferred and set for the deformed geometry in the downstream system.

3. Optionally, in the **Properties** pane:

a. Adjust the amount and direction of the deformation by setting a **Scale Factor** (a negative value reverses the direction of the deformation).

b. Adjust the upstream **Time** settings (for Transient, Static Structural, or Explicit Dynamics systems) or the **Mode** (for Eigenvalue Buckling or Modal systems) for each connection to a downstream Model cell.

    If you set a **User Defined Time**, the value is always interpreted as being in seconds.

c. Adjust the transfer input settings on downstream Model. The transfer settings are the same as those for Mesh-to-Mesh Connections (p. 203).

For more detailed information, see Geometry from Deformation Results in the *Mechanical User's Guide*.

Controlling Node and Element Numbering when Model Cells Share Data

When you connect the Model cell of a Mechanical system to the Model cell of a downstream Mechanical system, you can use the **Renumber Mesh Nodes and Elements Automatically** property in the downstream Mechanical Model cell to control whether mesh nodes and elements are automatically renumbered to prevent conflicts. You can set the **Renumber Mesh Nodes and Elements Automatically** property on a transfer-by-transfer basis. By default automatic renumbering occurs, but you can disable the property for as many of the transfers as you want—as long as their element numbers will not conflict downstream and the **Number Of Copies** is set to 0.

Preview Assembled Geometry

You can quickly preview the orientation of your parts in an assembly by right-clicking on the downstream model cell and selecting **Preview Assembled Geometry**. When using this feature, Mechanical assembles the geometry only (computationally expensive items such as the mesh are not assembled), allowing you to review the assembled geometry quickly. This feature is useful if you are unsure about the relative positioning of your subassemblies and some transformations are required for proper alignment.

Example Workflow

To start, you must have fully updated upstream **Model** cells connected to a downstream **Model** cell. The downstream **Model** cell is in a state of refresh required. In this demonstration, `Box1` and `Box2` are both the same model, a rectangular prism, and you would like to have them oriented into a "T" (one oriented 90 degrees from the other). However, a translation in the positive y-direction is also needed to ensure a smooth contact between the parts. To accomplish this, enter a value of 10 m in the **Properties** pane of the **Project Schematic** to translate the Y-origin of `Box1`. As opposed to issuing a full update on the downstream **Model** cell, which loads the mesh data as well as the geometry, you first perform a geometry-only assembly by right-clicking on the downstream model cell and selecting **Preview Assembled Geometry**.
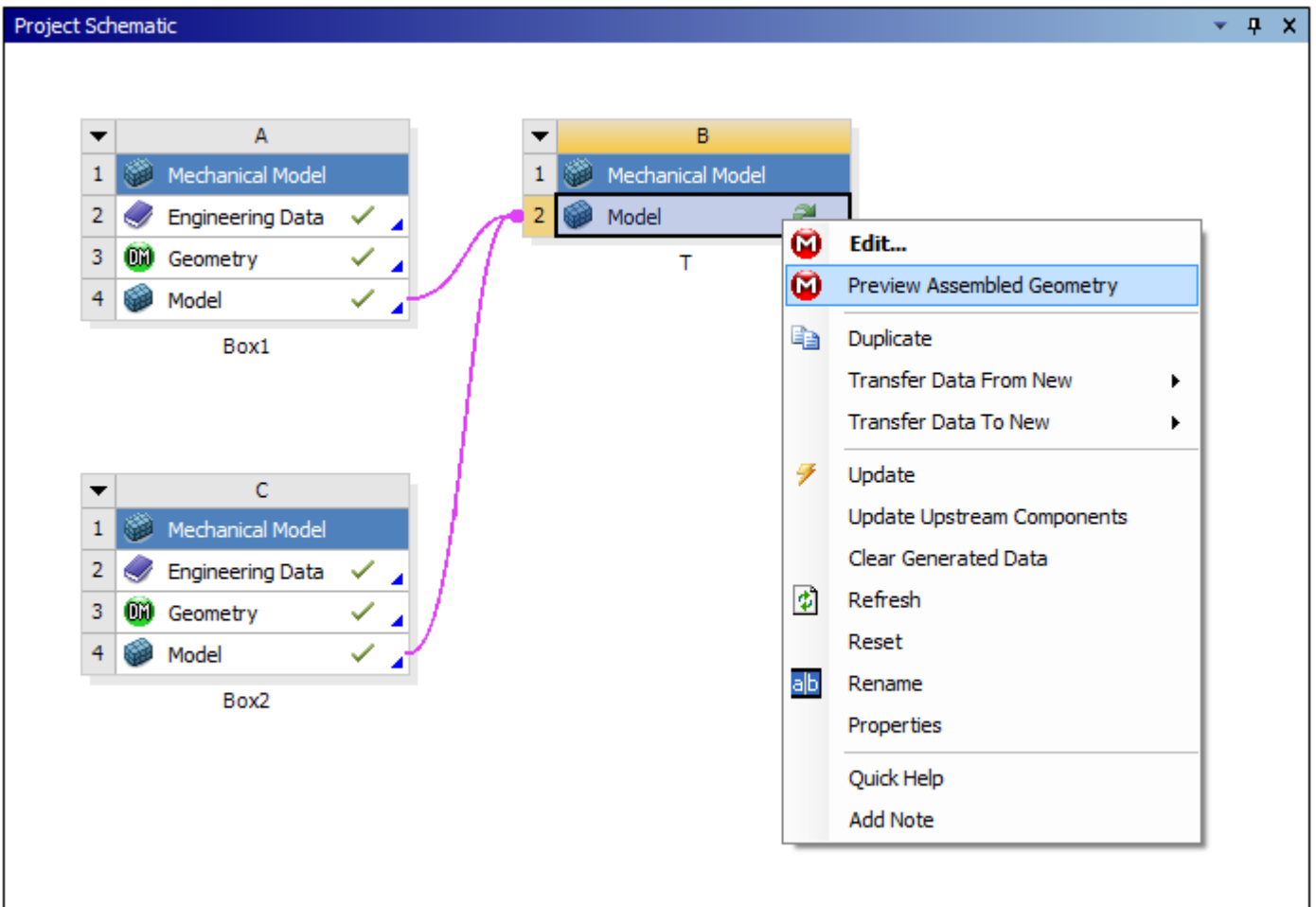
This opens the assembly in Mechanical for you to inspect. When you activate Mechanical in this way, no meshing or physics options are available.
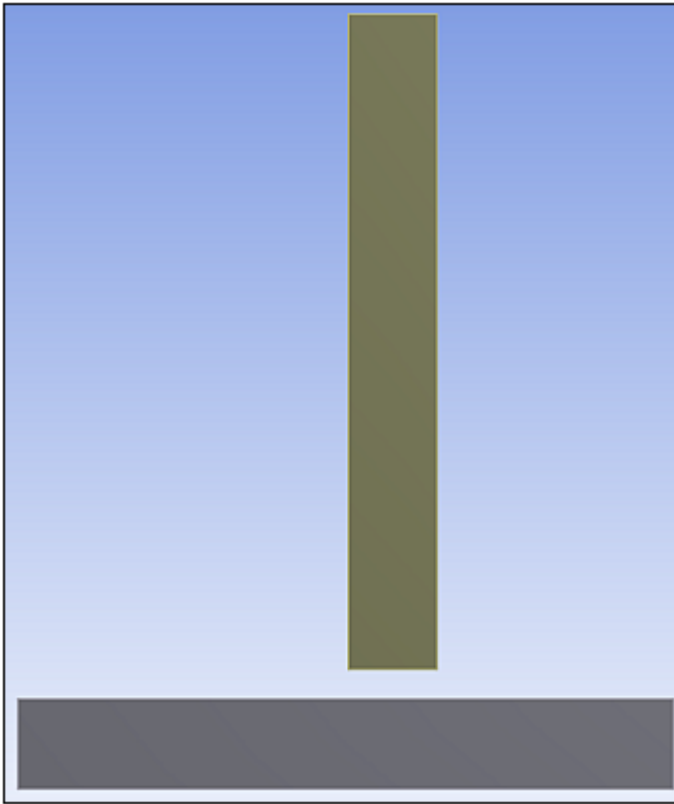
Upon viewing the preview, you discover that the geometry is not in the proper orientation. After inspecting the gap between the parts, you determine the proper y-translation is 9.35 m. After entering this data in the **Properties** pane of the **Project Schematic**, you can again right-click the downstream **Model** cell and select **Preview Assembled Geometry**. Mechanical will quickly update to show you the change in orientation.
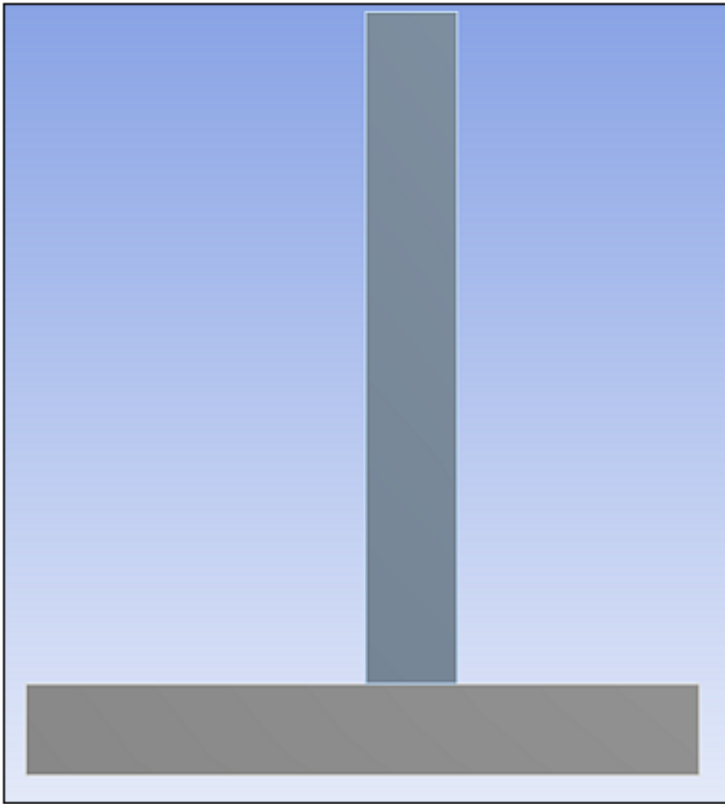
You confirm that the orientation of the two parts is correct. At this point, you can issue a full update so that Mechanical loads in the mesh data and you can continue your analysis.

Mesh

The **Mesh** system launches multiple meshing and geometry applications, allowing you to create and/or open geometry or mesh files.

To work through a **Mesh**, add it to the **Project Schematic** using any of these methods:

- Double-click the **Mesh** component system template in the **Toolbox**.

- Drag-and-drop the **Mesh** component system template onto the **Project Schematic**.

- Drag-and-drop a `.meshdat` or `.cmdb` file from Windows Explorer onto the **Project Schematic**.

- From the menu bar, select **File** → **Import** or click the **Import** button and select a `.meshdat` or `.cmdb` file.

Mesh Context Menu Options

For standard context menu options, see System Header Context Menu Options (p. 457) and Geometry (p. 448). The following table describes the context menu actions specific to the **Mesh** system.

Menu Item	Description
System Header	

Menu Item	Description
Replace With > Mechanical Model	Replaces the Mesh component system with a Mechanical Model component system.
Mesh Cell	
Edit	Opens the Meshing application and loads an existing geometry or mesh file.
Import Mesh File	Imports read-only meshes for downstream application use. For more information see Importing Read-only Meshes for Downstream Application Use
Duplicate	Duplicates the Mesh system. The **Geometry** cell is shared, and all data associated with the **Mesh** cell is copied to the second system.
Transfer Data To New	Transfers a mesh into one of the following downstream systems: • **Autodyn**: Inserts a downstream Autodyn system and generates a data transfer connection (`.cmdb` file) from the **Mesh** cell to the **Setup** cell of the Autodyn system. • **CFX**: Inserts a downstream CFX system and generates a data transfer connection (`.cmdb` file) from the **Mesh** cell to the **Setup** cell of the CFX system. • **External Model**: Inserts a downstream **External Model** system and generates a data transfer connection (`.cmdb` file) from the **Mesh** cell to the **Model** cell of the **External Model** system. • **Forte**: Inserts a downstream Forte system and generates a data transfer connection (`.msh` file) from the **Mesh** cell to the **Setup** cell of the Forte system. • **Mechanical APDL**: Inserts a downstream Mechanical APDL system and generates a data transfer connection (`.inp` file) from the **Mesh** cell to the **Analysis cell** of the Mechanical APDL system. • **Polyflow**: Inserts a downstream Polyflow system and generates a data transfer connection (`.poly` file) from the **Mesh** cell to the **Setup** cell of the Polyflow system.

Mesh-to-Mesh Connections

Multiple Mesh component systems can be merged together by creating a connection between the **Mesh** cells of each component system. This allows you to build up more complicated meshes from smaller, simpler meshes.
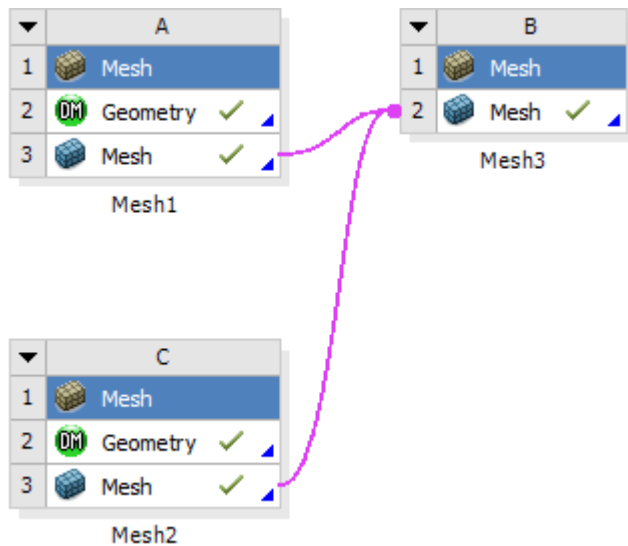
For every upstream **Mesh** cell connected to a downstream **Mesh** cell, a new set of properties called **Transfer Settings for [Mesh component name]** appears in the **Properties** pane of the downstream **Mesh** cell. In the above example, with two upstream Mesh components connected to single, downstream Mesh component, the **Properties** pane of the downstream Mesh (cell B3) has two new groups of properties.

Similarly, a **Renumber Mesh Nodes and Elements Automatically** property appears for each transfer. This property in the downstream mesh cell controls whether mesh nodes and elements are automatically renumbered to prevent conflicts. You can set the **Renumber Mesh Nodes and Elements Automatically** property on a transfer-by-transfer basis. By default, automatic renumbering occurs but you can disable the property for as many of the transfers as you want—as long as their element numbers do not conflict downstream and the **Number Of Copies** is set to 0.

	A		B	C	D
	Property		Value	Unit	P
1	Property		Value	Unit	P
2	⊟ General				
3	Component ID		Mesh 1		
4	Directory Name		SYS-4		
5	⊟ Notes				
6	Notes				
7	⊟ Used Licenses				
8	Last Update Used Licenses				
9	⊟ System Information				
10	Physics		Any		
11	Analysis		Any		
12	Solver		Any		
13	⊟ Mesh				
14	Save Mesh Data In Separate File		☐		
15	⊟ General Mesh Assembly Properties				
16	Length Unit		m ▼		
17	Object Renaming		Based on System... ▼		
18	Group Objects By Source		☑		
19	⊟ Transfer Settings for Mesh (Component ID: Mesh)				
20	Number of Copies		0		
21	Renumber Mesh Nodes and Elements Automatically		☑		
22	⊟ Rigid Transform				
23	Origin X		0	m ▼	☐
24	Origin Y		0	m ▼	☐
25	Origin Z		0	m ▼	☐
26	Theta XY		0	radian ▼	☐
27	Theta YZ		0	radian ▼	☐
28	Theta ZX		0	radian ▼	☐
29	⊟ Transfer Settings for Mesh (Component ID: Mesh 2)				
30	Number of Copies		0		
31	Renumber Mesh Nodes and Elements Automatically		☑		
32	⊞ Rigid Transform				

Table 5: Properties Pane: General Properties

Property	Description
Length Unit	This property sets the units for the downstream mesh assembly. After the meshes are initially assembled, this property becomes read-only. You must reset the downstream mesh cell in order to change the **Length Unit**.

Table 6: Properties Pane: Transfer Settings Section

Property	Description
Number	When set to zero (default), only the source mesh is transformed. If you specify a

Property	Description
Of Copies	number of copies greater than zero, these will be in addition to the source mesh. For example, if you import a .cdb file with a single part and set **Number Of Copies** to **2**, you will get three parts in Mechanical.
Renumber Mesh Nodes and Elements Automatically	When Mesh cells are connected, this property in the downstream Mesh cell controls whether mesh nodes and elements are automatically renumbered to prevent conflicts. If the **Number Of Copies** is set to **1** or greater, automatic renumbering occurs.
Transform Original	This property is available only when **Number Of Copies** is set to **1** or greater. Select the check box if you want to apply the specified transformation to the source mesh as well as any copies.
Origin X/Y/Z	These properties allow you to translate the origin of the model along the X, Y, or Z axis. If you specify any copies, the translation is applied relative to the previous copy (or source mesh in the case of the first copy).
Theta XY/YZ/ZX	These properties allow you to rotate the model about its origin in the XY, YZ, or ZX plane. If you specify any copies, the rotation is applied relative to the previous copy (or source mesh in the case of the first copy).

Any change in these properties will put the downstream mesh in a .

You can preview the mesh transformations by right-clicking on the **Mesh** cell and selecting **Preview Assembled Geometry**. Use this feature to confirm that the correct mesh transformations have been applied. For more details and an example workflow, see .

Microsoft Office Excel

The **Microsoft Office Excel** system launches Microsoft Office Excel 2013 or 2016, allowing you to use it as a calculator in Workbench This feature exposes Excel ranges as input and output parameters to Workbench. You can use these ranges to create design points and Design Exploration studies.

Note:

- The Excel add-in is included with Workbench. It does not require DesignXplorer or any other specific Ansys product, but it must connect to a locally installed seat of Microsoft Excel (purchased separately).

- In the **Trust Center** options for Excel, the **File Block Settings** page specifies the older versions of Excel files to block. If Workbench is unable to open an Excel file formatted in an older version, on the **File Block Settings** page, clear the check boxes for the older Excel versions that you use and click **OK**. After you close and reopen Workbench, it should be able to open Excel files formatted in these older versions.

Related topics:

Preparing the Excel File

To be exposed as parameters in Workbench, the ranges must be named in Excel. The names are filtered during the addition of the Excel file to the project; only the ranges matching the prefix defined by the **Named Range Key** property in the **Properties** pane of the **Analysis** cell are made visible in Workbench.

Note:

By default, no filtering prefix is defined at either the Workbench or the project level. You set a filter by either of the following methods:

- Set a default prefix that will be used for all new projects (see Microsoft Office Excel (p. 433)).

- Set a prefix at the project level by entering it in the **Named Range Key** property in the **Properties** pane of the Microsoft Office Excel **Analysis** cell.

To name a range in Excel:

1. Open the file in the Microsoft Office Excel application.

2. Right-click a cell and select **Define Name** from the context menu.
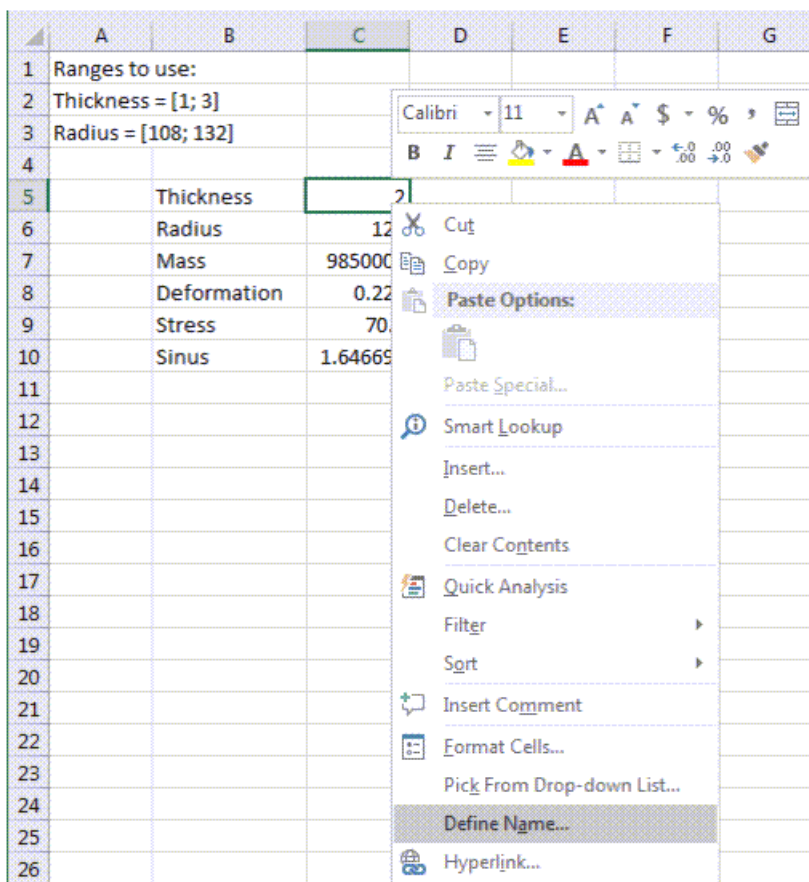
3.	Enter the name.

4.	To review and modify all the defined names in Excel, select **Formulas** → **Name Manager**.
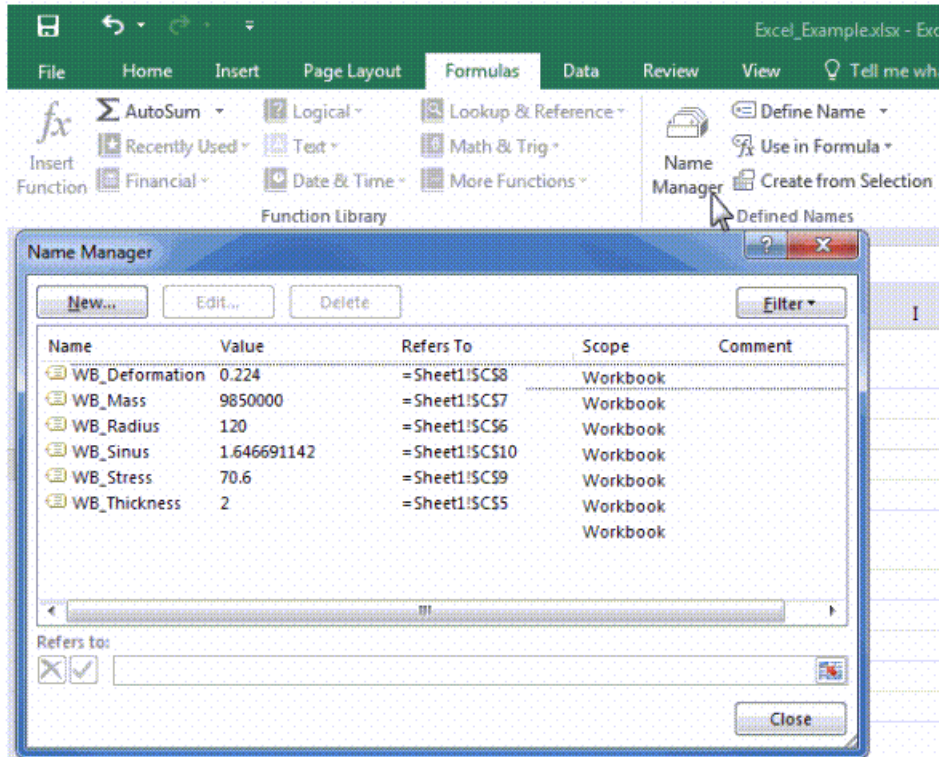
A named range can contain several cells. Workbench expects the named ranges to contain either one or two cells, the first cell being reserved for the value and the second cell being reserved for the unit string, if any. If a named range contains more than two cells, it is ignored.

Note:

If you added the file to the Excel system in Workbench before naming the ranges, or if you want to edit the names of the ranges after the addition of the file, you can right-click the **Analysis** component or the file node in the **Outline** and select **Open file in Excel**. This opens the Excel file used by Workbench in Excel.

All changes performed in Excel impact the state of the data in Workbench and invalidate the results; make sure you save your changes and reload the file afterwards (when editing the **Analysis** cell, right-click the file node and select **Reload**).

Setting Up the Excel Calculator

Complete the following procedure to set up Excel as a calculator in Workbench.

To add a file and set macro property usage:

1.	To add a **Microsoft Office Excel** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Right-click the **Analysis** cell and select **Add File** to the context menu.
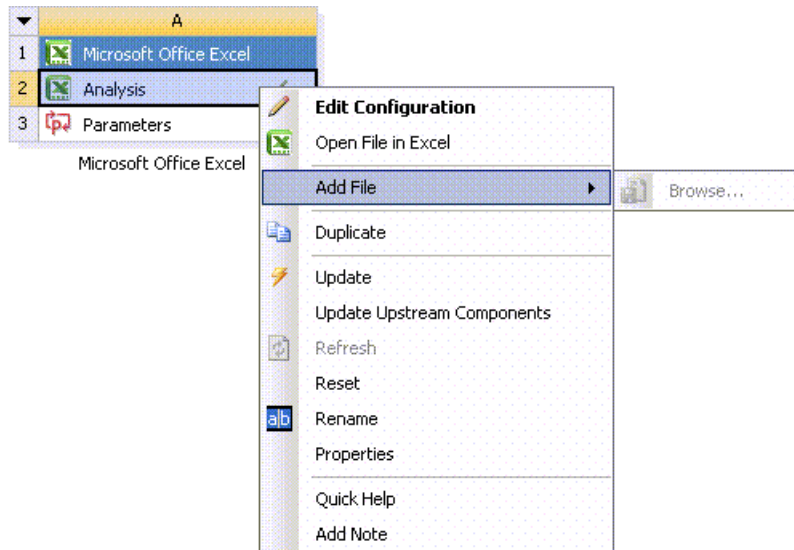
3. Browse to the Excel file that you have prepared.

4. Select the file and click **Open**.

5. To review the list of named ranges retrieved from the Excel file, right-click the **Analysis** cell and select **Edit Configuration** from the context menu.

6. For each listed named range, select the check boxes in the **Input** or **Output** column to publish the range as an input parameter or an output parameter in the Workbench project.
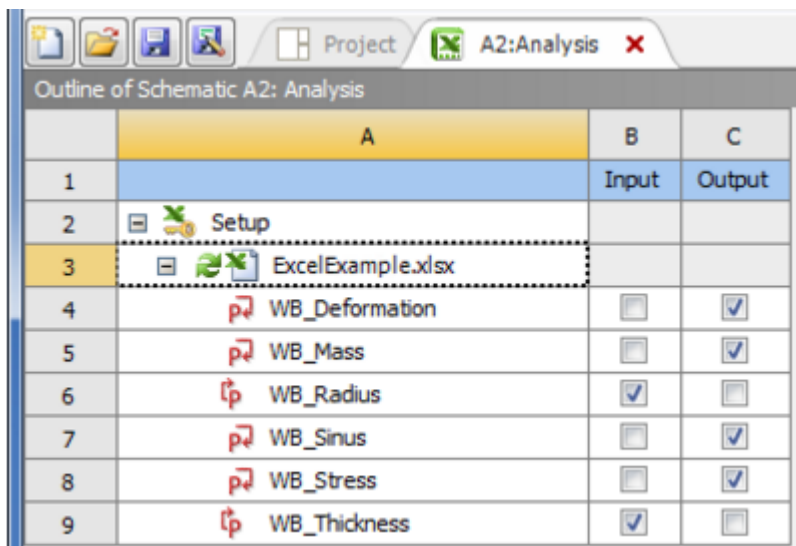
7. If the calculation in Excel requires the execution of a macro:

   a. In the **Outline** pane, select the added file.

   b. In the **Properties** pane, select the **Use a Macro** check box.

c. In the **Macro Name** field, enter the name of the macro.

Note:

If a button is used to start the calculation and you do not know the name of the macro associated with it, right-click the button and select **Assign Macro** to discover the name to use.
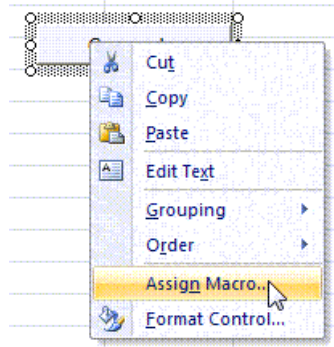
8. Switch to the **Project** tab.

On the **Project Schematic**, the **Microsoft Office Excel** system is linked to the **Parameter Set** bar.

The project is ready to create design points and Design Exploration studies.

Support of Units

Units are handled by the Excel system. A valid Workbench unit string must be used and included in the named range.

For instance, if an input parameter is a length in millimeters, you can name *WB_Length* the range A1:B1 where the cell A1 contains the length value (for example, 120.5) and the cell B1 contains the unit string (mm). When selecting the WB_Length range in the **Outline** pane, the **Properties** pane looks like the following image. In this case, the Quantity Name is automatically identified as a Length.

If there are several possible Quantity Names for the same unit string, you have to select the Quantity Name in the properties of the range for the unit conversion to be performed as expected. For instance, if A1:B1 was actually a temperature in degree Celsius, the Quantity Name could be a Temperature or a Temperature Difference.

For detailed information on working with units in Workbench, see Configuring Units in Workbench (p. 26)

File Management and Modification of the Worksheet

When the Excel file is added to the **Analysis** cell of the **Microsoft Office Excel** system, it is copied inside the Workbench project files. Any modification made to the original file is not seen by Workbench except if you delete and add the file again.

To modify the file copied in the Workbench project files, right-click the **Analysis** cell, or the file node in the **Outline** pane, and select **Open file in Excel**. Once modifications are done, save the file. The state of the file in Workbench changes to Refresh Required (p. 449), which indicates that data is not synchronized anymore. Results such as design points and Design Exploration systems in the Workbench project are outdated. Refresh the project to synchronize all the pieces of the project.

If a change in the Excel file was not detected by Workbench, you can force a reload of the file (right-click the file node and select **Reload**).

It is not necessary to close the workbook or the Excel application to proceed with design points or Design Exploration updates. Interaction with the Excel application are frozen, but you can see the performed calculations.

Performance Map

The **Performance Map** system launches Turbomachinery Performance Map, an application used to create performance maps for turbomachinery cases.

For more information, see Turbomachinery Performance Maps in the *TurboSystem User's Guide*

To work through a **Performance Map** system:

1. To add a **Performance Map** component template, drag the template from the **Toolbox** to the **Project Schematic** or double-click the template in the **Toolbox**.

2. To open Turbomachinery Performance Map, double-click the **Performance Map** cell.

Polyflow

The **Polyflow** system launches Ansys Polyflow, allowing you to analyze fluid flows with free surfaces, complex rheology (including non-Newtonian behavior with viscoelasticity), heat transfer, and chemical reactions. Using Polyflow involves specifying the computation models inside Polydata, which is the module for problem setup; and running the calculations using the Polyflow solver. Polyflow also

comes with several useful utilities, which can be accessed from the context menu on the **Setup** and **Solution** cells. For example, Polymat can be used to calculate material properties, including viscoelastic parameters based on experimental data. For more details, see the product documentation.

There are three Polyflow fluid-flow analysis systems available in Workbench:

- **Polyflow**: Provides the full simulation capabilities of Polyflow.

- **Polyflow - Blow Molding**: Provides only the application-specific capabilities of Polyflow that are suited to blow molding simulations.

- **Polyflow - Extrusion**: Provides only the application-specific capabilities of Polyflow that are suited to extrusion simulations.

To work through a **Polyflow** system:

1. To add a **Polyflow**, **Polyflow - Blow Molding**, or **Polyflow - Extrusion** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To import a mesh, right-click the **Setup** cell and select **Import Mesh** from the context menu.

3. To merge, scale, translate, and rotate the mesh, right-click the **Setup** cell and select **Polyfuse** from the context menu.

4. To define your preferences for Polydata, right-click the **Setup** cell and select **Preferences** → **Polydata** from the context menu.

5. To open Polydata, double-click the **Setup** cell.

   In Polydata, you can specify the computational models, materials, boundary conditions, and solution parameters.

   Alternatively, to import a previously saved Polyflow data file, right-click the **Setup** cell and select **Import Polyflow Dat...** from the context menu.

6. Optionally, to specify material data, right-click the **Setup** cell and select **Polymat** from the context menu.

7. To define your preferences for Polyflow, right-click the **Solution** cell and select **Preferences** → **Polyflow** from the context menu.

8. To run the Polyflow calculation, right-click the **Solution** cell and select **Update** from the context menu.

9. To check the status of the solver during or after the calculation, right-click the **Solution** cell and select **Polydiag** from the context menu.

10. To open the listing file to see what Polyflow has done during or after the calculation, right-click the **Solution** cell and select **Listing Viewer** from the context menu.

11. To generate plots of the solution data, right-click the **Solution** cell and select **Polycurve** from the context menu.

12. To statistically postprocess the results of the solution data, right-click the **Solution** cell and select **Polystat** from the context menu.

For detailed information on working with Polyflow, see the online documentation available under the **Help** menu within Polyflow. In addition, see Polyflow in Workbench User's Guide.

For information on submitting Polyflow jobs to Remote Solve Manager, see Submitting Polyflow Jobs to Remote Solve Manager (p. 338).

Results

The **Results** system launches Ansys CFD-Post, a flexible, state-of-the-art postprocessor that enables easy visualization and quantitative analysis of the results of CFD simulations.

For detailed information on working with Ansys CFD-Post, see CFD-Post in Ansys Workbench in the *CFD-Post User's Guide*.

To work through a **Results** system:

1. To add a **Results** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open CFD-Post, right-click the **Results** cell and select **Edit** from the context menu.

3. In CFD-Post, to load a results file from the Ansys CFX-Solver, Ansys Forte, or Ansys Polyflow, select **File → Load Results**.

Sherlock Pre and Post

The **Sherlock Pre** and **Sherlock Post** systems launch Ansys Sherlock, a Java-based application that allows you to analyze the reliability of circuit card assemblies based on their design files.

For more details on working with Sherlock, see the Sherlock User's Guide.

To work through a Sherlock system workflow:

1. To add a **Sherlock (Pre)** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Add a Sherlock project to the system.

   - **Import a Project**

     1. Right-click the **Project** cell and select **Import Project → Browse** from the context menu.

     2. Browse to the Sherlock project file, select it, and click **Open**.

        The Sherlock GRPC console opens as the project is loading.

   - **Create a Project in Sherlock**

     1. Right-click the **Project** cell and select **Edit** from the context menu.

     2. In Sherlock, in the **Project** menu, select **Add Project**.

3. Create a Sherlock project as described in the Sherlock User's Guide.

4. In Workbench, to sync the Sherlock project changes without closing Sherlock, right-click the **Project** cell and select **Sync** from the context menu.

    > **Note:**
    >
    > This step is optional. When you close Sherlock, Workbench automatically syncs the project.

5. Save the project and close Sherlock.

3. Right-click the **Setup** cell and select **Properties** from the context menu.

4. In the **Properties** pane, from the **Export Type** list, select the type of model to export.

5. From the **Board** list, select the board to use for the analysis.

6. From the **Analysis Type** list, select the analysis type to use.

    > **Note:**
    >
    > If the analysis selected is **Random Vibration**, set the following three options to `Yes` in Mechanical, under **Random Vibration** → **Analysis Settings** → **Output Controls**:
    >
    > - **Keep Modal Results**
    >
    > - **Calculate Velocity**
    >
    > - **Calculate Acceleration**
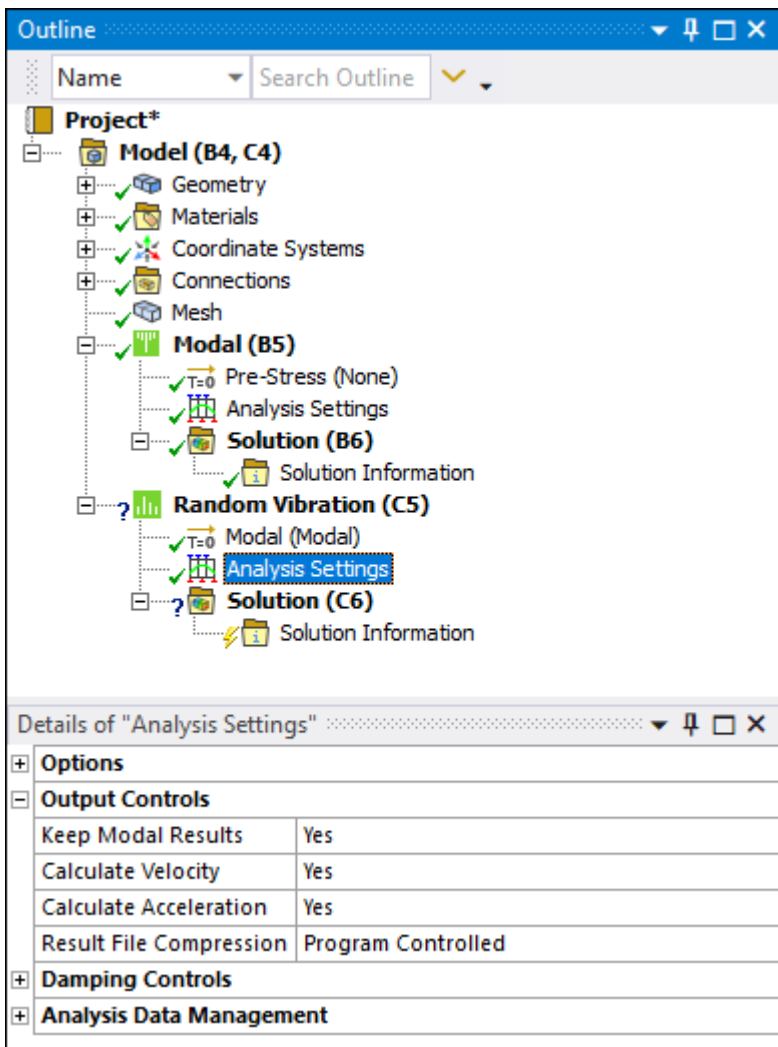
If these settings are not set correctly, the upstream Modal system's data is not transferred to the Sherlock (Post) system.

7. Right-click the **Setup** cell and select **Transfer to Mechanical System** from the context menu.

A Mechanical system corresponding to the analysis type selected in the previous step is added to the **Project Schematic**.
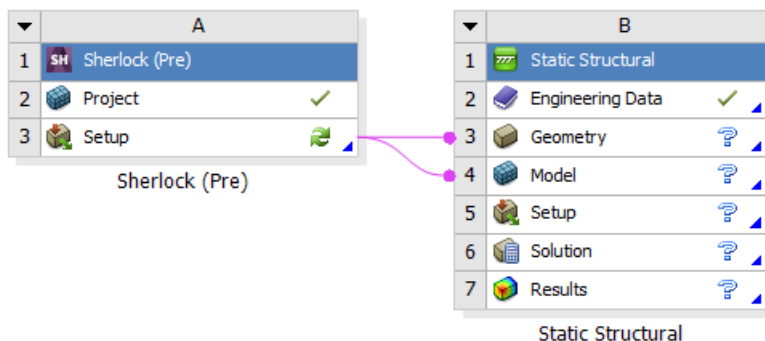
8. For multi-assembly projects, you can add other systems to the **Project Schematic** as required.

An example multi-assembly project is shown in the following image.
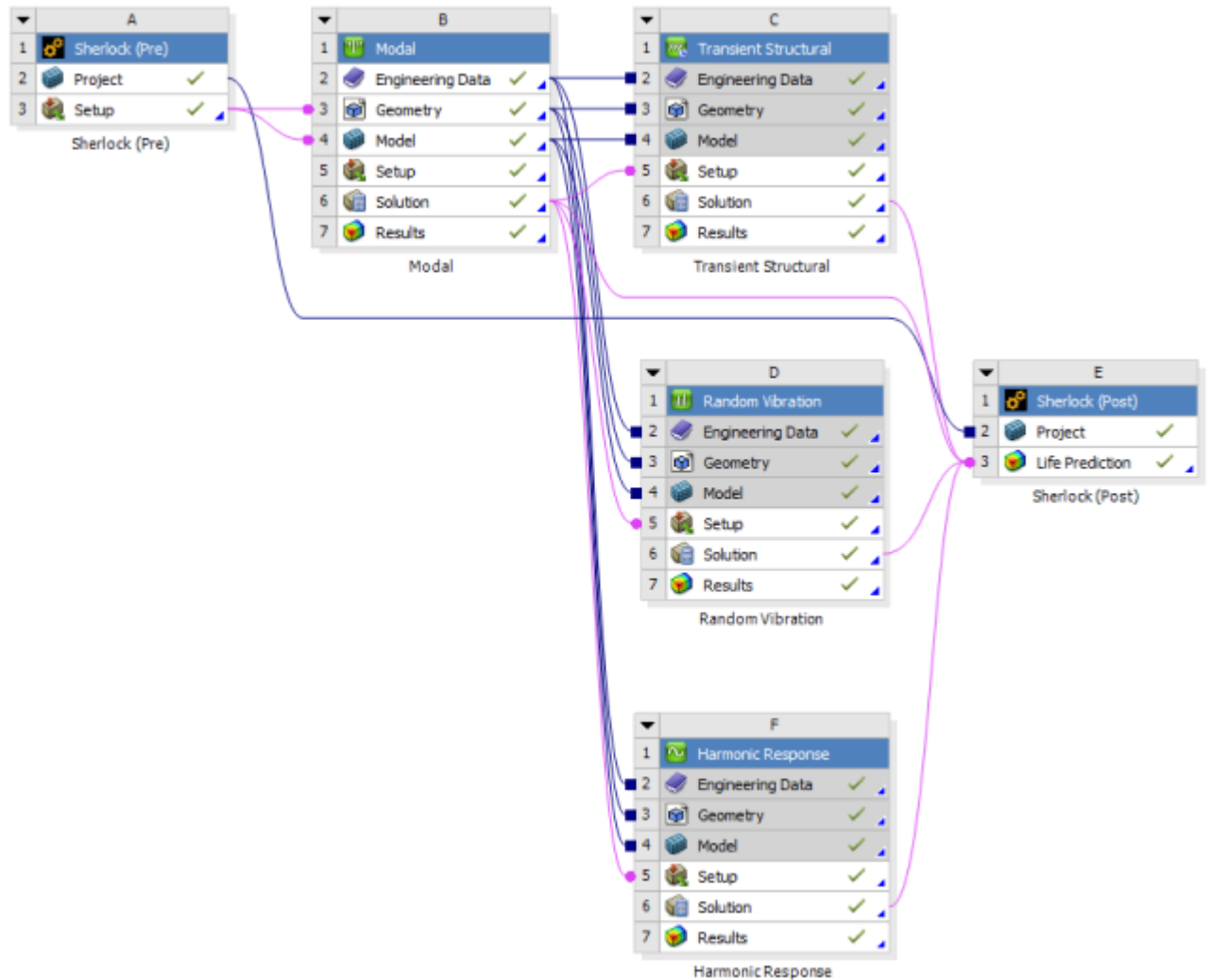
Note:

When you want to use the same base Sherlock project with different analysis or different boards, create the project in the **Sherlock (Pre)** system and then duplicate it. Use the **Properties** fields of each copy of the system to set up one analysis.

9. Drag a **Sherlock (Post)** system from the **Toolbox** and drop it onto the **Project** cell of the **Sherlock (Pre)** system
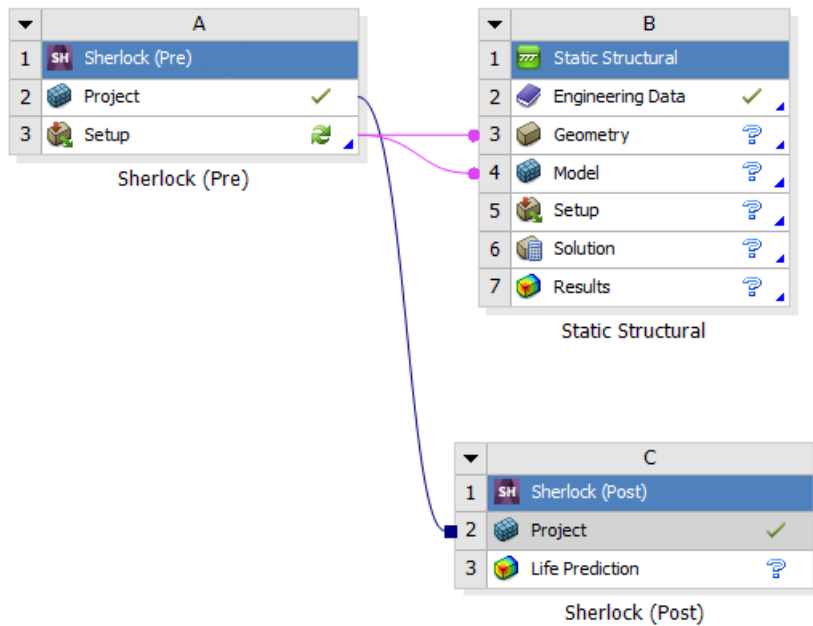
10. Drag the **Solution** cell of the Mechanical system onto the **Life Protection** cell of the **Sherlock (Post)** system.
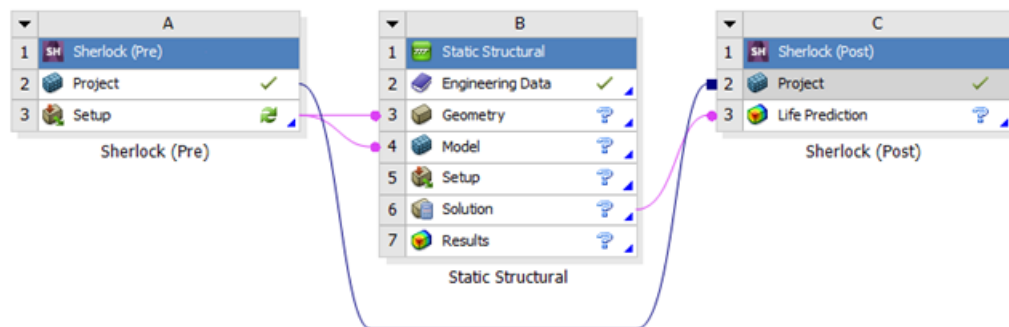
11. Right-click the **Setup** cell of the Sherlock (Pre) system and select **Update** from the context menu.

12. Right-click the **Geometry** cell of the Mechanical system and select **Update** from the context menu.

13. To view the Mechanical system geometry, right-click the **Model** cell and select **Edit** from the context menu or double-click the **Model** cell.

    Alternatively, you can complete this step using the **Setup** cell.

14. In the Mechanical application window, complete your analysis using the application's tools and features.

15. Right-click the **Project** cell of the Sherlock (Post) system and select **Update** from the context menu.

16. Right-click the **Life Prediction** cell of the Sherlock (Post) system and select **Update** from the context menu.

    The Sherlock GRPC console opens as the cell is updating.

17. Right-click the **Life Prediction** cell and select **Properties** from the context menu.

18. Select the appropriate Life Cycle events from the available lists.

	A	B
	Property	Value
1	Property	Value
2	General	
3	Component ID	Life Prediction
4	Directory Name	SHER-1
5	Update Condition Parameter (Beta)	None
6	ImportedResults	Item
8	Notes	
9	Notes	
10	Used Licenses	
11	Last Update Used Licenses	Ansys Sherlock
12	Environmental	
13	2 - Shopping Cart	None
14	3 - Earthquake	None
15	On The Road	
16	2 - Pothole	None
17	3 - Collision	None
18	5 - Harmonic Vibe	None
19	1 - Vibration	None
20	Default Life Cycle Events	
21	ICT Results File	ICT Analysis, Solution
22	Natural Frequency Results File	None

Properties of Schematic C3: [Life Prediction]

Note:

- The available Life Cycle events depend on what is created in the Sherlock project. They are also filtered by the analysis systems connected to the Sherlock (Post) system.

- If you have an older Workbench project created before 2022 R2, right-click the **Project** cell of the Sherlock (Pre) system and select **Edit** from the context menu to open Sherlock and pull in the new mappings.

- For more details on Sherlock Life Cycles, see Life Cycle Management in the *Sherlock User's Guide*.

19. To view the 2D or 3D results in Sherlock, right-click the **Life Prediction** cell and select one of the following options from the context menu:

   • **View 2D Results**

   • **View 3D Results**

   > **Note:**
   >
   > The **Sherlock 3D Viewer** may run slowly when analyzing large projects even if the computer has ample amounts of memory. This is likely a Java issue.
   >
   > If you encounter an error message that the **3D Viewer** has run out of memory, you can allocate more memory to the **3D Viewer** in the **Launcher Settings**. From Sherlock's **Main Menu**, click **Settings** → **Launcher**. Refer to Launcher Settings for additional information.

System Coupling

The **System Coupling** system launches the **System Coupling** tab, allowing you to model one-way and two-way multiphysics couplings for your project. To run a coupled analysis using System Coupling in Workbench, connect a **System Coupling** system to two coupling participant systems. The **System Coupling** system will synchronize and manage the participants' data transfers and the coupled solution.

> **Tip:**
>
> For detailed information on creating and running coupled analyses using the Workbench interface, see Using System Coupling in Workbench in the *System Coupling User's Guide*.

To work through a **System Coupling** system, perform the following steps:

1. Add two supported participant systems to the **Project Schematic** and complete their physics setups.

2. Add a **System Coupling** component system by dragging the system from the Toolbox and dropping it onto the **Project Schematic**.

3. Connect the System Coupling system to the other systems through their **Setup** or **Solution** cells.

   > **Note:**
   >
   > • The **Command Line Options** property located in the System Coupling **Solution** cell is for advanced users with a strong understanding of the appropriate command line arguments that can be used when starting System Coupling. For more information, see Command-Line Options in the *System Coupling Settings and Commands Reference*.
   >
   > • Once the systems are connected, you can export the files needed to run the coupled analysis in one of System Coupling's user interfaces. The export can be performed

> either before or after the coupling setup is completed in Workbench. For details, see Exporting a System Coupling Setup in the *System Coupling User's Guide*.

4. Open the **System Coupling** tab by double-clicking the **System Coupling** system's **Setup** cell. When asked is you want to read upstream data, click **Yes**.

   The **System Coupling** tab opens, with coupled analysis details shown in the **Outline** pane's tree view. To view or edit a given property, select it in the **Outline** to show its value in the **Properties** pane.

5. Complete the coupling setup, as follows:

   a. Add data transfers by multi selecting the participant coupling regions, right-clicking, and selecting **Create Data Transfer**.

   b. Specify data transfer details by selecting the data transfer and editing property values in the **Properties** pane.

   c. Specify execution controls by selecting a node under **Execution Controls** and editing property values in the **Properties** pane.

6. Once the setup is complete, run the coupled analysis by right-clicking the **Solution** node and selecting **Update**.

Turbo Setup

The **Turbo Setup** system launches Turbomachinery Data Setup, an application used to rapidly create the workflows needed for the analysis of centrifugal compressors.

The application offers no design calculation capability itself; it is purely a tool designed to facilitate the analysis of turbomachinery using the Vista TF and CFX applications.

For more information, see Turbo Setup in the *TurboSystem User's Guide*.

To work through a **Turbo Setup** system:

1. To add a **Turbo Setup** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open Turbomachinery Data Setup, double-click the **Turbo Setup** cell.

TurboGrid

The **TurboGrid** system launches Ansys TurboGrid, a tool that lets designers and analysts of rotating machinery create high-quality hexahedral meshes, while preserving the underlying geometry. These meshes are used in the Ansys workflow to solve complex blade passage problems.

For more information, see Ansys TurboGrid help.

To work through a **TurboGrid** system:

1. To add a **TurboGrid** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. Optionally, connect an upstream cell to the **Turbo Mesh** cell to provide the geometry data. To do this, right-click the **Turbo Mesh** cell and select **Transfer Data from New** from the context menu or drop the **TurboGrid** system on the **Geometry** cell of another system.

   If you do not complete this step, you can load geometry data using TurboGrid.

3. To open TurboGrid, double-click the **Turbo Mesh** cell.

Vista AFD

The **Vista AFD** system launches Vista Axial Fan Design, an application used for the preliminary design of axial fans. It creates axial fan geometry data for use in BladeGen or BladeEditor. It also provides estimates of the performance of the axial fan. It may be used to generate a preliminary fan design before moving rapidly to a full 3D geometry model and CFD analysis.

For more information, see Vista AFD help.

To work through a **Vista AFD** system:

1. To add a **Vista AFD** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To specify the solution parameters, generate a solution, and view the results of a meanline analysis, double-click the **Meanline** cell.

3. To perform the design (throughflow) calculation, double-click the **Design** cell.

4. Optionally, before creating a BladeGen or BladeEditor model, double-click the **Analysis** cell to perform an analysis calculation.

   This uses a similar throughflow method to the design calculation but simply analyses the design created in the previous step, rather than adjusting the geometry. A significant difference between the design and analysis results indicates a potentially flawed design.

Vista CCD and Vista CCD with CCM

The **Vista CCD** and **Vista CCD with CCM** systems launch Vista Centrifugal Compressor Design, an application used for the preliminary design of centrifugal compressors. It can be used in an iterative fashion to create a 1D design. The resulting geometry can be passed to BladeGen or BladeEditor. Vista CCD can be used to model an existing compressor and, if known, its measured performance at single operating points. An accurate 1D model can provide insight into the performance of the machine that goes beyond the test measurements.

For more information, see Vista CCD help.

To work through a **Vista CCD** system:

1. To add a **Vista CCD** or **Vista CCD with CCM** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To specify the solution parameters, generate a solution, and view the results, double-click the **Blade Design** cell.

3. For **Vista CCD with CCM** systems, to predict the overall performance of the design compressor state, double-click the **Performance Map** cell.

Vista CPD

The **Vista CPD** system launches Vista Centrifugal Pump Design, an application used for the preliminary design of pumps. It creates impeller geometry data for use in BladeEditor. It may be used to generate an optimized 1D pump impeller design before moving rapidly to a full 3D geometry model and CFD analysis.

For more information, see Vista CPD help.

To work through a **Vista CPD** system:

1. To add a **Vista CPD** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To specify the solution parameters, generate a solution, and view the results, double-click the **Blade Design** cell.

Vista RTD

The **Vista RTD** system launches Vista Radial Turbine Design, an application used for the preliminary design of radial inflow turbines. It can be used in an iterative fashion to create a 1D design. The resulting geometry can be passed to BladeGen, BladeEditor, and Vista TF. Vista RTD can also be used to model an existing turbine. An accurate 1D model can provide insight into the performance of the machine that goes beyond the test measurements.

For more information, see Vista RTD help.

To work through a **Vista RTD** system:

1. To add a **Vista RTD** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To specify the solution parameters, generate a solution, and view the results, double-click the **Blade Design** cell.

Vista TF

The **Vista TF** system allows you to rapidly evaluate radial blade rows (pumps, compressors and turbines) at the early stages of the design. The Vista TF program is a streamline curvature throughflow program for the analysis of any type of turbomachine, but has been developed in the first instance primarily as a tool for radial turbomachinery analysis.

For more information, see Vista TF help.

To work through a **Vista TF** system:

1. To add a **Vista TF** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open the cell properties and specify the solution parameters, double-click the **Setup** cell.

3. To generate a solution, double-click the **Solution** cell.

4. To view the results, double-click the **Results** cell.

Custom Systems

Workbench provides predefined custom templates for coupled systems, composed of multiple analysis systems with predefined data connections. Examples include one-way FSI and thermal-stress coupled analyses.

These templates are provided as a convenience; you can also manually create any of these systems using system drag-drop operations, context menu operations on cells, or individual cell linking. You can also create your own templates (p. 228) and add them to the **Toolbox** to easily build frequently-used projects.

To use a custom system, double-click the template. Drag and drop is not available.

The following custom systems are available:

- AM DED Process (p. 223)

- AM LPBF Inherent Strain (p. 224)

- AM LPBF Thermal-Structural (p. 224)

- FSI: Fluid Flow (Ansys CFX) > Static Structural (p. 225)

- FSI: Fluid Flow (Fluent) > Static Structural (p. 225)

- Pre-Stress Modal (p. 226)

- Random Vibration (p. 226)

- Response Spectrum (p. 227)

- Thermal-Stress (p. 227)

AM DED Process

This template creates two systems, a Transient Thermal analysis system and a Static Structural analysis system appropriate for use when performing a Directed Energy Deposition (DED) additive manufacturing simulation. The two systems share **Engineering Data**, **Geometry**, and **Model** cells.

Engineering Data is pre-populated with Ansys sample materials for additive manufacturing. Upon opening the Mechanical application, the project tree is pre-populated with the **AM Process for DED**, **Manual Clustering**, and **Build Settings** objects that are required for additive simulations, and loads the **DED Process Add-on**.

For more information, see Workflow Through the Project Tree in the *Workbench Additive Guide for DED Simulations*.

AM DED Thermal   AM DED Structural
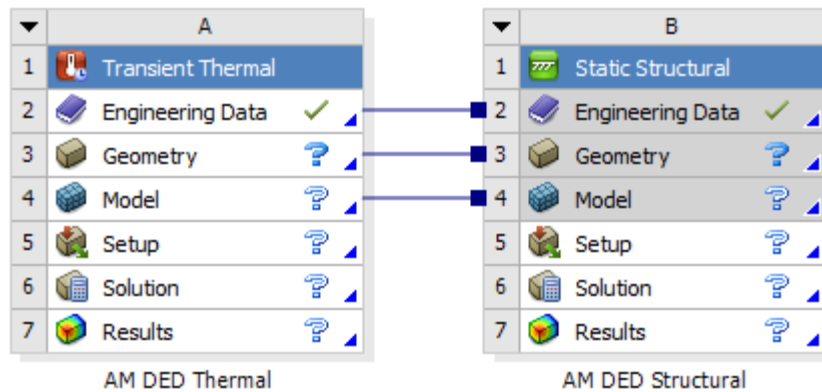
AM LPBF Inherent Strain

This template creates one Static Structural analysis system appropriate for use when performing a Laser Powder Bed Fusion (LPBF) additive manufacturing simulation assuming inherent strain. **Engineering Data** is pre-populated with Ansys sample materials for additive manufacturing. Upon opening the Mechanical application, the project tree is pre-populated with the **AM Process** and **Build Settings** objects that are required for additive simulations, and the **Inherent Strain** option (under AM Process, Build Settings) is set to Yes.

For more information, see Workflow Through the Project Tree in the *Workbench Additive Manufacturing Analysis Guide*.

AM LPBF Inherent Strain
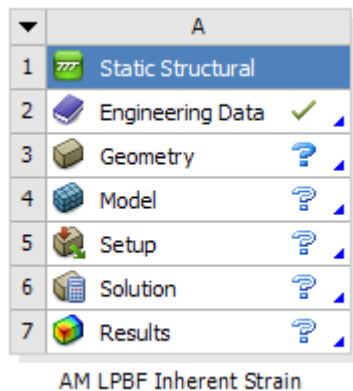
AM LPBF Thermal-Structural

This template creates two systems, a Transient Thermal analysis system and a Static Structural analysis system appropriate for use when performing a Laser Powder Bed Fusion (LPBF) additive manufacturing simulation. The two systems share **Engineering Data**, **Geometry**, and **Model** cells, and the **Solution** cell in the Transient Thermal system provides temperature load data to the **Setup** cell in the Static Structural system.

Engineering Data is pre-populated with Ansys sample materials for additive manufacturing. Upon opening the Mechanical application, the project tree is pre-populated with the **AM Process** and **Build**

Settings objects that are required for additive simulations, and the **Inherent Strain** option (under AM Process, Build Settings) is set to No.

For more information, see Workflow Through the Project Tree in the *Workbench Additive Manufacturing Analysis Guide*.

AM LPBF Thermal   AM LPBF Structural
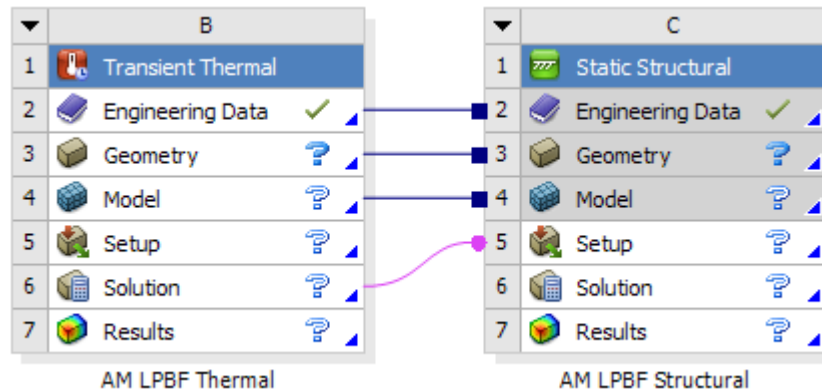
FSI: Fluid Flow (Ansys CFX) > Static Structural

This template performs analyses that couple the physics of Fluid Flow (Ansys CFX) and Static Structural analyses with results provided (one-way) from the former system to the latter. When this template is used, a Fluid Flow (CFX) (p. 84) analysis system and a Static Structural (p. 98) analysis system are automatically created. The two systems share a single geometry, and the fluid-flow **Solution** cell provides data that are treated as an Imported Load in the static structural **Setup** cell.
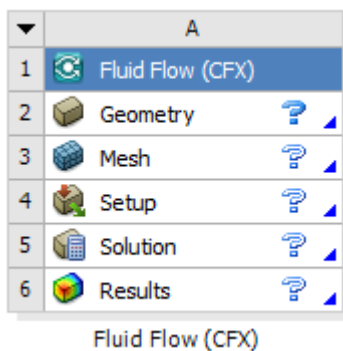
Fluid Flow (CFX)

Similar coupled systems may also be manually created between Fluid Flow (Ansys CFX) analysis and any of the following analysis systems:

• Transient Structural (p. 101)

• Steady-State Thermal (p. 99)

• Transient Thermal (p. 101)

FSI: Fluid Flow (Fluent) > Static Structural

This template creates two systems, a Fluid Flow (Fluent) analysis system and a Static Structural analysis system. The **Geometry** cells for the two systems share a single geometry, and the **Solution** cell

in the Fluid Flow (Fluent) system provides pressure load data to the **Setup** cell in the Static Structural system.

Fluid Flow (Fluent)
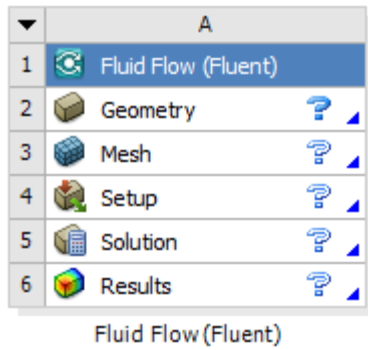
Pre-Stress Modal

This template creates two systems, a Static Structural analysis system that transfers data into a Modal analysis system. The two systems share **Engineering Data**, **Geometry**, and **Model** cells. The Static Structural solution provides the necessary solver files as input to the Modal **Setup** cell.

For more information, see Pre-Stress in the *Mechanical Object Reference*.

Static Structural   Modal
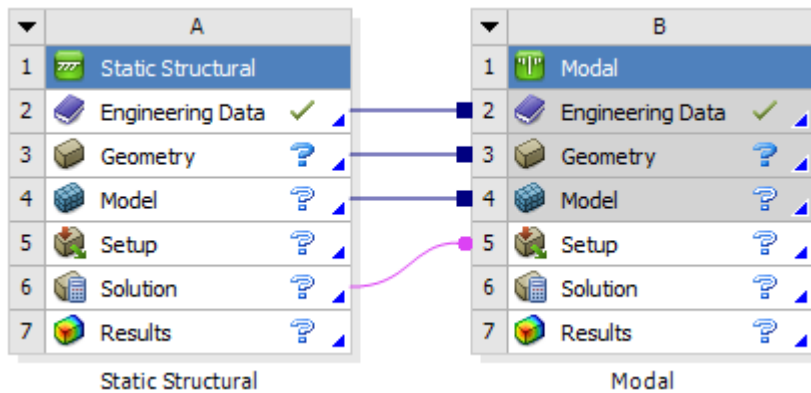
Random Vibration

This template creates two systems, a Modal analysis system that transfers data into a Random Vibration analysis system. The two systems share **Engineering Data**, **Geometry**, and **Model** cells. The Modal solution provides the necessary solver files as input to the Random Vibration **Setup** cell.

For more information, see Random Vibration Analysis in the *Mechanical User's Guide*.

226
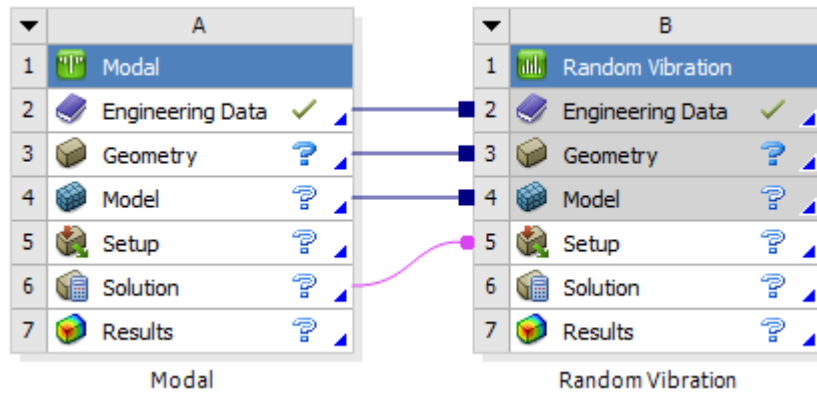
Response Spectrum

This template creates two systems, a Modal analysis system that transfers data into a Response Spectrum analysis system. The two systems share **Engineering Data**, **Geometry**, and **Model** cells. The Modal solution provides the necessary solver files as input to the Response Spectrum **Setup** cell.

For more information, see Response Spectrum Analysis in the *Mechanical User's Guide*.
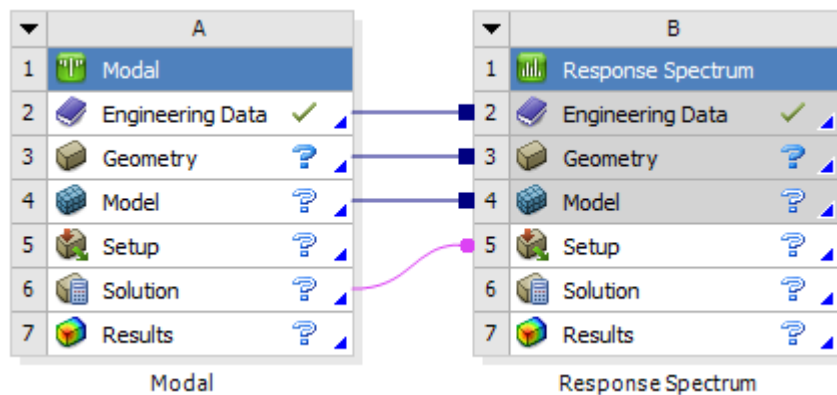
Thermal-Stress

This template creates two systems, a Steady-State Thermal analysis system that transfers data into a Static Structural analysis system. The two systems share **Engineering Data**, **Geometry**, and **Model** cells. The Steady-State Thermal solution provides temperature input to the Static Structural **Setup** cell.

For more information, see Steady-State Thermal Analysis in the *Mechanical User's Guide* and Initial Temperature in the *Mechanical Object Reference*.

Steady-State Thermal   Static Structural
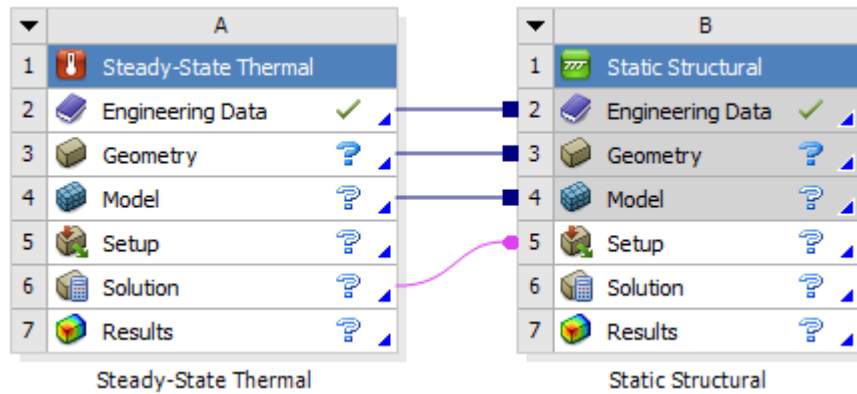
Creating a Custom System

You can create a custom system and add it to the **Toolbox** to easily build frequently-used projects.

To create a custom system:

1. Add systems to the **Project Schematic**.

2. Right-click the white space on the **Project Schematic** and select **Add to Custom** from the context menu.

3. In the **Name** field, type a name for the custom system.

4. Click **OK**.

   The new custom template appears in the **Toolbox** under **Custom Systems**.

   To delete the template, right-click the template name and select **Delete** from the context menu.

This animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
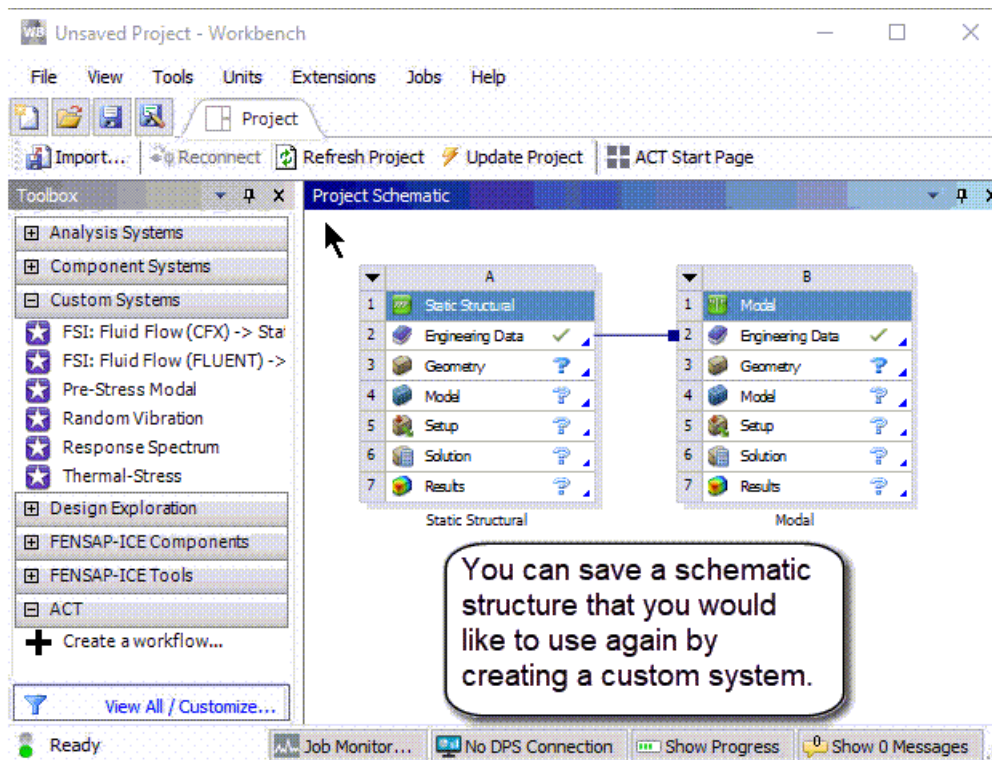
Design Exploration

DesignXplorer systems can be added beneath the **Parameter Set** bar, allowing you to perform in-depth Design Exploration studies.

DesignXplorer is a powerful approach for designing and understanding the analysis response of parts and assemblies. It uses a deterministic method based on Design of Experiments (DOE) and various optimization methods, with parameters as its fundamental components. These parameters can come from any supported system, DesignModeler, and various CAD systems. Responses can be studied, quantified, and graphed. Using a Goal Driven Optimization method, the deterministic method can obtain a multiplicity of design points. You can explore the calculated Response Surface and generate design points directly from the surface or transfer data from other analysis systems or components to a Direct Optimization system.

The following Design Exploration systems are available in Workbench:

- 3D ROM

- Direct Optimization

- Parameters Correlation

- Response Surface

- Response Surface Optimization

- Six Sigma Analysis

These systems are available only if you have installed the Ansys DesignXplorer product and have an appropriate license.

For an overview of these systems and links to more information about them, see What is Design Exploration?

Submitting Solutions

You can use the **Solution Process** properties of a system's **Solution** cell to control where you run the solution. You can choose from the update options listed here; the defaults for these options are determined by the settings specified in the **Solution Process** user preferences (p. 423).

To submit solutions:

1. Right-click the **Solution** cell of a system and select **Properties** from the context menu.

2. In the **Properties** pane, from the **Update Option** drop-down list, select from the following options:

    *   **Run in Foreground**: Runs the solution within the current Workbench session. This option is appropriate for quick-running solutions that fit within the resources of your workstation. This option is also the most robust as it is not possible to make changes that impact the current solution. When a solution is executing in the foreground, you cannot change or save the project, but you can interrupt or stop the solution.

        For Polyflow and CFX component or analysis systems, you can switch an update in progress on the local machine to background mode by right-clicking the **Solution** cell and selecting **Switch Active Solution to Background**.

    *   **Run in Background**: Runs the solution in the background on the local machine. This option is appropriate for solutions that fit within the resources of your workstation but will take longer to execute. When a solution runs in the background, the cell enters the Pending state (p. 449) and you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell is not in an up-to-date state when the solution completes.

        For Mechanical APDL, the **Download Distributed Files** property is not available and does appear in the **General Property** table during foreground and background updates.

        > **Note:**
        >
        > If you choose the **Run in Background** option for multiple solution components or for one solution component with multiple design points, your project and design point updates run multiple instances of the solver at the same time. If you are using the **Share single license between applications when possible** option in the license preferences, only one of the solver runs can succeed. The others fail because they cannot access the single license. There are two workarounds for this problem:
        >
        > – Change the license preference to be **Use a separate license for each application**. This preference allocates one license for each running solver.

       – Instead of using the **Run in Background** option, use the **Submit to Remote Solve Manager** option and select the **Local** RSM queue. This is provided that a basic ANSYS Cluster (ARC) has been configured to restrict the number of cores that can be used for a job. For more information, see Overview of an Ansys RSM Cluster (ARC) Configuration and Setting the Maximum Number of Cores to be Used on an Execution Node in the *Remote Solve Manager User's Guide*.

- **Use application default** (Mechanical application only): Runs the solution using the solver settings specified in the Mechanical application.

- **Submit to Remote Solve Manager**: Runs the solution in the background by submitting the solution to RSM. Use this option for long-running solutions that do not fit within your workstation's resources. When you submit a solution, RSM can send it to a remote compute cluster, or to a Cloud portal. RSM can also submit jobs to the local machine to allow the queuing of solutions on your workstation. When a solution is submitted to RSM, the **Solution** cell enters the Pending state (p. 449), similar to the **Run in Background** option.

   For more information on submitting solutions to RSM, see Submitting Solutions to Remote Solve Manager (p. 321).

3. To save the project, from the menu bar, select **File** → **Save**.

   If you choose not to save the project and any cells are configured to use RSM or run in the background, you must change the solution process settings of those cells to run in the foreground before you can update.

4. To start the solution, select the appropriate option from the application, or right-click the **Solution** cell and select **Update** from the context menu.

Opening Mechanical without a License (Read-Only Mode)

You can open the Mechanical application without consuming a license. Two common applications for using Mechanical in read-only mode are:

- Retrieving data during a Remote Solve Manager (RSM) run in order to monitor its progress.

- Reviewing results of an already solved project.

This option is not available when the cell state is Refresh Required (p. 449) or worse.

To open Mechanical without a license:

1. Right-click one of the following cells in a Mechanical system:

   - **Model**

   - **Section Data**

   - **Setup**

- **Solution**

- **Results**

2.   Select **Edit in Read-only Mode** from the context menu.

Updating Systems

Once the systems are placed on the **Project Schematic**, you can update them in various ways.

Related topics:

Naming and Renaming Systems

Duplicating Systems

Moving Systems

Deleting Systems

Replacing Systems

Creating Project, System, and Cell Notes

Updating and Refreshing System Components

Editing Shared Data

Clearing Generated Data

Resetting Data

Naming and Renaming Systems

In general, it is good practice to give each system a descriptive name that is meaningful for you and that indicates the details of the system. Workbench enables you to specify a name for each system, either initially when the system is added to your project, or at any time afterward.

Related Topics:

Naming Systems

Renaming Systems

Naming Systems

When a new system is added to the **Project Schematic,** the default name of the system is in focus (highlighted), and editable, as shown below.
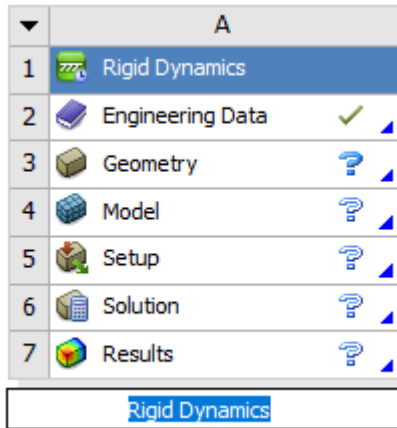
To name the system, select from one of the following options:

- To accept the default name for the system (the default name is usually the same as the system type), press **Enter** or select any other action in the user interface.

- To enter a name on a single line of text, type the new name and press **Enter**.
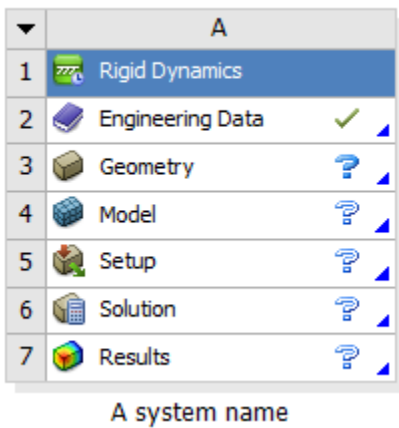
A system name

- To enter a name that spans multiple lines of text:

  1. Type the first line of text.

  2. To start a new line, press **Ctrl**+**Enter**.

  3. Repeat the previous two steps until the entire name is shown in the text field.

  4. To save the name, press **Enter**.
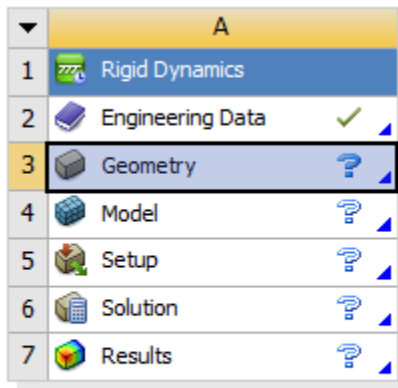
This is a system name
spanning multiple lines

After entering a new name or accepting the default name, the focus moves to the system cell requiring attention first (see Understanding Cell States (p. 449)). By applying focus in this manner, Workbench draws your attention to the cell where you will most likely begin working with your system (see Working Through a System (p. 67)).

Renaming Systems

You can change the system name again after it has been named.

To rename a system:

1. Double-click the system name or right-click the system header and select **Rename** from the context menu.
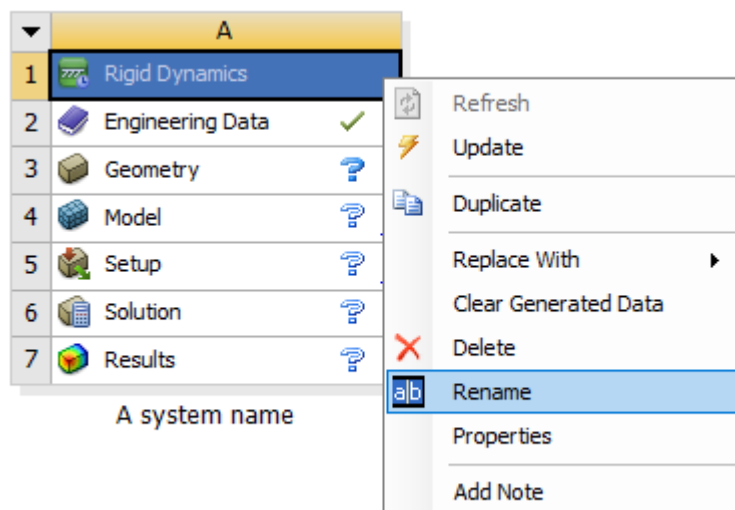
A system name

2. Type the new single or multiple line name.

3. Press **Enter**.

Duplicating Systems

Duplicating a system creates a copy of the selected system. All data associated with unshared cells in the system is copied to, or shared with, the duplicate system.

Related Topics:

Duplicating a Single System

Duplicating Multiple Systems

Duplicating a Single System

You can duplicate a single system so that the cells can either remain independent of the original system, or where the cell data is shared.

Creating Independent Duplicate Systems

To create a duplicate system where all cells can be edited independently of the original system, right-click the system header and select **Duplicate** from the context menu.

> **Note:**
>
> When a Mechanical system containing a **Results** cell is duplicated, the results are *not* copied to the new system.

The following Show-Me Animations are presented as animated GIFs in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
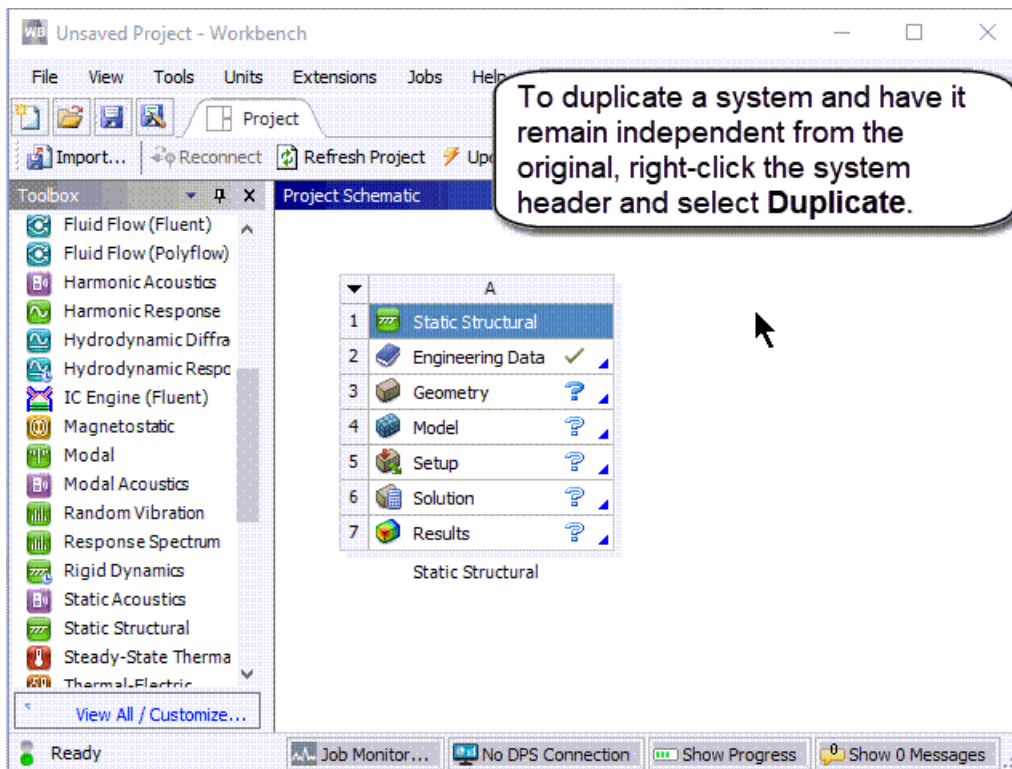
Creating Shared Duplicate Systems

To create a duplicate system where cell data is shared, right-click a cell and select **Duplicate** from the context menu.

All cells above the one selected for duplication are shared. The cell selected for duplication can be edited independently. For example:

- If you select **Duplicate** from the **Geometry** cell, the **Engineering Data** cell is shared, allowing you to edit the **Geometry** cell in the duplicate system to investigate a geometric alternative. Data from the **Model** cell and below is copied from the original system and can be modified independently.
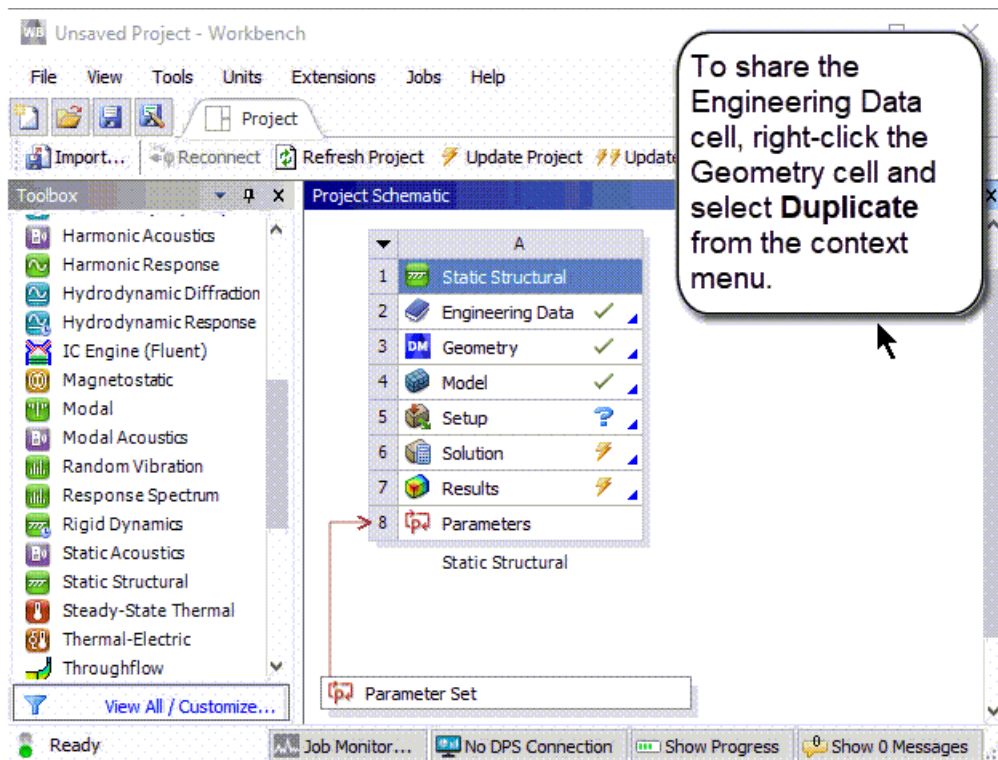
- If you select **Duplicate** from the **Model** cell, the **Engineering Data** and **Geometry** cells are shared, allowing you to edit the **Model** cell in the duplicate system to investigate an alternative modeling approach. All data in the **Setup** cell and below is copied from the original system and can be modified independently.

- If you select **Duplicate** from the **Setup** cell, the **Engineering Data**, **Geometry** and **Model** cells are shared, allowing you to edit the **Setup** cell in the duplicate system to investigate alternate loads and constraints. All data in the cells below **Setup** is copied from the original system and can be modified independently.

Duplicating Multiple Systems

You can duplicate more than one system at a time.

To duplicate multiple systems:

1. Select all systems to be duplicated using one of the following methods:

    - Box-select by clicking and dragging the selection box over the required systems.

    - Press **Ctrl** and click the required systems.

2. Right-click the header of one of the selected systems.

3. Select **Duplicate** from the context menu.

4. If you are duplicating systems linked to an upstream system by shared and/or data transfer connections, select from the following options:

- To keep the duplicate systems linked to the upstream systems, click **Yes**.

- To delete the upstream connections from the duplicated systems, click **No**.

Note:

System Coupling and TurboSystem's Performance Map systems do not work with multiple system duplication.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
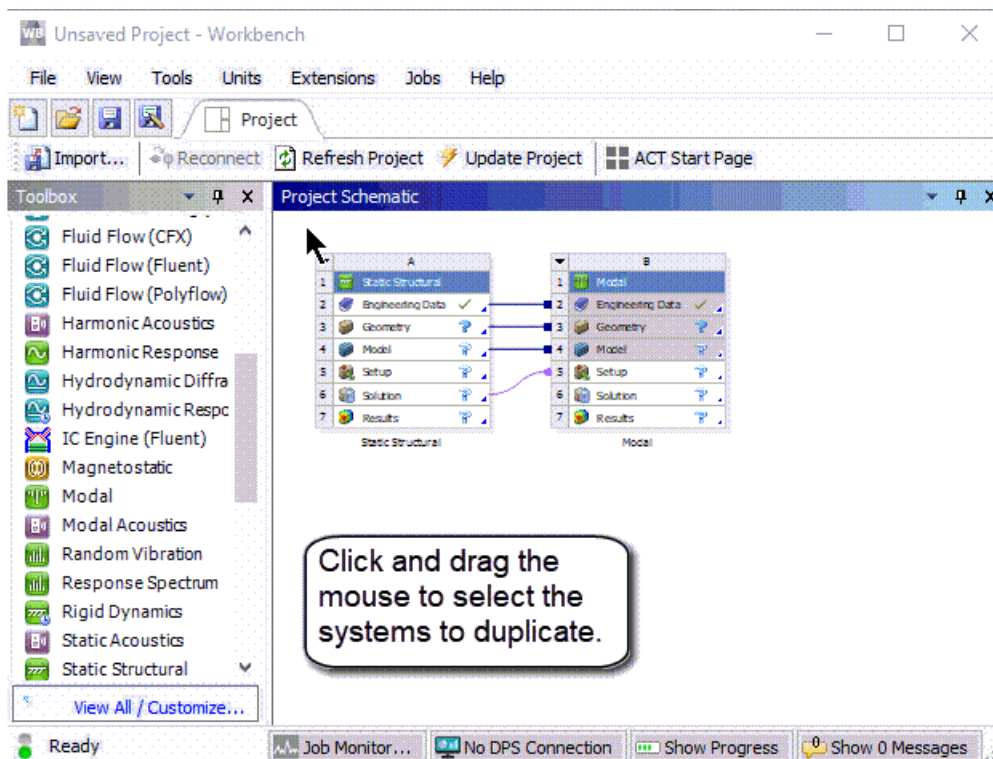
Moving Systems

You can move an existing system to another position on the **Project Schematic**.

To move a system:

1. Click the header cell of the system.

2. Drag the system to the new location.

   The preview indicates possible target locations for the system.

Deleting Systems

You can delete a single system or multiple systems on the **Project Schematic**.

Related Topics:

Deleting a Single System

Deleting Multiple Systems

Deleting a Single System

You can delete a system from the **Project Schematic**.

To delete a single system:

1. Right-click the system header and select **Delete** from the context menu.

2. To acknowledge the message and complete the deletion process, click **OK**.

Deleting Multiple Systems

You can delete more than one system at a time.

To delete multiple systems:

1. Select all systems to be deleted using one of the following methods:

   • Box-select by clicking and dragging the selection box over the required systems.

   • Press **Ctrl** and click the required systems.

2. Right-click the header of one of the selected systems.

3. Select **Delete** from the context menu.

4. To acknowledge the message and complete the deletion process, click **OK**.

   Note:

   If you delete an upstream system that is providing equivalent inputs, the system is deleted and its downstream systems automatically regenerate their deleted containers.

Replacing Systems

You can replace an existing system with different type of system. This functionality is available only from systems that use the Mechanical application.

> **Note:**
>
> If you use **Replace With** to replace a Mechanical system that has a **Section Data** cell with another Mechanical system, the new environment is added and the original Mechanical environment (including boundary conditions and results) is deleted.

To replace a system:

1. Right-click the system header and select **Replace With** from the context menu.

2. From the list of all system types that are available to replace the existing system, select a new system.

> **Note:**
>
> The units setting specified in an existing system is not maintained in the replacement system. In the replacement system, you must specify the units setting you want to use.

Creating Project, System, and Cell Notes

Notes allow you to annotate your projects, systems, and system cells. There is no limit to the amount of text you can type into a note; as you type, the pane increases in length. The content of notes is also included in project reports.

To create notes:

1. Right-click the area of the project you want to add a note to and select **Add Note** from the context menu.

   - To open the context menu for the **Project Schematic**, right-click the white space of the pane.

   - To open the context menu for the system, right-click the system header.

   - To open the context menu for a cell, right-click the cell.

2. In the yellow pane, type the note text.

3. To close the note, click outside the pane.

4. To view the note:

   - Right-click the area of the project containing the note and select **Properties** from the context menu.

   - Click the green triangle in the upper right corner of a system or cell to open the note.

Static Structural
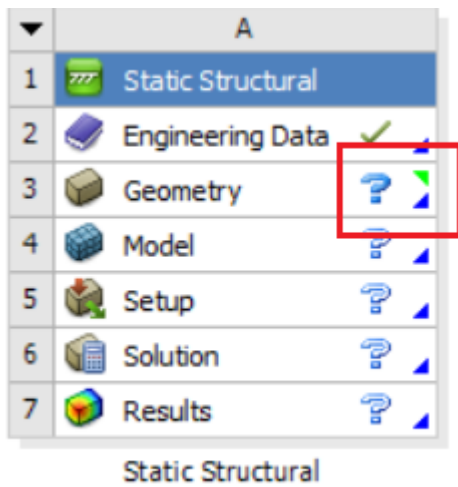
5. To edit the note text:

- Right-click the area of the project containing the note and select **Edit Note** from the context menu.

- Click the green triangle in the upper right corner of a system or cell to open the note.

Static Structural
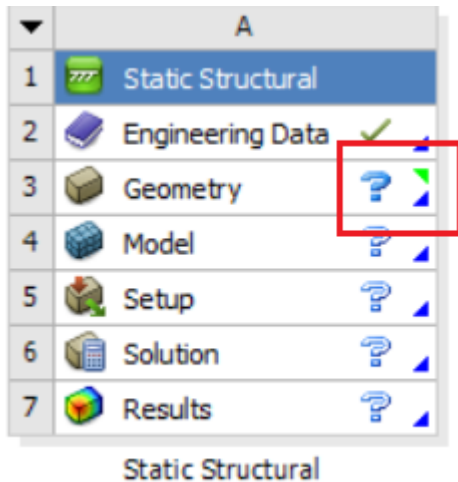
6. To delete a note, open the note in edit mode and delete all text from the yellow pane.

When you click outside of the yellow pane to close it, the note is deleted.

Updating and Refreshing System Components

Updating system components refreshes input data and generates required output data when one or more components of the project have been changed in some way.

242

Refreshing system components reads in all modified upstream data but does not necessarily regenerate the outputs of the component itself.

You can update or refresh an entire project, a system, or a cell within a system.

Note:

In some cases, the update, update upstream components, and refresh operations may be available while other operations, such as clear generated data, reset, delete, and duplicate, are still in progress. If the operation is in progress on a separate system, you can apply the Update, update upstream components, or refresh operation to the current system. However, if the operation is in progress on either the current system or a system connected to it, you cannot initiate the new operation until the in-progress operation has completed.

Related Topics:

Updating a Project

Updating a System

Updating a System Cell

Updating Upstream Components

Interrupting an Update Process

Refreshing System Components

Updating a Project

To update all cells in the project that are in an Update Required (p. 449) state, update the entire project.

If a project contains multiple design points, only the current design point is updated.

If a project contains parametric (DesignXplorer) systems, updating a parametric system updates the design points within that system, but does not update the design points in the parameter set.

Parametric systems are not updated when a project update is submitted to Remote Solve Manager (RSM). For more information, see Submitting Workbench Project Updates to Remote Solve Manager (p. 325).

To update a project:

1. To save the project, from the menu bar, select **File** → **Save**.

    If you choose not to save the project and any cells are configured to use RSM or run in the background, you must change the solution process settings of those cells to run in the foreground before you can update.

2. To update the project:

    • From the menu bar select **Tools** → **Update Project**.

    • On the **Project** tab toolbar click **Update Project**.

- Right-click the white space on the **Project Schematic** and select **Update Project** from the context menu.

Updating a System

You can update a system so that all cells within that system contain the most current data.

If a system contains a coupled cell, the update option on the coupled cell is disabled. To update the system that contains a coupled cell, you must update the coupled system so that both the coupled cell and all cells dependent on the coupled cell can be updated.

To update a system:

1. To save the project, from the menu bar, select **File → Save**.

    If you choose not to save the project and any cells are configured to use RSM or run in the background, you must change the solution process settings of those cells to run in the foreground before you can update.

2. To update the system, right-click the system header and select **Update** from the context menu.

Updating a System Cell

You can update a cell within a system to ensure it contains the most current data. When you update the cell, any upstream cells upon which the cell is dependent are also updated. You cannot update a cell if another update is already running on the cell or on any of the upstream cells.

If you update a cell in a Refresh Required (p. 449) state, the refresh operation is performed before the update operation.

To update a system cell:

1. To save the project, from the menu bar, select **File → Save**.

    If you choose not to save the project and any cells are configured to use RSM or run in the background, you must change the solution process settings of those cells to run in the foreground before you can update.

2. To update the cell, right-click it and select **Update** from the context menu.

Updating Upstream Components

Updating upstream components ensures that everything upstream of the selected cell is up-to- date so that the cell is ready to edit or update, without updating the cell itself.

The advantage to updating only the upstream components of a cell is that you are alerted to any errors or potential effects on downstream cells and can make necessary adjustments before launching a full update. This option is especially useful when you have a complex system and the cell update could require a significant time investment and/or computer resources.

To update upstream components:

1. To save the project, from the menu bar, select **File** → **Save**.

    If you choose not to save the project and any cells are configured to use RSM or run in the background, you must change the solution process settings of those cells to run in the foreground before you can update.

2. Right-click the required cell and select **Update Upstream Components** from the context menu.

Interrupting an Update Process

After starting an update, you can interrupt, or cancel the process. Be aware that not all processes can be interrupted and some processes that are interruptible may have periods where they cannot be interrupted. Because some processes can stop cleanly only at certain checkpoints, the delay between the interrupt request and the actual interruption can sometimes be lengthy.

Note:

The only Ansys Mechanical systems that can be interrupted are **Static Structural** and **Transient Structural**.

To interrupt an update process:

1. To open the **Progress** pane, select **View** → **Progress** from the menu bar.

2. To the right of the process you want to interrupt, click .

    The following dialog box displays:
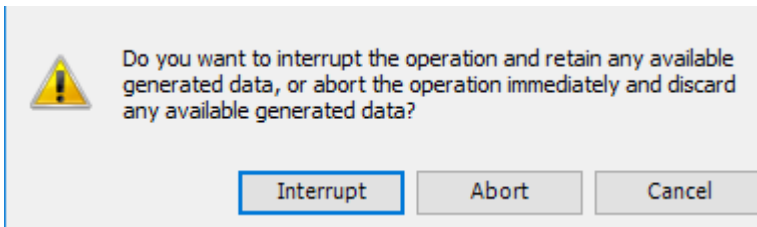
3. Select from the following options:

    *   **Interrupt**: Stops the update at the next point where data can be safely stored for later use.

    *   **Abort**: Stops the update immediately without checking if data associated with the current action can be stored.

    When an update is interrupted, the state of the **Solution** cell becomes Interrupted, Update Required (p. 449).

Refreshing System Components

Refreshing system components reads in all modified upstream data but does not necessarily regenerate the outputs of the component itself. For example, if the geometry changed, placing the Mesh cell in

a Refresh Required (p. 449) state, refreshing the Mesh cell updates the geometry without generating a new mesh (which could potentially be a lengthy operation).

To refresh system components:

- To refresh all cells in the project that are in a Refresh Required state, from the menu bar select **Tools → Refresh Project**, or on the **Project** tab toolbar click **Refresh Project**.

- To refresh a system, right-click the system header and select **Refresh** from the context menu.

- To refresh a cell, right-click the cell and select **Refresh** from the context menu.

Editing Shared Data

Links that are blue and drawn with a square terminator indicate that only one instance of the data exists, and it is shared between the connected systems. In order to edit the details of that data, you must edit the cell on the upstream system connected by these links.

In the example shown in the figure below, the **Geometry** cell from system A is shared with the **Geometry** cell in system B, which is in turn shared with the **Geometry** cell of system C. To edit the geometry for any of these systems, you must initiate the edit operation from the **Geometry** cell in system A. You can either double-click the cell or right-click the cell and select **Edit** from the context menu.
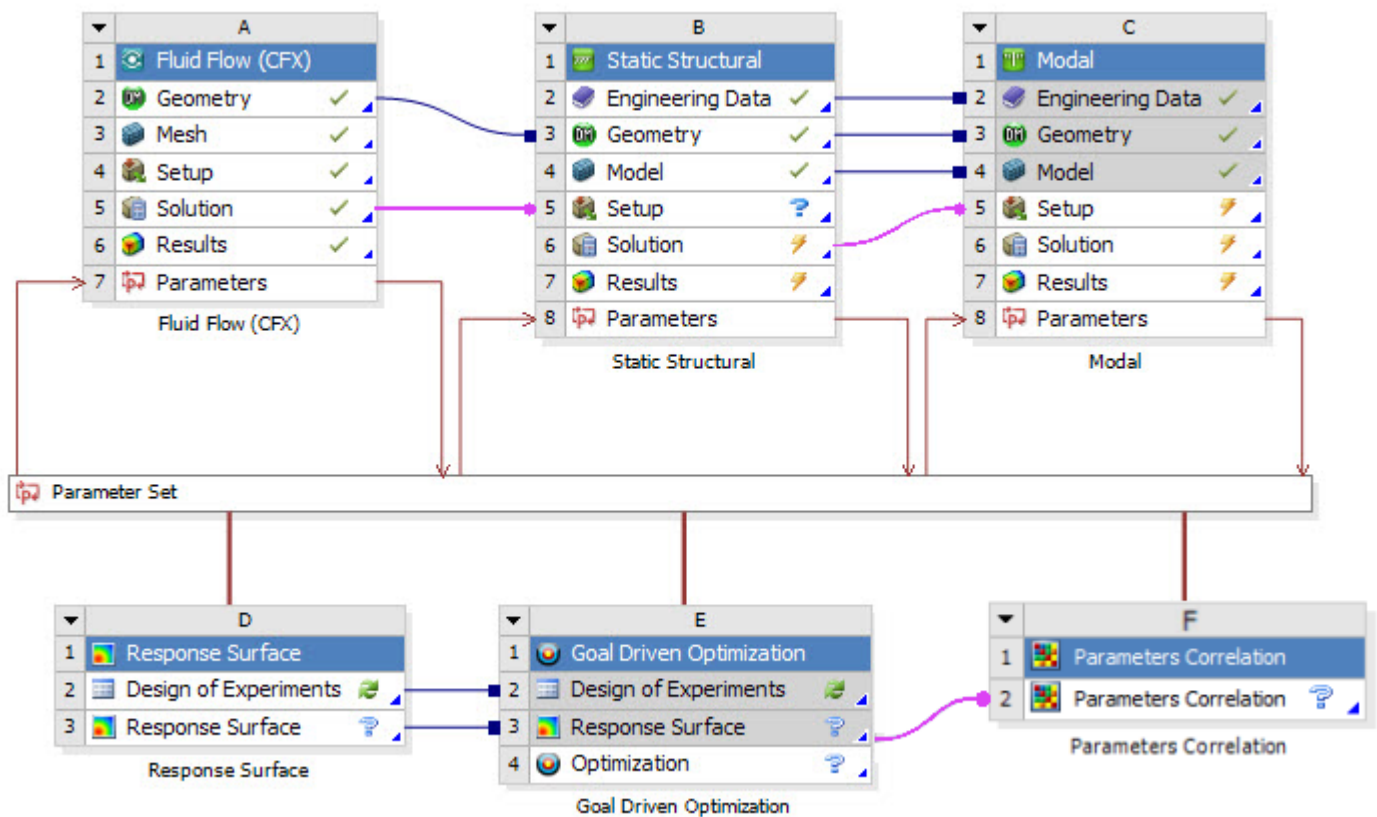
Clearing Generated Data

Clearing generated data removes or erases any data that the cell has generated, or is prepared to generate and store, if any such data is present. This data includes mesh, input files, solver files, and cached solution data. This operation may alter the state of the current cell and cells downstream from the selected cell.

Important:

For some systems, this operation results in specific behavior depending on the selected cell and in some cases specific system data (such as existing links). Refer to the following table for specific actions.

Cell	Data	Behavior Description
Mechanical systems		
Model		Deletes mesh, input files, and solver files. Cleans results.
Setup		Deletes input and solver files. Cleans results.
Solution		Deletes solver files. Cleans results. Input files (such as `ds.dat`, `CARep.xml`) are left intact.
Results		Cleans any solved results. Solver files are left intact.
Setup	Imported Temperature: One-way FSI input	Cleans imported load and deletes transfer xml file. User specifications (such as scoping) are intact.
Setup	Imported Temperature: Thermal stress	Cleans imported load and deletes transfer xml file. User specifications (such as scoping) are intact.
Model	Link to CFX	Deletes `SYS.cmdb`.
Model	Link to Mechanical APDL	Deletes `SYS.cdb`.
Setup	Link to Mechanical APDL	No additional action required (`ds.dat` is the natural output from the Setup cell).
Solution	Link to Mechanical APDL	No additional action required (`file.rst` is the natural output from the Solution cell).
Mechanical APDL		
Analysis		Deletes Ansys-generated files (created during Update or Edit). Files added using input links or **Add *xxx* File** are left intact.
Fluent		
Solution	Internal, link to initialization data	Closes the session without saving data. Deletes all files currently associated with the cell (latest available solution data). Any schematic input links remain intact.

To clear generated data:

1. Right-click the cell to clear and select **Clear Generated Data** from the context menu.

2. To complete the operation, click **OK**.

Resetting Data

Resetting data removes or erases both input and output data to the cell and sets the cell state back to the default. Any reference files are removed.

Reset may alter the state of cells downstream from the selected cell. If you have two systems that share cells, reset is not available from the cells that are shared, only from the source (the cell that is being shared from).

Reset clears all solution state data and cache data for all design points in the project.

> **Important:**
>
> For some systems, this operation results in specific behavior depending on the selected cell and in some cases specific system data (such as existing links). Refer to the following table for specific actions.

Cell	Data	Behavior Description
Mechanical systems		
Geometry		Clears the geometry source and resets geometry properties to the defaults.
Model		Closes the Mechanical session (if open) and deletes the `.mechdb` from disk. The system is in a state as if the geometry was never attached.
Setup		Deletes any objects under the Environment (such as loads and supports).
Solution		Sets the values in *Analysis Settings* back to the defaults.
Results		Deletes any results, probes, or post tools from Mechanical.
Setup	Imported Temperature: One-way FSI input	Deletes the Import Load from Mechanical and also deletes CFD result file from disk. State on Setup cell goes to Refresh Required.
Setup	Imported Temperature: Thermal stress	No additional action required.
Model	Link to CFX	No additional action required.
Model	Link to Mechanical APDL	No additional action required.
Setup	Link to Mechanical APDL	No additional action required.

Cell	Data	Behavior Description
Solution	Link to Mechanical APDL	No additional action required.
Mechanical APDL		
Analysis		Deletes all files in the Mechanical APDL `System` directory and resets any properties (as set in the **Properties** pane) back to the default. Any schematic input links remain intact and the needed files are copied back into the system upon Refresh. Any manually- added files (**Add Input/Reference**) are deleted and removed from the **Outline** pane.
Fluent		
Mesh		When the **Use Workflow** option enabled, a Reset on the Mesh cell removes the generated mesh (if any) and resets the workflow completely. The default template for the workflow (specified in the Mesh cell) is then initialized and all the workflow task states remain as Update Required. Since the workflow state is stored in a mesh file, the mesh file is not removed, it is present and is registered with the Mesh cell even after the Reset operation.
Setup	Link to imported case and other input files, settings, and possibly other input files	Closes the Fluent session without saving data. Deletes all internal files. Links to imported files are deleted (but the file is not deleted.) If Mesh is coming from an upstream simulation Mesh cell, the file is unregistered and might get deleted if the upstream Mesh cell no longer refers to it. All associated input parameters are deleted. Launcher settings are set to the default value.
Solution	Internal, link to initialization data	Closes the session without saving data. Deletes all files currently associated with the cell (the latest available solution data). Any schematic input links remain intact. Any imported initial solution data file is unregistered. (Only the link is removed; the file is not deleted.) All associated output parameters are deleted. Launcher settings are set to the default values.

To reset data:

1. Right-click the cell to reset and select **Reset** from the context menu.

2. To complete the operation, click **OK**.

Using Journals and Scripts

Workbench offers the ability to record the actions you perform in the user interface, also known as *journaling*. Journals are recorded as Python-based scripts. You can modify these scripts or create new ones by *scripting*. Together, these capabilities allow you to quickly and easily replay analyses you've already run using recorded journals, as well as to extend functionality, automate repetitive analyses, and run analyses in batch mode.

For detailed instructions on using journaling and scripting, as well as a complete list of all available data containers, namespaced commands, and data types, see the *Workbench Scripting Guide*.

Journaling

A journal is a record of all operations that have modified data during your session. Based on your preferences setting, a journal of your full session is automatically saved to the location you specify (see Setting Journaling Preferences (p. 251)). You can also choose to record part of a session to a journal file, capturing a specific set of actions. Playing back the journal recreates the recorded actions exactly. Journaling and scripting tools (including recording and playback) are available through the scripting menu and can be performed by anyone using Workbench. Use journaled sessions to restore work after a crash.

Journals are platform independent and portable, subject to file location consistency between accounts (see File Path Handling in Ansys Workbench for details on file path handling within journals and scripts). They can be used with any Ansys installation (release 12.1 or higher).

Scripting

A script is a set of instructions to be issued to Workbench. The script can be a modified journal, or it can be a completely new set of instructions that you write directly.

The creation of scripts requires a general understanding of programming constructs and paradigms. Workbench uses an object-based approach, similar to object-oriented programming.

Related topics:

Setting Journaling Preferences

Recording a Journal

Playing Back a Recorded Journal

Using the Command Window

Setting Journaling Preferences

You can set journaling preferences such as the default directory where journals will be written and how long to keep a journal file.

To set journaling preferences:

1.   From the menu bar, select **Tools** → **Options** → **Journals and Logs**.

2.   To have Workbench automatically write journal files, select **Record Journal Files**.

3.   Specify the default location where journal files are written.

      This is the location that the browser opens to automatically when you choose to begin recording a journal. You can also browse to a different location before saving a particular journal.

4.   Specify the number of days to keep a journal file.

5.   Specify how long (in seconds) to pause between each command when running a journal file.

6.   To save your settings, click **OK**.

Recording a Journal

Follow the steps described below to record a journal and then to playback a journal interactively. To use the command window, see Using the Command Window (p. 253).

To record a journal:

1.   From the menu bar, select **File** → **Scripting** → **Record Journal**.

2.   Specify the name and location of the journal file and click **Save**.

3.   Use the user interface to work through your analysis as you normally would.

> **Note:**
>
> Not all actions are journaled—only actions that change project data. Some examples of actions that are not journaled include:
>
> - Interface-only actions, such as:
>
>   > Interrupting a Solve operation
>   >
>   > Launching help (including quick help and the **Sidebar Help** pane)
>   >
>   > Running the View Solver Output option from VistaTF's **Solution** cell.
>
> - Actions taken in some data-integrated applications; see Scripting and Data-Integrated Applications.

4.   From the menu bar, select **File** → **Scripting** → **Stop Recording Journal**.

5.   A message appears informing you that the recording will stop. Click **OK**.

Playing Back a Recorded Journal

You can replay all of the previously recorded actions by playing back the journal file.

To play back a recorded journal:

1. From the menu bar, select **File** → **Scripting** → **Run Script File**.

2. Select the journal file to be played back and click **Open**.

Using the Command Window

The command window allows you to invoke commands, access data entity properties, and invoke data entity and data container methods interactively, one at a time.

> **Note:**
>
> The console window is the same as the command window but is present when running in batch mode to provide a way of working directly with commands outside of the user interface.

To use the command window:

1. From the menu bar, select **File** → **Scripting** → **Open Command Window**.

2. Enter the commands you want to run, one at a time.

   As you enter each command, the appropriate action is triggered in Workbench.

Working with the Chart Pane

The **Chart** pane is available for Workbench applications. Applications can display data using a number of different basic chart types. Each chart type has editable display properties.

Related topics:

Chart Types

Setting Chart Properties

Changing the Display of a 3D Chart

Using the Triad

Saving a Chart

Chart Types

Although the content of a chart will be tailored to the application that is displaying it, there are a standard set of charts that you might see in Workbench.

XY Plot
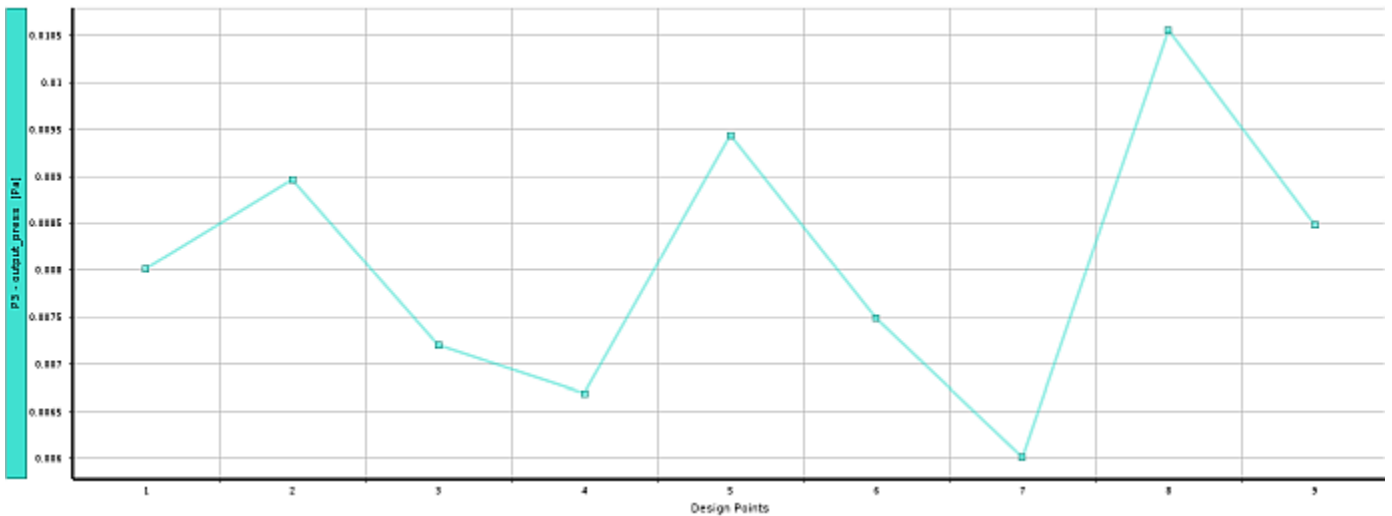
Plots lines, points, steps, bars, splines, scatter, or shaded regions in two dimensions.

XYZ Plot
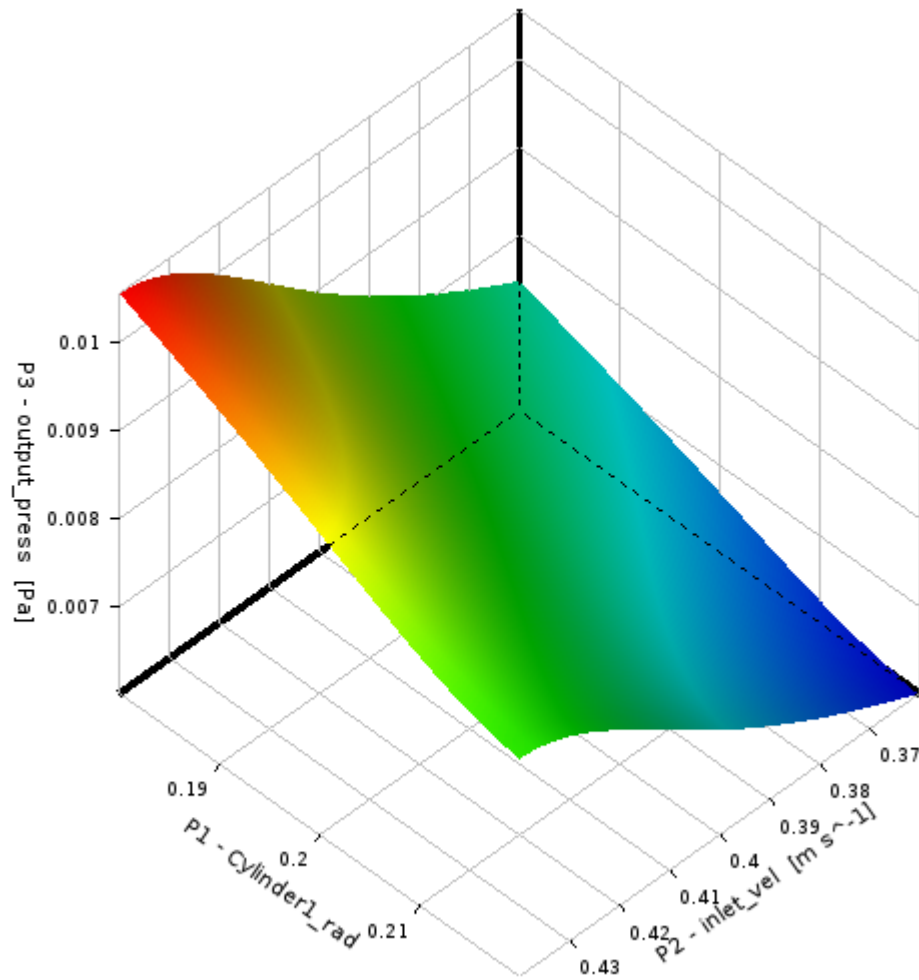
Plots lines, points, steps, bars, splines, scatter, or shaded regions in three dimensions.

Pie Chart
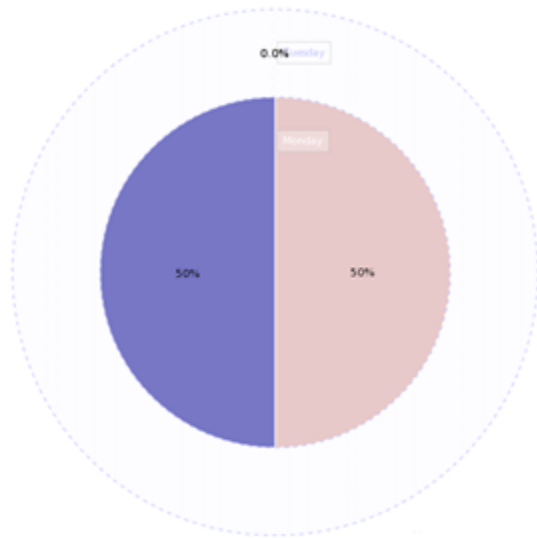
Displays variables as proportionally sized segments of a circle. If a variable to be displayed has no value, that variable displays as a ring around the circle.

Spider Chart
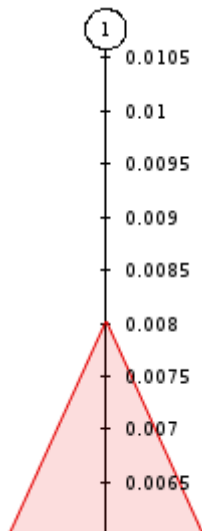
Displays variables as proportionally sized segments of a circle, in multiple dimensions. Good for displaying between three and ten variables. Can usually be displayed as a parallel coordinate plot also.

Parallel Coordinate Plot
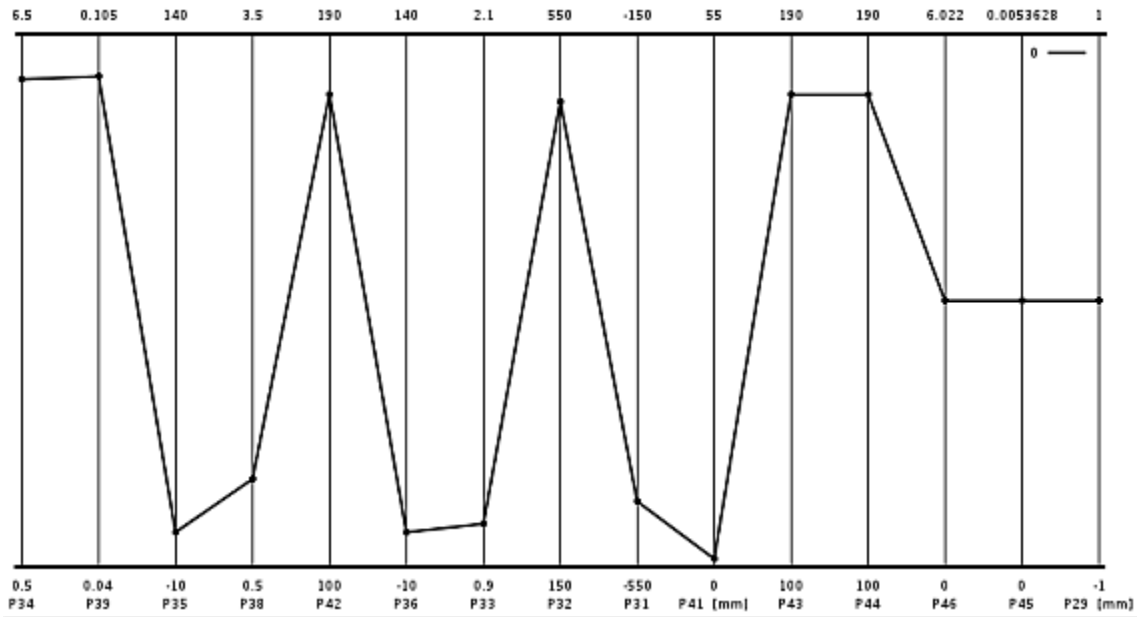
Displays variables (for example, design points.) using parallel Y axes to represent all of the inputs and outputs. Selecting an axis allows you to filter the variables shown by dragging the arrows at the ends of the axis. Variables with points outside of the axis range are eliminated from the chart. Can usually be displayed as a spider chart also.

Correlation Matrix
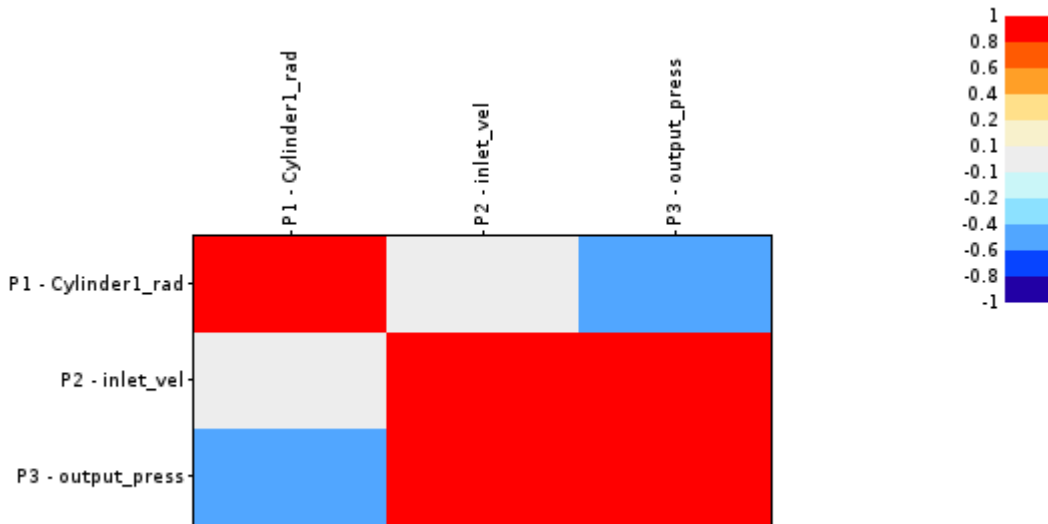
Displays how closely the various input and output parameters are coupled. The strength of correlation is indicated by color in the matrix.

Setting Chart Properties

Each chart has properties that can be set for chart data and chart display. This section describes chart display properties, common to charts displayed by any Workbench application. For more information about chart data properties, refer to the help for the individual applications displaying the chart.

The various chart properties are displayed in the **Properties** pane (p. 444).

Note:

Only the properties that are applicable for the current chart type are displayed when you edit a chart component (an axis, for example).

To set chart properties:

1. Right-click a part of a chart and select **Edit Properties** from the context menu.

    - To set axis properties, right-click on, or outside of, an axis.

    - To set legend properties, right-click the border or background of the legend.

    - To set variable properties, right-click the variable plot on the chart or right-click the variable name in the legend.

    - To edit chart properties, right-click the background of a chart.

2. Set the required properties.

Property	Description
Axis Properties	
The axis properties that are available depend on whether the chart axis is discrete or continuous.	
Title	Sets the name for the axis (defaults to variable name).
Title Background Color	Sets the background color used for the axis name.
Show Grid	• When selected, displays a grid for this chart axis • When cleared, hides the grid
Automatic Range	• When selected uses automatic scaling for the axis • When cleared uses **Range Minimum** and **Range Maximum** for axis scaling if they fall within the data bounds
Range Minimum	Sets the minimum range of the axis and is displayed when **Automatic Range** is cleared.
Range Maximum	Sets the maximum range of the axis and is displayed when **Automatic Range** is cleared.
Is Logarithmic	• When selected, sets axis scaling to be logarithmic

Property	Description
	• When cleared, sets axis scaling to be linear
Is Usability	• When selected, displays a line/bar plot of the cross-section data in a perpendicular direction to the axis direction • When cleared, hides the line/bar plot
Legend Properties	
Legend properties can also be set as part of general chart properties. If you click a legend variable, the variable properties are displayed instead of the legend properties.	

Visible	• When selected, displays the legend for the chart
	
	• When cleared, hides the legend for the chart
	
	    Note:
	
	    If there are too many entries in the legend, the legend does not display, even if **Visible** is selected.

Style	Displays the legend entries either horizontally or vertically and allows the legend to be expanded in the same direction, if it provides any benefit (defaults to vertical).
Foreground Color	Sets the color of the legend border. The legend border default is to have no color. You can apply a new color from the color wheel or click **More** and change the Alpha channel to **0** for transparent or **255** for opaque.
Background Color	Sets the color of the legend background. The legend background default is to have no color. You can apply a new color from the color wheel or click **More** and change the Alpha channel to **0** for transparent or **255** for opaque.
Variable Properties	
Label	Sets the name for the selected variable plot.
Is Included in Legend	When selected, adds the selected variable plot to the legend.
Display As	Sets the shape for the selected variable plot.
Automatic Range	• When selected uses automatic scaling for the variable values • When cleared uses **Range Minimum** and **Range Maximum** for variable value scaling if they fall within the data bounds
Range Minimum	Sets minimum range of variable values displayed.
Range Maximum	Sets maximum range of variable values displayed.
Style Display Properties	
When plot rendering is controlled per variable, the style display properties appear when you edit the variable properties. When plot line rendering is controlled generally rather than per variable (for example, spider charts), the style display properties appear when you edit the chart properties.	

Property	Description
Smoothed Edges	Renders lines and surfaces with anti-aliasing set on so that the line or surface appears to be smooth.
Line Style	Sets the type of line used for the plot.
Fill Style	Sets the fill style for objects in the plot that use a fill.
Line Colors	Sets the color sequence for multiple lines displayed on the chart.
Fill Colors	Sets the color sequence for sequential symbols or gradient used for plot display.
Number of Color Bands	When set to 0, the gradient is displayed as a smooth graduation between values, rather than banded, where a single color is shown for a range of values.
Relative Bar Width	Sets the width of a bar as a proportion of available space [0 - 1], determined by the maximum size bar that can be displayed without overlapping any adjacent bars of the same variable (if other variables appear between bars of this variable, making the bar wider may overlap those intervening variables).
Relative Bar Offset	Sets the start position of a bar, proportional to the minimum point where the bar could be placed to the maximum point where the bar could be placed [0 - 1], with the maximum size based on the same criteria as **Relative Bar Width**.
Show Linear Interpolation of Lines	When selected, causes the ends of a line plot to extend to the edge of the chart, when the plot is not against a discrete axis. **Note:** In most cases, properties are not shown if they are not applicable to the item selected. However, Show Linear Interpolation of Lines is displayed whether the axis is discrete or continuous, but it only applies in cases of continuous axes. The option will have no effect if you set it when the plot is displayed on a discrete axis.
Symbol Outline Colors	Sets the color used to outline displayed symbols.
Symbol Size	Sets the size of displayed symbols in pixels [1 - 16].
Line Width	Sets the width of lines (including lines outlining bars), in pixels [1 - 10].
Chart Properties	
When plot line rendering is controlled generally rather than per variable (for example, in spider charts), the style display properties also appear when you edit the chart properties.	
Title	Sets the chart title that appears in the **Chart** pane title bar.
Chart Type	Allows some charts to be displayed as another chart type.
Display Percentages	Turns on percentage values on pie charts.

Changing the Display of a 3D Chart

You can manipulate the display of a 3D chart using the zoom, pan, and rotate features.

To change the display of a 3D chart:

*   To zoom in or out of the entire chart, click the chart and use the scroll wheel or press **Shift** + middle mouse button.

    The chart expands or shrinks as you roll the wheel (or move the mouse) toward you or away from you, remaining centered in the pane. The chart can be panned or rotated when it is magnified.

*   To magnify a particular area of a chart, right-click and drag a box top to bottom over the area you want to view.

    The contents of the box you draw are magnified, so the smaller the box you draw, the closer the magnification on a particular area.

*   To reduce the chart to a particular size, right-click and draw a box bottom to top over the chart.

    The chart zooms out to roughly correspond to the size of the box that you have drawn, so a very small box gives you a tiny chart.

*   To pan, press **Ctrl** + middle mouse button and move the mouse left or right.

Using the Triad

On three dimensional charts, the *triad* appears in the lower left corner showing the orientation of the three axes for the current view of the chart. The x axis is red, the y axis is green, and the z axis is blue. There is a light blue ball in the triad that indicates the orientation ISO z axis up position of the chart. If you click this ball, it sets your chart to be ISO z axis up, fit to window.

If you move your cursor around the triad, a yellow arrow shows the direction that corresponds to the position of your cursor (+x, -x, +y, -y, +z, -z). If you click the arrow, it changes your chart so that the chart axis indicated by the arrow is facing out.

There are several shortcut keys that can be used when you are viewing a 3D chart:

*   f – fits the chart to the window

*   x – displays the x+ view, fit to window

*   y – displays the y+ view, fit to window

*   z – displays the z+ view, fit to window

*   i – displays the chart in the ISO z axis up position, fit to window

Saving a Chart

You can save the chart that you are viewing as a graphic.

To save a chart:

1. Right-click the background of the chart and select **Save Image As** from the context menu.

2. From the **Size** drop-down list, select the image resolution to use when saving the chart.

3. To the right of the **File** field, click the ellipses .

4. In the **File Name** field, type a name for the image.

5. From the **Save as type** drop-down list, select either **PNG image** or **BMP image**.

6. To save the settings, click **Save**.

7. To save the image, click **OK**.

Working with Project Reports

Workbench project reports generate a visual snapshot of your project. The contents and organization of the report reflect the layout of the **Project Schematic**, with sections for global project information, analysis system information, system cell information, and where applicable, content provided by applications in the project.

The report contains basic project information, including:

- Export time and date

- Workbench version number

- A graphic of the systems as shown in the **Project Schematic**

- File information

- Parameter and design point information

- System and cell information

- Contents of notes

The specific information provided varies depending on the contents of the project.

> **Note:**
>
> At this time, DesignXplorer and CFD-Post are the only applications that provide the project report with detailed report content. Content for other applications is limited to the data visible at the project level (for example, in the **Properties** pane for the associated cell in the **Project Schematic**).

The report is written to the `user_files` directory under the project directory by default.

If the project includes parameters and report content has been provided by an application in the project during a design point update, then the design points table contains links to sub-reports for each design point. Detailed report content for each design point can be accessed by a hyperlink in the **Report** column of the design points table in the project report. Clicking the link opens a sub-report that contains the application-specific content for that design point (if available).

Related topics:

Setting Project Report Options

Including CFD-Post Data in the Project Report

Exporting a Project Report

Editing Project Reports

Viewing Report Images

Setting Project Report Options

You can control whether the report opens by default using the project reporting settings.

To configure project report options:

1. From the menu bar, select **Tools** → **Options**.

2. In the **Options** dialog, select **Project Reporting** from the navigation tree.

3. To launch your default browser and load the report immediately after it is generated, select the **After exporting report, automatically open in default browser** check box.

   If you do not select this option, you must navigate to the report file and open it manually.

4. Click **OK**.

Including CFD-Post Data in the Project Report

You can set whether or not to include CFD-Post data in the project report.

To include CFD-Post data:

1. In the **Project Schematic**, right-click the **Results** cell for an Ansys CFD analysis system or Results system and select **Properties**.

2. In the **Properties** pane, select the **Publish Report** check box.

3. Repeat the previous two steps for each **Results** cell to be included in the project report.

Note:

- Only 2D content such as graphs and figures are supported in the project report. If interactive 3D content exists, it is displayed in 2D format.

- The format/style of CFD-Post reports produced from Workbench are consistent with general Workbench-produced reports; reports from standalone CFD-Post use the standard CFD-Post format.

Exporting a Project Report

You can export a report of the current project in .html/.htm format.

To export a project report:

1. From the menu bar, select **File** → **Export Report**.

2. To change the default file name, type a new name in the **File name** field.

3. Click **Save**.

Editing Project Reports

Once you have generated a project report, you can edit its contents.

To edit a project report:

1. To open the report file with an HTML-adapted editor (such as Microsoft Word), right-click the file and select **Open with**.

2. Edit the report contents and formatting as needed.

3. Save the file to a location outside Workbench.

Viewing Report Images

When you have a solved project that has design points, if a project report has been generated, you can preview design point images from the report in the **Parameter Set** tab.

To view report images:

1. In the **Project Schematic**, right-click the **Results** cell and select **Properties**.

2. In the **Properties** pane, select the **Publish Report** check box.

3. In the toolbar, click **Update All Design Points**.

4. When the project has updated, click the **Parameter Set** bar.

5. In the **Properties** pane, under **Design Point Report**, note the list of PNG files that are available for design points in the **Report Image** field.

   The image files are located in the following directory:

   `[project_location]\dp[current_design_point]\[system_type]\Post\Report\Report`

6. Double-click the **Parameter Set** bar.

7. In the design points table, click a thumbnail in the **Report Image** column to open the image file.

To clear the images, in the **Properties** pane, set **Report Image** to **None**.

> **Note:**
>
> The image in the **Report Image** column is the default image for a CFX or Fluent report (Generic Report Figure 1). To have a different image:
>
> 1. Set up CFD-Post to display the image you want in the report.
>
> 2. Click the Figure icon in the toolbar.

---

3. In the **Outline** pane, disable Generic Report Figure 1 and enable Figure 1.

4. In Workbench, click **Update All Design Points**.

Saving and Managing Project Files

The file management system in Workbench stores several different files under a single project, using directory trees to organize files relevant to each system and the applications used in the system.

When the project file (`<filename>.wbpj`) is created, Workbench creates a project folder named `<filename>_files` where `<filename>` is a name you provide. All files relevant to the project are saved within this folder.

The primary subdirectories within the project folder are `dp0`, `dpall`, and `user_files`.

We strongly recommend that you use caution when directly modifying any of the content in any of the Workbench project directories or subdirectories other than `user_files`. You should work through the Workbench user interface to manage your project as much as possible. Workbench may not recognize or be aware of any changes that you make directly in the file system (such as adding or removing a file).

> **Important:**
>
> If you are resuming a project in Mechanical on the Linux platform, there is a restriction that the path to the project, as well as the project name, include ASCII characters only, otherwise, the project cannot open.

Related topics:

Saving a Project File

Archiving a Project

Saving an Archive File

Exporting Systems to a New Project

Working in the Files Pane

Recovering Projects

Unlocking Project Files

Project Directories

Understanding Project File Management

Saving a Project File

To create a project file and folder, save the project.

To save the project:

1. Select one of the following options:

- To save the current project, from the menu bar, select **File** → **Save** or from the main toolbar, click .

- To save the current project under a different name or location, from the menu bar, select **File** → **Save As** or from the main toolbar, click .

2. If prompted:

    a. Browse to the location where you want to save the file.

    b. Type a name for the project.

    c. Click **Save**.

Archiving a Project

If you want to send a project to a colleague or to Ansys Technical Support, or need to package all of the files, you can archive the project.

To archive a project:

1. Optionally, specify the compression level (p. 418) for the archive file.

2. If you have imported an external file into the project, and that external file refers to other files (for example, when a CAD assembly is linked to the CAD parts), copy the referenced files into the `user_files` directory to ensure they are added to the archive.

3. To save the project, from the menu bar, select **File** → **Save**.

    If you do not save the project before archiving, you are prompted to save before proceeding.

4. From the menu bar, select **File** → **Archive**.

5. In the **Save Archive** dialog box, navigate to the directory where you want to save the file.

    > **Note:**
    >
    > The initial default save location depends on your project management (p. 418) user preferences. Also, the location you select is remembered and used as the default location the next time you archive a project. Review the default location carefully before saving the archive file.

6. From the **Save as type** list, select the archive type:

    - **Archive Files (*.wbpz)**

    - **Zip File (*.zip)**

7. Click **Save**.

8. In the **Archive Options** dialog box, select the optional items to archive:

    - **Result/solution and retained design point files**

    - **Imported files external to project directory**

    - **Items in the User_files folder**

    > **Note:**
    >
    > – If your project contains Imported Boundary Conditions in the Mechanical application, you must include result/solution items so that the necessary upstream files are archived. Failure to archive these files prevents you from importing data or accessing features that involve reading upstream data, when the project is restored.
    >
    > – If you are using an External Model system, select the **Imported files external to project directory** option. External Model systems do not copy files to the project folders of downstream systems.

9. Click **Archive**.

Saving an Archive File

On Windows systems, you can double-click `.wbpz` archive files from the directory they are saved in to open the archive. If you open an archive file this way and then make changes to the project, when you save the file you are prompted to either overwrite the archive, create a copy of the archive, or cancel the save operation.

To save an archive file:

1. From the menu bar, select **File → Save**.

2. To overwrite the archive file:

    a. In the dialog box, click **Overwrite**.

    b. In the **Archive Options** dialog box, select the optional items to archive.

        - **Result/solution and retained design point files**

        - **Imported files external to project directory**

        - **Items in the User_files folder**

        > **Note:**
        >
        > If your project contains Imported Boundary Conditions in the Mechanical application, you must include result/solution items so that the necessary upstream files

are archived. Failure to archive these files prevents you from importing data or accessing features that involve reading upstream data, when the project is restored.

    c. Click **Archive**.

        After saving, you are returned to Workbench, working in the updated archive. Use **Save As** to restore the project to a `.wbpj` file.

3. To save a copy of the archive file:

    a. In the dialog box, click **Save a Copy**.

    b. Browse to the location to save the file.

    c. Type a name for the archive copy.

    d. Click **Save**.

        After saving, you are returned to Workbench, working in the new (copy) archive. The original archive remains unchanged. Use **Save As** to restore the project to a `.wbpj` file.

Exporting Systems to a New Project

You can reuse systems from a previously created project file by exporting one or more of them to a new project.

Important:

To export systems, Workbench temporarily removes the systems not being exported from the Project Schematic, saves the resulting schematic as a new project, and then restores the original project. These systems are **not** being deleted, they will be restored after the export project is saved.

Depending on how many systems are contained in the original project, and how many you are exporting, this process can take a long time.

To create the new project:

1. If you have not saved your current project, from the menu bar, select **File** → **Save**.

2. Select all systems to be exported using one of the following methods:

    • Click the header cell of a single system.

    • Box-select by clicking and dragging the selection box over the required systems.

    • Press **Ctrl** and click the required systems.

3.   Right-click the header of one of the selected systems.

4.   Select **Export System(s)** from the context menu.

5.   If you did not save your project in step 1, the **Unsaved Project** dialog box is displayed:

  a.   Click **Save and Proceed**.

  b.   In the **File name** field, type a name for the current project.

  c.   Click **Save**.

6.   In the **Export As** dialog box, type the name of the new export project into the **File name** field.

7.   From the **Save as type** drop-down list, select **.wbpj** to save the export project as a new Workbench project, or **.wbpz** to save the export project as an archive file.

8.   Click **Save**.

9.   If you are saving the export project as an archive file:

  a.   In the **Archive Options** dialog box, select the optional items to archive.

    • **Result/solution and retained design point files**

    • **Imported files external to project directory**

    • **Items in the User_files folder**

    > **Note:**
    >
    > If your project contains Imported Boundary Conditions in the Mechanical application, you must include result/solution items so that the necessary upstream files are archived. Failure to archive these files prevents you from importing data or accessing features that involve reading upstream data, when the project is restored.

  b.   Click **Archive**.

The export project can be opened directly, or imported into another project (p. 49).

Working in the Files Pane

The **Files** pane shows a list of all files associated with the project. It enables you to see the name and type of file, the IDs of the cells the file is associated with, the size of the file, the location of the file, and other information.
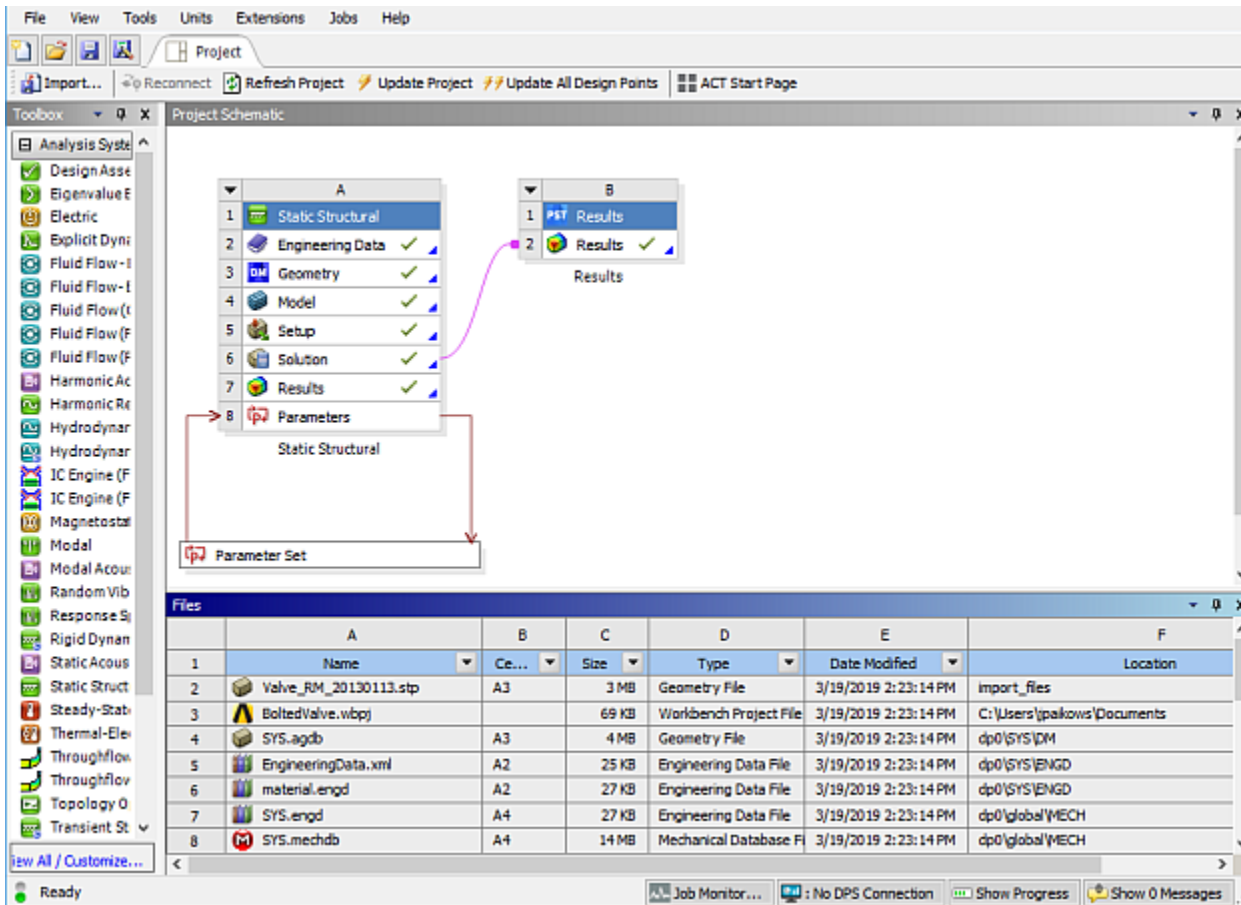
Workbench File Types

The following file types are created by the various Ansys applications and are shown in the Files pane.

Database Files

Ansys applications create the following types of database files:

- Ansys Workbench project database file (`.wbpj`)

- Mechanical APDL (`.db`)

- Fluent (`.cas`, `.dat`, and `.msh`)

- CFX (`.cfx`, `.def`, `.res`, `.mdef` and `.mres`)

- DesignModeler (`.agdb`)

- CFX-Mesh (`.cmdb`)

- Mechanical (`.mechdb`)

- Meshing (`.cmdb`)

- Engineering Data (`.eddb`)

- Mesh Morpher (`.rsx`)

- Ansys Autodyn (`.ad`)

- DesignXplorer (`.dxdb`)

- BladeGen (`.bgd`)

Workbench Journal and Log Files

Workbench writes journal and log files for each Workbench session. For more information on journal and log file options, see Journals and Logs (p. 421).

Design Point Log Files

During a design point update, the parameter values of each successfully updated design point are written to a CSV log file in the `user_files` directory. For more information, see Exporting Design Point Parameter Values to a Comma-Separated Values File (p. 316).

Sorting and Filtering the Files List

Files added to the project appear in the **Files** list.

To sort and filter the files list:

1. To sort individual column data, click the arrow (▼) to the right of a column name and select from the following options:

    - To sort the column data in ascending order, select **Ascending**.

    - To sort the column data in descending order, select **Descending**.

    - To cancel the current sorting option, select **Cancel sorting**.

2. To sort multiple columns at the same time:

    a. Click the arrow (▼) to the right of a column name and select **Sort Settings**.

    b. From the **Column** drop-down list, select the name of the column to sort.

    c. To sort the column in ascending order, leave the Ascending check box selected. To sort the column in descending order, clear the check box.

    d. Repeat the previous steps until you have created a list of all of the columns you need to sort.

        To remove all of the columns from the list, click **Remove All**.

    e. To save the changes, click **OK**.

3. To select the types of files that appear in the **Files** list:

a.	Right-click any of the cells and select **File Type Filter** from the context menu.

b.	From the **Select to Display** column, select or clear any check boxes to display or hide the indicated file type.

c.	To save the changes, click **OK**.

Repairing or Removing Deleted Files

Files missing or deleted from the project are shown in red text and are marked with a deleted icon.

You can either repair the missing files by adding them back into the project from another location, or you can remove them from the **Files** list.

To repair or remove deleted files:

1.	In the **Files** list, select a cell on the line containing the deleted file or press **Ctrl** and click on multiple cells to select multiple files.

2.	To repair the deleted files:

a.	Right-click the selection and select **Repair _filename_** from the context menu.

b.	Browse to the location where a copy of the files are stored.

c.	Click **Open.**

The deleted files are copied into their original directory, and the name of the files are shown in black text.

3.	To remove the deleted files from the list, right-click the selection and select **Remove _filename_ from List** from the context menu.

Opening the Folder Containing Project Files

You can select a file from the **Files** list and open the operating system's file manager to the folder containing that file. In most cases, you should not edit, add, or delete a file from the operating system file manager. Workbench does not recognize or is aware of any changes that you make directly in the file system (such as adding or removing a file). However, if used with caution, this pane can be a useful way to edit files such as application input files (for example, the Mechanical APDL application input files).

To open the folder containing project files, right-click a cell and select **Open Containing Folder** from the context menu.

Copying Cell Text

You can copy the text from one or more of the **Files** list cells to paste into a different program, such as Microsoft Excel.

To copy cell text:

1.   In the **Files** list, select the cell you want to copy the text from or press **Ctrl** and click multiple cells to select them.

2.   Right-click the selection and select **Copy** from the context menu.

Recovering Projects

Workbench creates backup files of projects that are currently active and in progress. In the event of a crash, Workbench can use these backup files to restore your project to the last saved event.

As with any computer program, it's important that you save your work frequently to minimize data loss in the event of a crash. Do not move or otherwise alter the backup directory.

If a project save operation fails (for example, an application is busy and cannot execute the save), you will be given the following options:

- Revert to the last saved project.

- Make a copy of the last saved project before continuing with the partially saved project. The project is copied into a new location that you specify.

- Exit Workbench and decide later. Use this option if you want to handle the save failure manually. As a result, the backup directory is not cleared so that you can manually recover files from that directory later.

- Continue with the partially saved project, discarding the last saved project (not recommended). This option results in the backup directory being cleared. Use this option with caution, as it could result in corrupt project files.

> **Note:**
>
> Recovering from a save failure occurs only when saving a previously saved file and *not* when saving as a new file, or when saving the project for the first time.

Unlocking Project Files

Workbench implements a project locking mechanism in order to help prevent the project from being loaded into more than one session at a time. When the project is opened, a `.lock` file is created in the project files directory. The project is unlocked by deleting the `.lock` file.

An improperly unlocked project can occur in situations such as program crash where the `.lock` file is not deleted, or if the project files directory is duplicated and the `.lock` file is copied with it.

If you open a locked project, you are asked how to proceed:

- If the file is locked because of an abnormal termination, such as a program crash, click **Unlock**.

- If the file is locked because the project is already open in another Workbench session, click **Cancel**.

    Opening the same project in multiple sessions can result in corrupted project files.

Project Directories

The project directory structure includes the follow folders:

- dp0 Folder (p. 278)

- dp*n* Subdirectories (p. 279)

- user_files Folder (p. 279)

> **Important:**
>
> If you are resuming a project in Mechanical on the Linux platform, there is a restriction that the path to the project, as well as the project name, include ASCII characters only, otherwise, the project cannot open.

dp0 Folder

Workbench designates the active project as design point 0 and creates a `dp0` folder that always corresponds to the active project files. For more information on design points, see Working with Design Points (p. 303).

Within the folder are system folders for each system in the project. Within each system folder are folders for each application used in the system (for example, the Mechanical application, Fluent, and so on). These folders contain application-specific files and folders, such as input files, model directories, engineering data, and resources. System folders for each system type are named as follows.

System Type	Folder name
Autodyn	ATD
BladeGen	BG
Design Exploration	DX
Engineering Data	ENGD
Fluid Flow (Fluent)	FFF (analysis system), FLU (component system)
Fluid Flow (Polyflow)	PFL (Polyflow), PFL-BM (Blow Molding), PFL-EX (Extrusion)
Fluid Flow (CFX)	CFX
Geometry	Geom
optiSLang	ETK
Mesh	SYS (top level) / MECH (subdirectory)

System Type	Folder name
Mechanical	SYS (top level) / MECH (subdirectory)
Mechanical APDL	APDL
System Coupling	SC (top level) / SC (subdirectory) / SyC (subdirectory)
TurboGrid	TS
Vista TF	VTF
Icepak	IPK

The Mechanical application and Mesh system folders are labeled SYS. Both the Mechanical application and Mesh files are written to `MECH` subdirectories, because both are generated by Mechanical-based applications.

In addition to the system folders, the `dp0` folder also contains a global folder. This folder contains subdirectories for all systems in the project. These subdirectories may be shared by more than one system in the project and contain all database files, as well as any files that are associated directly with the database files. For example, the Mechanical application writes figures, images, and contact tool data to the appropriate system subdirectory under the global folder.

dp*n* Subdirectories

While a design point update is running, Workbench creates temporary design point folders. If the Retain check box has been selected for a design point, its data is saved to a `dpn` subdirectory (where `dpn` indicates the number of the design point) that is a sibling to the `dpn` folder and contains the same sub-structure. For more information on retaining design point data, see Retaining Design Point Data (p. 312).

user_files Folder

Under the project folder is a `user_files` folder. This folder contains any files (such as input files and referenced files) that you supply to a project or any output (images, charts, movie clips, and so on) generated by Workbench that you want to have associated with the project. In most cases, you are responsible for placing required files into this folder. In other cases, such as the export of design point update data from a design exploration system to a CSV log file, data is written directly to a file created in this folder. For details, see Exporting Design Point Parameter Values to a Comma-Separated Values File (p. 316).

Along with other project files, all of the files contained in the `user_files` folder appear in the Files pane (p. 443) in Workbench.

Workbench also protects this folder and ensures that it is managed and archived appropriately with the rest of the project; therefore, you can safely store additional files (such as PowerPoint or Excel files, or other files from separate applications that are associated with this project) here without the risk of losing data. If you save files in any other folder in the project structure and then exit without saving, Workbench deletes any files saved there since the last Workbench save.

Example Project

A finished project that includes a Fluid Flow (Polyflow) system (FFF), a Mechanical application system (MECH), and parameters (DesignXplorer) might look like this:
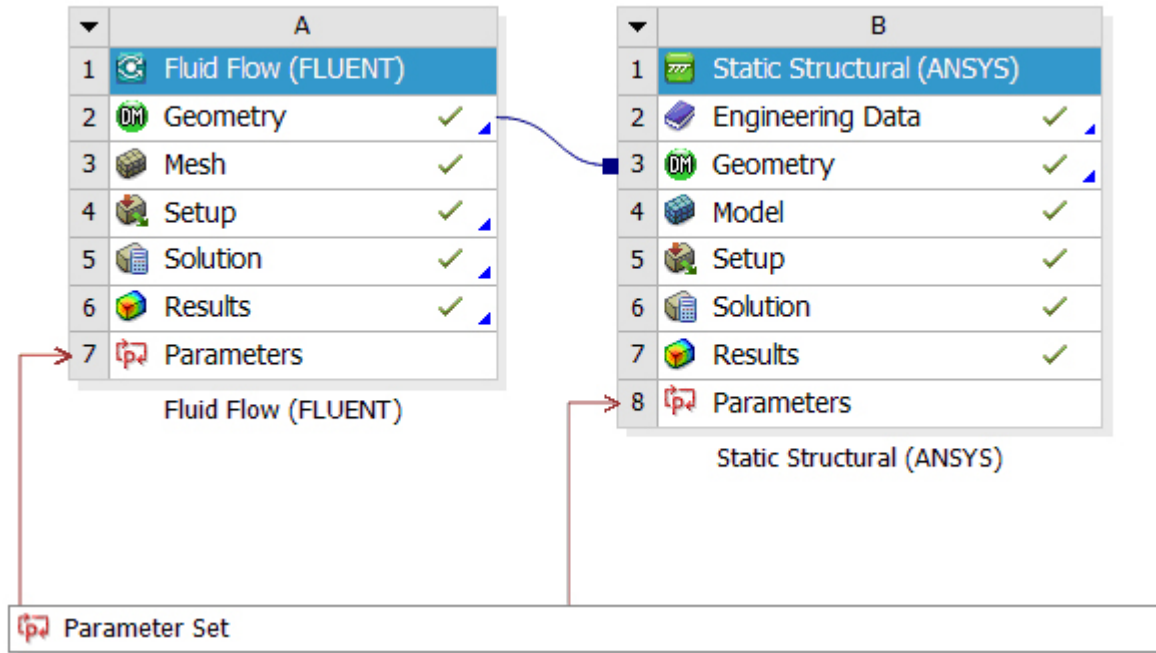
The corresponding directory structure would look like this:

```
Myworkbenchproject_files
 dp0
    FFF
       DM
       Fluent
       MECH
       Post
    global
       MECH
          FFF
          SYS
    SYS
       ENGD
       MECH
 dpall
    global
       DX
```

Understanding Project File Management

The project system may delete, back up, or restore files when:

- Deleting, duplicating, or replacing systems in the schematic

- Opening an application for a cell

- Closing a project without saving

- Archiving a project

- Switching to the next design point when updating all design points

While the project system is performing one of these operations, you cannot have project files open in other applications (such as a text editor) or have the directories open in Windows Explorer. Doing so may cause these file management operations to fail.

You cannot move a project or any of its associated files to another machine while a background run is in progress. File information for the background run is, by necessity, machine-specific. You cannot package or modify the background run while it is in progress.

Moving a project that has references to files outside of the project directory to a different machine or location creates errors. By opening the project from a different machine or location, those file references do not resolve, unless the file is still available under the same absolute path.

Copied images exist in only one location on disk that is referenced and do not exist as physical copies. If you delete an image that has been copied, all pointers to the copies of that image will contain broken links.

Working with Parameters and Design Points

In Ansys applications you can define key simulation properties to be parameters. You can then manipulate the parameters at the project level in Workbench to investigate design alternatives. A set of parameter values representing one design alternative is called a *design point*. You can create a set of design points in tabular form and run them automatically to perform a what-if study.

For the most part, you work with parameters and design points on the **Parameters** tab and the **Parameter Set** tab. However, you can also use parameters and design points in Ansys DesignXplorer for automated design exploration studies. DesignXplorer supports the insertion of the following **Design Exploration** systems from the Workbench **Toolbox** into the **Project Schematic**:

- **3D ROM**

- **Direct Optimization**

- **Parameters Correlation**

- **Response Surface**

- **Response Surface Optimization**

- **Six Sigma Analysis**

Note:

For large parametric studies, you should use AnsysDesign Point Service (DPS), which is part of Distributed Compute Services. DPS provides for quickly and efficiently evaluating tens of thousands of design points using multiple compute resources. For more information, see the DCS for Design Points Guide.

Related topics:

Parametrization Basics

Working with Parameters

Working with Design Points

Parametrization Basics

Parameters can be defined not only in Workbench applications but also in numerous external CAD packages. Applications that allow for parametrization include the following:

- DesignModeler

- Mechanical

- Mechanical APDL

- Fluent

- CFX

- Polyflow

- SpaceClaim

- ExplicitSTR

- CFD-Post

- Vista TF

- Meshing

- Engineering Data

- Microsoft Excel

Parameterizing CAD Dimensions

To parameterize CAD dimensions, you must first enable parameters in Workbench:

1. From the Workbench menu, select **Tools** → **Options** → **Geometry Import**.

2. For **Filtering Prefixes and Suffixes**, specify any keys that must appear at the beginning or end of a CAD parameter name for it to display in Workbench.

    The default is **ANS;DS**. However, you can specify any number of prefixes or suffixes as keys. For example, to expose a CAD parameter named **Length** in Workbench, you could rename it to something like **dslength**, **Lengthds**, **ds_Length**, or **Length_ds**. The order, underscore, and case do not matter. All of these would be recognized as matching the key **DS**.
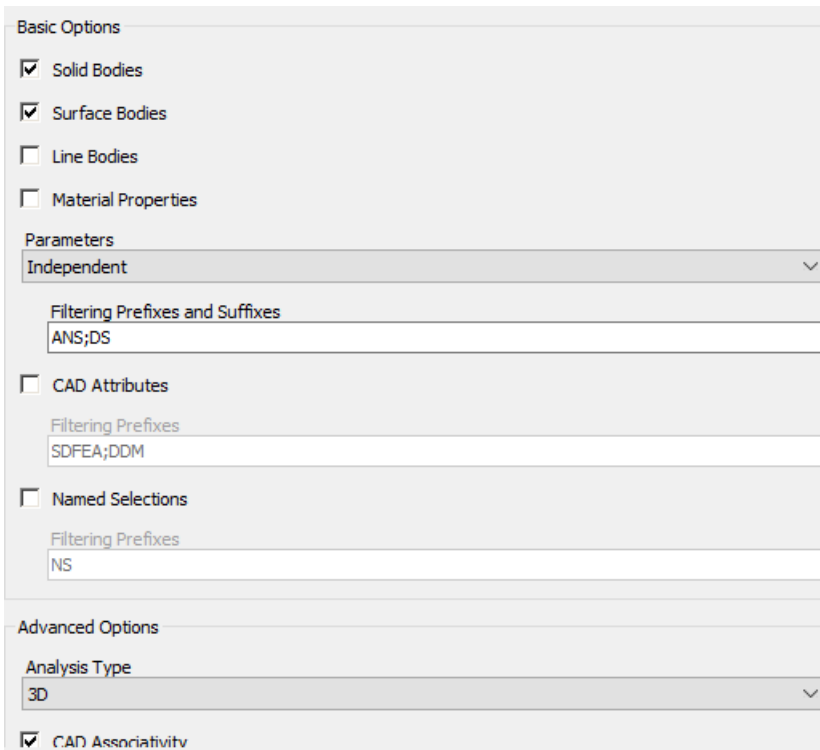
Parameterizing Workbench Dimensions

The process for parameterizing Workbench dimensions depends on the application. The following figures show different methods for parameterizing model data. For more information, see the documentation for the application that you are using.

Input and Output Parameters Created in DesignModeler and Mechanical
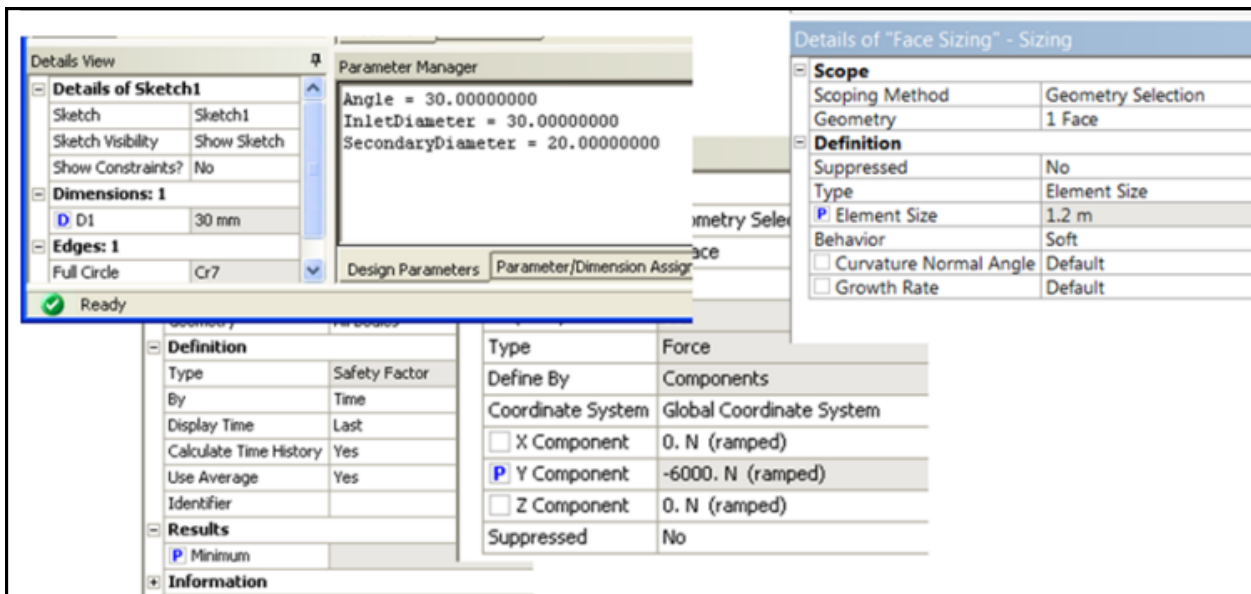
Parameter Definition Based on a Mechanical APDL Input File
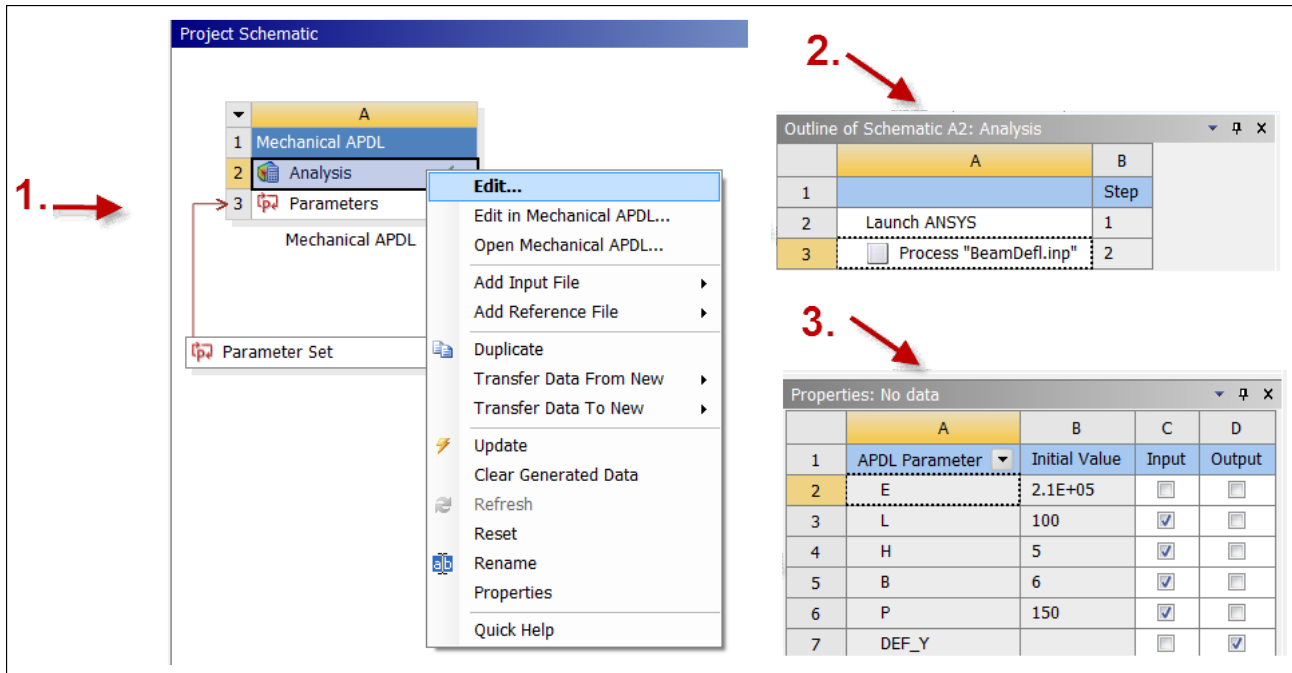
Input Parameters Created from an Expression in CFX
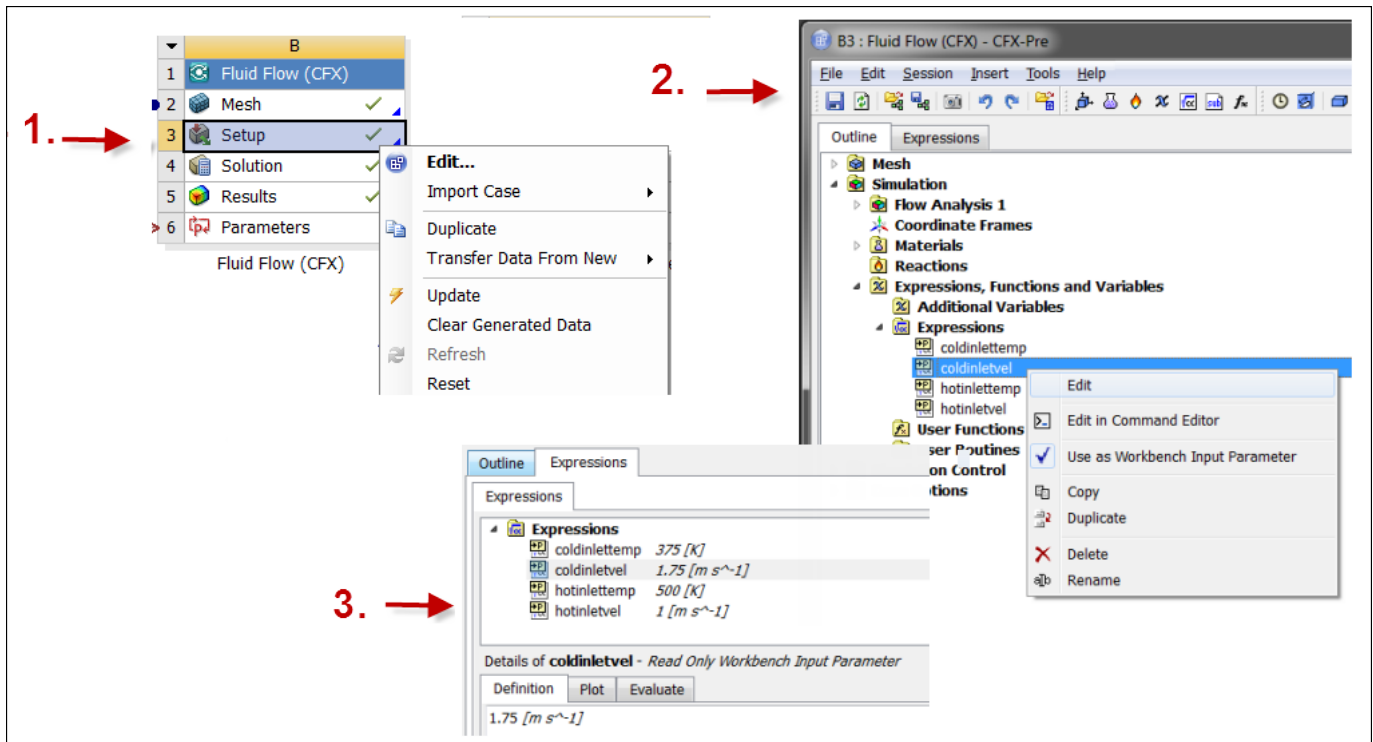
Input Parameter Created from an Expression in Fluent

Output Parameters Created from an Expression in CFD-Post

Derived Parameters

Derived parameters are created using analytical expressions composed of input parameters, output parameters, or both input and output parameters. As the definition suggests, derived parameters are calculated from other parameters by using equations that you provide.

Some examples of derived parameters include:

· **Cost Function**: Product of mass and cost per mass

· **Normalized Stress**: Stress response divided by an applied stress

· **Average Value**: Average of the first three frequencies

· **Mesh Sizing**: Mesh parameter set as a function of a geometric parameter

> **Note:**
>
> Derived parameters cannot reference other derived parameters.

Defining Derived Parameters

You can define derived parameters using various built-in arithmetic, trigonometric, and statistical functions. They are created in the analysis system and passed into DesignXplorer as output parameters.

To create a derived parameter:

1. Double-click the **Parameter Set** bar to open it.

2. Enter the expression in either of these locations:

   - **Value** field in the **Outline** pane

   - **Expression** field in the **Properties** pane (after selecting the parameter in the **Outline** pane)

Note:

You can add both derived parameters and constant values in the **Outline** pane. However, once derived parameters are added, they become read-only in the **Outline** pane. To edit them, you must use the **Properties** pane.

Working with Parameters

A *parameter* is a numerical or other measurable factor that helps to define a particular system. A parameter is linked to a data model property within an application. You can have input and output parameters. Parameter values can be numeric or non-numeric (string or Boolean). Non-numeric parameters are ignored for charting purposes. An input parameter can be modified at the project level and drives

a change within the data model. The value of an output parameter is set by the application, based on the current results or state.

Input parameters define the geometry or inputs to the analysis for the system under investigation. Input parameters have predefined values or ranges that may be changed. These include CAD parameters, analysis parameters, and DesignModeler parameters. CAD and DesignModeler input parameters include length and radius. Analysis input parameters include pressure, material properties, and sheet thickness.

Output parameters result from the geometry or are the response outputs from the analysis. These include volume, mass, frequency, stress, velocities, pressures, forces, and heat flux.

Custom Parameters are project-level parameters that are not directly associated with a data model property. They can be a custom input or custom output parameter that is defined by a constant value, such as `12.5 [cm]` or `sin(pi/2)`. Or, they can be a derived parameter, defined by an expression of other parameters, such as `P2+3*P3`. For more information, see Expressions, Functions, Quantities, and Units (p. 291).

All parameters have a quantity, preferably with a quantity name. The quantity name is used to define preferred and available units for the quantity. If the value does not have a quantity name defined, Workbench displays the value without units.

You can add parameters to or delete them from the current project. However, this may set existing design points and design exploration systems to an out-of-date state and can result in several hours of recalculation time, depending on the project. Be aware that deleting a parameter referenced in the expression of another parameter invalidates the driven data model, resulting in an error.

Note:

- Negative dimension values can invert the direction vector of SpaceClaim operations with which they are associated. This change is applied to the current and subsequent design point updates. As a result, when a Workbench input parameter is used as a driving dimension for a SpaceClaim geometry, negative dimension values can result in unexpected geometric changes.

- When dimensional geometry parameters are imported into Workbench from SpaceClaim or a bi-directional CAD interface, design point updates may fail when dependencies exist between the parameters or when the parametric update in the CAD system or SpaceClaim cannot be realized with the parameter values specified in Workbench.

- When you define a derived parameter in the **Parameter Set** bar, Workbench keeps the unit specified at the moment of its creation as the native unit. You can select **Units** → **Display Values as Defined** to see the native unit. For example, if **meter** is the unit specified when a derived parameter is created, **meter** is the native unit. If you want the derived parameter to use **inch** as the native unit, you must specify **inch** before you create the derived parameter.

Related topics:

Expressions, Functions, Quantities, and Units

Creating or Editing Parameters

Chaining an Output Parameter to an Input Parameter

Expressions, Functions, Quantities, and Units

The Workbench expression parser supports standard math functions and operators, as well as units for quantities. Dimensional quantities are defined in units that are a combination of one or more separate units.

Workbench expression and mathematical function evaluation is based on the Python 2.6 programming language (See the Python website) and inherits some behavior as described here. All Python numeric and function capabilities can be used. For example, Python provides support for hexadecimal (base 16) numbers. Appending a zero and an x to the front of a number tells Python to treat the number as a hexadecimal numeric literal.

Expressions

When entering expressions, you must use the list and decimal separators defined in your locale settings. For example, if a comma is defined as a decimal separator and a semicolon defined as a list separator, use these when you type in an expression.

The expression defined for a custom Boolean parameter can be the Python values **True** or **False**, or it can be a Python logical expression such as `P1 > P2` or `P1 == 10 and P2 == 10`. For a parameterized Boolean parameter, you can select **True/False** from the drop-down in the **Value** column.

The expression defined for a custom string parameter must be quoted with single or double quotes, such as 'string value' or "string value".

Do not start expression definitions with an **=** operator. Given existing parameters P1, P2, and P3 to define a derived parameter P4 such that **P4=P1*P2*P3**, type the expression **P1*P2*P3** in the **Value** field.

Outline of All Parameters				
	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	⊟ Input Parameters			
3	⊟ 🔲 Geometry (A1)			
4	℗ P1	PCB_thickness	2	mm ▼
5	℗ P2	PCB_depth	1.5	mm ▼
6	℗ P3	PCB_width	18	mm ▼
7	℗ P4	PCB_vol	P1*P2*P3|	
*	℗ New input parameter	New name	New expression	
9	⊟ Output Parameters			
*	℗ New output parameter		New expression	
11	Charts			

When you click away from the field, the expression is solved.

Outline of All Parameters				
	A	**B**	**C**	**D**
1	ID	Parameter Name	Value	Unit
2	⊟ Input Parameters			
3	⊟ 📦 Geometry (A1)			
4	📌 P1	PCB_thickness	2	mm ▼
5	📌 P2	PCB_depth	1.5	mm ▼
6	📌 P3	PCB_width	18	mm ▼
7	📌 P4	PCB_vol	5.4E-08	m^3
*	📌 New input parameter	New name	New expression	
9	⊟ Output Parameters			
*	📝 New output parameter		New expression	
11	Charts			

Functions

Expressions support the following intrinsic functions, which support both standard numeric values and quantities as arguments.

abs(arg)	cos(arg)	int2(quantity, reference)	nint2(quantity, reference)	sinh(arg)
acos(arg)	cosh(arg)	log(arg)	pow(value, exponent)	sqrt(arg)
asin(arg)	exp(arg)	loge(arg)	round(arg)	tan(arg)
atan(arg)	fabs(arg)	log10(arg)	round2(quantity, reference)	tanh(arg)
atan2(arg1,arg2)	floor(arg)	max(arg list)	sign(arg)	PI, pi − constant
ceil(arg)	floor2(quantity, reference)	min(arg list)	sign2(quantity, reference)	E, e − constant
ceil2(quantity, reference)	int(arg)	nint(arg)	sin(arg)	

Rounding Functions

The following rounding functions are designed to perform with, and restricted for use with, dimensionless quantity arguments for which the results are in the same unit system:

- ceil

- floor

- int

- nint

- round

- trunc

If these functions are used on quantities with units, an error message is displayed.

To perform rounding operations on quantities, use the following functions:

- ceil2

- floor2

- int2

- nint2

- round2

- trunc2

These functions support both numeric values and quantities. When used for quantities, the format is `func(quantity, reference)`. The quantity argument is converted to the unit of the reference argument, then the resultant quantity value is rounded to the nearest reference value.

For example, to calculate a ceiling value rounded up to the nearest whole meter, enter **`ceil2(quantity, 1[m])`**.

Sign Function

Like the rounding functions, the sign function is designed for dimensionless quantities and numbers. To perform this operation on quantities, use the sign2 function.

The format is `sign2(quantity, reference)`, but in this case the value of the reference argument has no significance in the calculation, only the unit used for conversion matters.

For example, both `sign2(3 [cm], 1 [in])` and `sign2(3 [cm], 10 [in])` return 1, indicating that the value of 3 cm converted into inches is a positive number.

Trigonometric Functions

The arguments for trigonometric functions are evaluated as follows:

- If the argument is a real number or integer, the argument is evaluated as radians.

- If the argument is a quantity (which has a value and units), the argument must be of type **`Angle`**. The evaluation is based on the supplied units (**`[deg]`** or **`[rad]`**).

Quantities

Expressions that involve quantities must be dimensionally consistent. The + and − operators require that the two operands have compatible units. For example, you cannot add an **`Area`** parameter to a **`Length`** parameter. Both units must be **`Length`** or both units must be **`Area`**. The **`*`** and **`/`** operators do not have this limitation. They allow one operand to be a quantity with a unit and the other operand to be a dimensionless factor. Or, they allow both operands to be quantities with units where the result is a different quantity type. For example, **`Length/Time`** results in a quantity with a **`Velocity`** unit.

If the expression defining a custom parameter results in a quantity, Workbench infers its quantity name from the value produced by the expression evaluation. Consequently, if the expression is a sum of multiple terms, every term must be using the same quantity name. For example, if the resulting quantity is `3.4[m^2]`, the new custom parameter must have its **Quantity Name** property set to **Area**. In some cases, more than one quantity name may be valid for the expression. In these cases, the **Quantity Name** property is not set, but you can select from a list for the **Quantity Name** property in the **Properties** pane for the parameter. Once the quantity name is determined from the unit of the value, it can be subsequently changed only by changing the **Quantity Name** property for the parameter. For example, you cannot change the expression from **Area** to **Volume** without changing the **Quantity Name** property.

Once the value quantity name (such as **Area**) is determined from the unit of the value, it can only be subsequently changed by changing the **Quantity Name** property for the parameter. You cannot change the expression (for example, from area to volume) without changing the **Quantity Name** property. Automatic unit conversion is only done when the quantity name is known. If you want to express a temperature difference, start with temperature in absolute units.

Units

You can include units, assuming that the unit makes sense in context of the expression. For example, `P2+3[mm]*P3` is valid if `mm` is a valid unit expression for P2 and P3. For example, `mm` is a valid unit expression if P2 is torque and P3 is force, or if P2 is area and P3 is length.

The project unit system is used to evaluate the expressions. For temperatures, absolute temperature values are used in expression evaluation. All quantity values in an expression are converted to the project unit system.

The general units syntax in Workbench is defined as `multiplier|unit|^power`, where:

- `multiplier` is a multiplying quantity or its abbreviation, such as mega (M) or pico (p)

- `unit` is the unit string or abbreviation, such as gram (g), pound (lb), foot (ft), or meter (m)

- `power` is the power to which the unit is raised

The following tables are examples of multipliers and commonly used units

Table 7: Unit Multipliers

Multiplier Name	Multiplier Value	Multiplier Abbreviation
exa	10^{18}	E
peta	10^{15}	P
tera	10^{12}	T
giga	10^{9}	G
mega	10^{6}	M
kilo	10^{3}	k
hecto	10^{2}	h
deca	10^{1}	da
deci	10^{-1}	d

Multiplier Name	Multiplier Value	Multiplier Abbreviation
centi	10^{-2}	c
milli	10^{-3}	m
micro	10^{-6}	u
nano	10^{-9}	n
pico	10^{-12}	p
femto	10^{-15}	f
atto	10^{-18}	a

Table 8: Example Quantities and Units

Quantity	Dimensionality	Example Units
Acceleration	Length Time^-2	m s^-2 ft s^-2 in s^-2 um ms^-2
Current	Current	A mA pA
Density	Mass Length^-3	kg m^-3 g cm^-3 lb ft^-3 slug in^-3 slinch in^-3
Electric Charge	Current Time	A s coulomb pA s
Energy	Mass Length^2 Time^-2	J BTU erg lbf ft slug in^2 s^-2
Force	Mass Length Time^-2	dyne N pdl lbf slug in s^-2

Quantity	Dimensionality	Example Units
Length	Length	m cm foot in mm micron ft um yard
Pressure	Mass Length^-1 Time^-2	Pa MPa N m^-2 bar torr mm Hg psi psf atm dyne cm^-2
Power	Mass Length^2 Time^-3	W BTU s^-1 HP erg s^-1 lbf ft s^-1
Temperature	Temperature	K C R F
Temperature Difference	Temperature	K C R F
Temperature Variance	Temperature^2	K^2 C^2 R^2 F^2

When typing units in an expression, you must enclose the units in square braces `[...]`. You generally do not see the braces when selecting units from a list of commonly used units. In general, units declarations must obey the following rules:

- A units string consists of one or more units quantities, each with an optional multiplier and optional power. Each separate units quantity is separated by one or more spaces.

- Abbreviations for multipliers and unit names are typically used, but full names are also supported.

- Powers are denoted by the ^ (caret) symbol. A power of 1 is assumed if no power is given. A negative power is typically used for unit division. For example, `[kg m^-3]` corresponds to kilograms per cubic meter.

- If you enter units that are inconsistent with the physical quantity being described, then an expression error occurs.

- Units do not have to be given in terms of the fundamental units, which are mass, length, time, temperature, angle, and solid angle. For instance, Pa (Pascals) and J (Joules) are both acceptable as parts of unit strings.

- Units strings are case sensitive. For example, `Kg` and `KG` are invalid units strings; `kg` is correct.

Caution:

When the specified project unit system uses the relative temperature units (C or F), the evaluation of expressions involving temperature, temperature differences, or temperature variances is a special case.

For the unit conversion of a specific temperature value, `1 degC = 274.15 K`. However, the unit conversion for a temperature interval (delta T) is `1 degC = 1 K`. The expression evaluator takes any temperature value and treats it as a specific temperature (not a temperature interval) by converting it to the absolute unit of the project unit system (either K or R). If the intent is to perform the evaluation in terms of temperature intervals, you need to start with temperatures in absolute units.

Similarly, any expressions to be evaluated in terms of temperatures or temperature differences are converted to absolute units for calculation. In an expression with a temperature unit raised to a power other than 1 or a unit involving both temperatures and other units (for example a temperature gradient [C/m]), the temperature is assumed to be a temperature difference.

Table 9: Scenarios in which temperature-related units are converted before and during expression evaluation

Scenario	Example
Temperature unit conversion	10[C] * 2 = 566.3[K] = 293.15[C]
Temperatures appearing as part of a mixed unit are converted as temperature intervals	10[C/m] * 50 [m] = 500[K]
Temperatures raised to a power other than 1 are converted as temperature intervals	10[C^3] * 10[C] = 2831.5[K^4]

Sound Pressure Level and A Weighted Sound Pressure Level are dimensionless quantities with units dB and dBA, respectively.

- Math operations between these two units or either of these and a numerical value will result in a value with no unit. For example, 10 db x 10 = 100. (no unit)

- Math operations between either of these units and a dimensional unit will result in a value with the dimensional unit. For example, 10 db x 10 m = 100 m.

Creating or Editing Parameters

The process of working with parameters begins when you define a parameter in an application, such as your CAD system, Mechanical, or Fluent. For information on defining parameters, refer to the documentation for each application in which you are working.

Workbench recognizes parameters defined in the individual applications and exposes them in the **Parameter Set** bar, which can be shared by multiple systems. The **Parameter Set** bar is the visual representation of the project's full parameter set. Double-clicking it opens the **Parameter Set** tab, which displays all parameters defined for all systems in your project. Each parameter is identified by its system of origin.
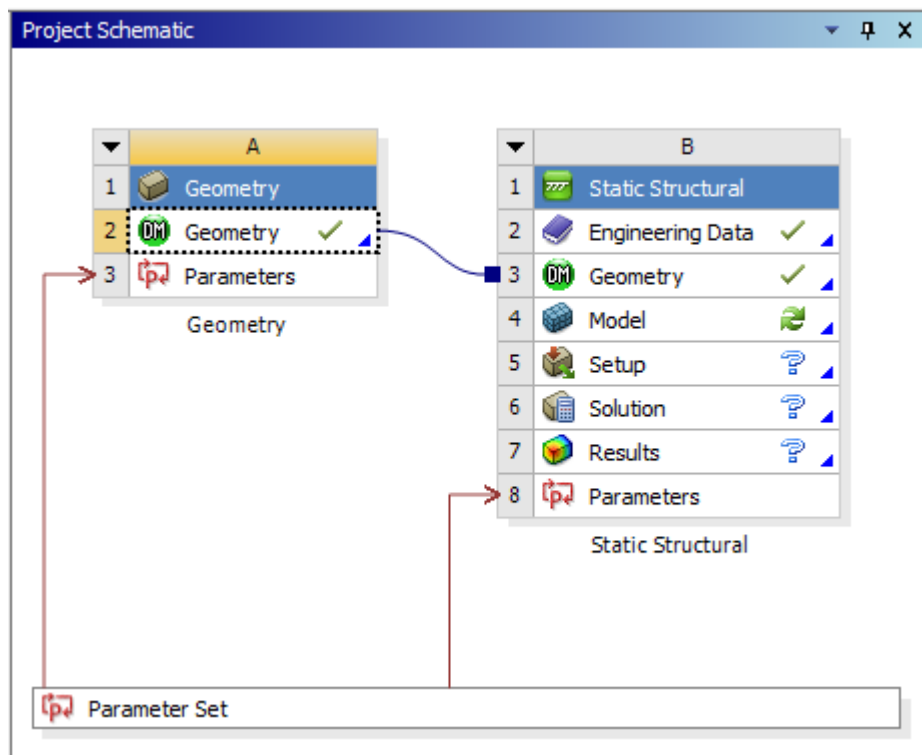
Additionally, each system with parameters has a **Parameters** cell, which you can double-click to open the corresponding **Parameters** tab. The **Parameters** tab for a given system displays all of the parameters defined for this system.

Each of these tabs has an **Outline** pane and a **Properties** pane for viewing and working with your parameters.

The **Outline** pane lists the parameters, grouping them into **Input Parameters** and **Output Parameters**. Input parameters affect the definition of the data model. Output parameters are analysis results that are quantities of interest for the design.

> **Note:**
>
> On a **Parameters** tab, the title in the header for the **Outline** pane displays the ID of the corresponding **Parameters** cell to indicate the source of the parameters. On the **Parameter Set** tab, the title is **Outline of All Parameters**.

For each parameter, the **Outline** pane shows an ID, name, current value, and unit system. You can edit most of these properties. The exceptions are the parameter IDs and the units for parameters with quantity values. When you select a parameter, its properties are shown in the **Properties** pane.

The **Properties** pane displays information for the object selected in the **Outline** pane. The objects available for selection include parameters, expressions, and charts.

To create or edit parameters:

1. To view the parameters, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. To add a new custom input parameter:

   a. In the **Outline** pane, to the right of **New input parameter**, click the **New name** cell and type a name for the parameter.

b. Click the **New expression** cell and type either a constant value or expression for the para-
    meter.

3. To add a new custom output parameter, in the **Outline** pane, to the right of **New output para-
    meter**, click the **New expression** cell and type either a constant value or expression for the
    parameter.

Note:

When you define a derived parameter in the **Parameter Set** bar, Workbench keeps
the unit specified at the moment of its creation as the native unit. You can select
Units → Display Values as Defined to see the native unit. For example, if **meter** is
the unit specified when a derived parameter is created, **meter** is the native unit. If
you want the derived parameter to use **inch** as the native unit, you must specify **inch**
before you create the derived parameter.

4. If you have parameterized the **Assignment** property of the Body or Part objects or the **Material Name** property of a Material Assignment object in the Mechanical application, select the desired material from the drop-down list in the **Value** cell next to the parameter name.

	A	B	C
1	ID	Parameter Name	Value
8	P33	mp_1_sublaminate.number_of_layers	1
9	P34	mp_2_ud.number_of_layers	6
10	P35	mp_1_sublaminate.ply_angle	0
11	P36	mp_2_ud.ply_angle	0
12	P38	mp_2_ud.ply_material	1
13	P39	UD.thickness	0.1
14	P42	CylindricalRule.left_binding.radius	180
15	P43	CylindricalRule.right_binding.radius	180
16	P47	001-02-103-p__foam_top_milled	Corecell_A550
17	P48	001-02-103-p__foam_top_m	Epoxy_Carbon_UD_230GPa_Pre
18	P44	Binding_Radius	Epoxy_Carbon_UD_230GPa_We
			Epoxy_Carbon_UD_395GPa_Pre
*	New input parameter	New name	SAN_Foam_103kgm3
			Corecell_A550
20	Output Parameters		

5. To edit the name of a parameter, in the **Outline** pane, click the **Parameter Name** cell and type the new name.

6. To delete custom parameters:

   a. In the **Outline** pane, click the **ID** cell for a parameter to select it, or press **Ctrl** and click the **ID** cell of multiple parameters to delete.

   b. Right-click the selection and select **Delete Selected Unlinked Parameters** from the context menu.

7. To show the properties for a parameter, click the **ID** cell for that parameter.

8. To add or edit an expression, in the **Properties** pane, click the **Expression Value** cell and type the new expression.

9. For an expression, once the quantity name (such as area) is determined from the unit of the value, it can only be changed by changing the **Quantity Name** property for the parameter. In the **Properties** pane, from the **Quantity Name** drop-down list, select a new value.

10. To add or edit the parameter description, in the **Properties** pane, click the **Description Value** cell and type the new description.

11. Once you have finished adding and editing parameters, to register the changes and return the values of the output parameters. Note that an update operation can be lengthy, depending on the analysis details.

Chaining an Output Parameter to an Input Parameter

You can chain an output parameter to an input parameter to allow an input to be driven directly from the current value of an output, provided that the chaining does not create a circular dependency.

To chain an output parameter to an input parameter:

1. To view the parameters, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. In the **Outline** pane, to the right of **New input parameter**, click the **New name** cell and type a name for the parameter.

3. Click the **New expression** cell and type either a constant value or expression for the parameter.

4. Update the project, system, or Parameters cell (p. 242) to register the changes and return the values of the output parameter. Note that an update operation can be lengthy, depending on the analysis details.

Working with Design Points

Each design point is a single set of parameter values representing one design alternative. Basically, you can think of a design point as a snapshot of your design given a set of input parameter values, where output parameter values are calculated by an update of the project. Design points allow you to perform what-if studies and can be created in the **Parameter Set** bar or generated with ANSYS DesignXplorer.

You manage design points in the **Table** pane for the **Parameter Set** tab or on a **Parameter** tab. The **Table** pane displays the design points.

From the **Table** pane, you can see existing design points and perform the following operations:

• Add new design points

• Duplicate existing design points

• Enter or change the input parameter values for design points

• Delete existing design points except for the current one

• Specify the design point update order

• Update design points, either locally or through Ansys Remote Solve Manager

• Retain design point data within the project for design points other than the current, which by default has retained data

- Preview the image from the , assuming that the project has design points, is solved, and a saved project report exists

- Set a design point with retained data as the current design point

- Export selected design points to separate projects

- Export a CSV file of the design points table

The design point that you interact with on the **Project Schematic** is always the current design point. Initially, the current design point is DP 0. However, you can set any design point that has retained results to be the current design point. You cannot rename or delete the current design point.

If you add a new design point, change its input parameter values, and update the project, you calculate output parameter values for this new design point. You can then perform what-if studies to compare this design to other designs.

Related topics:

Multiple Design Points Workflow

Adding a New Design Point by Duplicating an Existing Design Point

Adding a New Design Point by Entering Values

Updating Design Points

Retaining Design Point Data

Reviewing the Retained Data Column

Retaining Parameter Data for Solved Design Points with DesignXplorer

Setting a Different Design Point as Current

Copying Input Parameter Values to the Current Design Point

Exporting Design Points to New Projects

Exporting Design Point Parameter Values to a Comma-Separated Values File

Performing and Retaining Partial (Geometry-Only) Updates

Design Point States

Multiple Design Points Workflow

Workbench allows you to create multiple design points and generate comparison studies (what-if studies) of input and output parameters.

To analyze your simulation across several design points:

1. In Ansys applications, generate input parameters for the current project (and, if appropriate, specify output parameters to be created).

   A **Parameters** cell is added to the relevant systems and a **Parameter Set** bar is added to the project. The current project is designated as design point 0, or **DP 0**.

2.

3. Create multiple design points by duplicating an existing design point or by entering values.

4. Set whether you want to retain the files generated during the design point update for further analysis.

    Workbench saves the data for the current design point, any retained design points, and the values of the output parameters computed for each design point.

5. Update the design points.

Adding a New Design Point by Duplicating an Existing Design Point

When you duplicate a design point, all parameters, parameter values, design point states, and design point export settings are copied. You can then modify any of these values as required.

To duplicate an existing design point:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set Bar**.

2. Right-click a design point in the **Table** pane and select **Duplicate** from the context menu.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
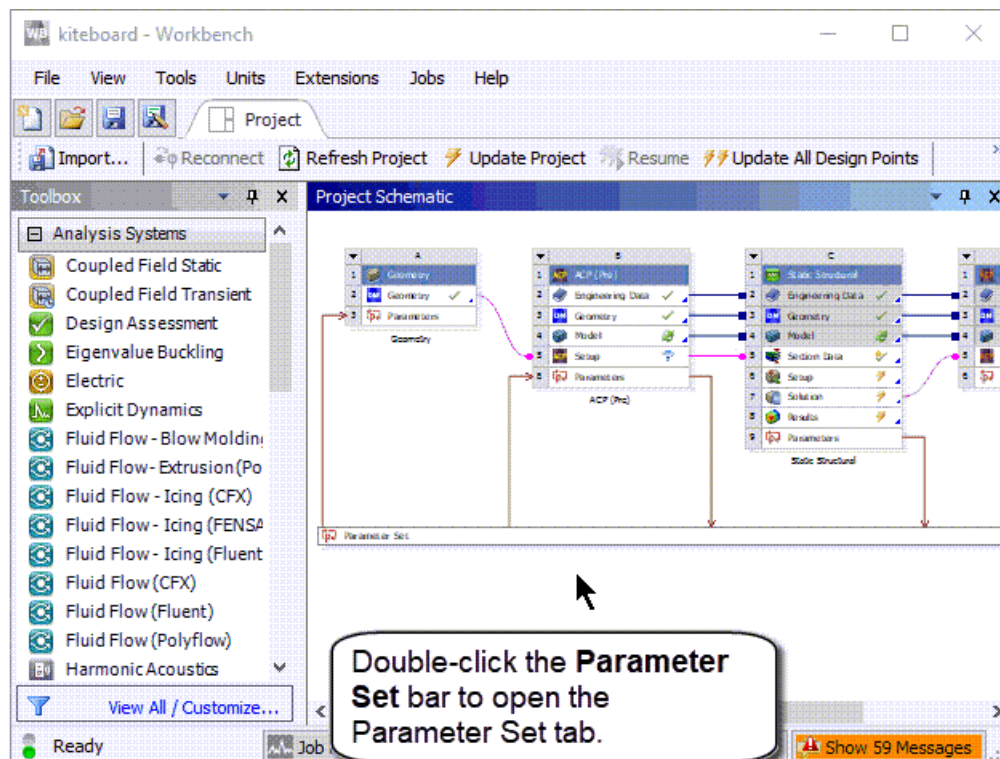
Adding a New Design Point by Entering Values

You can add a new design point by entering parametric values in the **Table** pane.

To enter values:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. In the bottom row of the table, below the existing design points, type a parametric value in one or more of the cells.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.
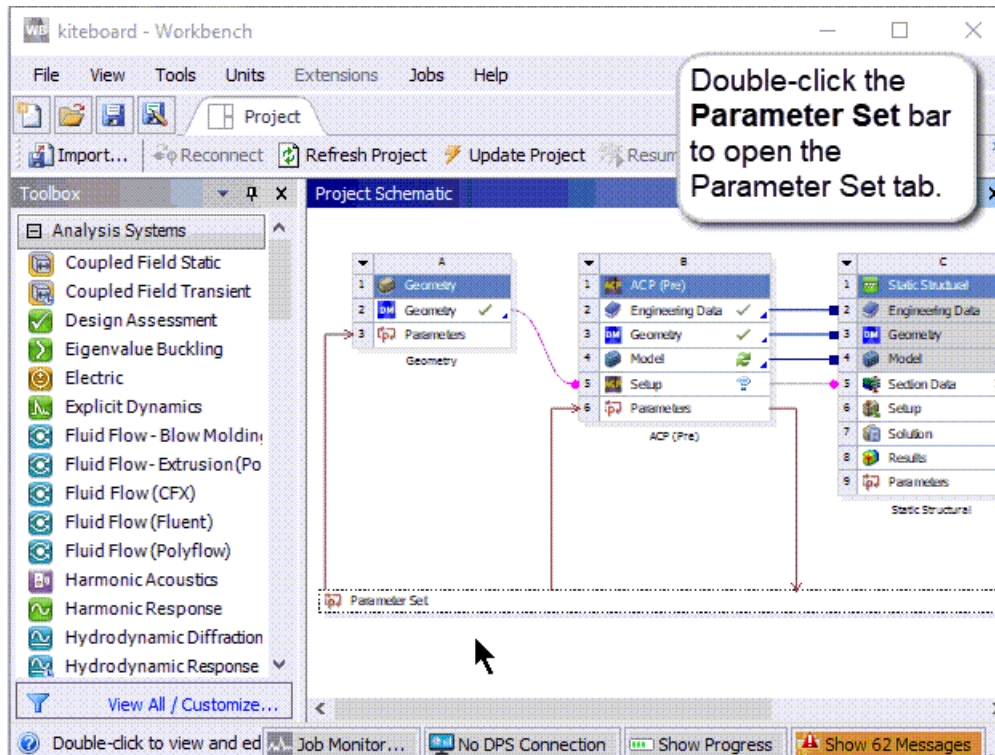
Updating Design Points

Output parameter values for a design point are calculated when you update the design point. You can update only the current design point, a set of selected design points, or all design points. Updating design points updates solution data only where output parameters have been defined.

You have several options for updating design points. The settings specified by the **Design Point Initialization** property apply. For more information, see Specifying the Initialization Conditions for a Design Point Update (p. 311).

When you update multiple design points, you can choose the order in which design points update. You can also choose the initial conditions with which they update.

You may want to specify the order for design point updates to improve efficiency. For instance, if several design points use the same geometry parameter values, it is more efficient to process them together so as to update the geometry only once.

By default, when each design point is updated, the design point is initialized with the data of the design point designated as current. Retained design points with valid retained data are exceptions

because they do not require initialization data. In some cases, it may be more efficient to update each design point starting from the data of the previously updated design point, rather than restarting from current design point each time.

To update design points in Remote Solve Manager (RSM), see Updating Design Points in Remote Solve Manager (p. 352).

Non-parametric changes made to a project with up-to-date design points may cause all existing design points to go out-of-date, if the change is related to the parametric study being performed. Any change not relevant to the parametric study, such as adding a standalone system or making a change downstream of the study, should not cause design points to go out-of-date. For changes that invalidate the design points table, you must update the project, which can require significant time and computing resources. You should always save your project after updating all design points and before further modifying the project.

In most cases, a design point update applies only to cells affected by parameter changes and to cells downstream. Cells without associated parameters or cells whose associated parameters did not change are not updated.

Related topics:

Setting the Design Point Update Option

Updating Only the Current Design Point

Updating Selected Design Points

Update All Design Points

Fixing an Interrupted Design Point Update

Changing the Design Point Update Order

Specifying the Initialization Conditions for a Design Point Update

Setting Systems using CFD-Post to Update with Design Points

Setting the Design Point Update Option

Before updating design points, you must set the location where the updates are processed.

To set the design point update option:

1. On the **Project Schematic**, right-click the **Parameter Set** bar and select **Properties** from the context menu.

2. In the **Properties** pane, from the **Update Option** drop-down list, select from the following options:

   • **Run in Foreground**: Runs the design point update within the current Workbench session. This option is appropriate for quick-running updates that fit within the resources of your workstation. This option is also the most robust as it is not possible to make changes that impact the design points. When a design point update is executing in the foreground, you cannot change or save the project, but you can interrupt or stop the update.

   • **Remote Solve Manager**: Runs the design point update in the background by submitting it to RSM using HPC shared licensing features.

Use this option for long-running updates that do not fit within your workstation's resources. When you submit an update, RSM can send it to a remote compute cluster or to a Cloud portal. RSM can also submit jobs to the local machine to allow the queuing of updates on your workstation.

For more information on submitting design point updates to RSM, see Updating Design Points in Remote Solve Manager (p. 352).

- **Submit to Design Point Service (DPS)**: Runs the design point update in the background by submitting it to DPS. Use this option to quickly and efficiently evaluate large numbers of design points simultaneously using multiple compute resources, including ad-hoc networks, Amazon web services, and Remote Solve Manager.

  For more information on submitting design point updates to DPS, see Working with Design Point Service (p. 363).

Updating Only the Current Design Point

To update just the current design point, update the project (p. 243).

Updating Selected Design Points

You can choose a set of design points from the table to update.

To update selected design points:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. Press **Ctrl** and click each design point in the table to update.

3. Right-click the selection and select **Update Selected Design Points** from the context menu.

308

If the update for one of the design points fails, the **Update Failed, Update Required** ( ) icon is displayed for the failed design point, but the next design point update begins immediately. The **Messages** pane opens, showing the error message for the one or more design points that failed. When the update process is complete, a failure summary dialog box opens. Review the error messages for details on which design points failed to update.

Update All Design Points

You can update all design points in the table.

To update all design points:

1.   To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2.   To update the design points:

   •   From the menu bar select **Tools** → **Update All Design Points**.

   •   On the Project tab toolbar click **Update All Design Points**.

   •   Right-click the white space on the **Project Schematic** and select **Update All Design Points** from the context menu.

If the update for one of the design points fails, the **Update Failed, Update Required** ( ) icon is displayed for the failed design point, but the next design point update begins immediately. The **Messages** pane opens, showing the error message for the one or more design points that failed.

When the update process is complete, a failure summary dialog box opens. Review the error messages for details on which design points failed to update.

Fixing an Interrupted Design Point Update

If the update of a design point is interrupted, the partially updated icon (⬤) is displayed beside the values of the output parameters for the interrupted component. The next time that this design point is selected for update, a dialog box opens.

To continue, select from the following options:

- **Use Partially Updated**: Accepts output parameters that are only partially updated as up-to-date, using the existing results to update design points.

- **Update All**: Restarts the interrupted update, recalculating results for all output parameters that are partially updated and then updating all design points. The dialog box notes that this can be a lengthy process.

Changing the Design Point Update Order

By default, design points are updated in the order in which they appear in the design points table. The **Update Order** column shows the order number for each design point. To view the column, right-click in the table and select **Show Update Order** from the context menu.

To change the order that design points are updated in, select from the following options:

- Right-click the table and select **Optimize Update Order** from the context menu.

  An optimal update order is determined by an analysis of parameter dependencies in the project and a scan of parameter values across all design points. The **Update Order** value for each design point is updated automatically.

  > **Note:**
  >
  > The primary goal of **Optimize Update Order** is to reduce the number of geometry and mesh system updates. In general, the optimized order is driven by the order of the update tasks, which means that the components are updated depending on their order in the system, their data dependencies, their parameter dependencies, and the state of the parameters. This works best in a horizontal schematic with geometry as the first separate system. In a single vertically-integrated system project, the parameters defined in the upper components are considered the most important. In some cases, because of the order of parameter creation or the presence of engineering data parameters, you may find that the first columns are not geometry parameters. In this case, we recommend that you use the **Optimize Update Order** option to sort by geometry parameters.

- Edit the values in the **Update Order** column.

- Sort the table by one or several columns and then right-click in the table and select **Set Update Order by Row**.

The **Update Order** value for each design point is regenerated to match the sorting of the table.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, access this section in the online help. The interface shown may differ slightly from that in your installed product.

	A	B	C	D	E
1	Name	Update Order	P31 - ParallelRule.1.neg_distance	P32 - ParallelRule.1.pos_distance	P33 - mp_1_sublaminate.number_of_layers
2	Units				
3	DP 9	0	-223.93	166.22	2
4	DP 10	0	-169.32	164.07	2
5	DP 11	0	-170.57	163.71	2
6	DP 12	0	-171.98	165.06	2
7	DP 13	0	-409.53	161.56	2
8	DP 14	0	-172.41	165.75	2
9	DP 15	0	-250.93	191.62	2
10	DP 0 (Current)	1	-500	500	1
11	DP 1	2	-500	500	1
12	DP 2	3	-500	500	1
13	DP 3	4	-500	500	1
14	DP 4	5	-500	500	1
15	DP 5	6	-500	500	1
16	DP 6	7	-500	500	1
17	DP 7	8	-500	500	1
18	DP 8	9	-500	500	1
*					

> To sort the design points for the most efficient update, right-click the Update Order column and select **Optimize Update Order** from the context menu.

Specifying the Initialization Conditions for a Design Point Update

When a design point is updated, the solver is initialized with data. Design points with valid retained data use that as their initialization data. Other design points either:

- Initialize with the data of the design point designated as current (**From Current**)

- Initialize starting from the data of the previously updated design point (**From Previous Updated**).

Using the **From Previous Updated** initialization value works best when:

- Sequential design points share the same geometry. For example, if DP 2 has the same geometry as DP 1, but both of these differ from DP 0, changing the default behavior saves the computational cost of updating both the geometry and the mesh for DP 2.

- You have modified the design point update order in the **Update Order** column, either by editing the values manually or by generating them with the **Set Update Order by Row** option.

- The design point update order has been optimized, either by the **Optimize Update Order** option or by a DesignXplorer optimization.

To set how design points are initialized:

1. On the **Project Schematic**, right-click the **Parameter Set** bar and select **Properties** from the context menu.

2. From the **Design Point Initialization** drop-down list, select the required initialization value.

Setting Systems using CFD-Post to Update with Design Points

For systems using CFD-Post results, you can specify that the results cell is always updated with a design point update operation, even when the update does not affect parameter values.

To set systems using CFD-Post to update with design points:

1. On the **Project Schematic**, right-click a CFD-Post **Results** cell and select **Properties** from the context menu.

2. In the **Properties** pane, select the **Always Include in Design Points Update** check box.

Retaining Design Point Data

By default, Workbench does not retain design points after they are solved. It saves the calculated data only for the current design point. For other design points, only the parameter values are saved.

In some cases, you might want to switch back and forth between multiple designs within the same project. To do this, select one or more design points and retain their calculated data within the project.

To retain design points data:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. For each design point you want to retain data for, select the **Retained** check box.

   > **Note:**
   >
   > When this check box is selected, data for all systems inside the Workbench project are stored for the design point, regardless of whether a system has connection to the parameters. When only a small portion of the system is parameterized, retaining the data is not efficient. This is a known limitation.

3. To generate retained design point data, .

Once design point data has been retained:

- You can verify that the data has been retained and see the availability and state of the retained data. For more information, see .

- You can view multiple designs by setting different design points as current. For more information, see .

Reviewing the Retained Data Column

You can verify that data has been retained by viewing the icon in the **Retained Data** column for each design point. This icon indicates the availability, validity, and state of the retained data for the design point.

Different kinds of changes to the project have different impacts on retained data. Changes to the parameter values for the design point cause the retained data to become out-of-date but still valid. Non-parametric changes cause the retained data for non-active design points to become invalid.

Note:

During a local design point update, only the design point being updated is active. When no update is in progress, only the current design point is active.

The state of the design point and the validity of its retained data do not necessarily match. It is possible for a design point to be up-to-date while its retained data is invalid or vice versa.

When used in the **Retained Data** column, icons have slightly different meanings than they do elsewhere in Workbench. The following table describes the meaning of each icon within the context of retained design point data.

Icon	State/Validity	Description
✔	Valid and up-to-date	The design point has valid retained data that is up-to-date. The retained data is available to be exported.
✔	Valid but out-of-date	The design point has valid retained data, but the retained data is out-of-date because of a change to the design point's parameter values. The retained data is available to be exported.
⚡	Invalid or non-existing	The design point either does not have retained data or previously retained data has been invalidated by a non-parametric change. The retained data is not available to be exported.

Each state is illustrated in the image below, as follows:

- **DP 0 (Current)** has retained data that is valid and up-to-date. The current design point always has valid retained data and the **Retain** check box is disabled.

- **DP 1** has valid retained data that is current and up-to-date.

- **DP 2** either does not have retained data or has retained data that is invalid.

- **DP 3** has retained data that is valid but out-of-date.

	A	B	C	D	E	F	G
Table of Design Points							
1	Name ▼	P1 - Pressure Magnitude ▼	P2 - Equivalent Stress Minimum ▼	P3 - Equivalent Stress Maximum ▼	☐ Retain	Retained Data	Note ▼
2	Units	Pa ▼	Pa	Pa			
3	DP 0 (Current)	2000	19.512	1.0622E+05	☑	✓	Harmonic testing
4	DP 1	5000	48.779	2.6556E+05	☑	✓	
5	DP 2	3000	29.267	1.5934E+05	☑	⚡	Harmonic testing
6	DP 3	4000	⚡ 58.535	⚡ 3.1867E+05	☑	✔	Harmonic testing
7	DP 4	6000	39.023	2.1245E+05	☐		
*					☐		

Retaining Parameter Data for Solved Design Points with DesignXplorer

When design points are updated, DesignXplorer provides the following methods of retaining parameter data for solved design points:

- It saves design point data to the design point cache. DesignXplorer reuses data from the cache when you preview or update a **Design Exploration** system. For more information, see Cache of Design Point Results in the *DesignXplorer User's Guide*.

- It exports design point data into design point log files. You can import the log file back into DesignXplorer as needed. For more information, see Design Point Log Files in the .

- It allows you to export all design point values to an ASCII file. This functionality is available for DesignXplorer charts or tables containing design point data, including the design points table. You can then use the parameter values in the file with other programs for further processing or import into DesignXplorer.

> **Note:**
>
> Both the design point log files and the ASCII file are formatted in the extended CSV file format. For details, see Exporting Design Point Parameter Values to a Comma-Separated Values File (p. 316).

Setting a Different Design Point as Current

Once multiple design points have retained data, you can switch back and forth between these multiple designs within the project. To view the design associated with a given design point, you set it as the current design point. Only design points with retained data can be set to current, and only one design point can be set to current at a time.

To set a design point as the current design point:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. In the **Table** pane, right-click a design point that has retained data (p. 312) and select **Set as Current** from the context menu.

The design point becomes the current design point for the project. In the **Name** column, **(Current)** displays after the name of the current design point. Setting a design point as current does not automatically update it.

> **Note:**
>
> In a project that includes a Microsoft Office Excel system and uses retained design points, switching to a different up-to-date design point automatically updates the Excel system. This can be time-consuming if the spreadsheet contains macros or extensive calculations.

Copying Input Parameter Values to the Current Design Point

An alternate method to switching design points is to copy a different design point's input values to the current design point. Copying the input values does not automatically update the design points.

To copy input parameter values to the current design point:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. In the **Table** pane, right-click a design point and select **Copy inputs to Current** from the context menu.

Exporting Design Points to New Projects

To work with the calculated data for a design point other than the current design point, you can export the design point to a separate project. A new project is created for each design point that is exported.

When you export a design point, if the design point has already been exported, the existing exported files are overwritten.

> **Note:**
>
> DesignXplorer data is not a part of design points and is not exported. DesignXplorer is a consumer of design points but does not define design points. Consequently, it is not involved when a design point is updated, or by extension, exported.

To export design points:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. In the **Table** pane, select the design points.

3. Right-click the selection and select **Export Selected Design Points** from the context menu.

The data of each design point is saved to a new project, named `filename_dpn` (where `dpn` indicates the number of the exported design point), with a project file named `filename_dpn.wbpj`. Working files are saved to a `filename_dpn_files` directory that is a sibling to the original `filename_files` project folder.

The content of the new project depends on the state of the design point's retained data at the time of the export:

- If valid retained data is available for an exported design point, the retained data is used to create the new project. This retained data can be either up-to-date or out-of-date.

- If no valid retained data is available for an exported design point, the project exported is an out-of-date project based on the current design point with the parameter values for this design point applied. No valid retained data exists if the design point is either not selected to retain data or is selected but retained data is not yet generated or has become invalid. You are asked to confirm before the export of such a design point is performed. When you open the exported project, it is out-of-date and requires an update to solve for the exported design point.

After an export, you must save the parent project before you can export design points again.

If a design point fails to export, the files for this design point remain in the `project_files\dpn` directory until you delete them manually or attempt to update the design point again.

Exporting Design Point Parameter Values to a Comma-Separated Values File

From the design points table, you can export the design point parameter values to a Comma-Separated Values (CSV) file, which you can then use with other software tools for further processing. This file is created in an extended file format. In addition to supporting standard CSV formatting, it supports several Workbench-specific formatting conventions. The values are always exported in units as defined in Workbench.

The file format includes the following components:

- Values are separated by commas.

- The optional header line indicates the name of each column.

- Each line is an independent record made of fields separated by commas.

- The format is not dependent on the locale, which means that the real number 12 and 345 one-thousandths is always written as **12.345**, regardless of the regional settings of the computer.

- If a line starts with the character #, it is considered a comment line rather than a header or data line and is ignored.

- The header line is mandatory. It is the line where each parameter is identified by its ID (P1, P2, …, Pn) to describe each column. The IDs of the parameters in header line match the IDs of the parameters in the project.

- The first column is used to indicate a name for each row.

- A file can contain several blocks of data, with the beginning of each block being determined by a new header line.

To export design point parameter values to a CSV file:

1. To open the **Table** pane, double-click the **Parameters** cell of a system or the **Parameter Set** bar.

2. In the **Table** pane, right-click a cell and select **Export Table Data as CSV** from the context menu.

A file similar to the following is generated:

```
#
# 10/1/2012 10:38:01 AM
# The parameters defined in the project are:
# P1 - WB_B [mm], P2 - WB_D [mm], P3 - WB_L [mm], P4 - WB_P [N], P5 - WB_E [MPa],
# P10 - WB_SIG [MPa], P8 - WB_DIS [mm], P9 - WB_BUCK [N]
#
# The following header line defines the name of the columns by reference to the parameters.
Name, P1, P2, P3, P4, P5, P10, P8, P9
DP 0, 2, 5, 100, 1000, 200000, 12000, 80, 1028.91145833333
DP 1, 3, 5, 150, 1000, 200000, , ,
DP 2, 5, 5, 20, 1000, 200000, , ,
```

Performing and Retaining Partial (Geometry-Only) Updates

To check your geometry for parametric failures, you can perform a geometry-only design point update. When the partial results have been calculated, you can save them to the project to be used in subsequent design point updates. You might do this if you want to generate the geometry updates on one machine and then move the project to a machine that cannot update geometry.

To perform a geometry-only update:

1. On the **Project Schematic**, right-click the **Parameter Set** bar and select **Properties** from the context menu.

   If the **Properties** pane is already visible, click the **Parameter Set** bar to refresh the **Properties** pane with the design point settings.

2. In the **Properties** pane, from the **Update Option** drop-down list, select **Run in Foreground**.

3. From the **Partial Update** drop-down list, select **Geometry**.

4. To retain the results from the partial update, from the **Retain Partial Update** drop-down list, select **Geometry**.

   > **Note:**
   >
   > This setting causes the geometry results to be saved whether you perform a partial update or a full update.

Partial update data can exist from either a partial or full design point update. To remove existing partial update data, set **Retain Partial Update** to **None**. The partial retained data is deleted from the project the next time that design points are updated.

Design Point States

On the **Parameters Set** tab and any **Parameters** tab, the **Outline** and **Table** panes display icons on output parameters to indicate their states.

- An output parameter that is up-to-date displays no icon. Only the value of the output parameter is shown.

- An output parameter that is out-of-date displays the **Update Required** icon ( ). Either the output parameter has not been solved or requires an update because local data has changed.

- An output parameter that has failed to update displays the **Update Failed, Update Required** icon ( ).

- An output parameter that has partially updated values because of an interruption to either a project update or a design point update displays the **Partially Updated** icon ( ).

During cell, system, project, and both local and remote design point updates, the **Outline** and **Table** panes update the states of the output parameters.

Changes elsewhere in the project could cause the state of some, but not all, output parameters to go out-of-date and require update. In those cases, only those output parameters affected by the changes are shown as requiring an update. Output parameters that are not affected by the changes remain up-to-date.

Changes to derived parameters or associated expressions are recalculated immediately rather than requiring an update. If you add or change derived parameters, other parameters are not affected by this change and so do not require an update.

Design points that are being updated using RSM display the **Pending** icon ( ) in the design points table while the remote design point update is in progress. When each updated design point is retrieved, associated output parameters reflect the results of the update.

Working with Ansys Remote Solve Manager

Ansys Remote Solve Manager (RSM) provides the central framework for job submission to an established cluster or third-party Cloud. This enables you to access powerful compute resources when needed.

To enable job submission to a cluster or Cloud portal, an administrator must create *configurations* in RSM. An RSM configuration contains information about the HPC resource where jobs are submitted (such as the name of the cluster submit host, or URL of the web portal), file transfer methods, and the queues that are available for job submission. (See RSM Configuration in the *Remote Solve Manager User's Guide*.)

RSM configurations enable RSM to integrate with the following types of HPC resources:

- **Commercial clusters**. Includes Microsoft HPC, LSF, PBS Pro, UGE (SGE), TORQUE with Moab, and custom clusters.

- **Ansys Cluster (ARC)**. An ARC operates in the same way that a commercial cluster does, running Ansys applications in local or distributed mode, but uses its own scheduling capability rather than that of a third-party job scheduler.

  Every RSM installation includes a default `localhost` ARC configuration that enables you to run solutions and updates on your local machine, essentially making your machine a single-node cluster.

- **Third-party Cloud**. Through RSM customization, provides access to Cloud-hosted compute services.

  > **Note:**
  >
  > - For a Microsoft HPC cluster, PSSH is not supported on Microsoft Windows for job submission.
  >
  > - PBS, LSF, UGE and TORQUE clusters on Windows are not supported.

Once the configurations have been created, the administrator must share them with users. You must then either copy the configurations into your default RSM configuration directory, or, if they reside in a shared directory, change the path of your RSM configuration directory to the path of the shared directory. (See Sharing and Accessing RSM Configurations in the *Remote Solve Manager User's Guide*.)

Once RSM configurations have been made available to your local machine, the *RSM queues* that are defined in those configurations are listed in the **RSM Queue** drop-down menu when you select the **Submit to Remote Solve Manager** option for updates and solutions. Each RSM queue is associated with a specific *HPC queue* that has been created on the cluster/portal side. An HPC queue determines the machine(s) where jobs are run.

Depending on the RSM queue chosen, jobs can be run locally in the background on your local machine, sent to a remote machine, or distributed across multiple machines.

Note:

The naming scheme for jobs sent to RSM is as follows:

- For Mechanical jobs:

  [client machine name]-[Workbench project name]-[design point name]-[Model node name (including cell ID)]-[Environment node name (including cell ID)]-[Solution node name (including cell ID)]

  Example: MyWorkstation-MyProject-DP0-Model (A4)-Static Structural (B5)-Solution (B6)

- For CFX/Fluent/Polyflow/Mechanical APDL jobs:

  [client machine name]-[Workbench project name]-[design point name]-[System name]-[Component name (including cell ID)]

  Example: MyWorkstation-MyProject-DP0-CFX-Solution (C3)

Jobs submitted to RSM can be monitored using the Workbench **Job Monitor** (see Monitoring and Controlling Remote Solve Manager Jobs in Workbench (p. 348)).

This chapter assumes that RSM has been set up according to the instructions in the *Remote Solve Manager User's Guide* (see RSM Installation and Startup and RSM Configuration).

Related topics:

Specifying Credentials for Remote Solve Manager Job Submission

Submitting Solutions to Remote Solve Manager

Submitting Workbench Project Updates to Remote Solve Manager

Submitting System Coupling Jobs to Remote Solve Manager

Submitting Mechanical Application Jobs to Remote Solve Manager

Submitting Mechanical APDL Jobs to Remote Solve Manager

Submitting Fluids Jobs to Remote Solve Manager

Monitoring and Controlling Remote Solve Manager Jobs in Workbench

Updating Design Points in Remote Solve Manager
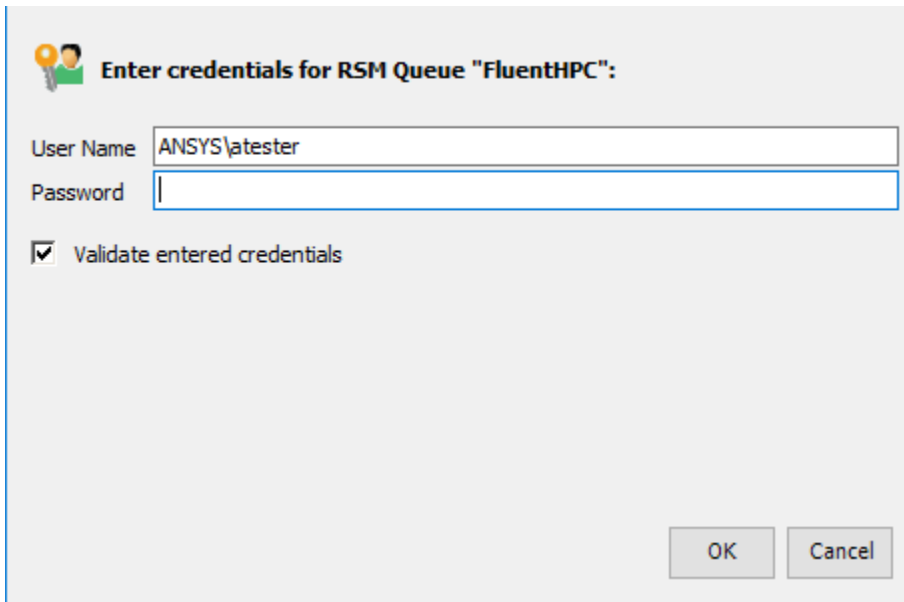
Specifying Credentials for Remote Solve Manager Job Submission

Remote Solve Manager (RSM) needs to acquire and cache your credentials to be able to submit jobs to an HPC resource on your behalf. RSM's direct integration with Workbench and built-in caching capability automates this process to a great extent.

When you submit a job to RSM from Workbench for the first time, and your credentials have not yet been cached for the RSM queue in RSM, or the cached password does not validate because your OS password has changed or expired, you are prompted to specify credentials for that queue:

If the credentials validate, the job is submitted to the HPC resource. If the credentials do not validate, the job is aborted.

Submitting Solutions to Remote Solve Manager

You can use the **Solution Process** properties of a system's **Solution** cell to submit the solution to Remote Solve Manager (RSM).

Use this option for long-running solutions that do not fit within your workstation's resources. When you submit a solution, RSM can send it to a remote compute cluster, or to a Cloud portal. RSM can also submit jobs to the local machine to allow the queuing of solutions on your workstation.

Note:

- When submitting solve jobs to the remote cluster, the license preferences set for your local machine (the RSM client) may not be the same as the license preferences set for the cluster. In this case, the cluster license preferences are used for all jobs.

- Solution cell updates via RSM are not supported for System Coupling; System Coupling jobs must be sent to RSM either as project updates or design point updates. For more information, see .

To submit solutions to RSM:

1. Right-click the **Solution** cell of a system and select **Properties** from the context menu.

2.   In the **Properties** pane, from the **Update Option** drop-down list, select **Submit to Remote Solve Manager**.

Important:

When design points are configured to be updated in RSM, the **Solution** cell cannot also be updated in RSM. In this case, change the **Solution** cell update settings by setting the **Update Option Solution Process** property to **Run in Foreground**.

---

3.   Set the following additional properties:

Properties of Schematic A3: Solution		
A	**B**	**C**
1 Property	Value	Unit
2 ⊟ General		
3 Component ID	Solution	
4 Directory Name	FLU	
5 Use Setup Launcher Settings	☐	
6 Precision	Single Precision ▼	
7 Display Mesh After Reading	☑	
8 Embed Graphics Windows	☑	
9 Use Workbench Color Scheme	☑	
10 Load ACT Start Page	☐	
11 Environment Path		
12 Initialization Method	Program Controlled ▼	
13 Solution Monitoring	☐	
14 Generate Solution Monitor Plots for Report	☐	
15 Data Interpolation	☑	
16 ⊟ Notes		
17 Notes		
18 ⊟ Used Licenses		
19 Last Update Used Licenses	ANSYS CFD	
20 ⊟ Parallel Run Settings		
21 Interconnect	default	
22 MPI Type	default	
23 ⊟ Solution Process		
24 Update Option	Submit to Remote Solve Manager ▼	
25 RSM Queue	HIGH-MEM ▼	
26 ⊟ RSM Queue Details		
27 HPC Configuration	WinHPC-HQ	
28 HPC Queue	XLM	
29 HPC Type	Windows HPC	
30 Job Name	Workbench	
31 Download Progress Information	☑	
32 Progress Download Interval	30	s ▼
33 Execution Mode	Parallel ▼	
34 Number of Processes	2	

- **Solve Process Setting** (Mechanical application only): Select the solving configuration defined in Mechanical.

- **RSM Queue**: Specifies a queue defined in RSM that maps to a specific HPC queue on the cluster/portal side. Any RSM queues currently defined in RSM appear in this drop-down list. The **RSM Queue Details** section displays the **HPC Configuration**, **HPC Queue**, and **HPC Type** (for example, ARC, WinHPC, or LSF) that the RSM queue is associated with.

  If a queue has been recently added in RSM and you do not see it in the list, select **Refresh Queues** to update the queue list.

- **Job Name**: Displays the name reported in the RSM job log. The default name for a Workbench job is `Workbench`, or whatever is set for the **Default Job Name** in your Workbench options. The default name for a Mechanical job submitted from the Mechanical application is `Mechanical`. The job name provides a traceable piece of data that administrators can use to track job submissions to a cluster (for billing purposes, for example).

  To override the default job name, double-click the **Job Name Value** cell and type the required name.

  Job name overrides must adhere to the following conventions:

  – Can only use alphanumeric characters and ~ # % ^ , & * ' ( ) - _ + = / < >

  – Cannot exceed 15 characters

  – The first character must be one of the alphanumeric characters (a-z, A-Z, 0-9)

- **Download Progress Information**: Specifies that the solver monitor should periodically query RSM for output files in order to display progress (where applicable). Queries begin when you initiate an action to display the solution progress (such as choosing the **Display Monitors** option in CFX). This option, available for Fluent and CFX systems, is enabled by default.

  For CFX systems, select from the following options:

  – **Always Download Progress Information**: Causes queries to RSM to begin immediately. This option is enabled by default.

  – **Download Progress Information on Demand**: Causes queries to RSM to begin when you initiate an action to display the solution progress (such as choosing the **Display Monitors** option in CFX).

  For all other systems, select or clear the **Download Progress Information** check box.

- **Progress Download Interval**: Specifies the periodic time interval, in seconds, that the solver queries RSM for output files to display progress. The default is **30**. Setting this value to **0** results in continuous queries; that is, as soon as files are downloaded from the execution node, Workbench immediately queries again. This option is available for Fluent and CFX systems.

- **Execution Mode**: Specifies the serial or parallel solver execution mode. The parallel option is available only if the selected solver supports parallel execution mode. This option may not be available with all systems.

  When performing a design point update in RSM with component update in the foreground, the **Parameter Set** properties override the parallel-process settings in individual components. For details on updating design points, see Updating Design Points in Remote Solve Manager (p. 352).

- **Number of Processes**: Specifies the number of processes to use for the solution if parallel execution mode is selected. The default is **2**. Must be set to a number greater than or equal to 2.

4. To save the project, from the menu bar, select **File** → **Save**.

   If you choose not to save the project and any cells are configured to use RSM or run in the background, you must change the solution process settings of those cells to run in the foreground before you can update.

5. To start the solution, select the appropriate option from the application, or right-click the **Solution** cell and select **Update** from the context menu.

   The **Solution** cell enters the .

Submitting Workbench Project Updates to Remote Solve Manager

You can submit a project to Remote Solve Manager (RSM) for remote update of the project. Depending on how RSM is configured, the update can be submitted to an HPC cluster or Cloud portal for execution (see).

Note:

If you are sending solve jobs to a remote cluster, the license preferences set for your local machine (the RSM client) may not be the same as the license preferences set for the remote cluster. In this case, the cluster license preferences are used for all jobs.

To submit a project update to RSM:

1. To open the project properties, from the menu bar select **View** → **Properties** or right-click the white space of the **Project Schematic** and select **Properties**.

2. Under **Solution Process**, set the **Update Option** property to **Submit to Remote Solve Manager**.

	A	B
	Property	Value
1	Property	Value
2	⊟ Notes	
3	Notes	
4	⊟ Solution Process	
5	Update Option	Remote Solve Manager
6	RSM Queue	Local
7	⊟ RSM Queue Details	
8	HPC Configuration	localhost
9	HPC Queue	local
10	HPC Type	ARC
11	Job Name	Workbench
12	⊟ Project Update	
13	Pre-RSM Foreground Update	Geometry
14	Component Execution Mode	Serial
15	Remote Job Status Check Interval in Seconds	20

3.   Additional properties are displayed. Use these properties to specify your remote submission settings.

   - **RSM Queue**: Select the queue to be used for the remote solution. Any queues currently defined for RSM appear in the drop-down list. If you have added a queue and it does not appear in the list, click **Refresh Queues**.

     Every RSM queue is associated with a particular configuration that is defined in RSM. In the RSM configuration, the RSM queue is mapped to a specific HPC queue that has been defined on the cluster/portal side.

   - **Job Name**: Displays the name reported in the RSM job log. The default name for a Workbench job is `Workbench`, or whatever is set for the **Default Job Name** in your Workbench options. The default name for a Mechanical job submitted from the Mechanical application is `Mechanical`. The job name provides a traceable piece of data that administrators can use to track job submissions to a cluster (for billing purposes, for example).

     To override the default job name, double-click the **Job Name Value** cell and type the required name.

     Job name overrides must adhere to the following conventions:

     – Can only use alphanumeric characters and ~ # % ^ , & * ' ( ) - _ + = / < >

     – Cannot exceed 15 characters

     – The first character must be one of the alphanumeric characters (a-z, A-Z, 0-9)

   - **Pre-RSM Foreground Update**:

     – To update your geometry locally before submitting project updates to RSM, select **Geometry**.

       Select this option if the execution node does not have a license to perform geometry updates.

     – To submit project updates to RSM without first updating the geometry locally, .select **None**.

   - **Component Execution Mode**: Specifies the serial or parallel solver execution mode. The parallel option is available only if the selected solver supports parallel execution mode. This option may not be available with all systems.

     When performing a design point update in RSM with component update in the foreground, the **Parameter Set** properties override the parallel-process settings in individual components. For details on updating design points, see Updating Design Points in Remote Solve Manager (p. 352).

   - **Remote Job Status Check Interval in Seconds**

     Sets the number of seconds between each query to Remote Solve Manager for the Mechanical solution update job status. The default is 20. This setting only takes effect when the Mechanical solution is submitted to RSM.

The **Project Update** properties specified here are shared with the **Parameter Set** as **Design Point Update Process** properties. Changes to the values of these properties here are reflected in the **Parameter Set** properties, and vice versa.

> **Note:**
>
> In most cases, the **Update Project** button or menu item updates all systems and cells in a project. When a project update is submitted to RSM as described, however, only those systems above the **Parameter Set** bar are sent to RSM for remote update. If needed, DesignXplorer systems can be further updated once the remote project update is completed.

Submitting System Coupling Jobs to Remote Solve Manager

> **Tip:**
>
> For remote execution of a co-simulation created in Workbench, the recommended workflow is to export the coupling setup from Workbench and then run the co-simulation in System Coupling's GUI or CLI, using System Coupling's parallel processing capabilities to submit jobs for remote execution.

A System Coupling co-simulation may be sent to Remote Solve Manager (RSM) for remote execution either as a full **project update** or a **design point update**.

> **Note:**
>
> - Solution cell updates via RSM are not supported for System Coupling; for a **System Coupling** system's **Solution** cell, **Run in Foreground** is the only option available for the **Solution Process | Update Option** property. As such:
>
>   – If you attempt to solve the coupled analysis by updating System Coupling's **Solution** cell, then the update will not be sent to RSM, regardless of the **Update Option** setting for the project or the **Parameter Set**.
>
>   – Because coupling participant solutions are tied to the update of System Coupling's **Solution** cell, participant setups should be consistent with System Coupling's setup. Ensure that all participant **Solution** cells are also set to solve in the foreground.
>
> - When submitting a job on a machine that uses Microsoft HPC Pack 2019, you may receive an error message indicating that the coupling process failed to start. To address this issue, submit the job on another resource, either by using an earlier version of HPC Pack or by submitting the job on a Linux system.

For more information on working with RSM, see:

Submitting a Coupled Analysis Project Update to RSM

Submitting a Coupled Analysis Design Point Update to RSM

Distribution of Computing Resources Across System Coupling Participants

Restricting Solve Processes for System Coupling Updates

Submitting a Coupled Analysis Project Update to RSM

To send a coupled analysis project update to RSM, perform the following steps:

1. Set the project's **Solution Process** and **Project Update** properties as described in Submitting Workbench Project Updates to Remote Solve Manager (p. 325).

2. Click the **Update Project** button in the Workbench toolbar.

All the systems on the **Project Schematic**, including the **System Coupling** system, are updated through RSM.

Submitting a Coupled Analysis Design Point Update to RSM

To send a coupled analysis design point update to RSM, perform the following steps:

1. On the **Project Schematic**, right-click the **Parameter Set** bar and select **Properties**.

   Corresponding settings are shown below in the **Properties** pane.

2. Use the steps outlined in Updating Design Points in Remote Solve Manager (p. 352) to set the following properties:

   a. **Solution Process**

   b. **Design Point Update Process**

   > **Note:**
   >
   > Because design point updates are performed only on output parameters, a **System Coupling** system that does not affect outputs may not be updated during a design point update. However, when updating retained design points (including Current), you can ensure that all project systems are updated. To do so, set the **Retained Design Point** property to **Update full project**, then right-click the Current design point and select **Update selected design points**.

3. Save the project.

   If you are working in an archived project, save the project to a permanent location.

4. Update the design point(s), using any of the methods documented in Updating Design Points (p. 306) (that is, updating the Current design point, selected design points, or all design points).

The project is archived and submitted to a cluster using RSM. The remote data is retrieved periodically as the design point updates complete. Should a design point update fail, an error is reported to the **Messages** pane.

Distribution of Computing Resources Across System Coupling Participants

In a project with System Coupling, the individual components (such as Fluent, Mechanical, or CFX) share resources. By default, when design points from a project containing a **System Coupling** system are submitted to RSM, available computing resources are distributed equally across all coupling participants according to the number of compute nodes available and the number of processes requested. Component-level parallel settings (such as the **Execution Mode** and **Number of Processes** properties) are disregarded during the design point update.

Example 1: Six Processes

If a coupled simulation requests six processes to be executed on a cluster that has six nodes with one processor per node, then participant solvers will start six processes, with one on each compute node. The participant solvers alternate their processing, so that only one participant is actively using computing resources at a time.

Node	Solver Processes for Participant 1	Solver Processes for Participant 2
1	1	1
2	1	1
3	1	1
4	1	1
5	1	1
6	1	1

Example 2: Ten Processes

If a coupled simulation requests 10 processes to be executed on a cluster that has 4 nodes with 8 processors per node, then each participant solver will start 10 processes.

Node	Solver Processes for Participant 1	Solver Processes for Participant 2
1	8	8
2	2	2
3	0	0
4	0	0

Restricting Solve Processes for System Coupling Updates

When submitting an update for parallel processing, you can limit the number of solve processes that a coupling participant can use when updates are submitted to Remote Solve Manager (RSM). To do so, set the **Restriction for Project / Remote Design Point Update** properties for the participant's **Solution** cell, as described in Updating Design Points in Remote Solve Manager (p. 352).

The solve process limitations are applied only to that component, and resources are otherwise allocated equally across participants. Resource allocation for the restricted participant is as follows:

- If the component's **Specify Number of Processes Restriction** is enabled and the **Number of Processes Used Not to Exceed** (or **Max Number of Utilized Cores**) value is less than or equal to the number of processes requested by the project, then the component is limited to the component-specified number of processes.

- If the component's **Specify Number of Processes Restriction** is enabled and the **Number of Processes Used Not to Exceed** (or **Max Number of Utilized Cores**) value is set to a value greater than the number of processes requested by the project, then the component is limited to the project-specified number of processes.

- If the component's **Serial Execution Only** is enabled, then the component is limited to a single process.

Monitoring System Coupling Solution Information

As the remote design point update progresses, a single System Coupling Transcript/Log file (`scLog.scl`) is generated for the coupling project and displayed in the **Solution Information** pane of the System Coupling workspace. It is updated intermittently so you can use it to monitor the design point update.

> **Note:**
>
> When multiple design points are being updated simultaneously, you cannot switch between the log files for the different design points. During and after completion of the updates, System Coupling displays the log file for the design point with the highest ID. To display the log file for the Current design point, you must save and reopen the project.

For more information, see the following sections in the System Coupling User's Guide:

- Monitoring Solution Progress

- Transcript and Log File

Restarting an Interrupted Remote Update

When a remote **System Coupling** update has been interrupted:

- For a project update, the **Solution** cells in your project are given a state of *Interrupted, Up to Date* ():

- For a design point update, the output parameters for the interrupted component are given a state of *Partially Updated* ().

To restart the update:

1. In the Workbench toolbar, click the **Update Project** or **Update All Design Points** button.

2. In the dialog box that opens, click one of the following buttons:

- **Use Partially Updated**: Accepts the partially updated solution as *Up-to-Date* (✔) and uses it to restart the update.

- **Update All**: Restarts the interrupted run, updating all solutions to completion before restarting the update.

Submitting Mechanical Application Jobs to Remote Solve Manager

Use the following steps to submit Mechanical jobs to a cluster or Cloud portal using Remote Solve Manager (RSM).

1. Configure your remote solution settings in Remote Solve Manager.

   See RSM Configuration in the *Remote Solve Manager User's Guide*.

2. To launch the Mechanical application, on the **Project Schematic**, double-click the **Solution** cell in your Mechanical system.

3. In the Mechanical application, on the **Home** tab, select **Solve Process Settings**.
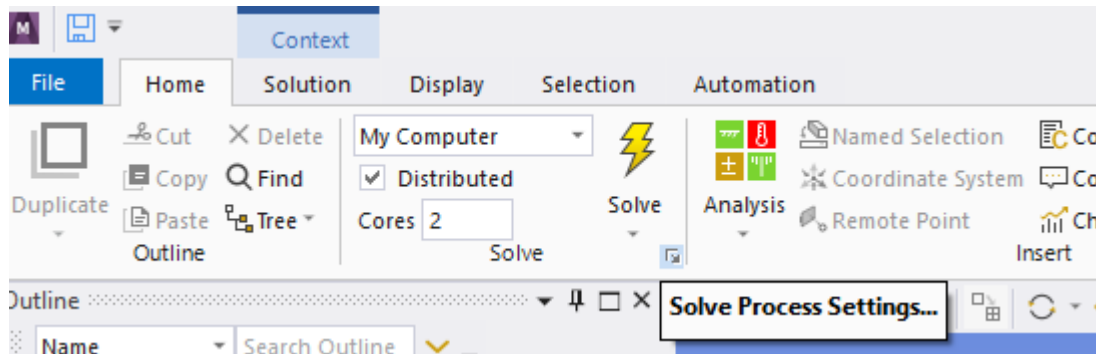
   Alternatively, switch to the **File** tab and select **Solve Process Settings**.

4. Follow the instructions in Using Solve Process Settings to configure your solution settings.

   The settings established here are visible in the **Solution Process** properties in Workbench.
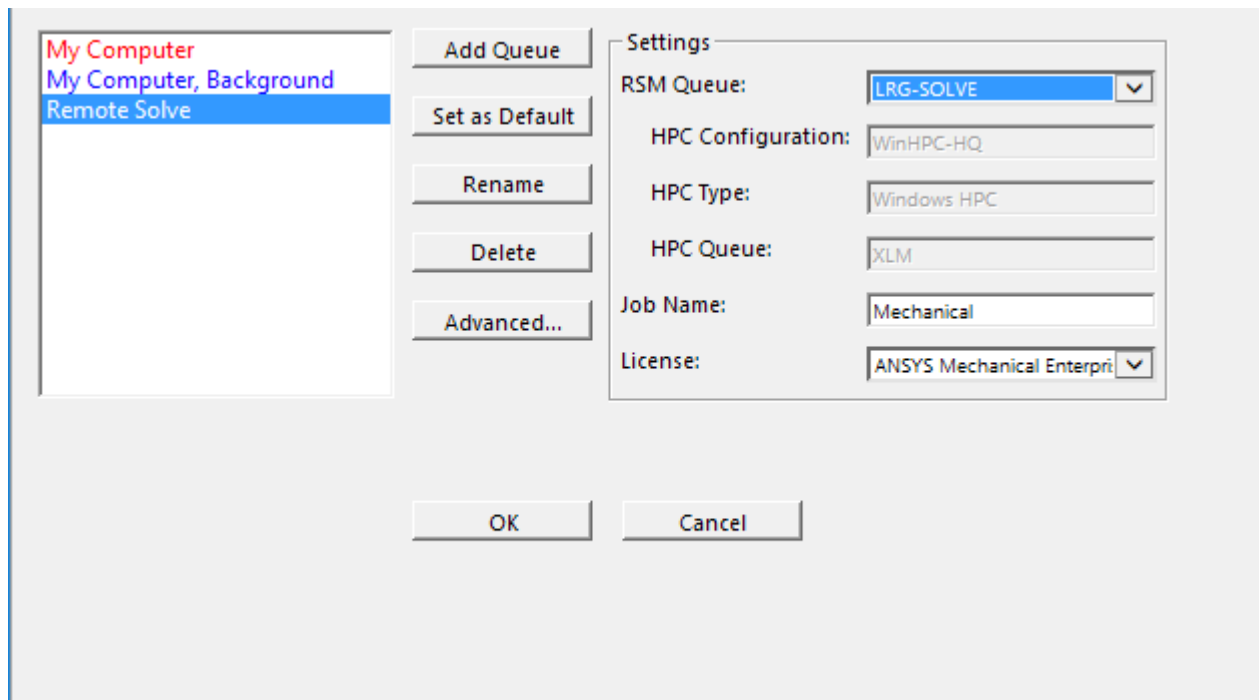
5. In Workbench, select the **Solution Process** settings that you want to use for this solution and update the **Solution** cell (p. 244).

Alternatively, in the Mechanical application, on the **Home** tab select **Solve** then select a configuration from the drop-down list.
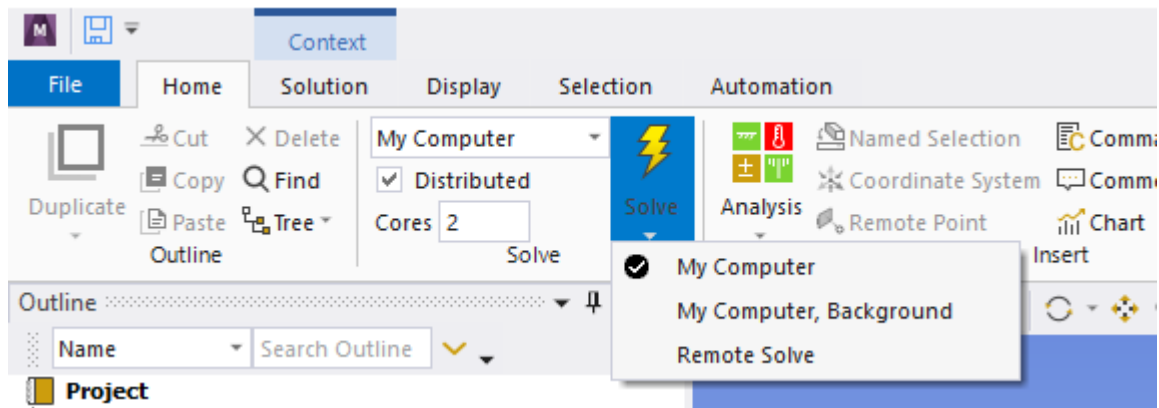
Note:

- You can interrupt or abort an update by right-clicking the **Solution** cell. These options are available during foreground, background, Portal and RSM updates.

- The Workbench **Options** allow you to control the Mechanical application's license handling. As described in Mechanical (p. 430), you can configure the license to always be released during batch run or to be released when you perform an **Update All Design Points** run. A third option is **On Demand**, which makes a **Release License**

option available on the **Solution** cell's context menu when the cell is in the pending
state during a batch run.

Workbench/RSM Integration with the Mechanical Application

The properties associated with a Mechanical system's **Solution** cell include a **Solution Process** section.
This section allows you to specify the **Solve Process Setting** for the Mechanical application to use
when an update operation is initiated from Workbench. You can select only an existing solve process
setting, but new solve process settings can be added (and existing solve process settings can be
modified) by selecting **Tools** → **Solve Process Settings** in the Mechanical application.

> **Note:**
>
> Any solve initiated in the Mechanical application ignores the solution component properties
> selected in Workbench and continues to work as it has in previous versions.

Example 3: Default Update with *My Computer* Set as the Default Solve Process Setting

In this case, the default Solve Process Setting in the Mechanical application is *My Computer*.

In Workbench, the **Solution Process** property **Update Option** is set to **Use application default**,
which causes Mechanical to use its default solve process setting.

	A	B
	Property	Value
2	▣ General	
3	Component ID	Solution
4	Directory Name	SYS
5	▣ Notes	
6	Notes	
7	▣ Used Licenses	
8	Last Update Used Licenses	
9	▣ System Information	
10	Physics	Structural
11	Analysis	Static Structural
12	Solver	Mechanical APDL
13	▣ Solution Process	
14	Update Option	Use application default
15	Solve Process Setting	My Computer
16	Queue	

Properties of Schematic A6: Solution

Job Monitor... Show Progress Show 0 Messages

Results:

Solve Initiated From	Solve Process Setting Used
Workbench	My Computer
Mechanical	My Computer

Example 4: Default Update with *My Computer, Background* Set as the Default Solve Process Setting

In this case, the default solve process setting in the Mechanical application is *My Computer, Background*.
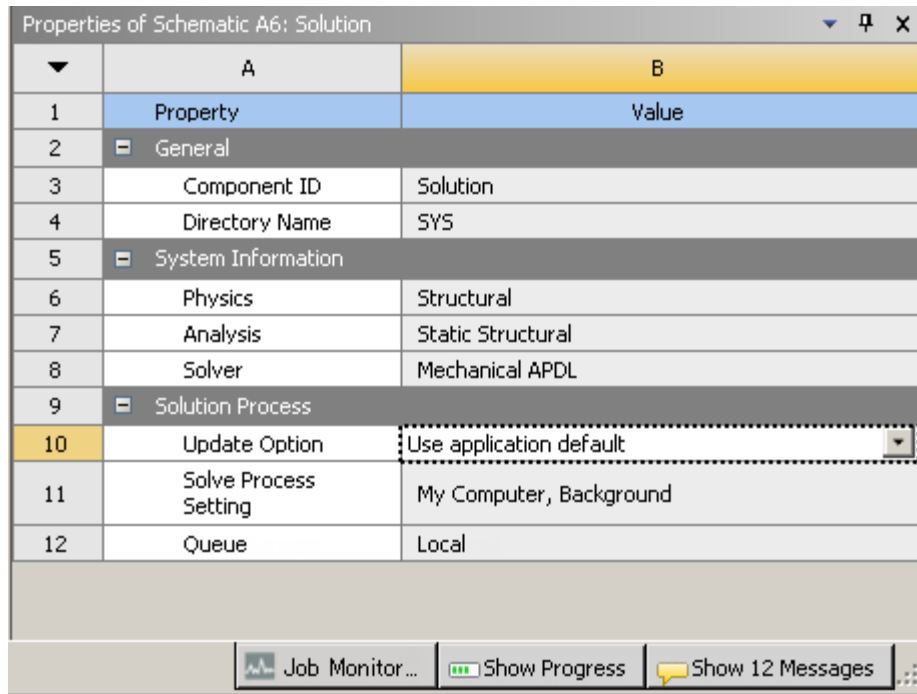
In Workbench, the **Solution Process** property **Update Option** is set to **Use application default**, which causes Mechanical to use its default solve process setting.

Results:

Solve Initiated From	Solve Process Setting Used
Workbench	My Computer, Background
Mechanical	My Computer, Background

Example 5: RSM Update Using *My Computer, Background*, with *My Computer* Set as the Default Solve Process Setting

In this case, the default solve process setting in the Mechanical application is *My Computer*.
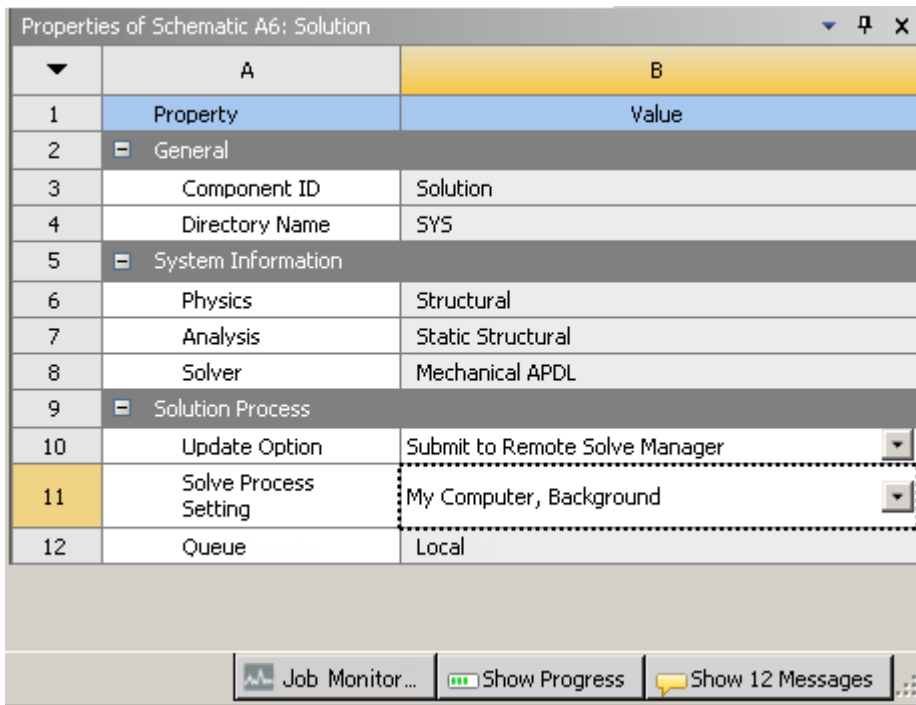
In Workbench, the **Solution Process** property **Update Option** is set to **Submit to Remote Solve Manager**. The solve process setting selected in Workbench is **My Computer, Background**.

Results:

Solve Initiated From	Solve Process Setting Used
Workbench	My Computer, Background
Mechanical	My Computer

Submitting Mechanical APDL Jobs to Remote Solve Manager

You can use Remote Solve Manager (RSM) as a serial or parallel solution for Mechanical APDL jobs. If your RSM configuration directory contains configurations for a cluster or a Cloud portal, you can access cluster/portal queues that act as a parallel solution option.

> **Note:**
>
> Mechanical APDL has the following limitations when used with Remote Solve Manager:
>
> - Reconnect does not work after moving a project to another machine.
>
> - Only one copy of a saved project that is in the pending state can reconnect successfully.
>
> - You must manually save a project after a reconnect.
>
> - Reference files must be in the same directory as the Ansys input file.
>
> - Restarts of solves to a cluster are not supported from Workbench.

In the **Solution Process** pane:

1. Configure your remote solution settings in Remote Solve Manager. For more information see RSM Configuration in the *Remote Solve Manager User's Guide*.

2. In Workbench, right-click the **Solution** cell and select **Properties**.

3. In the **Solution Process** settings, set the **Update Option** to **Submit to Remote Solve Manager**.

4. Select a **RSM Queue**.

5. Set the **Execution Mode** to either **Serial** or **Parallel**.

6. Set the number of **Utilized cores**. For a serial solution, select **1**; for a parallel solution, specify the number of partitions.

   You should have at least 100,000 elements or nodes per partition to compensate for the overhead associated with the partition. The cluster job scheduler automatically controls where the partitions are solved.

7. Update the project (p. 243).

   When the Mechanical APDL job is submitted to RSM, some options in the **Properties** pane (**Processors**, **Distributed**, **MPI Type**, **Machine List**, and **Custom Executable Path**) become read-only and their values are ignored.

   > **Note:**
   >
   > If **Download Distributed Files** is checked, all files involved in a distributed solve are downloaded.

Submitting Fluids Jobs to Remote Solve Manager

Use the following instructions to submit various fluid jobs to Remote Solve Manager (RSM).

Submitting CFX Jobs to Remote Solve Manager

Submitting Polyflow Jobs to Remote Solve Manager

Submitting Fluent Jobs to Remote Solve Manager

Submitting Icepak Jobs to Remote Solve Manager

Submitting Forte Jobs to Remote Solve Manager

Exiting a Project During a Remote Solve Manager Solution Cell Update

Submitting CFX Jobs to Remote Solve Manager

You can use Remote Solve Manager (RSM) as a serial or parallel solution for CFX jobs. If your RSM configuration directory contains configurations for a cluster or a Cloud portal, you can access cluster/portal queues that act as a parallel solution option.

Note:

CFX has the following limitations when used with Remote Solve Manager:

- RIF (Flamelet) cases are not supported.

- Reconnect does not work after moving a project to another machine.

- Only one copy of a saved project that is in the pending state can reconnect successfully.

- You must manually save a project after a reconnect.

- You cannot edit a run that is in progress.

- You cannot perform a remote backup.

- For runs that are submitted to a remote machine:

  - Serial and local parallel runs are always supported.

  - Other local parallel modes must be supported on the job host.

  - Distributed parallel is supported only for multi-node clusters.

- The default update interval for display monitors is 120 seconds. Use the **Download Progress Information** and **Progress Download Interval** options under **Tools** → **Options** → **Solution Process** to enable/disable polling or to change the polling interval. The settings established here are also visible in the **Solution Process** properties pane in Workbench.

In the **Solution Process** pane:

1. Configure your remote solution settings in Remote Solve Manager. For more information see RSM Configuration in the *Remote Solve Manager User's Guide*.

2. In Workbench, right-click the **Solution** cell and select **Properties**.

3. In the **Solution Process** settings, set the **Update Option** to **Submit to Remote Solve Manager**.

4. Select a **RSM Queue**.

5. Set the **Execution Mode** to either **Serial** or **Parallel**.

   For a parallel solution:

- You can specify the **Number of Processes** that you want to use to create the results file. You should specify a number that is less than or equal to the number of cores available in the cluster, and you should ensure that each process contains at least 100,000 nodes or elements.

- UGE and PBS clusters must be configured to use the SSH communication protocol; RSH is not supported. If you are using the **Submit to Remote Solve Manager** option, see Configuring RSM to Use SSH for Job Submission and/or File Transfers to a Remote Linux Cluster in the *Remote Solve Manager User's Guide*.

6. Update the project (p. 243).

    You can interrupt or abort an update by right-clicking the **Solution** cell. These options are available during foreground, background and RSM updates.

    Note:

    The monitoring of solution updates submitted to RSM has the following limitations:

    - When you use the **Solution** cell's properties field to set the frequency at which you poll the data, this represents a maximum frequency. In cases of network congestion or where large files are involved, the observed monitor update frequency is less.

    - Solution monitor data is transferred using the CFX Solver-Manager, which has a limited capacity to handle monitor data transfer requests. In situations where multiple users are using the same CFX Solver-Manager to monitor runs, or where multiple runs are being monitored by a single user, the update frequency of monitor data may decrease or become sporadic.

    - When monitoring RSM updates, CFX Solver-Manager does not report run completion. The **Out File** window displays the completed run information; however, CFX Solver-Manager's workspace still displays *Running*. Note that in these cases, Workbench accurately reports update completion.

Submitting CFX Jobs with Design Points to Remote Solve Manager

When performing a Design Point update with a CFX system using Remote Solve Manager (RSM), the **Component Execution Mode** and **Max. Number of Processes Per Job** settings on the **Parameter Set Properties** pane control the parallel-processing settings that the CFX Solver uses. For details, see Updating Design Points in Remote Solve Manager (p. 352).

Submitting Polyflow Jobs to Remote Solve Manager

Use the following steps to submit Polyflow jobs to Remote Solve Manager (RSM).

Note:

Polyflow has the following limitations when used with Remote Solve Manager:

- Reconnect does not work after moving a project to another machine.

- Only one copy of a saved project that is in the pending state can reconnect successfully.

- You must manually save a project after a reconnect.

- The RSM interrupt option performs an abort operation, not an interrupt.

- Parallel design point submissions to batch queue clusters only run on the head node and with the number of cores allocated by the batch queue scheduler.

In the **Solution Process** pane:

1. Configure your remote solution settings in Remote Solve Manager. For more information see RSM Configuration in the *Remote Solve Manager User's Guide*.

2. In Workbench, right-click the **Solution** cell and select **Properties**.

3. In the **Solution Process** settings, set the **Update Option** to **Submit to Remote Solve Manager**.

4. Select a **RSM Queue**.

5. Update the project (p. 243).

   You can interrupt or abort an update by right-clicking the **Solution** cell. These options are available during foreground, background and RSM updates.

Submitting Fluent Jobs to Remote Solve Manager

You can use this feature to queue multiple jobs to run on the local machine (overnight, for example, or during other low-usage times), or to submit a job to a remote cluster or Cloud portal.

> **Note:**
>
> Fluent has the following limitations when used with Remote Solve Manager:
>
> - Only one copy of a saved project that is in the pending state can reconnect successfully.
>
> - System Coupling is not supported.
>
> - UDFs are supported but you must have a supported compiler on Windows 64-bit machines. Supported compilers for Windows are Microsoft Visual Studio 2008 Standard and Microsoft Visual Studio 2010 Professional.
>
> - On Linux, UDFs are supported. You can always send UDFs between Linux machines and make use of the auto-compile feature; if the machines have compatible compilers, you can send precompiled UDFs.
>
> For additional information about compiling Fluent UDFs, see Compiling UDFs in the *Fluent Customization Manual*.

In the **Solution Process** pane:

1. Configure your remote solution settings in Remote Solve Manager. For more information see RSM Configuration in the *Remote Solve Manager User's Guide*.

2. In Workbench, right-click the **Solution** cell and select **Properties**.

3. In the **Solution Process** settings, set the **Update Option** to **Submit to Remote Solve Manager**.

4. Select a **RSM Queue**.

5. Set the **Execution Mode** to either **Serial** or **Parallel**.

   For a parallel solution, specify the **Number of Processes** that you want to use to create the results file. You should specify a number that is less than or equal to the number of cores available in the cluster, and you should ensure that each process contains at least 100,000 nodes or elements.

6. Update the project (p. 243).

   An update of the **Solution** cell submits the job to RSM, moves into Pending mode for the duration of the solution, and then automatically reconnects at the end of the run.

   You can interrupt or abort an update by right-clicking the **Solution** cell. These options are available during foreground, background and RSM updates.

Solution Properties: Use Setup Launcher Settings

The **Properties** pane of the Fluent **Solution** cell has a toggle that controls whether the solution uses settings from the setup launcher. When **Use Setup Launcher Settings** is checked, the launcher settings are copied from **Setup** component to **Solution** component. This copying also happens any time you change the launcher settings in setup. When **Use Setup Launcher Settings** is cleared, the **Solution** cell's launcher settings are available:

Setting Name	Default	Applicable to Remote Solve Manager (RSM)
General		
Precision	**Single Precision**	Yes
Show Launcher at Startup	Enabled	No
Display Mesh After Reading	Enabled	Yes
Embed Graphics Window	Enabled	Yes
Use Workbench Color Scheme	Enabled	Yes
Set up Compilation Environment for UDF	Enabled	No
Use Job Scheduler	Disabled	No
Run Parallel Version	The initial default execution mode (Serial / Parallel) is based on the	Yes

Submitting Fluids Jobs to Remote Solve Manager

Setting Name	Default	Applicable to Remote Solve Manager (RSM)
	run settings. The initial number of processes in RSM is based on the **Number of Processors** option in the run settings: if it is > 1, that value is used, otherwise 2 is used. If **Run Parallel Version** is not enabled, then serial mode is forced.	
UDF Compilation Script Path	**$(FLUENT_ROOT)\$(ARCH)\udf.bat**	No
Use Remote Linux Nodes	Enabled	No
Remote		
Remote Polyflow Root Path	None	No
Use Specified Remote Working Directory	Enabled	No
Remote Working Directory	None	No
Remote Spawn Command	**RSH**	No
Use Remote Cluster Head Node	Enabled	No
Remote Host Name	None	No
Parallel Run Settings		
Number of Processors	1	No
Interconnect	None	No **Note:** You can enable **Interconnect** to be available for Polyflow RSM runs by enabling **Tools → Options → Solution Process → Show Advanced Solver Options**. This setting requires you to ensure that the remote Compute servers can accept the interconnect that you

Setting Name	Default	Applicable to Remote Solve Manager (RSM)
		specify—there is no automatic checking for such compatibility. To learn what values are available, see Starting Parallel Ansys Fluent on a Windows System Using Command Line Options in the *Fluent User's Guide*.
MPI Type	None	No **Note:** You can enable **MPI Type** to be available for Polyflow RSM runs by enabling **Tools** → **Options** → **Solution Process** → **Show Advanced Solver Options**. This setting requires you to ensure that the remote Compute servers can accept the MPI Type that you specify—there is no automatic checking for such compatibility. To learn what values are available, see Starting Parallel Ansys Fluent on a Windows System Using Command Line Options in the *Fluent User's Guide*.
Use Shared Memory	Disabled	No
Machine Specification	File containing the machine list	No

Setting Name	Default	Applicable to Remote Solve Manager (RSM)
Machine Filename	None	No
Solution Process		
The **Solution** cell's **Solution Process** setting is always displayed; when **Use Setup Launcher Settings** is checked, the option is read-only:		
Update Option	You can choose to **Run in Foreground**, **Run in Background**, or **Submit to Remote Solve Manager**. When **Submit to Remote Solve Manager** is chosen, options that are not applicable to RSM are hidden. Also, `fluent-launcher.txt` does not contain options that conflict with RSM.	

Submitting Icepak Jobs to Remote Solve Manager

You can use this feature to queue multiple jobs to run on the local machine (overnight, for example, or during other low-usage times), or to submit a job to a remote cluster or Cloud portal.

> **Note:**
>
> Icepak has the following limitations when used with Remote Solve Manager:
>
> - You cannot submit jobs containing parametric trials to RSM.
>
> - Convergence data is not plotted in Icepak on the local computer. The `.res` and `.uns_out` files are written at the frequency specified in RSM settings. You must open them in Icepak to view them.

In the **Solution Process** pane:

1. Configure your remote solution settings in Remote Solve Manager. For more information see RSM Configuration in the *Remote Solve Manager User's Guide*.

2. In Workbench, right-click the **Solution** cell and select **Properties**.
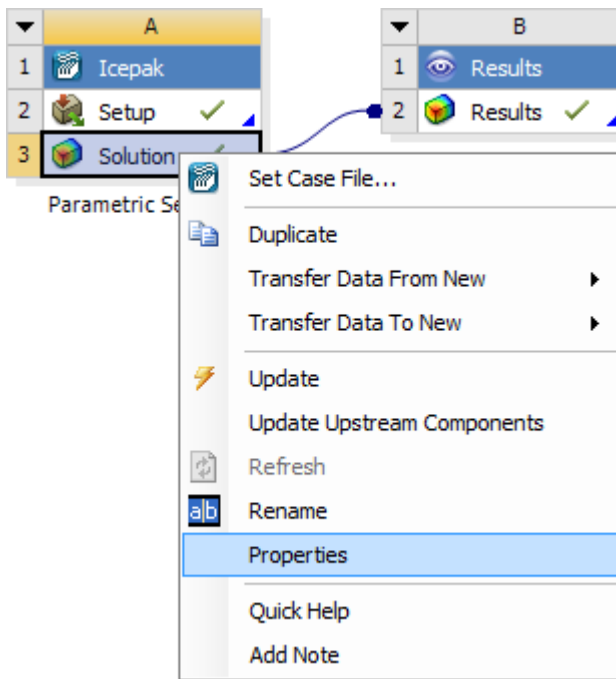
An update of the **Solution** cell submits the job to RSM, moves into Pending mode for the duration of the solution, and then automatically reconnects at the end of the run.

3. In the **Solution Process** settings, set the **Update Option** to **Submit to Remote Solve Manager**.

4. Select a **RSM Queue**.

5. Set the **Execution Mode** to either **Serial** or **Parallel**.

   For a parallel solution, specify the **Number of Processes** that you want to use to create the results file. You should specify a number that is less than or equal to the number of cores available in the cluster, and you should ensure that each process contains at least 100,000 nodes or elements.

	A	B	C
Properties of Schematic A3: Solution			
1	Property	Value	Unit
2	⊟ General		
3	Component ID	Solution	
4	Directory Name	IPK	
5	⊟ Notes		
6	Notes		
7	⊟ Used Licenses		
8	Last Update Used Licenses		
9	⊟ Solution Process		
10	Update Option	Submit to Remote Solve Manager ▼	
11	RSM Queue	High Mem ▼	
12	⊞ RSM Queue Details		
16	Job Name	Workbench	
17	Download Progress Information	✓	
18	Progress Download Interval	120	s ▼
19	Execution Mode	Parallel ▼	
20	Number of Processes	2	

Note:

RSM updates Icepak with convergence plot data at the **Progress Download Interval** specified in Workbench. Set the **Progress Download Interval** as needed prior to starting the solution.

6. Update the project (p. 243).

An update of the **Solution** cell submits the job to RSM, moves into Pending mode for the duration of the solution, and then automatically reconnects at the end of the run. To re-run a solution with the same model, run another solution in the same project, or run a restart solution, right-click the **Setup** cell in Workbench and select **Enable Update**.
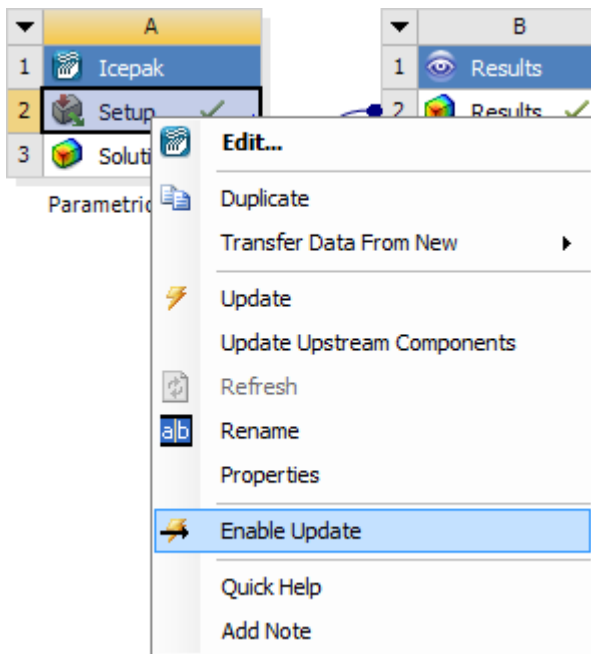
You can interrupt or abort an update by right-clicking the solution in the Workbench Job Monitor. These options are available during background and RSM updates. To display the **Job Monitor**, select **Jobs → Open Job Monitor**.

Submitting Forte Jobs to Remote Solve Manager

Use the following steps to submit Forte jobs to Remote Solve Manager (RSM).

> **Note:**
>
> Forte has the following limitations when used with Remote Solve Manager:
>
> - If solving on a Linux RSM queue, upstream geometry changes must be configured to occur before the RSM job. Select Pre-RSM Foreground Update to Geometry in the Project Update settings of the **Project Schematic**.
>
> - Forte's Monitor component does not provide live updates.
>
> - You must specify at least two parallel processes.

1. Configure your remote solution settings in Remote Solve Manager. For more information see RSM Configuration in the *Remote Solve Manager User's Guide*.

2. In Workbench, right-click the white space of the **Project Schematic** and select **Properties**.

3. In the **Solution Process** settings, set the **Update Option** to **Submit to Remote Solve Manager**.

4. Select a **RSM Queue**.

5. Set the **Execution Mode** to **Parallel**.

6. Specify the **Number of Processes** that you want to use to create the results file.

 The number of process must be equal to or greater than two.

7.

Exiting a Project During a Remote Solve Manager Solution Cell Update

You can exit a project while a **Solution** cell update using Remote Solve Manager (RSM) is in progress (one or more RSM jobs are running in the background and the **Progress** pane displays a project status of **Waiting for background task**).

Note:

You cannot exit Workbench while job files are being uploaded. You can abort the job during the upload by using the **Stop** button in the **Progress** bar.

The project must be saved at least once after the **Solution** cell update job was initiated for RSM jobs to continue to run after you exit the project.

If you attempt to exit a project while a **Solution** cell update job is still running, the following scenarios cause a dialog box to open, allowing you to specify whether you want to save the project before exiting:

• You have never saved the project at any time after the same **Solution** cell update job was initiated.

• You have saved the project at least once after the same **Solution** cell update was initiated, but results have been retrieved since your last save.

In either of these cases, if you do not save the project before exiting:

• All results retrieved since the last save operation are lost. You must save the project before exiting if you want to keep any of results retrieved since the last save of the background **Solution** cell update.

• If the project has never been saved after the **Solution** cell update job was initiated, all RSM jobs are aborted and display a status of **Cancelled** and the cancelled icon (🔴) in the RSM **List** pane; the asterisk on the icon indicates that the job has also been released. When you reopen the project, it is in the state of your last manual save.

• If the project has been saved at least once after the **Solution** cell update job was initiated, RSM jobs that are queued and running continue to run after you exit. When you reopen the project, you can resume the update of the pending jobs to reconnect and download the results.

If you do save the project before exiting:

• Retrieved results are saved to the project.

• RSM jobs that are queued and running will continue to run after you exit.

• Jobs where the results have been saved display a **Status** of **Finished** and the finished icon (🎉) in the RSM **List** pane; the asterisk on the icon indicates that the job has also been released.

- Jobs where results have not yet been saved are not released upon exit. When you reopen the project, you can resume update of the pending jobs to reconnect and download the results.

Monitoring and Controlling Remote Solve Manager Jobs in Workbench

If you have chosen to submit project updates, design point updates, or component updates using the **Submit to Remote Solve Manager** option, you can monitor their progress by displaying the **Job Monitor** in Workbench.

To display the **Job Monitor**, select **Jobs** → **Open Job Monitor**, or click [Job Monitor...] on the status bar.

The **Job Monitor** provides a live view of what is happening in Remote Solve Manager (RSM) with the current Workbench project.
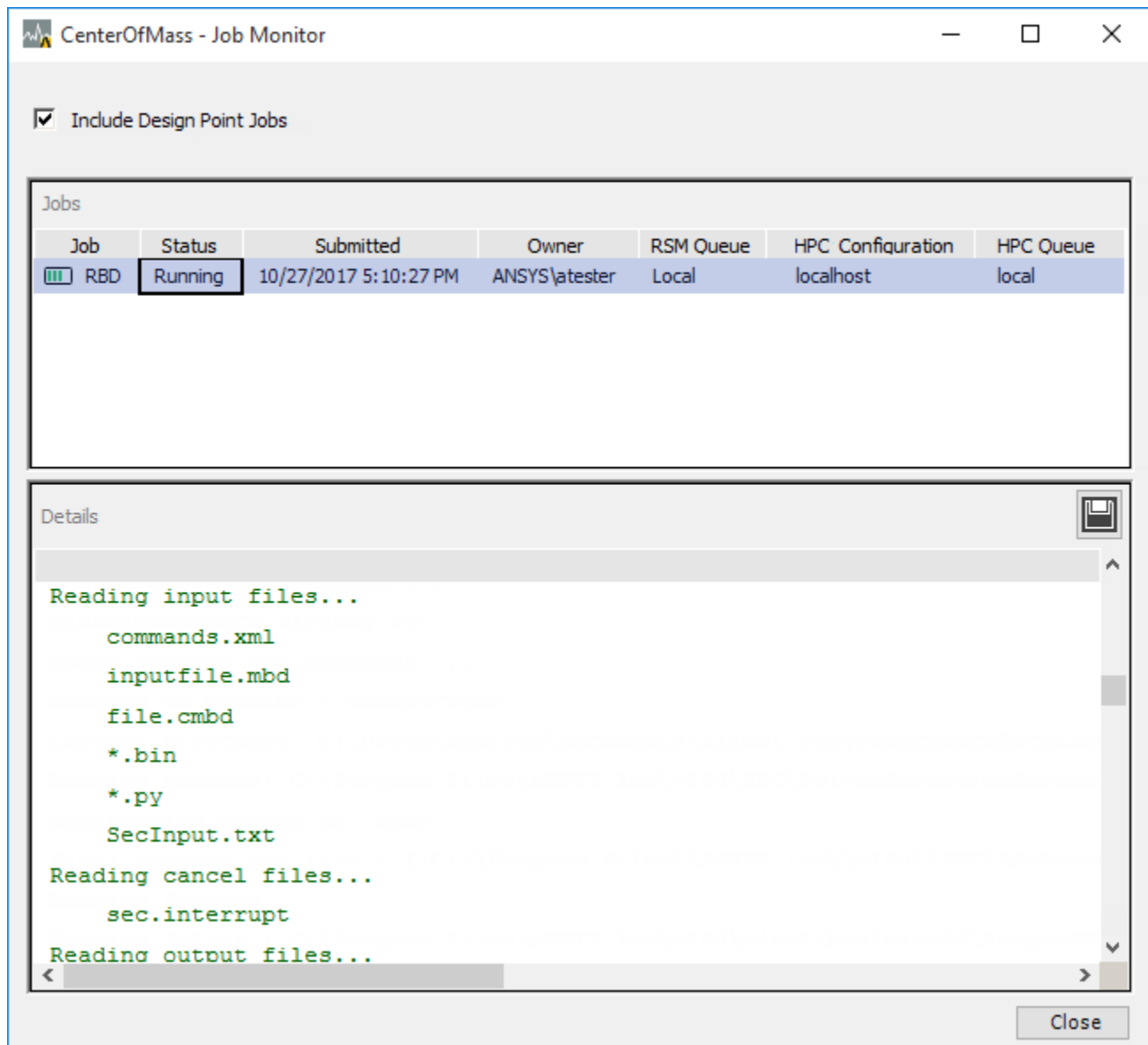
The following information for each job is displayed:

• Job name

• Status (Running, Finished, Failed, and so on)

• Submission date/time

• Owner

• RSM queue

• HPC configuration

• HPC queue

Clicking on a job displays a detailed job log in the lower pane.

You can perform the following actions:

• To hide design point updates in the job list, disable the **Include Design Point Jobs** check box.

• To view a job report for a specific job, select the job in the **Jobs** pane. The report is displayed in the **Details** pane.

• To save a job report to a file, click [icon] in the **Details** pane, then accept or specify the save location, filename, and content to include:
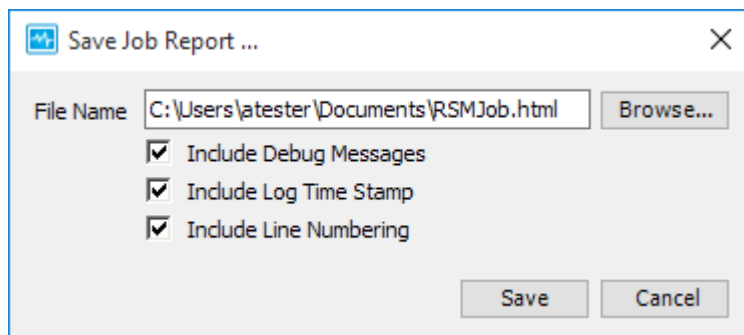
You can use the context-sensitive menus in the **Jobs** pane and **Details** pane to perform a variety of actions. For example, you can customize the job list display, and perform actions on a job such as **Abort** or **Interrupt**.

Related topics:

Using the Taskbar Icon

Viewing the Status of Jobs

Controlling the Display of the Job List

Performing Job Actions

Controlling the Display of Job Details

Using the Taskbar Icon

When you launch the **Job Monitor**, a job monitor icon is also displayed on your Windows taskbar. If you subsequently minimize the **Job Monitor** window, you can quickly restore it by clicking the taskbar icon.

Viewing the Status of Jobs

The status of each job is indicated in the **Status** column in the **Jobs** pane, and by a unique icon at the beginning of the job entry. For jobs that have completed, the **Status** column and icon indicate the final status of the job. The addition of an arrow symbol to the final status icon indicates that the job has been released.

Status	Description	Icon	Released Icon
Input Pending	Job is being uploaded to the cluster.		
Queued	The job has been placed in the cluster queue and is waiting to run.		
Running	Job is running.		
Cancelled	Job has been terminated using the **Abort** option. Also applies to jobs that have been aborted because you exited a project without first performing one of the following actions: • Saving the project since the update was initiated • Saving results retrieved since your last save		
Finished	Job has completed successfully. Also applies to jobs that have been terminated using the **Interrupt** option or for which you have saved results prior to exiting the project.		
Failed	Job has failed. May be applied to jobs that cannot be cancelled due to fatal errors.		

Controlling the Display of the Job List

You can sort the job list by job name, status, owner, and so on by clicking on the appropriate column header in the **Jobs** pane. Clicking on the same column header again reverses the sort order.

If you right-click any column header, a context menu is displayed which contains the following options:

Select All (Ctrl+A)	Selects all jobs in the list.
Scroll to Top	Returns you to the beginning of the job list.
Scroll to Bottom	Takes you to the end of the job list.
View Line Numbers	Toggles the display of numbers at the beginning of each entry in the job list.

Performing Job Actions

If you right-click a job in the **Jobs** pane, a context-sensitive menu is displayed that contains the following options:

Abort	Immediately terminates a running job. Enabled only if a running job is selected.
	Jobs terminated via this option will have a **Status** of **Canceled** in the **Jobs** pane.
Interrupt	Terminates a running job. Enabled only if a running job is selected.
	Jobs terminated via this option will have a **Status** of **Finished** in the **Jobs** pane.
Remove	Deletes the selected job or jobs from the **Jobs** pane. Enabled only if a completed job is selected.
	Cannot be used on a running job.

Controlling the Display of Job Details

When you select a job in the **Jobs** pane, a corresponding job log is displayed in the **Details** pane. The log automatically scrolls to the bottom to keep the most recent messages in view.

If you right-click the job log, a context-sensitive menu is displayed which contains the following options:

Copy	Copy selected text in the **Details** pane.
	Alternatively, you can use the **Ctrl**+**C** key combination.
Select All	Select all of the text in the **Details** pane.
	Alternatively, you can use the **Ctrl**+**A** key combination.
Save Job Report...	Save the job log to a file.
	The job report includes job details and the contents of the job log shown in the **Details** pane.
	To save the job report:
	1. Specify the following report preferences:

	• **Include Debug Messages**: adds debugging messages to the report • **Include Log Time Stamp**: adds a log time stamp to the report • **Include Line Numbering**: displays line numbering in the report 2. Click **Browse** 3. In the **File name** field, type in the name of the report (`RSMJob.html` by default). 4. Select the report format (HTML or text) 5. Click **Save**.
View Line Numbers	Toggle the display of line numbers at the beginning of each line in the job log.
View Time Stamps	Toggle the display of time stamps at the beginning of each line in the job log.
View Exclusively File Messages	Only display file-related messages in the job log, such as those relating to file transfers.
View Debug Messages	Toggle the display of debugging information in the job log.
SubTask Filters	Filters the content of the job log by design point sub-task. This option is only displayed when the job contains design points.
Scroll to Top	Go to the top of the job log.
Scroll to Bottom	Go to the bottom of the job log.

Updating Design Points in Remote Solve Manager

Remote Solve Manager (RSM) can integrate with established compute clusters or a third-party Cloud using *configurations* defined in RSM. When these configurations are made available to the RSM client (your local computer), the RSM queues defined in those configurations are available in Workbench Each RSM queue is associated with a specific HPC queue defined on the cluster/portal side, making it possible for the update to be submitted to remote HPC resources. For more information see Working with Ansys Remote Solve Manager (p. 319).

Important:

- If you are sending design point update jobs to a remote compute cluster, the license preferences set for your local machine (the RSM client) may not be the same as the license preferences set for the remote cluster. In this case, the cluster license preferences are used for all jobs. For more information, see Setting HPC User Preferences.

- When design points are configured to be updated using RSM, the **Solution** cell cannot also be updated using RSM. To update design points in RSM, change the **Solution** cell update settings by setting the **Update Option** property to **Run in Foreground**.

Related topics:

Submitting Design Points to Remote Solve Manager

To submit design points to Remote Solve Manager (RSM):

1. To view the **Design Point Update Process** settings, right-click the **Parameter Set** bar and select **Properties**.

   If the **Properties** pane is already visible, click the **Parameter Set** bar to refresh the **Properties** pane with the design point settings.

2. In the **Properties** pane, from the **Update Option** drop-down list, select **Remote Solve Manager**.

3. Specify an available **RSM Queue**.

4. From the **Design Point Initialization** drop-down list, select one of the following options:

   • **Update From Current**: Causes each design point to be initialized from the Current design point.

   • **Update Design Points in Order**: Causes each design point to be initialized from the previous design point.

5. To update your geometry locally before submitting the design points to RSM, from the **Pre-RSM Foreground Update** drop-down list, select **Geometry**.

   Otherwise, leave the option as **None**.

6. If you selected **Geometry** in the previous step, to retain the results from the geometry update, from the **Retain Partial Update** drop-down list select **Geometry**.

7. For the **Component Execution Mode** property, select one of the following options:

   • **Serial**: All components participating in the update are run in serial mode.

   • **Parallel**: All components participating in the update that support this setting are run in parallel mode.

   Submitting a design point update to RSM is supported by the **Solution** (or **Analysis**) component update for the Mechanical APDL, Mechanical, Fluent, CFX and Polyflow solvers. These settings override any parallel or serial settings defined at the component level. When updating a compon-

ent, system, or project, the **Parameter Set** properties for a design point update are ignored. The following product specific settings are overridden for a design point update in RSM:

Product	Ignored Properties
CFX-Solver Manager or CFX-Pre **Solution** cell	• All settings under the **Parallel Environment** tab. • Run mode • All host information and partition weighting
Fluent **Solution** cell	• Use Job Scheduler • Run Parallel Version • Number of Processes • Use Shared Memory • Machine Specification • Machine List **Note:** The following properties remain enabled in all situations, if set: • Interconnect • MPI Type
Advanced section of a Mechanical Solve Process Setting	• Distributed Solution (if possible) • Max number of utilized cores
Mechanical APDL **Analysis** cell	• Processors • Distributed • MPI Type • Machine list
Polyflow **Options** accessed by **Solution** cell preferences	Number of Processes

8. In the **Number of Jobs to be Created** field, type the number of RSM jobs to create. The default
    is **1**.

    > **Note:**
    >
    > License sharing only works within a job. If you create multiple jobs, you cannot share
    > licenses across them, so we recommend creating multiple tasks within one job to take
    > advantage of license sharing.

9. In the **Number of Tasks Per Job** field, type the number of design point update tasks to run per
    RSM job. The default is **1**.

    The numbers you enter in steps 8 and 9 divide the total number of design points you are updating
    into individual tasks, with each task running a separate Workbench session. Then the tasks are
    divided up into one or more RSM jobs.

    For example, you are updating 20 design points and request two RSM jobs and five tasks per
    job. Each job will contain five tasks, and each task will update two design points. A total of 10
    Workbench sessions will be created.

10. If you selected **Parallel** in step 6, in the **Number of Cores Required Per Task** field, type the
    number of cores to use on the cluster machine for each task. The default is **2**.

    The total number of cores requested to run the entire update is calculated using the formula
    (Number of tasks * Number of Cores)+1. The additional core is for the head task, which is used
    to create all of the sub-tasks.

11. For the **Retained Design Point** property, select one of the following options:

    - **Update parameters** (default): Only parameters are updated for retained design points. If a
      component is not needed to get the value of an output parameter, it is not updated.

    - **Update full project**: Full project is updated for retained design points. Use this setting if you
      want to generate reports, or other content, from components that do not produce output
      parameters.

12. The **Remote Job Status Check Interval in Seconds** field sets the number of seconds between
    each query to RSM for the design point update job status. To increase or decrease this interval,
    enter the number of seconds into the field. The default is 20.

13. To set upfront license checkouts, click **Select Licenses**. For more details, see Using Upfront License
    Checkouts for Design Point Updates (p. 358).

14. Save the project.

    If you are working in an archived project, you must save the project to a permanent location.

15. Update the design points.

The project is archived and submitted to a cluster using RSM. The remote data is retrieved periodically as the design point updates complete. Should a design point update fail, an error is reported to the **Messages** pane.

Updating Geometry Locally Before Submitting to Remote Solve Manager

You can update your geometry locally before submitting design point updates to Remote Solve Manager (RSM). Do this when:

- Design points are being updated simultaneously in RSM. The geometry update infrastructure does not support simultaneous design point updates in the same source geometry.

- The compute cluster cannot access the required CAD licenses.

- The compute cluster does not have a CAD program installed

Performing a Pre-RSM Geometry Update

This is the most common workflow used to update geometry locally.

To perform a pre-RSM geometry update:

1. On the **Project Schematic**, right-click the **Parameter Set** bar and select **Properties** from the context menu.

   If the **Properties** pane is already visible, click the **Parameter Set** bar to refresh the **Properties** pane with the design point settings.

2. Set the **Update Option** to **Remote Solve Manager** and then specify an available **RSM Queue**.

3. In the **Design Point Update Process** settings, set **Pre-RSM Foreground Update** to **Geometry**.

   > **Note:**
   >
   > If the project includes a Geometry component and you have set Remote Solve Manager (Legacy) **Job Submission** to either **One Job for Each Design Point** or **Specify Number of Jobs** (with the value set to greater than **1**), the setting does not apply to the geometry update. The geometry is updated locally in the foreground before your design points jobs are sent to RSM.

4. Update the remaining **Design Point Update Process** settings as required.

   For detailed setting descriptions, see Submitting Design Points to Remote Solve Manager (p. 353).

5. Save the project (p. 269).

   If you are working in an archived project, you must save the active project.

6. Update the design points (p. 306).

The project is archived and submitted to a cluster using RSM. The remote data is retrieved periodically as the design point updates complete. Should a design point update fail, an error is reported to the **Messages** pane.

Performing a Geometry-Only Design Point Update

This is a non-standard workflow, but it allows you to move the geometry updates to the compute cluster manually.

To perform a geometry-only design point update:

1. On the **Project Schematic**, right-click the **Parameter Set** bar and select **Properties** from the context menu.

    If the **Properties** pane is already visible, click the **Parameter Set** bar to refresh the **Properties** pane with the design point settings.

2. In the **Properties** pane, from the **Update Option** drop-down list, select **Run in Foreground.**

3. From the **Partial Update** drop-down list, select **Geometry**.

4. To retain the results from the partial update, from the **Retain Partial Update** drop-down list, select **Geometry**.

    > **Note:**
    >
    > This setting causes the geometry results to be saved whether you perform a partial update or a full update.

5. Save the project (p. 269).

    If you are working in an archived project, you must save the active project.

6. Update the design points (p. 306).

7. Transfer the project files from the local machine to the RSM compute cluster.

Design Point License Sharing

When Workbench design point updates are submitted to Remote Solve Manager on a cluster, you can specify the number of tasks in a RSM job to update concurrently. You can also specify multiple jobs to be submitted to reduce the individual job size.

Licenses are not shared between different RSM jobs.

However, licenses are shared between concurrent tasks within the same job using HPC licenses, allowing you to leverage standard HPC licenses instead of extra solver licenses. Each task requires eight HPC increments or one HPC Pack increment. Similar to non-parametric simulations, each task includes four parallel cores. Additional HPC parallel cores are shared across all tasks, using the same HPC rules. All licenses that are consumed during a RSM job are held for the duration of the job, based on the maximum count of each license.

You can use the Ansys Parametric Licensing Calculator (https://ansys.com/parametric) to determine the number of HPC Workgroup or HPC Pack licenses required.

Tracking Licenses

Workbench tracks licenses that are used during an update. You can see what licenses were used for any component by viewing that component's properties. The **Last Update Used Licenses** property displays which licenses were used. You may find it useful to run the update of at least one design point as you normally would and then review the licenses that were used so that you know which licenses are needed for a future study. License usage is tracked only for an update operation in a component. It is not tracked if a component becomes up-to-date as the result of an edit operation. License usage is not tracked for DesignXplorer components or any parametric components.

License tracking may not return the information in some cases, for example if a component editor was open for viewing or editing at the time of the update. In this case, **Not Applicable** is shown for **Last Update Used Licenses**, even though a license is used. To ensure that tracking captures information for all components, close all component editors before updating.

> **Note:**
>
> License tracking can be turned off by your corporate license administrator. If you run a study and do not see which licenses were used, contact your license administrator.

Using Upfront License Checkouts for Design Point Updates

Ansys Common Licensing eliminates the need to reserve licenses ahead of the update. The license sharing based on HPC context automatically starts when design points are updated using Remote Solve Manager. In some situations, you may need to have your design point RSM job fail on startup if certain set of licenses are not available. To do this, you can specify licenses that are checked out at the start of RSM job execution, so that the job fails early if there are insufficient licenses. If more licenses are required during the update than were checked out upfront, they are checked out automatically based on HPC context.

To use upfront license checkouts:

1. To view the **Design Point Update Process** settings, right-click the **Parameter Set** bar and select **Properties**.

    If the **Properties** pane is already visible, click the **Parameter Set** bar to refresh the **Properties** pane with the design point settings.

2. In the **Properties** pane, from the **Update Option** drop-down list, select **Remote Solve Manager**.

3. Click **Select Licenses**.

4. In the **Select Licenses** dialog box:

    a. From the **Available Licenses** list, select one or more licenses.

        You can select multiple licenses at one time by holding down the **Ctrl** key as you click each license.

        The Available Licenses list displays licenses currently available on the license server connected to the Workbench session, which may or may not be the license server connected to the RSM cluster.

    b. Click **Add**.

        The licenses you have selected appear in the **Selected Licenses for Upfront Checkout** pane.

    c. To see which licenses were used for this Workbench project in previous updates, switch to the **Used Licenses** tab.

        You can add licenses to the selection from this list as well.

    d. To save the selection and close the dialog, click **OK**.

        The selected licenses are not checked out at this time. License checkout occurs only when you begin the update.

You may need to include a Geometry license as part of upfront-checkout licenses even when the Geometry component is not parameterized and is up-to-date, if either of the following situations is true:

•   If the project contains CAD geometry or any other geometry that is not managed by the Geometry component but rather by a downstream Model or Mesh component, and the geometry is parameterized using the Model or Mesh component.

•   If an Engineering Data component is parameterized and shares a model or mesh downstream with a non-parameterized Geometry component.

Aborting or Interrupting an RSM Design Point Update

You can abort or interrupt an RSM update that is in progress.

To abort or interrupt the update:

1.   In the **Progress** pane, while the **Status** column displays the message `Waiting for back-ground task`, click the stop button () in the **Progress** column.

2.   In the dialog box, select one of the following options:

   •   To stop the RSM job and return no data to the project, click **Abort**.

   •   To interrupt the RSM job and return any available data to the project, click **Interrupt**.

   •   To let the update continue and close the dialog box, click **Cancel**.

Exiting a Project during an RSM Design Point Update

You can exit a project while an RSM design point update is in progress. During the update, while the one or more RSM jobs are running in the background, the **Progress** pane displays a project **Status** of **Waiting for background task**.

> **Note:**
>
>   •   If you have submitted the **Update All Design Points** operation to RSM and are unable to exit the project because Workbench remains busy, see Suspending and Resuming Collection of RSM Design Point Results (p. 361).
>
>   •   You cannot exit Workbench while job files are being uploaded. However, you can abort the job during the upload by clicking the stop button () in the **Progress** bar.

For RSM jobs to continue to run after you exit the project, the project must be saved at least once after the design point update job was initiated.

If you attempt to exit a project while a design point update job is still running, the following scenarios cause a dialog box to open, allowing you to specify whether you want to save the project before exiting:

•   You have never saved the project at any time after the same design point update job was initiated.

•   You have saved the project at least once after the same design point update was initiated, but design point results have been retrieved since your last save.

In either of these cases, if you do not save the project before exiting:

•   All design point results retrieved since the last **Save** operation are lost. However, if the project has been saved at least once since the update job was initiated, the results can be retrieved again when the project is reopened.

•   If the project has never been saved after the update job was initiated, all RSM jobs are aborted and show a **Status** of **Cancelled**. The cancelled icon () displays in the RSM **List** pane. The asterisk on the icon indicates that the job has also been released. When you reopen the project, it is in the state of your last manual save.

If you do save the project before exiting:

- Retrieved design point results are saved to the project.

- RSM jobs that are queued and running continue to run after you exit.

- Jobs where the results have been saved show a **Status** of **Finished**. The finished icon (🏆) displays in the RSM **List** pane. The asterisk on the icon indicates that the job has also been released.

- Jobs where results have not yet been saved are not released upon exit. When you reopen the project, you can resume update of the pending jobs to reconnect and download the results.

Suspending and Resuming Collection of RSM Design Point Results

When the **Update All Design Points** operation has been submitted to RSM, Workbench may be collecting intermediate and final results and not allow you to exit the session. To pause the data collection so you can exit the Workbench session, select **Tools → Suspend Collection of RSM Results**.

When you exit the project, results are saved according to the criteria described in Exiting a Project during an RSM Design Point Update (p. 360).

> **Note:**
>
> The **Suspend Collection of RSM Results** option is also available during a project update in RSM. However, because only the current design point is being updated and RSM collects results only at the end of the update process, this option has no impact on the project update.

To resume the collection of design point data, select **Tools → Resume Collecting RSM Results**. The project reopens in the same state it was in when it closed. Once all the design points have been updated, all of the suspended results are collected and then updated to the project at the same time.

It is not necessary to use the **Resume Collecting RSM Results** option. The collection and update happen automatically when all jobs have been completed.

Product-Specific Limitations

Some products have additional limitations when submitting design points updates to RSM:

Ansoft

Projects that include Ansoft systems return updated parameter values but do not return the detailed solution for the current design point.

Mechanical

Design point updates in RSM can fail for a class of problems (typically involving a Structural system linked to a Modal or other type of Mechanical system) that meet the following criteria:

- A non-parameterized upstream Mechanical system provides solution data to a parameterized downstream Mechanical system and both systems share the same geometry/model.

- Updates are performed via a pre-RSM local update.

The parameters for some of the design points are not computed and are marked as errors in the design points table.

Workarounds:

- Select the **Enable Legacy Solve** check box (**Tools** → **Options** → **Mechanical**).

- In the **Parameter Set** properties, set **Pre-RSM Foreground Update** to **None** and **Job Submission** to **One Job for All Design Points**.

Third-Party CAD Systems

Projects that include geometry parameters that rely on third-party CAD systems do not update the geometry unless the CAD system is accessible on the execution node.

Working with Design Point Service

Design Point Service (DPS) is part of Distributed Compute Services (DCS). DPS can quickly and efficiently evaluate tens of thousands of design points using multiple compute resources. It supports submission of design point updates on local machines and on HPC clusters using Remote Solve Manager (RSM).

- For information about DPS, see the DCS for Design Points Guide.

- For information about RSM, see Working with Ansys Remote Solve Manager (p. 319).

The following steps provide a quick start for a single-user desktop scenario. They summarize how to set up a Workbench project to use DPS for design point updates.

1. If DCS is not installed on your machine, install it using the Ansys installer. For more information, see DCS Installation in the *DCS for Design Points Guide*.

2. Create a Workbench project, identify input and output parameters for the simulation, and insert a few design points.

    The remaining steps summarize the detailed information given in Setting Up a Workbench Project to Use DPS (p. 364) and Submitting Design Point Updates to DPS (p. 365).

3. In the **Project Schematic**, right-click the **Parameter Set** bar and select **Properties**.

4. Under **Solution Process**, set **Update Option** to **Submit to Design Point Service (DPS)**.

    In a single-user desktop scenario, you can use the defaults for the other properties under this heading to have a local evaluator update the design points that are sent to DPS.

5. Under **Design Point Update Process**, set **Component Execution Mode** to **Parallel** and **Number of Processes** to **2** or more, depending on your machine.

6. Click **Update All Design Points** and, when prompted, enter DCS credentials to sign in and send the design points to DPS for evaluation. Default system administrator account information for all DCS services is given in DCS Installation.

7. Either view results as they are written back to the Workbench project or, in the status bar, click the button for opening the DPS web app (🖥), where you can view, filter, sort, and compare the design points in the connected DPS project. For more information, see DPS Web App in the *DCS for Design Points Guide*.

Related topics:

    Setting Up a Workbench Project to Use DPS

    Submitting Design Point Updates to DPS

    Submitting Project Updates to DPS

    Accessing the DPS Web App from a Workbench Project

Setting Up a Workbench Project to Use DPS

In the properties for the Workbench **Parameter Set** bar or **Project Schematic**, you set **Update Option** to **Submit to Design Point Service (DPS)**. When you update design points or the project, the Workbench project definition and either unsolved design points or project updates are sent to DPS for evaluation.

1. In the **Project Schematic**, right-click the **Parameter Set** bar or the white space of the **Project Schematic** and select **Properties**.

2. In the **Properties** view, set **Update Option** to **Submit to Design Point Service (DPS)** and then set the other properties appropriately.

	A	B
1	Property	Value
2	☐ Solution Process	
3	Update Option	Submit to Design Point Service (DPS) ▼
4	Server URL	https://localhost/dcs
5	Database Name	BoltedValve
6	Auto-Start Local Evaluator	☑
7	Execution Time Limit (seconds)	300
8	Update Design Points in Steps	☐
9	☐ Design Point Update Process	
10	Component Execution Mode	Parallel ▼
11	Number of Processes	2

See the following descriptions for DPS-related properties:

Property	Description
Update Option	When set to **Submit to Design Point Service (DPS)**, the request for updating one or more design points in the **Parameter Set** bar, any cell in a DesignXplorer system, or a project update is sent to DPS. For this setting, Workbench checks to see if the URL setting for the DCS server is valid and if the server is responding. If a problem exists, a dialog box opens, asking whether you want to start a local server or supply a valid URL to a remote server.
Server URL	URL for starting the DCS server. • The URL for starting a local DC local server is `https://localhost/dcs` on Windows or `https://localhost:8443/dcs  on Linux`. • The URL for starting a remote DCS server on Windows is `https://host-name/dcs` or `https://IP_Address/dcs`. • The URL for starting a remote DCS server on Linux is `https://host-name:port/dcs` or `https://IP_Address:port/dcs`.

Property	Description
Database Name	Name of the DPS project to which this Workbench project connects. The DPS project stores and manages the project definition and design points. If you leave this property blank, the DPS project is given the same name as the Workbench project.
Auto-Start Local Evaluator	Whether to automatically start an evaluator on the local machine when updates are requested. This check box is selected by default. If you clear this check box, you can start the evaluator on the local machine by going to the Windows **Start** menu and selecting **Ansys 2022 R2** → **DC Evaluator (DCE) 2022 R2**. You can also start it from the command line by typing `dclauncher.exe start evaluator`.
Execution Time Limit (seconds)	Maximum execution time in which to update one design point or run the project update. If the update does not complete within this time, the evaluation is terminated. In the DPS web app, the design point is then assigned to the **timeout** status filter. For more information, see DPS Web App in the *DCS for Design Points Guide*.
Update Design Points in Steps	Whether to distribute the computing for different tasks between different applications and evaluators. For example, the project can use Ansys Discovery SpaceClaim to update the geometry and Ansys Fluent to generate results. SpaceClaim could be running on a Windows machine while Fluent is running on a Linux cluster, cloud service, or local machine. For more information, see Evaluating Design Points or Project Updates in Multiple Process Steps (p. 369).
Component Execution Mode	Specifies whether to run the solution in serial or in parallel. • When **Serial** is selected (default), updates are executed on only one CPU core at a time. • When **Parallel** is selected, updates are executed on multiple CPU cores.
Number of Processes	Specifies the default number of processes to use if you are running updates in parallel. This property must be set to **2** or greater.

Note:

In **Tools** → **Options** → **Distributed Compute Services**, you must set a default URL for starting the DCS server and specify whether to auto-start a local evaluator. For more information, see Distributed Compute Services (p. 427). Here in the **Properties** pane for the **Parameter Set**, you can change these inherited properties for this particular project.

Submitting Design Point Updates to DPS

When the **Update Option** is set to **Submit to Design Point Service (DPS)**, any update of design points from either the **Parameter Set** bar or a DesignXplorer system cell will send the Workbench project definition and unsolved design points to DPS for evaluation. The Workbench project definition includes the parameter definitions, workflow definition, and project files.

Clicking **Update All Design Points** in the **Parameter Set** bar opens a dialog box. The prompt depends on if there are design points that require updating and if the DPS project exists.

• If there are no design points that require updating, you are told that the output parameters for all design points are up-to-date.

- If the Workbench project has unsaved changes, you are told that you must save the Workbench project to proceed. Clicking **Save and Proceed** saves the project and sends the project definition and unsolved design points to the DPS project.

- If the DPS project exists but is not the same as the last one synced from the Workbench project, you are asked if you would like to replace the project and its design points, update the project but keep its design points, or create a new project. If you click **Create new**, a dialog box opens for entering a different name for the DPS project. The name can contain only letters and digits.

- If the DPS project cannot be found because it has either been deleted or moved to a different server, you are asked if you want to create a new DPS project with the same name. Clicking **Create** sends the Workbench project and design points to the new DPS project.

As unsolved design points are sent to the DPS project, evaluators pick them up and run the simulation tasks, writing results back to the Workbench project as the evaluations are completed.

Throughout this process, Workbench displays processing information in the status bar and the **Progress** view. In the status bar, you can click the button for opening the DPS web app (🖥). The button displays the name of the connected DPS project. Or, from the **Jobs** menu, you can select **Open Design Point Service Web App**.

In the DPS web app, you can view project information, configure the project, and filter, sort, and compare design points. For more information, see DPS Web App in the *DCS for Design Points Guide*.

> **Note:**
>
> If you exit Workbench before the update finishes, evaluators continue to update the design points in the DPS project. When you next open the Workbench project, it automatically syncs the design points in the Workbench and DPS projects. If you later make parameter or project changes in the Workbench project, selecting **Jobs → Sync with Design Point Service** pushes these changes to the DPS project.

Submitting Project Updates to DPS

When the **Update Option** is set to **Submit to Design Point Service (DPS)** and you update the project, the Workbench project definition and project updates are sent to DPS for evaluation. The Workbench project definition includes the workflow definition and project files.

To submit project updates to DPS:

1. To save the project, from the menu bar, select **File → Save**.

2. Update the project:

   - From the menu bar select **Tools → Update Project**.

   - On the **Project** tab toolbar click **Update Project**.

   - Right-click the white space on the **Project Schematic** and select **Update Project** from the context menu.

The project update starts. Throughout this process, Workbench displays processing information in the status bar and the **Progress** view. In the status bar, you can click the button for opening the DPS web app (🖥️). The button displays the name of the connected DPS project. Or, from the **Jobs** menu, you can select **Open Design Point Service Web App**.

In the DPS web app, you can view project information and configure the project. For more information, see DPS Web App in the *DCS for Design Points Guide*.

Note:

If you exit Workbench before the update finishes, evaluators continue the process in the DPS project. When you next open the Workbench project, it automatically syncs the c in the Workbench and DPS projects. If you later make parameter or project changes in the Workbench project, selecting **Jobs → Sync with Design Point Service** pushes these changes to the DPS project.

Accessing the DPS Web App from a Workbench Project

When a Workbench project has an established DPS project connection, you can access the DPS web app to view DPS project information using one of these methods:

- In the status bar, click the button for opening the DPS web app (🖥️). The button displays the name of the connected DPS project. If the Workbench project is not connected to a DPS project, the button displays **No DPS Connection**.

- Select **Jobs → Open Design Point Service Web App**.

You are taken to the **Design Points** tab for this DPS project so that you can view design point information.

Note:

- If you try to access the DPS web app before a DPS project connection is established, you are taken to the DPS web app's **Projects** page, where you can click the name of a DPS project to go to its **Design Points** tab.

- If you want to sign out of DCS from a Workbench project with a DPS connection, select **Jobs → Sign out from Distributed Compute Services**.

Importing DPS Design Points into a Workbench Project

If you have evaluated design points in a DPS project, you can import any design points of interest into the Workbench project for further analysis or review:

1. On the **Project Schematic**, open either the **Parameter Set** bar or the DesignXplorer cell into which to import DPS design points.

2. In the **Table** pane, right-click and select one of the following, depending on your location:

- **Import from Design Point Service**: Select to import DPS design points into the design points table for the **Parameter Set** bar or into a DesignXplorer custom DOE or parameters correlation.

- **Import Refinement Points from DPS**: Select to import DPS design points into the re-finement points table in a DesignXplorer response surface.

- **Import Verification Points from DPS**: Select to import DPS design points into the verification points table in a DesignXplorer response surface.

- **Import Custom Candidate Points from DPS**: Select to import DPS design points into a custom candidate points table in a DesignXplorer optimization.
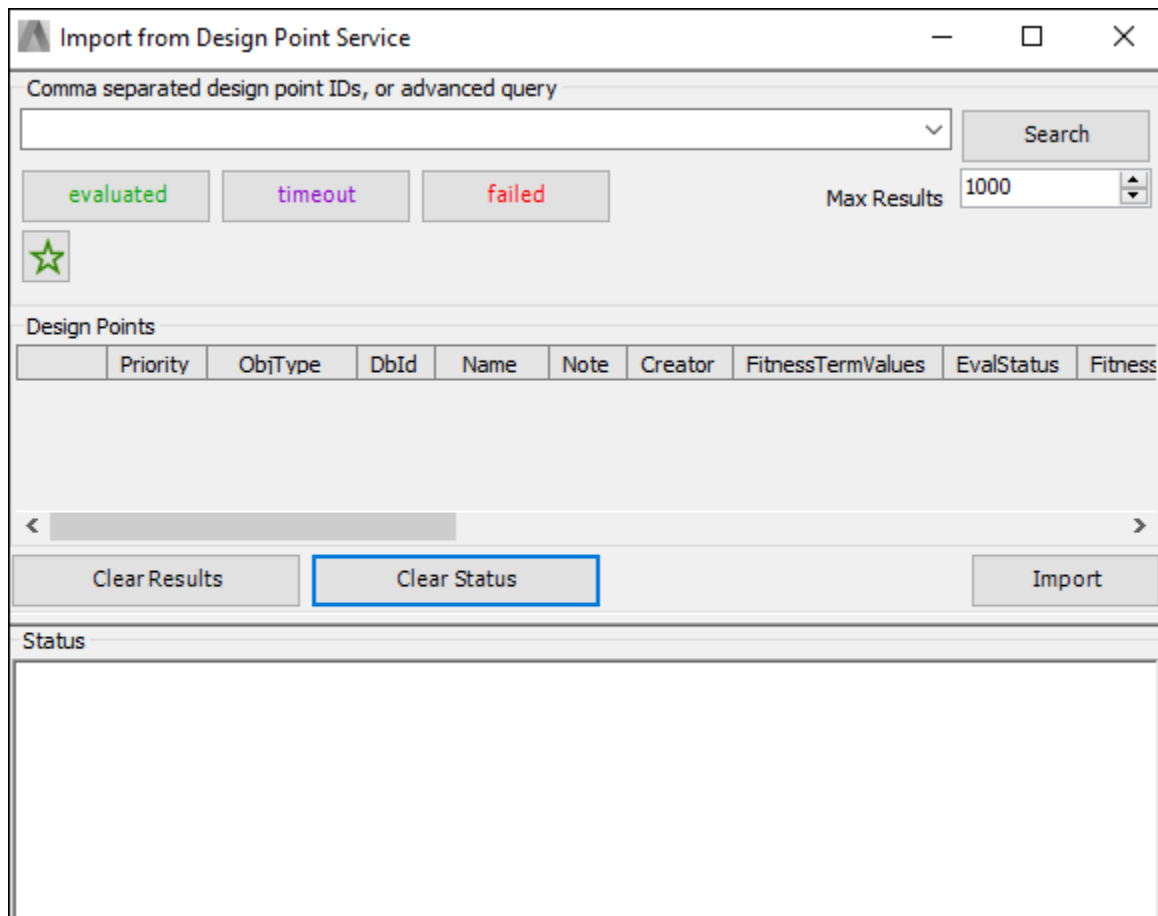
> **Note:**
>
> The above options are also available on the context menu for the **Outline** pane if you right-click a node that supports importing design points from DPS.

The **Import from Design Point Service** window opens.

To select the DPS design points to import, you can specify design point IDs, enter a query string, or enable filters. When you place the mouse cursor over the description for the field in which you enter IDs or a query string, the tooltip displays usage examples.

By default, the maximum number of design points returned from a query or from enabled filters is **1000**. However, before querying or filtering, you can change the value for **Max Results**. This field appears in the upper right corner, below the **Search** button.

3. Select the design points to import using one of these methods:

   - Specify design points IDs, separating them with commas, and click **Search**.

   - Enter or select a previously entered query string and click **Search**.

   - Click the buttons for filters to switch between enabling and disabling them. Choices are for the **evaluated**, **timeout**, and **failed** status filters and the star filter.

   The heading for the **Design Points** pane displays the total number of design points selected in parentheses. The table displays these design points. Below the table, the buttons on the left clear the table of results and clear the **Status** pane of processing information. On the right, the **Import** button imports the selected design points into the **Parameter Set** bar.

4. Once the table displays the design points that you want to import, click **Import**.

   The window closes and a progress bar appears.

If the DPS project and Workbench project have differences beyond just design points, a dialog box indicates that imported design points will be set out-of-date and asks if you want to continue. If you click **Yes**, before it imports the design points, DesignXplorer parses and validates the design point data. For more information, see Parsing and Validation of Design Point Data in the *DesignXplorer User's Guide*.

The design points that are imported into the Workbench project from DPS are not linked to the design points in DPS. You can perform standard operations on an imported design point, such as setting it as the current design point.

Although output parameter values for imported design points are read-only, you can set them as editable. For more information, see Editable Output Parameter Values in the *DesignXplorer User's Guide*.

Evaluating Design Points or Project Updates in Multiple Process Steps

You can set up a Workbench project to have DCS evaluate design points or project updates in multiple process steps (tasks) on multiple machines. For example, you might be using standalone SpaceClaim, third-party geometry and CAE tools plus an Ansys solver tool, and a Python postprocessing tool.

When running simulations in a mixed environment:

1. In the **Project Schematic**, right-click the **Parameter Set** bar or the white space of the **Project Schematic** and select **Properties**.

2. In the **Properties** pane, select the **Update Design Points in Steps** check box.

3. Select **Jobs** → **Sync with Design Point Service** to send the updated simulation workflow definition to the DPS project.

4. In DPS, go to the project's **Design Points** table and click the **Configurations** tab.

5. Open the appropriate configuration's details view.

In the task pane on the right, two tasks are now visible:

- Workbench_Geometry

- Workbench_Solution

6. Define each of these tasks.

Example

Assume that the geometry step requires SpaceClaim (SCDM), which runs on a Windows machine, and the solution step requires a custom solver (`my_solver`), which runs on a Linux machine.

Only an evaluator with the SCDM capability can update the geometry, and only an evaluator with the `my_solver` capability can update the solution.

For the geometry step, the default settings sent from the Workbench project should be correct. For the solution step, because only an evaluator with the `my_solver` capability on the Linux machine can update the solution, you would have to add `my_solver` as a capability and click **Save**.

After you submit a design point update to DPS, in the design point's Viewing Design Point Details, the **Evaluation** area will display information for the two tasks that have been run on two different evaluators.

> **Note:**
>
> While there cannot be more than two tasks when design points are updated from a Workbench project, you can use the lightweight DCS Python Client to script any number of process steps.

When an evaluation has multiple tasks, they run sequentially by default. For example, based on the example, the geometry is updated first, and the solution is updated second. However, when defining a task, you can use **Execution Level** in the **Requirements** area to specify the sequence for executing the task when multiple tasks are defined. All tasks set to **0** can be evaluated at the same time, supporting an interesting workflow if there are multiple unrelated load cases.

Working with Ansys optiSLang

The Ansys Workbench optiSLang Extension optimizes your product designs by combining the powerful parametric modeling capabilities of Workbench with the Robust Design Optimization (RDO) methods of optiSLang. The optiSLang toolbox includes modules for sensitivity analysis, optimization, and robustness evaluation that you can easily drag and drop into a Workbench project to form an interactive process chain.

Installing the Ansys Workbench optiSLang Extension

There are three methods to install the extension:

- Installed with optiSLang using the optiSLang standalone installer. For installation details, see the optiSLang Installation and Licensing Guide.

- Installed using the Ansys unified installer (p. 371) and then activated.

- Installed manually (p. 372) when you have the extension file but no installer.

Installing and Activating the Extension Using the Ansys Unified Installer

You can install optiSLang and the Ansys Workbench optiSLang Extension through the Ansys unified installer.
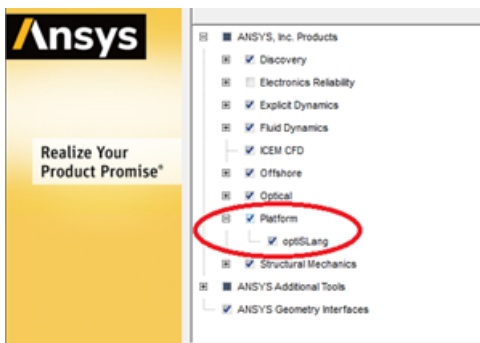
When you first open Workbench, the extension is not active. To activate the extension:

1. From the menu bar, select **Extensions → Manage Extensions**.

2. To load the extension, select the check box to the left of **optiSLang**.

3. To have the extension load each time you open Workbench, right-click **optiSLang** and select **Load as Default** from the context menu.
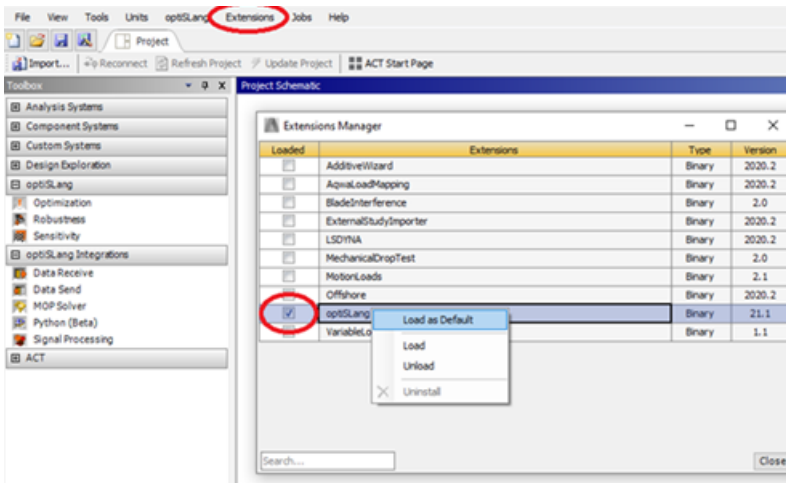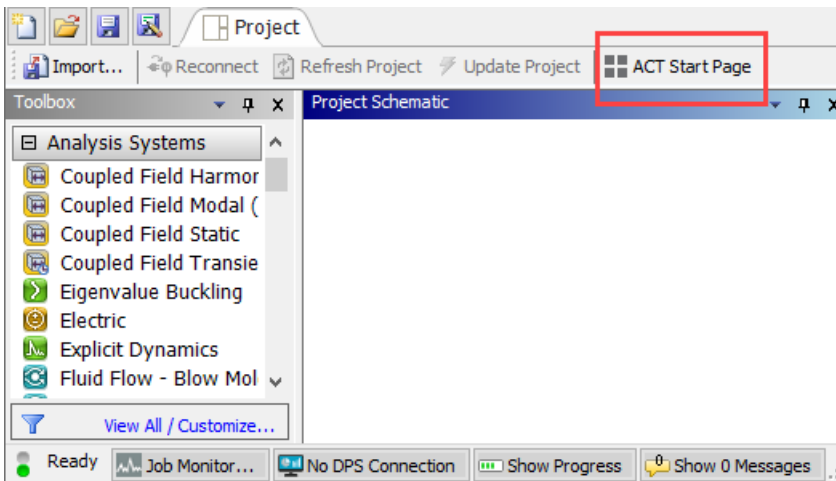
Manually Installing Using the Extension File

If you do not have an optiSLang standalone installer or the Ansys unified installer, but you have the `optiSLang version_number.wbex` extension file, you can install it manually.
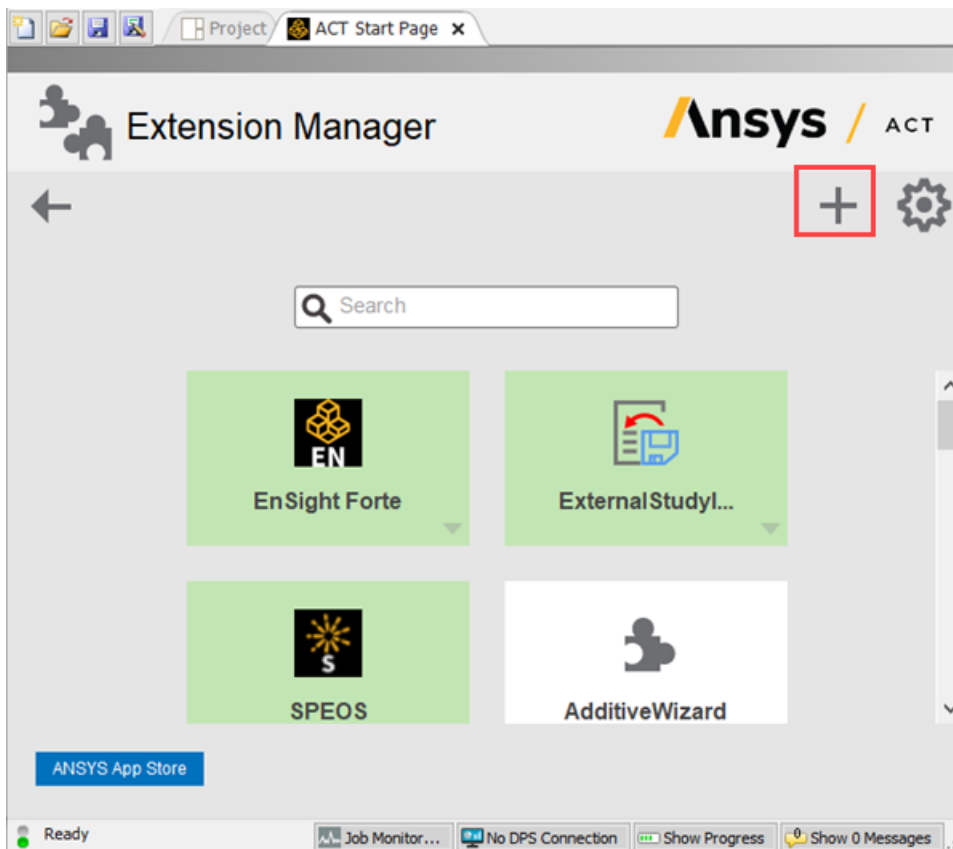
To install and activate the extension manually:

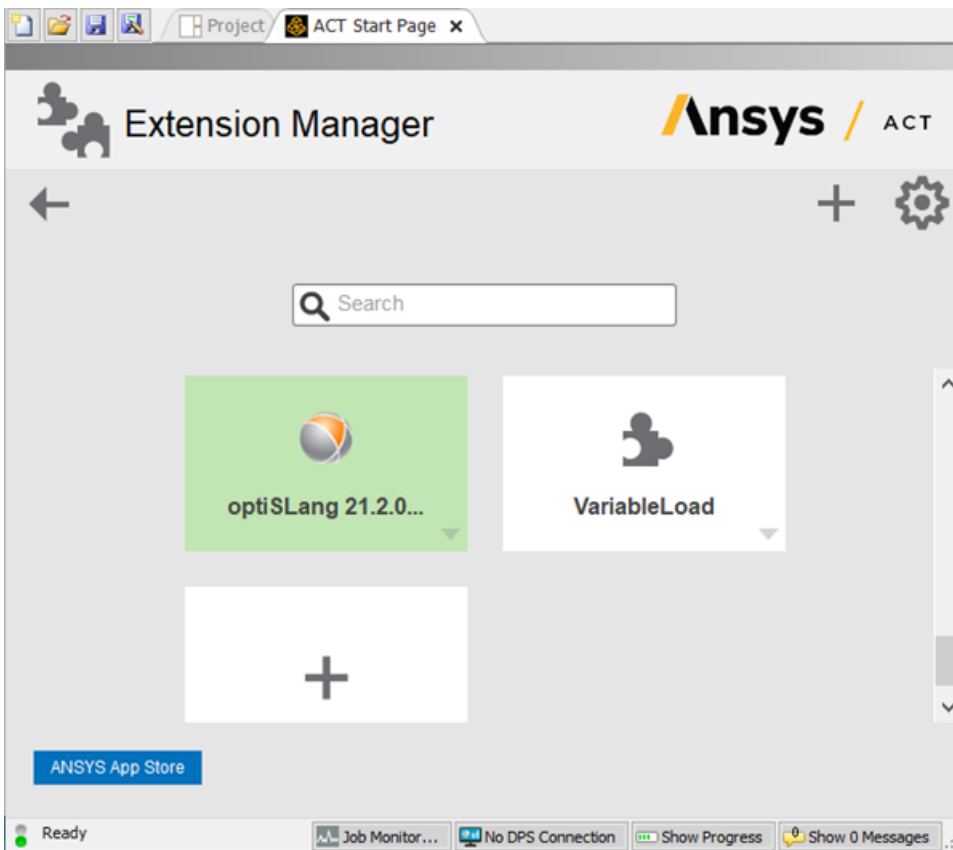1. On the **Project** tab toolbar, click **ACT Start Page**.

2. On the **ACT Start Page** tab, click **Manage Extensions**.
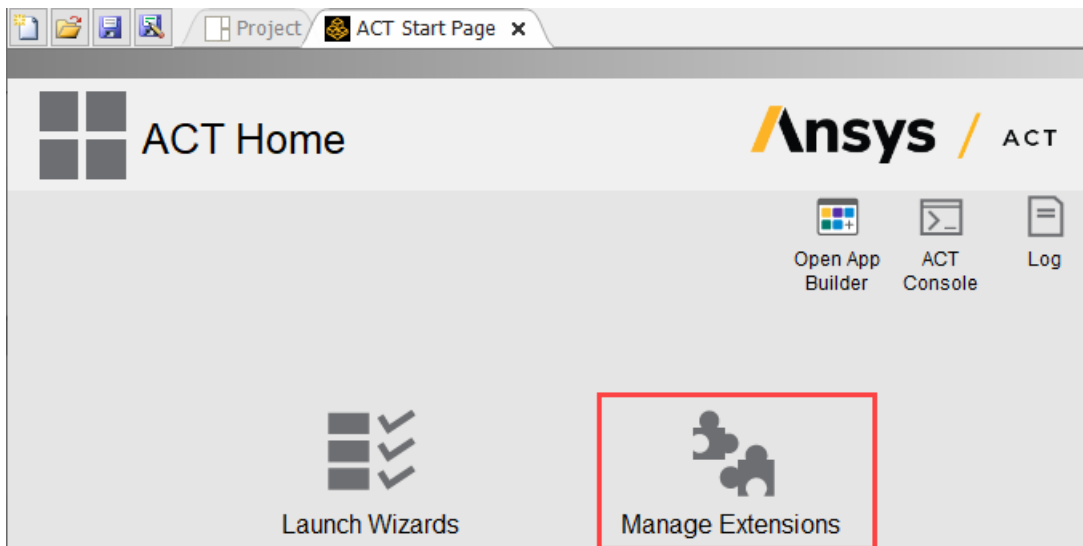
3. To install the extension, click **+**.
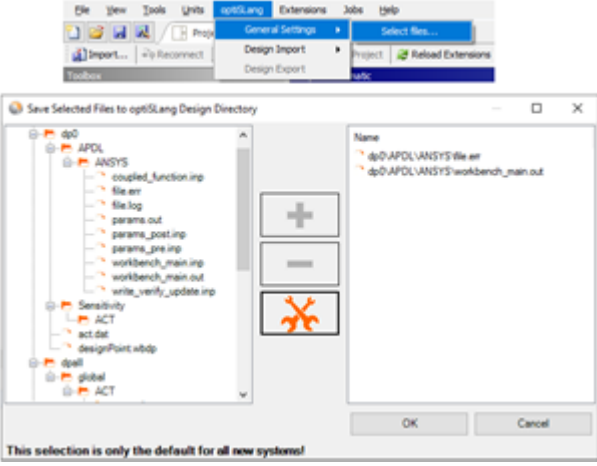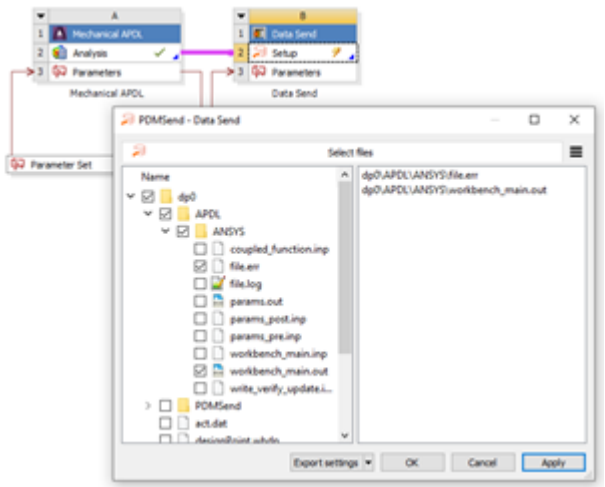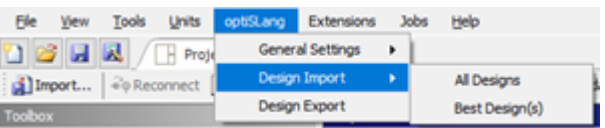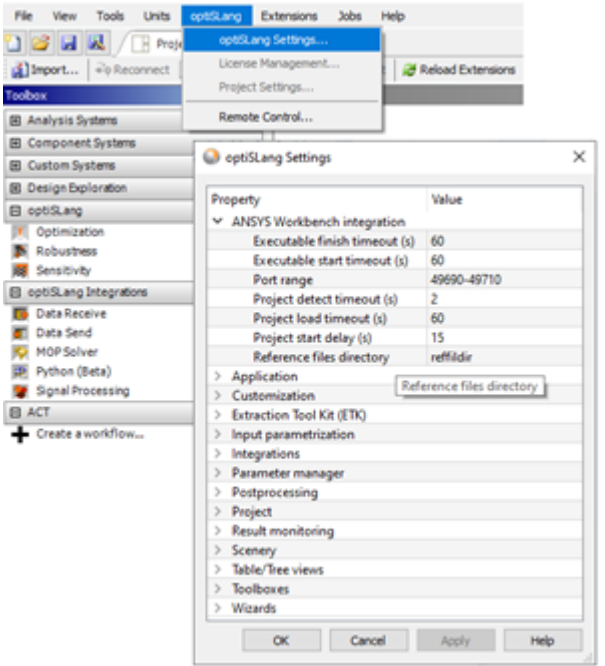
4. Browse to the location of the `optiSLang version_number.wbex` and click **Open**.

5. Activate the extension by clicking it. The box turns green to indicate it is active.
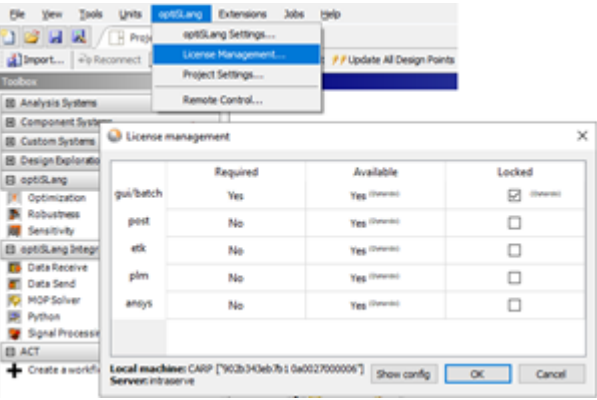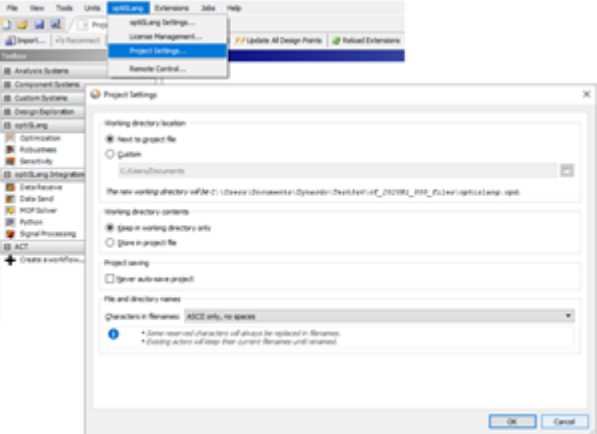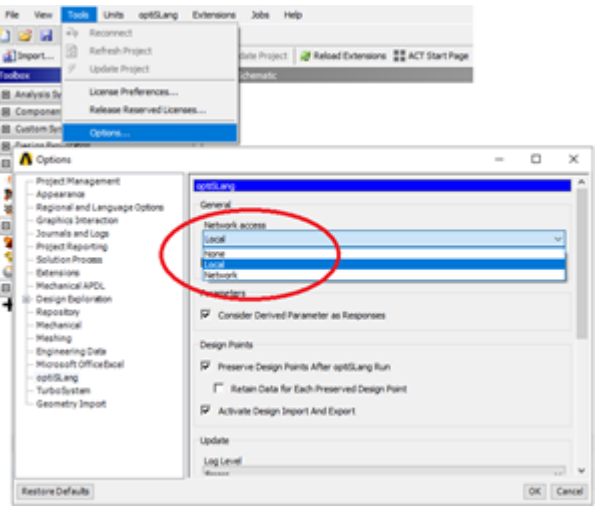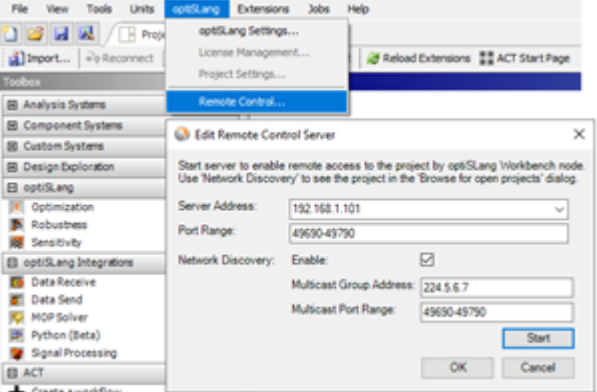
Migrating Workbench Projects from the SDK to ACT Extension
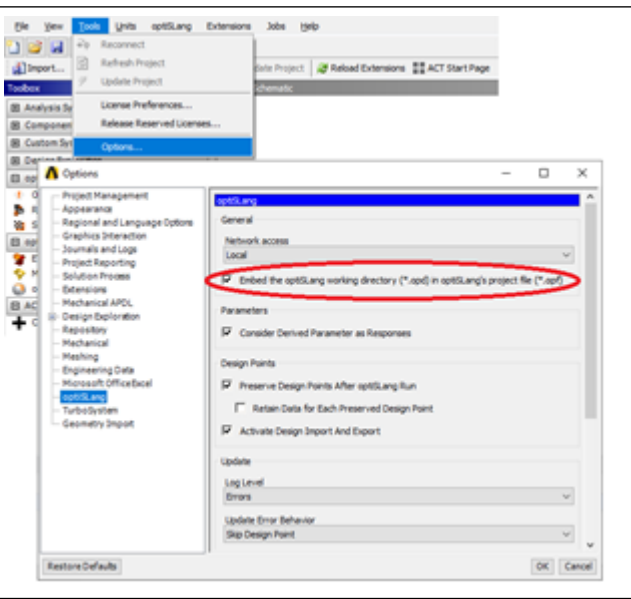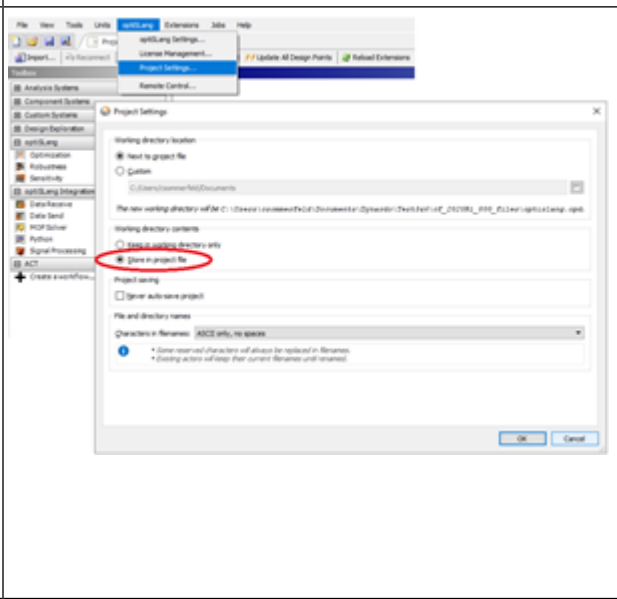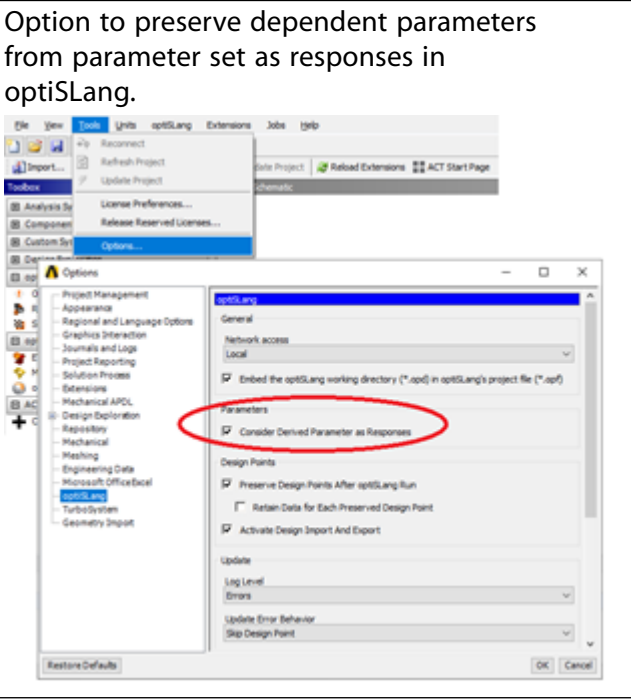
Earlier versions of the extension were written using an SDK, while the latest version has been written using Ansys ACT. The following table demonstrates the differences between the two extension versions.
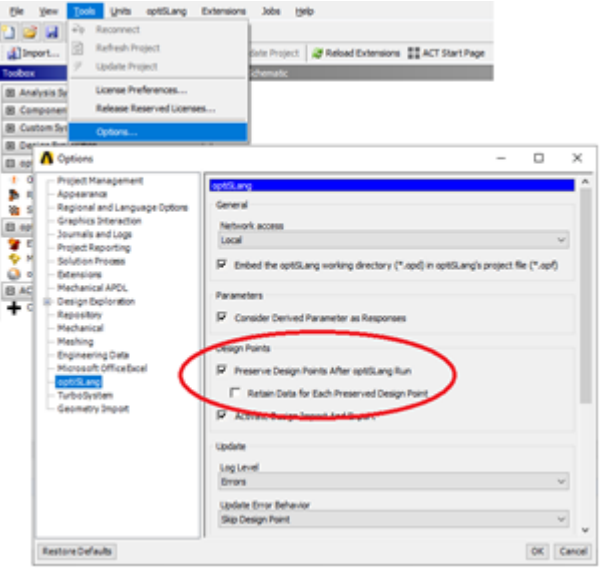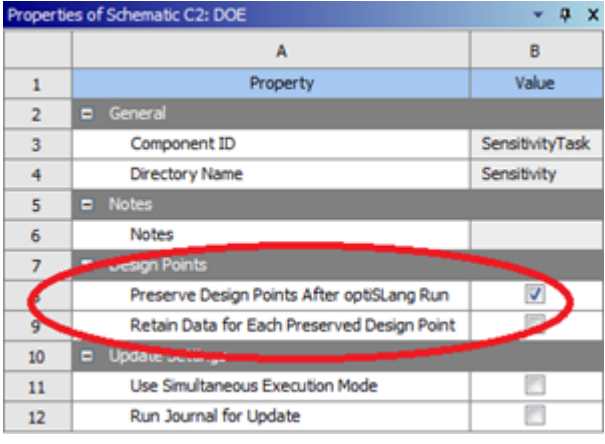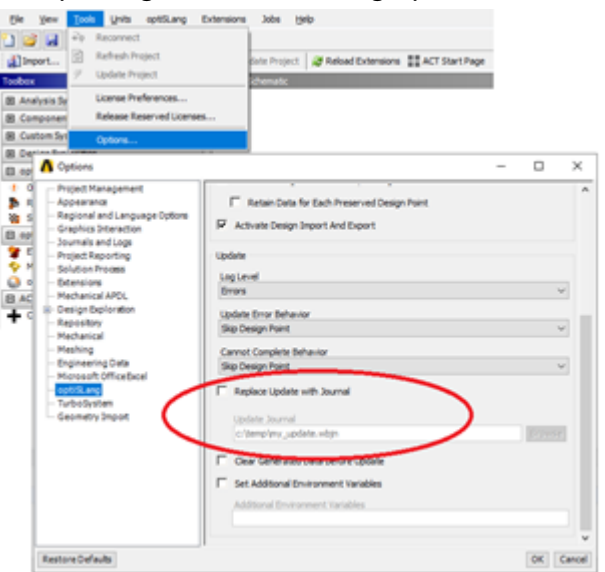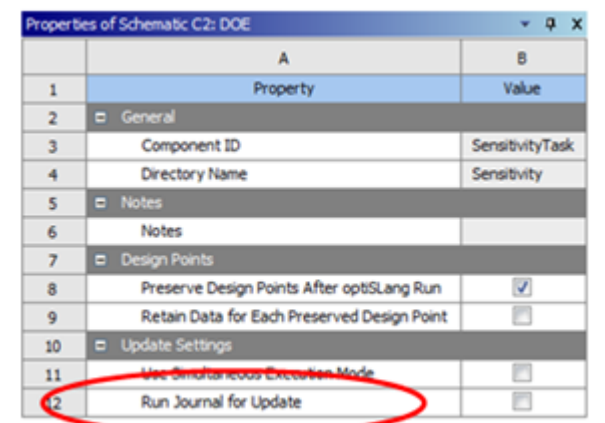
Function	SDK	ACT
Select files	Save project files generated during design point update to the optiSLang design directory.	Save project files generated during design point update to the optiSLang design directory with a separate **Data Send** component system.
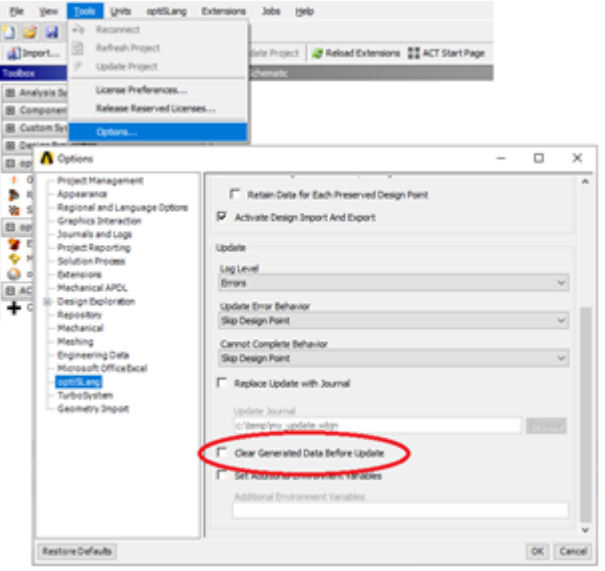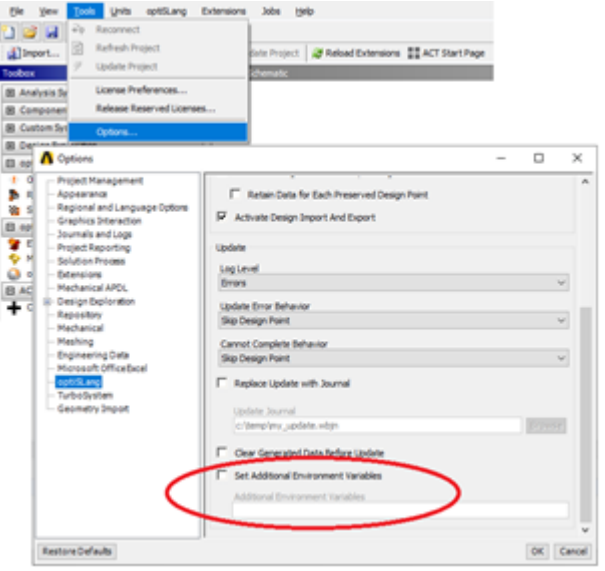
Function	SDK	ACT
Data import and export	Exchange of designs between Workbench design table and optiSLang database file (*.omdb). 	Will be provided in a future version.
optiSLang configuration	N/A	Displays the general application settings in optiSLang.
License management	N/A	Displays all available and used licenses in optiSLang.
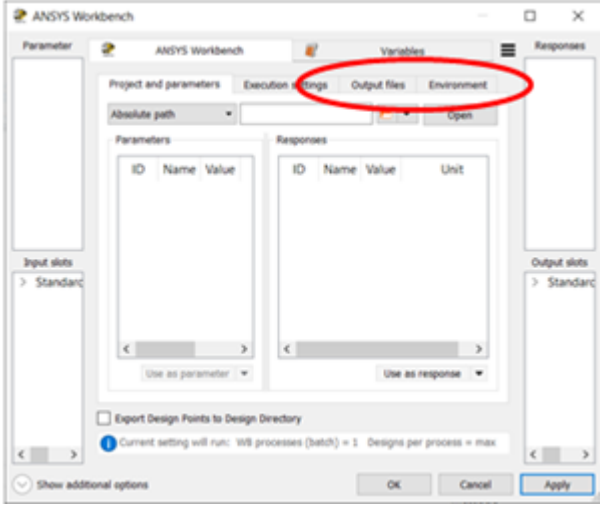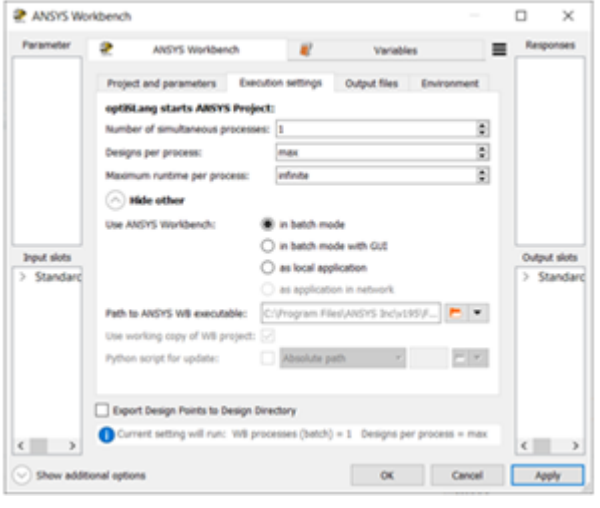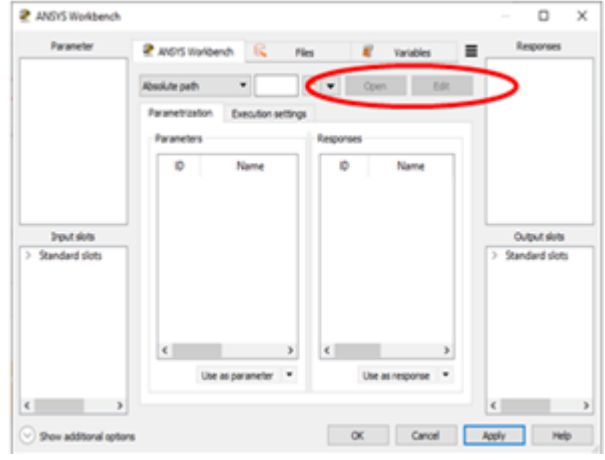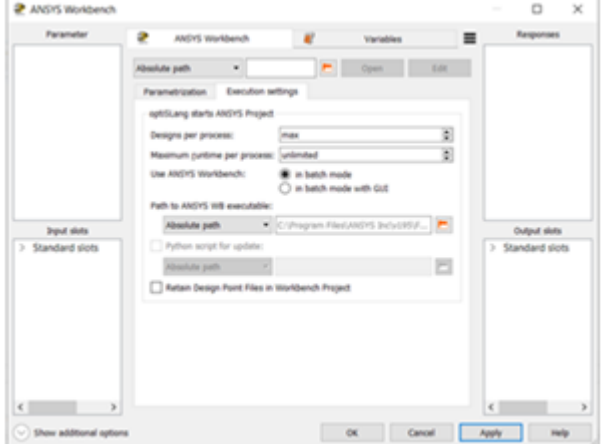
Function	SDK	ACT
optiSLang project settings	N/A	Displays the settings of the currently open project in optiSLang.
Network access	Defines the scope for external access to open projects. 	Starts and stops the Network Discovery service to publish the currently opened project in the local network.
Embed *.opd files in *.opf files	Option to archive or not archive the optiSLang project directory (*.opd) in the optiSLang project file (*.opf).	Option to archive or not archive the optiSLang project directory (*.opd) in the optiSLang project file (*.opf).

Function	SDK	ACT
Derived parameters	Option to preserve dependent parameters from parameter set as responses in optiSLang. 	Preserves dependent parameters from parameter set as responses by default.
Preserve and retain data	Options to preserve the design points and the generated design point data after an update.	Options to preserve the design points and the generated design point data after an update.

Function	SDK	ACT
Update with journal	Option to submit a customized Python script for updating the table of design points. 	Option to submit a customized Python script for updating the table of design points. Can only be set directly on the system properties.
Clear generated data	Option to delete all result data before the next update.	Deletes all result data before the next update by default.

Function	SDK	ACT
Set additional environment variables	You can set additional environment variables that are used in the next Workbench process call. 	Not required.
Workbench node in optiSLang	• Input and output files located on the **Ansys Workbench** tab • Cannot edit files • **Environment** tab available	• Input and output files located on the **Files tab** • New edit function • **Environment** tab will be added in a future version

Function	SDK	ACT
	  • **Designs per process** option located on the **Execution Settings** tab 	 • **Designs per process** option located in **Additional options** • **Use Ansys Workbench as local application** is no longer available • **Use working copy of WB project** is no longer supported

To migrate a Workbench project:

1. Load the SDK project with the appropriate Workbench version.

2. Right-click a system and select **Edit** from the context menu.

3. Export the system settings (not results).
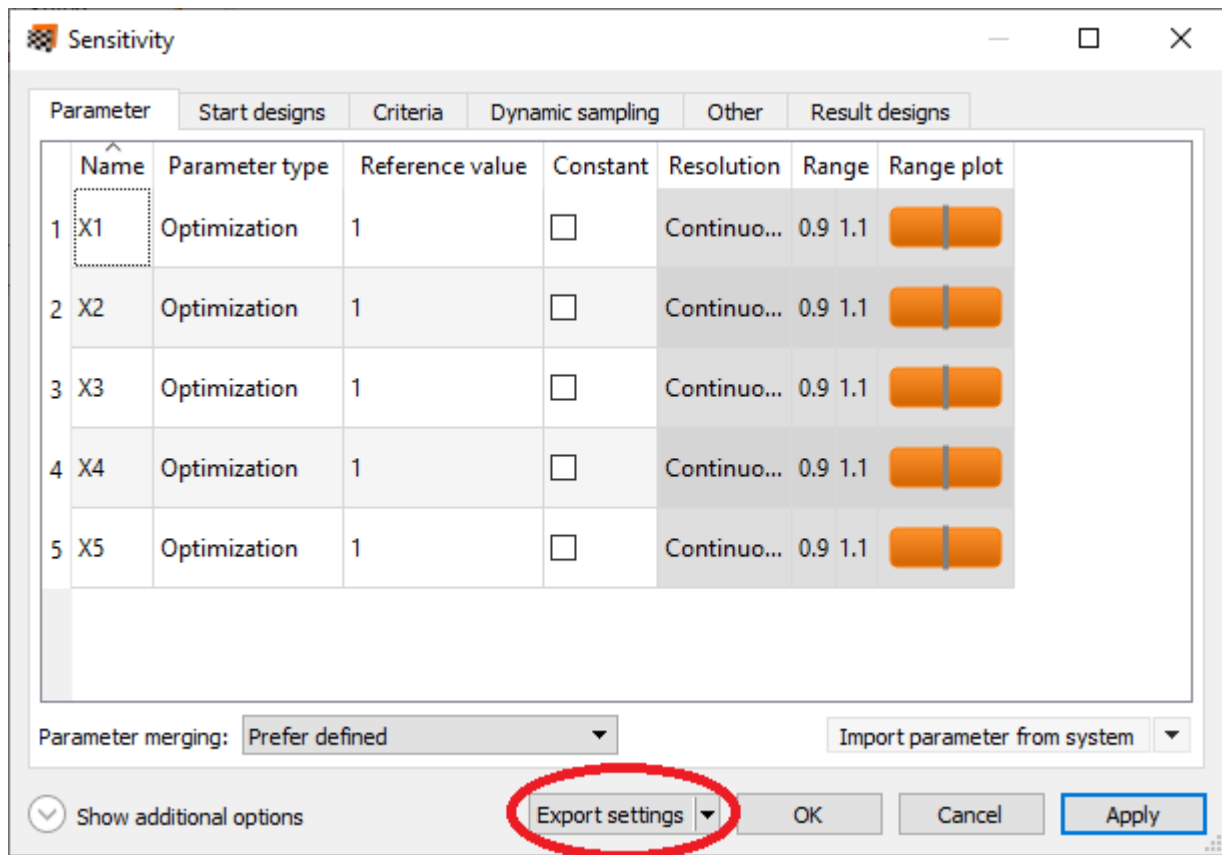
Additional steps are required for the following components:

Data Receive / Data Send

1. Open the **Files** tab.

2. Click the multi-action button to export the files.
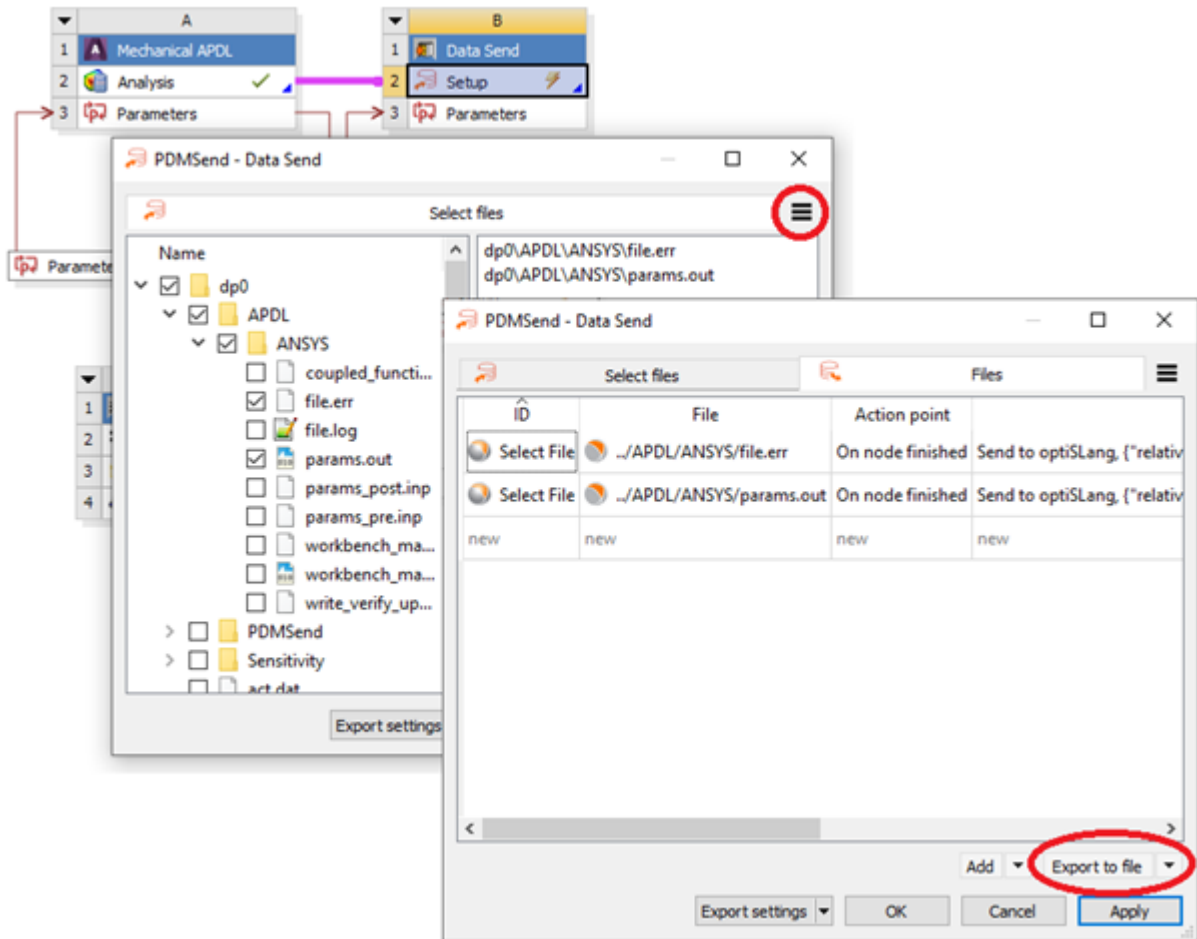
Signal Processing

1. Switch to the **Variables** tab.

2. Right-click the pane and select **Export variables** from the context menu.
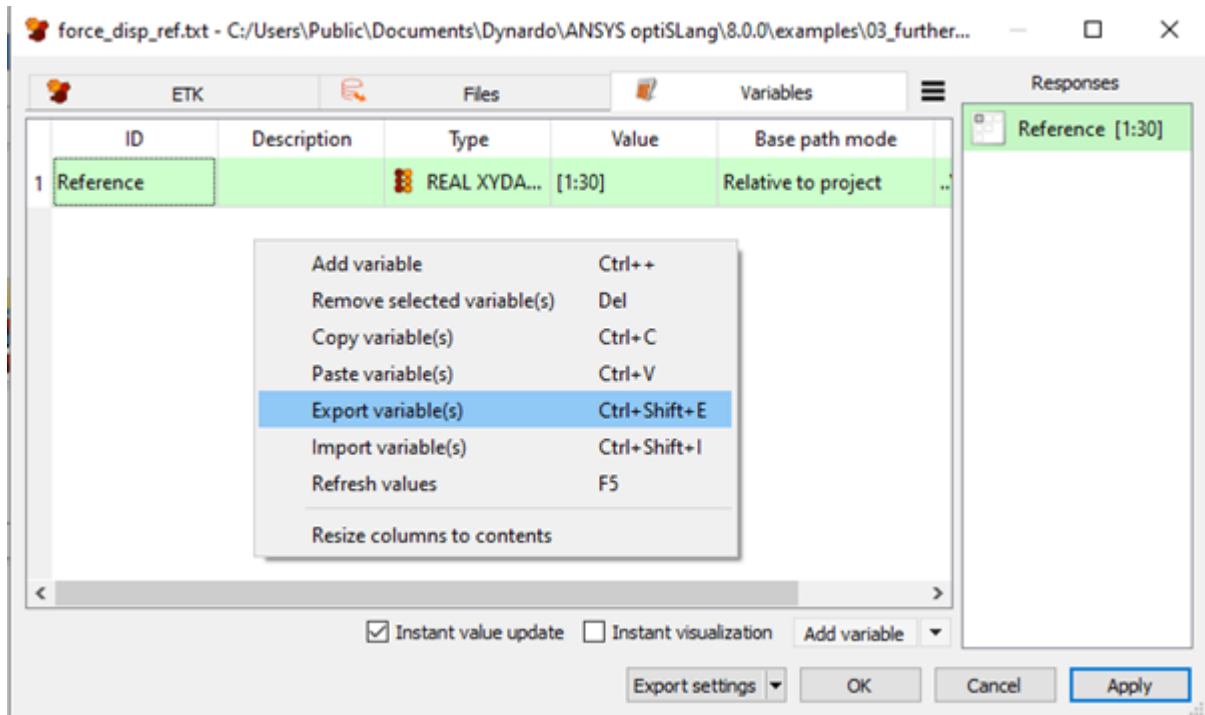
4. Repeat steps 2 and 3 for all other systems in the project.

5. Create a new project in the most recent version of Workbench.

6. Recreate the projects using the ACT extension components available in the Toolbox.

7. Import the corresponding settings exported in step 3.

Using the Ansys Workbench optiSLang Extension

The Ansys Workbench optiSLang Extension is an ACT extension that offers wizards for sensitivity studies, multidisciplinary optimization, robustness and reliability analysis as well as Robust Design Optimization supporting all kinds of Ansys design parameters.

The extension uses much of the same functionality as Workbench. To use the extension tools:

1. From the **optiSLang** and **optiSLang Integrations** sections of the **Toolbox**, add (p. 55) optiSLang systems and components to the **Project Schematic**.

   See System Connections (p. 387) for descriptions of optiSLang system connections.

2. Complete the optiSLang system wizards (p. 384), configure the optiSLang components (p. 384), and work through (p. 67) any Workbench systems created.

3. Save the Workbench project (p. 269).

4. Update systems or the project (p. 242).

Algorithm Systems

Each algorithm is represented by a system consisting of parameters, criteria, algorithm settings, start designs, nominal designs, and results. The first five components are combined into a wizard that is launched when you add the system to the **Project Schematic**. Results can be opened with a post-processor during or after calculations.

Optimization

The Optimization system prepares an optimization workflow using the parameter sets provided by Workbench systems.

For details on the Optimization wizard, see Optimization Wizard. For an example tutorial, see Optimization of a Damped Oscillator in Ansys Workbench in the *optiSLang Tutorials*.

Robustness

The Robustness system prepares a robustness or reliability analysis workflow using the parameter sets provided by Workbench systems.

For details on the Robustness wizard, see Robustness Wizard. For example tutorials, see Robustness Evaluation of a Damped Oscillator in Ansys Workbench and Reliability Analysis of a Damped Oscillator in Ansys Workbench in the *optiSLang Tutorials*.

Sensitivity

The Sensitivity system prepares a sensitivity study workflow using the parameter sets provided by Workbench systems.

For details on the Sensitivity wizard, see Sensitivity Wizard. For an example tutorial, see Sensitivity Analysis of a Coupled Function in Ansys Workbench in the *optiSLang Tutorials*.

Integration Components

You can add the following optiSLang integration components to the **Project Schematic**.

Data Receive

Manages the data exchange of external files. Registered files can be received from different external locations like file systems or versioning databases.

Each file entry is made available using a appropriate connection. Supported transfer types are:

- Geometry

- Engineering Data

- Mesh

- optiSLang

• Turbo Geometry

For more information, see Data Receive.

Data Send

Manages the data exchange of external files. Registered files can be sent to different external locations like file systems or versioning databases.

In the standard view, the files from the Workbench design point directory are offered hierarchically on the left, which can be transferred to the optiSLang design working directory. The selected files are listed with their relative path on the right. This structure is retained during the file transfer.
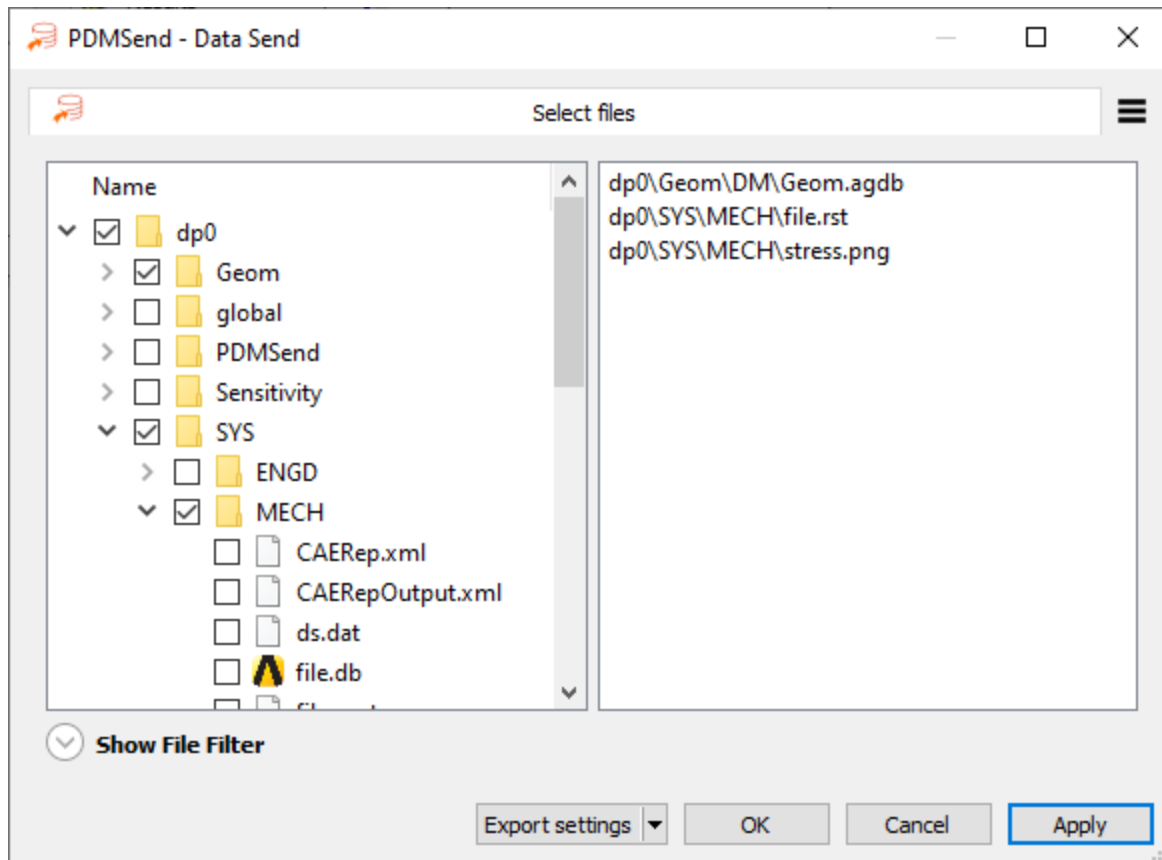
If the files to be transferred are not known, or not available at the time the workflow is set up, a file filter can be used. The definition of the filter is identical to that of the purge dialog.
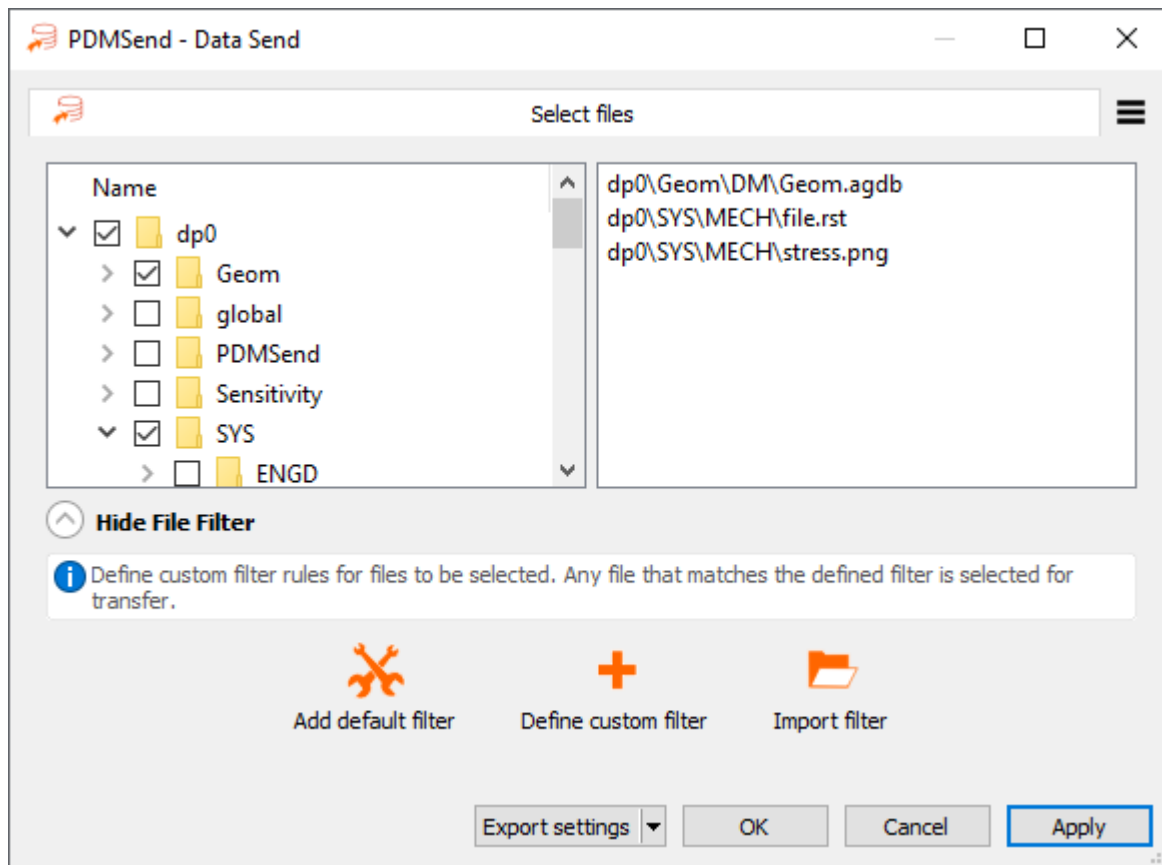
For more information, see Data Send.

MOP Solver

Uses Metamodel of Optimal Prognosis surrogate models for CAE calculations.

Starting from a reduced number of simulations, a surrogate model of the original physical problem can be used to perform various design configurations without computing any further analyses. These well-distributed results are used to create the metamodels. The metamodel with highest Coefficient of Prognosis (CoP) is taken as the optimal by optiSLang. The MOP results are stored in an optiSLang monitoring database `*.omdb` file or in an optiSLang `*.bin` file. The MOP Solver calls can be used to replace the original solver process.

For more information, see MOP Solver.

Python

Provides Python scripting for automating tasks. For more information, see Python.

Signal Processing

Extracts scalar and non-scalar response values from Ansys result files and text files and provides the non-scalar responses to optiSLang algorithm systems and the Ansys Workbench component in optiSLang if the programs are connected locally. Signal processing in the extension requires access to the file system so remote connections are not supported.

The extraction may reduce data, as only the extracted optiSLang raw value (`*.orv`) files are saved, not the result file. The raw value files are saved automatically to the respective optiSLang design directories and are used for providing responses within the optiSLang systems. If you are defining scalar responses, they are added to the Ansys parameter set.

For more information, see ETK.

System Connections

To create a specific workflow and to exchange data between different systems connections, add the optiSLang algorithmic systems and integration components to the **Project Schematic** using the instructions provided in Adding Systems to the Project Schematic (p. 55).

You can make connections between the following optiSLang systems and components.

MOP Solver to Optimization

Finds improved designs as a result of a metamodel-based sensitivity analysis. Data exchanged is the information about parameters. You can establish multiple connections between the two systems. This connection can be established for all optimization strategies except for ARSM which uses its own metamodeling approach internally.

Sensitivity to Sensitivity

Enhances an existing sampling by adding further samples. Data exchanged is the information about parameters, criteria, and the already sampled designs.

Sensitivity to Optimization

Finds improved designs as a result of a sensitivity analysis. If an objective function is defined in the Sensitivity system, the best design is exchanged as a start design for the optimization. Otherwise, a connection is not established.

Optimization to Optimization

Checks if any further improved designs can be found in the existing best designs. Data exchanged is the information about parameters, criteria, and the already sampled designs.

Optimization to Robustness

Analyzes the robustness of optimization results. The best design of the optimization is exchanged to set the nominal design for the robustness analysis. You establish connections from an Optimization system to multiple Robustness systems.

optiSLang Extension User Interface Reference
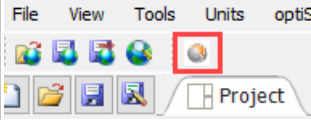
The following components are specific to the Ansys Workbench optiSLang Extension. For general Workbench interface details, see Workbench Interface Reference (p. 439).

Menu Bar

File Menu

Menu Item	Description
Load Archive From	Loads a project from an external data source.
Save Archive To	Saves the current project to an external data source.

optiSLang Menu

Menu Item	Description
optiSLang Settings	Opens the application settings dialog.
License Management	Opens the License Management dialog box and display an overview of all available and used licenses.
Project Settings	Opens the Project Settings dialog box.
optiSLang Help	Opens and displays the optiSLang help.
About optiSLang	Opens the About Ansys optiSLang dialog box containing various product details.
Export Designs	Exports the table of design points to a JSON file and starts the optiSLang postprocessing with it.
Export DX Data	Exports the parameters, designs and optimization criteria of a Design Explorer system to be selected in an optiSLang readable JSON file. This menu item is only displayed if there is a Design Explorer system in the **Project Schematic**.
Remote Control	Opens a dialog to register remote server connection data, allowing remote access to the project from the Ansys Workbench node in optiSLang. Enable or disable the connection by clicking **Start** or **Stop**.
Go To optiSLang	Opens the optiSLang application where you can continue to work on the project. Also available as a shortcut button.

Context Menu

Menu Item	Description
Edit	Opens a dialog to edit specific system or component settings.
Finalize	If an algorithm component was stopped, it can be finalized to continue the calculation of further systems.
Refresh Parameter	Reloads the parametric data from Workbench. If parameters are added to or removed from the parameter set in Workbench, select Refresh Parameter in all optiSLang systems before updating the project. All new parameters are added to the system without changing the user-defined properties of the existing ones.

Menu Item	Description
	Parameters removed in Workbench are also removed for the optiSLang system.
Results	Opens the postprocessing dialog for the system.
DoE Results	Opens the statistics postprocessing of the Design of Experiment or sampling component for this system.
Import DX Data	Opens a selection dialog for a DX system and transfers the parameters, designs (as start designs) and optimization criteria to the optiSLang system.
Export FMU	Exports the metamodel to Functional Mock-up Interface.

Properties Pane

Property	Description
Preserve Design Points After optiSLang Run	Preserves designs belonging to the currently updated system in the table of design points.
Retain Data for Each Preserved Design Point	Retains generated design point files during the update in the Workbench design point directory.
Use Simultaneous Execution Mode	Submits multiple design points in parallel to Remote Solve Manager (RSM). When selected, optiSLang submits the number of design points entered in *Number of Design Points sent to Parameter Set* at the same time. If this is not selected, the design points are submitted to RSM sequentially. For more details on submitting design points to RSM, see Updating Design Points in Remote Solve Manager (p. 352).
Number of Design Points sent to Parameter Set	Submits the listed number of design points to RSM, either in parallel if *Use Simultaneous Execution Mode* is enabled, or sequentially if it is not. The size is automatically adjusted if fewer designs need to be evaluated. Set this number based on the number of free clients in your RSM setup.
Run Journal for Update	Controls the project, system, or cell update using the selected Python script or Workbench journal.

Situational Dialogs

The following dialogs can appear, depending on the project state.

Dialog	Description
Interrupt/Abort (p. 245)	Allows you to interrupt the calculation of a project.
Recovering Projects (p. 277)	Allows you to continue a crashed session.
Reset	Defines which data can be discarded when resetting the project or component.
Remove Working Directory	Confirms the decision when selecting **Remove design directories before calculating**.
Confirm Modifications	Applies modifications to a component. This might cause a loss of calculated data. Different scenarios are offered for resetting the component.

Project File Structure

optiSLang projects in Workbench use the following directory structure and files.

`*.wbpj/*.wbpz`—Workbench project file or archive

`*_files`—Workbench working directory

- `dp0`—The current design point (reference design)

    - `ETK`—Signal Processing (p. 384) working directory

File/Folder	Description
`*.opd`	optiSLang working directory
`*.opf`	optiSLang project file
`*.orv`	optiSLang raw value file
Input file	Input file for signal processing
Output file	Output file for signal processing

- `user_files`—User or program provided files

    - optiSLang protocol file containing project session information (for example, calculations or errors)

- `*.opf`—optiSLang project file

- `*.opd`—optiSLang working directory

    - Algorithm systems

        → `*.omdb`—optiSLang monitoring database file

        → Design directories—Created for every run started by optiSLang. The numbers of these directories are equal to the numbers of the designs shown in the history plots. Stores and processes the modified input files and the result files.

File/Folder	Description
Input file	Input file for design
Output file	Output file for design
Solver Output	Solver output files for design
`*_files`	Workbench working directory

    - `*.omdb`—optiSLang monitoring database file

`*.optislang`—A special folder that optiSLang creates when the Workbench project information is read for the first time. It contains a JSON file with all relevant data, such as parameters, responses, re-

gistered files and a hash value. Reloading the project information is only necessary if the hash value of the project has changed.

System Environment

The Ansys Workbench optiSLang Extension uses the system environment and registry to find the compatible optiSLang application in the following order:

1. Environment variable OPTISLANG_HOME*XXXX*

   where *XXXX* is the Workbench version number, with leading zeros if applicable. For example OPTISLANG_HOME0810 or OPTISLANG_HOME2110.

2. OPTISLANG_HOME*XXXX* entry in

   - **Windows**: `%APPDATA%\Dynardo\optiSLangWBExtension.ini`

   - **Linux**: `%HOME%/.config/Dynardo/optiSLangWBExtension.ini`

3. Under AWP_ROOT of the currently running Workbench version.

4. Registry (Windows only)

   - HKEY_CURRENT_USER\SOFTWARE\Dynardo\ANSYS optiSLang entry of the Windows registry if optiSLang was installed for the **local user**

   - HKEY_LOCAL_MACHINE\SOFTWARE\Dynardo\ANSYS optiSLang entry of the Windows registry if optiSLang was installed for **all users**

Working with Workbench and Ansys Minerva

Ansys Minerva is a simulation process and data management (SPDM) software system that enables you to work with simulation data in a repository. Within Workbench, you can create a connection to an Ansys Minerva site so that you can transfer projects to and from Ansys Minerva repositories.

You can use Ansys Minerva to:

- Archive completed projects or store works-in-progress in a central repository

- Share your projects and collaborate on those of others

- Retrieve your own projects or those shared by other users

- Change the permissions of projects or place them under version control

- Search projects based on names, dates, simulation type, or other criteria

- Create reports

For more details, see the *Ansys Minerva User's Guide*.

Related topics:
Creating a Connection to an Ansys Minerva Site
Displaying the Ansys Minerva Toolbar
Launching Ansys Minerva with a Web Browser
Working with Workbench Projects Saved in Ansys Minerva

Creating a Connection to an Ansys Minerva Site

To be able to save a Workbench project to or retrieve a project from Ansys Minerva (directly from Workbench), you must create a connection between your local Workbench installation and an Ansys Minerva site.

A connection contains information about the server hosting Ansys Minerva, the database to use, and the authentication type.

To create a connection in Workbench:

1. From the menu bar, select **Tools** → **Options**.

2. From the list, select **Ansys Minerva**

3.   Specify a **Authentication Type** for the connection.

> **Note:**
>
> Select the method that matches what is configured for the Ansys Minerva server you are connecting to.

- **Basic**—Uses a username and password to authenticate using the internal mechanism within Ansys Minerva.

- **Browser**—Uses the browser to authenticate. If you are logged into Ansys Minerva in the browser, those credentials are used.

- **Windows**—Uses Active Directory to pass the credentials of the user currently logged into the Ansys Minerva server.

4.   Enter the **Server URL** address of the Ansys Minerva server.

5.   Enter the Ansys Minerva **Database** to access.

6.   Click **OK**.

Displaying the Ansys Minerva Toolbar

The Ansys Minerva functions available from the Workbench **File** menu are also available in the Ansys Minerva toolbar.

To display the toolbar, select **View** → **Toolbars** → **Ansys Minerva Toolbar**.

Launching Ansys Minerva with a Web Browser

To access Ansys Minerva directly, open the web client in your default browser by doing one of the following:

- Select **File** → **Ansys Minerva** → **Launch Minerva Web Client**.

- Click on the Ansys Minerva toolbar.

Working with Workbench Projects Saved in Ansys Minerva

When working with an Ansys Minerva repository, you must understand the relationship between your local working version of the project and the copy stored in the repository. Even if a project has been stored to or opened from a repository, Workbench always operates directly on the local working copy of the project.

Workbench is aware of the relationship between the local project and the repository copy, and helps you manage your project and maintain consistency. You can save the local copy of a project to the re-

pository at any point. If you open a local project, and the copy in the repository is more recent, you have the option of downloading and updating to the repository version.

When you save a Workbench project to an Ansys Minerva repository, Ansys Minerva always stores it as a virtual project archive (with the *.wbpz extension) so that you can act on a single object, rather than on a set of objects.

Related topics:

Saving the Current Workbench Project to an Ansys Minerva Repository

Saving the Current Workbench Project to a New Location in the Repository

Opening a Project from an Ansys Minerva Repository

Claiming a Project

Unclaiming a Project

Saving the Current Workbench Project to an Ansys Minerva Repository

You can save your current Workbench project to an Ansys Minerva repository for archive storage, or to share with other users.

> **Note:**
>
> The first time the project is saved, all of the files are sent to Ansys Minerva. In subsequent saves, only the items in the project that have changed are sent.

To save the current project to an Ansys Minerva repository:

1. Save any project changes locally.

2. From the menu bar, select **File** → **Ansys Minerva** → **Claim** or on the Ansys Minerva toolbar, click
   .

3. From the menu bar, select **File** → **Ansys Minerva** → **Save** or on the Ansys Minerva toolbar, click
   .

4. If you have not authenticated against an Ansys Minerva site, the **Connect** dialog box appears.
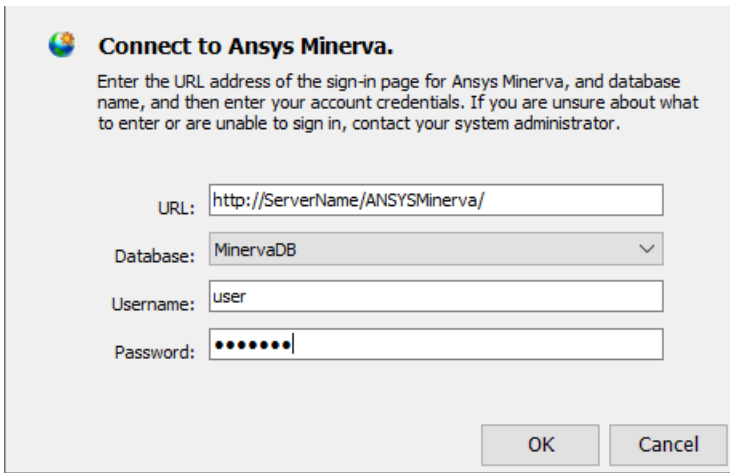
a. Enter your **Username** and **Password**.

b. Click **OK**.

5. If this is the first time you have saved this project to Ansys Minerva:

a. Browse to the location in the repository to save the files.
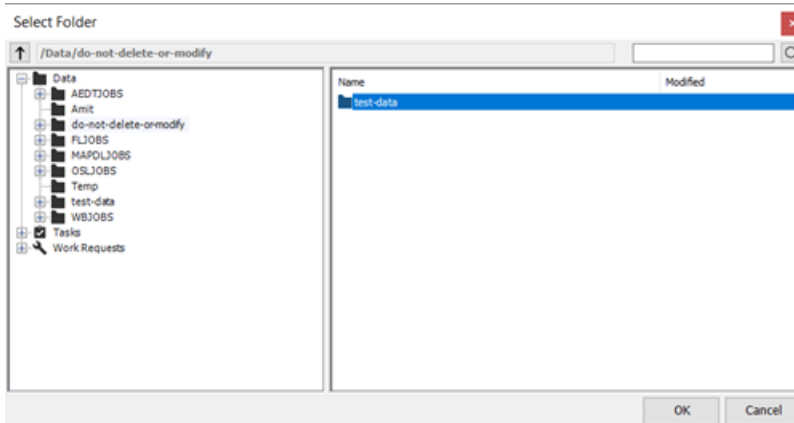
- To search for the location, enter the text in the search field. You can include wildcards in the search terms.
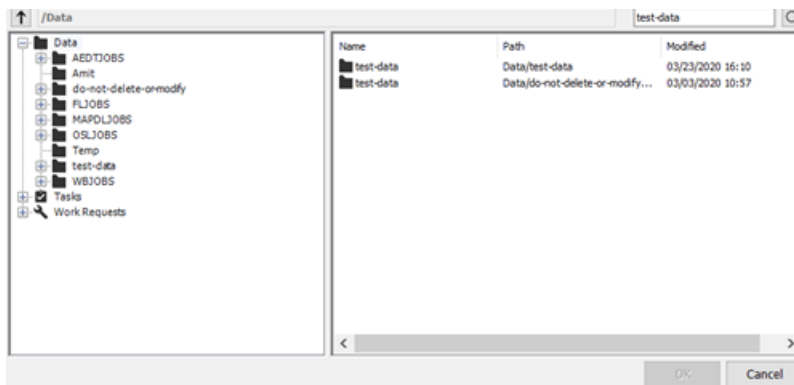
· To clear the search, right-click the right pane and select **Clear Search** from the context menu.
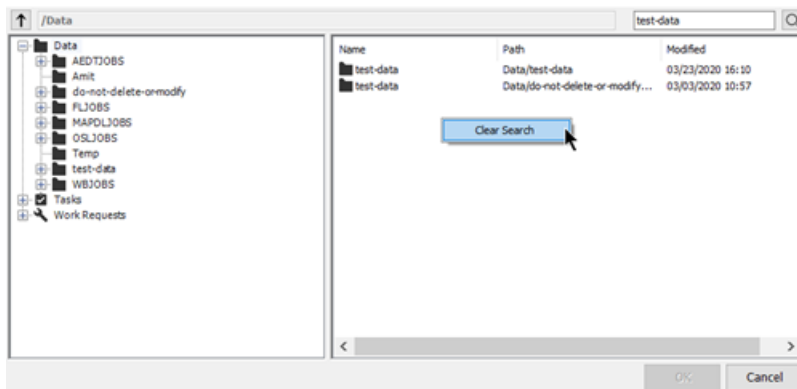
You can also clear the search by clicking on a different location in the tree in the left pane.

· To add a new folder, right-click the right pane and select **New Folder** from the context menu.
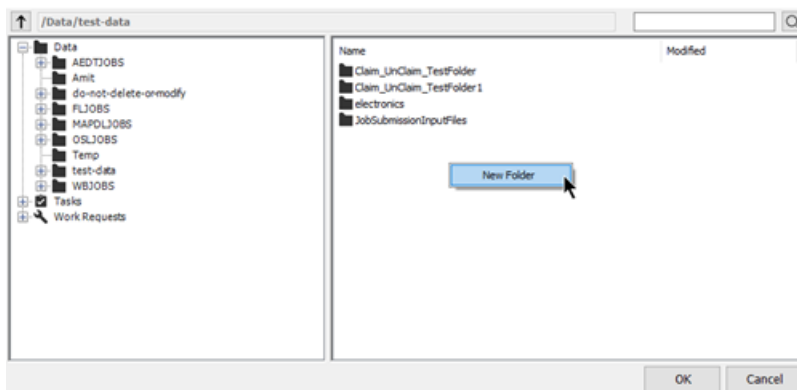
Note:

This location will be automatically selected in subsequent saves. To save a copy of the project to a new location, complete a save to operation (p. 397).

---

b.   Click **OK**.

Saving the Current Workbench Project to a New Location in the Repository

Once you have saved a project to Ansys Minerva, Workbench stores the path to the repository location and subsequent saves are made to the same location. To switch to a new storage area, you must complete a save to operation.

To save the project to a new location:

1.   Save any project changes locally.

2. From the menu bar, select **File** → **Ansys Minerva** → **Claim** or on the Ansys Minerva toolbar, click

    .

3. From the menu bar, select **File** → **Ansys Minerva** → **Save To** or on the Ansys Minerva toolbar, click .

4. If you have not authenticated against an Ansys Minerva site, the **Connect** dialog box appears.
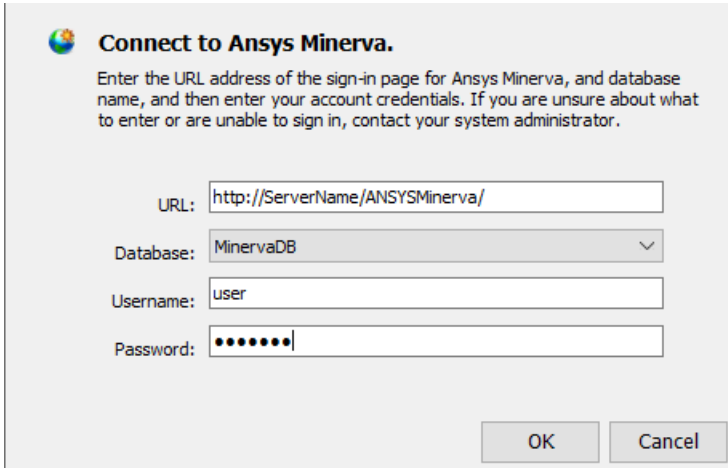
    a. Enter your **Username** and **Password**.

    b. Click **OK**.

5. Browse to the location in the repository to save the files.
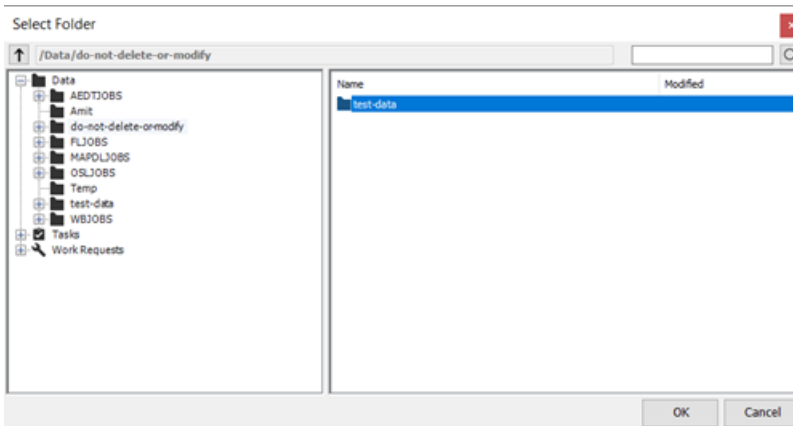
    • To search for the location, enter the text in the search field. You can include wildcards in the search terms.
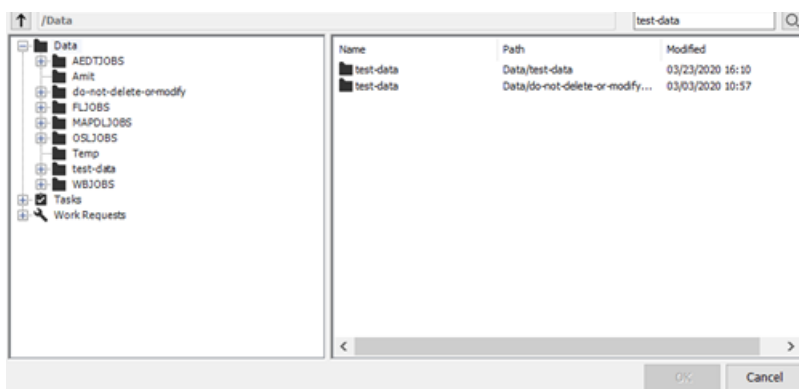
- To clear the search, right-click the right pane and select **Clear Search** from the context menu.
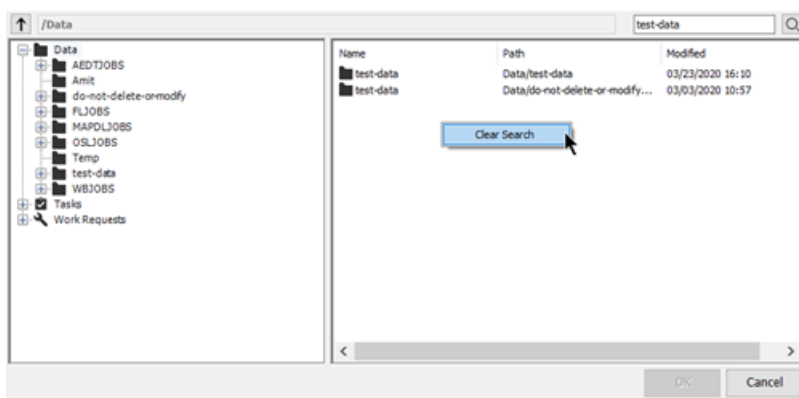
You can also clear the search by clicking on a different location in the tree in the left pane.

- To add a new folder, right-click the right pane and select **New Folder** from the context menu.
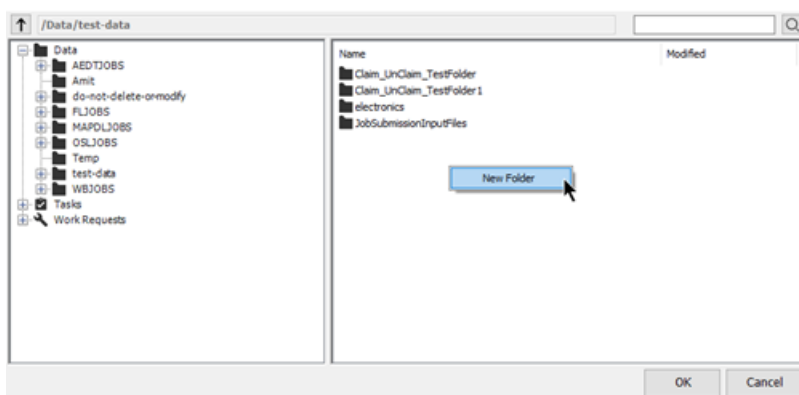
6. Click **OK**.
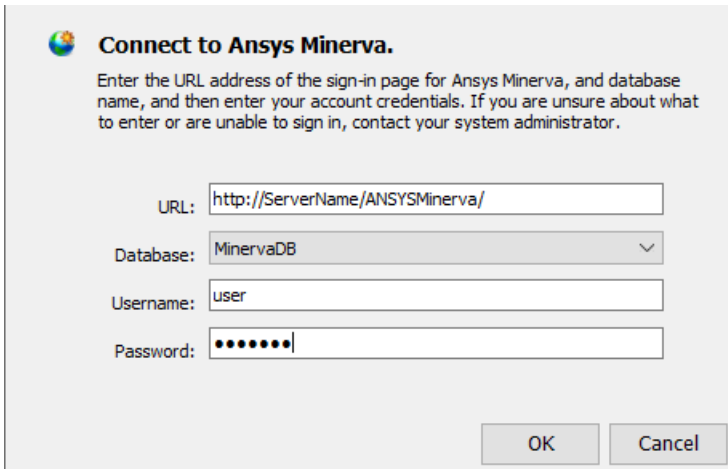
Opening a Project from an Ansys Minerva Repository

When you open a project from an Ansys Minerva repository, the archive file is downloaded, extracted, and opened in Workbench.

The files are saved automatically to the location you specify in your Workbench user preferences, on the Project Management page. See *Project Management* in the *Workbench User's Guide* for more information.

To open a project from an Ansys Minerva repository:

1. From the menu bar, select **File** → **Ansys Minerva** → **Open** or from the Ansys Minerva toolbar, click .

2. If you have not authenticated against an Ansys Minerva site, the **Connect** dialog box appears.

   a. Enter your **Username** and **Password**.

   b. Click **OK**.

3. Browse to the project file location in the repository.

   • To search for the location, enter the text in the search field. You can include wildcards in the search terms.
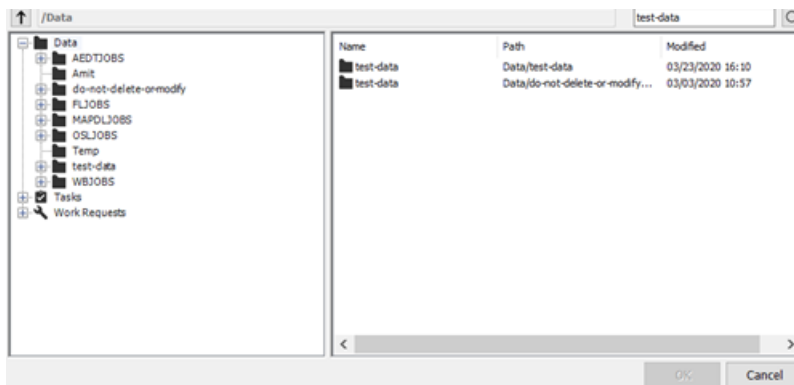
   • To clear the search, right-click the right pane and select **Clear Search** from the context menu.
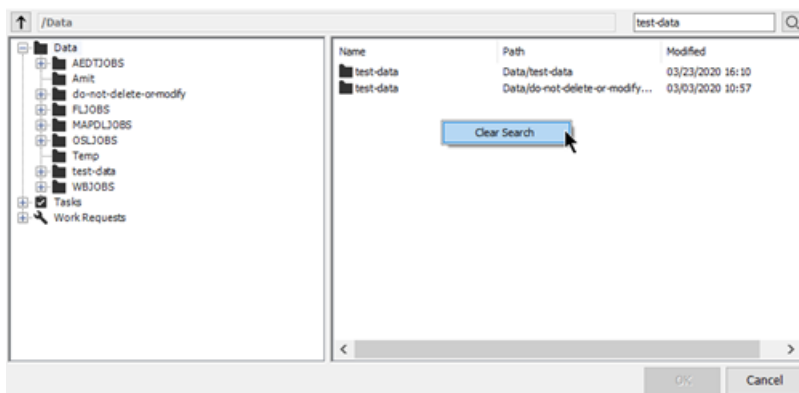
You can also clear the search by clicking on a different location in the tree in the left pane.

4.   Select the project file and click **OK**.
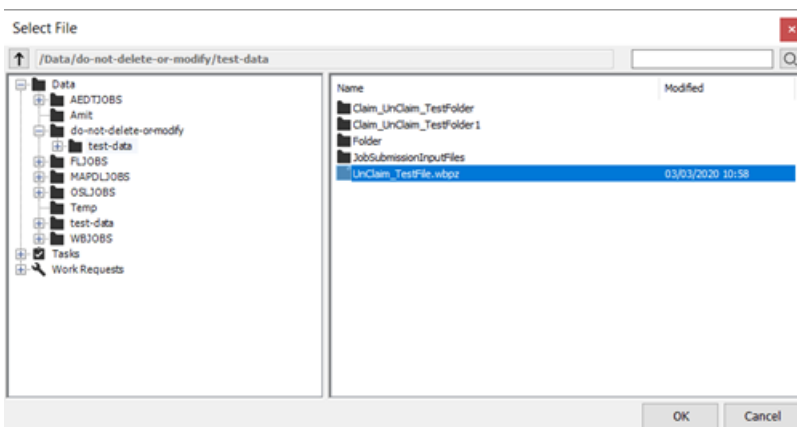
The archive file is downloaded and opened in Workbench.

Claiming a Project

When you have a project opened from the repository that is under version control, you can claim it. This gives you exclusive control over the project and prevents other users from making any modifications to it while you are working with it.

To claim a project:

1.   Open the project from the repository (p. 399).

2.   From the menu bar, select **File** → **Ansys Minerva** → **Claim** or on the Ansys Minerva toolbar, click

      .

The project is claimed. A flag appears beside the project name in Ansys Minerva.
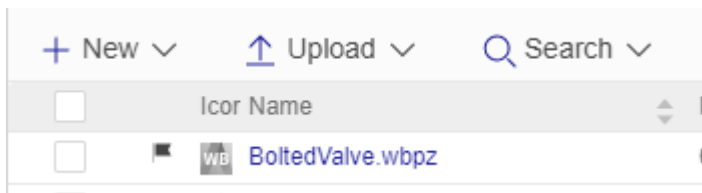
Unclaiming a Project

When you are finished editing a project, you release it back to Ansys Minerva by unclaiming it.

To unclaim a project:

1. Open the project from the repository (p. 399).

2. From the menu bar, select **File** → **Ansys Minerva** → **Unclaim** or from the Ansys Minerva toolbar, click .

    The project is released. In Ansys Minerva, the claimed flag is removed from the project name.

Troubleshooting Workbench

The following topics list issues and error messages that you may encounter while running Workbench.

These topics do not include troubleshooting for data-integrated applications. See the help for the specific application for troubleshooting information.

For additional troubleshooting information on native applications, see the following:

- Troubleshooting in the *DesignXplorer User's Guide*

- DesignModeler Frequently Asked Questions in the *Ansys DesignModeler User's Guide*

- Troubleshooting in the *optiSLang User's Guide*

Related topics:

General Workbench Issues

Product-Related Issues

Startup or Graphics Issues

Error Messages

Known Issues and Limitations

General Workbench Issues

This topic lists issues that you may encounter in various areas of Workbench.

Error Messages Appear During Workbench Setup

During setup, if you encounter any errors containing `0x8000FFFF`, you must install the required installation prerequisites.

1. To open the Product & CAD Configuration Manager, select **Start > Ansys 2022 R2 > Product & CAD Configuration 2022 R2**.

2. Click **Install Required Prerequisites**.

CAD System Plug-In Menus Do Not Appear for NX or Creo Parametric

On Windows platforms, Workbench appends its information to an existing customization file. If a customization file does not exist, a new one is created. For NX, Workbench looks for the `custom_dirs.dat` file in the directory specified in the **UGII_CUSTOM_DIRECTORY_FILE** environment variable. For Creo Parametric, Workbench looks for the `config.pro` file in the `%HOMEDRIVE%%HOMEPATH%` directory.

When the Creo Parametric Geometry Interface is set up, Workbench also appends its information to the `config.pro` file located in the Creo Parametric installation path, under the `\text` directory (for example, `Proewildfire2\text\config.pro`).

If Workbench encounters a read-only file, it cannot write the necessary information to the file. To display the Ansys menu in NX or Creo Parametric, you must change the permissions on the file and manually add the appropriate Workbench-specific information.

Script Errors When Running Workbench

If you encounter script errors such as `Error: Unable to create object microsoft.XMLDOM`, you may need to install the latest version of Microsoft Core XML Service (MSXML).

See the MSXML Download Page for more information on downloading and installing MSXML.

Charts Do Not Appear

If you do not see the appropriate charts created after a design point run or other updates (such as response surface updates), from the menu bar select **View → Reset Workspace**.

Applications Do Not Start After Crash

On Linux machines, if Workbench applications do not launch after an abnormal exit such as a crash, run the following utility:

```
$INSTALL/v222/aisol/wbcleanup
```

Wobbly Windows Desktop Effect Causes Crashes

The wobbly windows desktop effect on Linux machines may cause Workbench or its applications to crash. This effect is on by default on some Linux platforms. Make sure the effect is turned off.

Linux Threading Limit

By default, many Linux systems limit the total number of threads each user can run at once, to avoid multiple processes overloading the system. If you are running many Ansys jobs concurrently as the same user (for example, submitting them through Remote Solve Manager), you may hit this limit, producing errors such as `Thread creation failed`.

To view the current thread limit, run the following command:

- `ulimit -u` (bash)

- `limit maxproc` (csh)

To increase the thread limit, run the following command:

- `ulimit -u [limit]` (bash)

- `limit maxproc [limit]` (csh)

Where *[limit]* is a numerical value (for example, 4096).

To view the administrator-imposed hard limit, run the following command:

- `ulimit -H -u` (bash)

- `limit -H maxproc` (csh)

You cannot increase the limit amount beyond this hard limit value. Contact your system administrator to increase the hard limit.

Communication Errors with Hyper-V Virtual Ethernet Adapter

Running Workbench projects containing System Coupling components through a virtual machine or VPN connection can cause various errors, for example:

```
Updated failed for the Solution component in System Coupling. The coupling service failed to start.
```

This is caused by a Hyper-V virtual ethernet adapter preventing the Message Passing Interface (MPI) (a method for managing multiple processes) from working correctly.

To solve this issue:

1. Open Control Panel.

2. Click **Network and Sharing Center**.

3. Click **Change adapter settings**.

4. Find the adapter that has *Hyper-V Virtual Ethernet Adapter* in the description.

    vEthernet (Default Switch)
    Enabled
    Hyper-V Virtual Ethernet Adapter
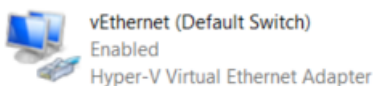
5. Right-click the adapter and select **Disable** from the context menu.

6. Repeat step 5 for all other Hyper-V adapters.

Design Point Update Subtask Error When Using UGE Scheduler

When submitting design point updates to Remote Solve Manager, the design points are allocated to one or more tasks and these tasks are allocated to one or more RSM jobs. This method of job submission can create an error similar to the following when using an UGE scheduler queue:

```
executing task of job 3119 failed: execution daemon on host
"testhost.domain.com" didn't accept task
```

To solve this issue, open the `sge_pe` file on the cluster and change the `daemon_forks_slaves` parameter to **FALSE**.

Using 4K Displays with Workbench

Currently, Workbench does not support 4K displays. You may see one or more of the following issues if you use Workbench on a 4K display:

- Initially launching Workbench displays a tiny window and text is cut off.

- Titles are cut off.

- You must maximize the Workbench window twice before it is set to full screen.

- Various parts of the user interface are scaled incorrectly, including tab titles, drop-down menu text, toolbars, and the ACT start page.

To improve these issues, complete the following procedure:

1. On the Windows toolbar, select **Start > Ansys 2022 R2**

2. Right-click the **Workbench 2022 R2** icon and select **More** → **Open file location** from the context menu.

3. In the **Ansys 2022 R2** folder, right-click the **Workbench 2022 R2** icon and select **Properties** from the context menu.

4. Switch to the **Compatibility** tab.

5. Under **Settings**, click **Change high DPI settings**.

6. Select the **Override high DPI scaling behavior. Scaling performed by:** check box.

7. Select **System** from the drop-down menu below the check box.
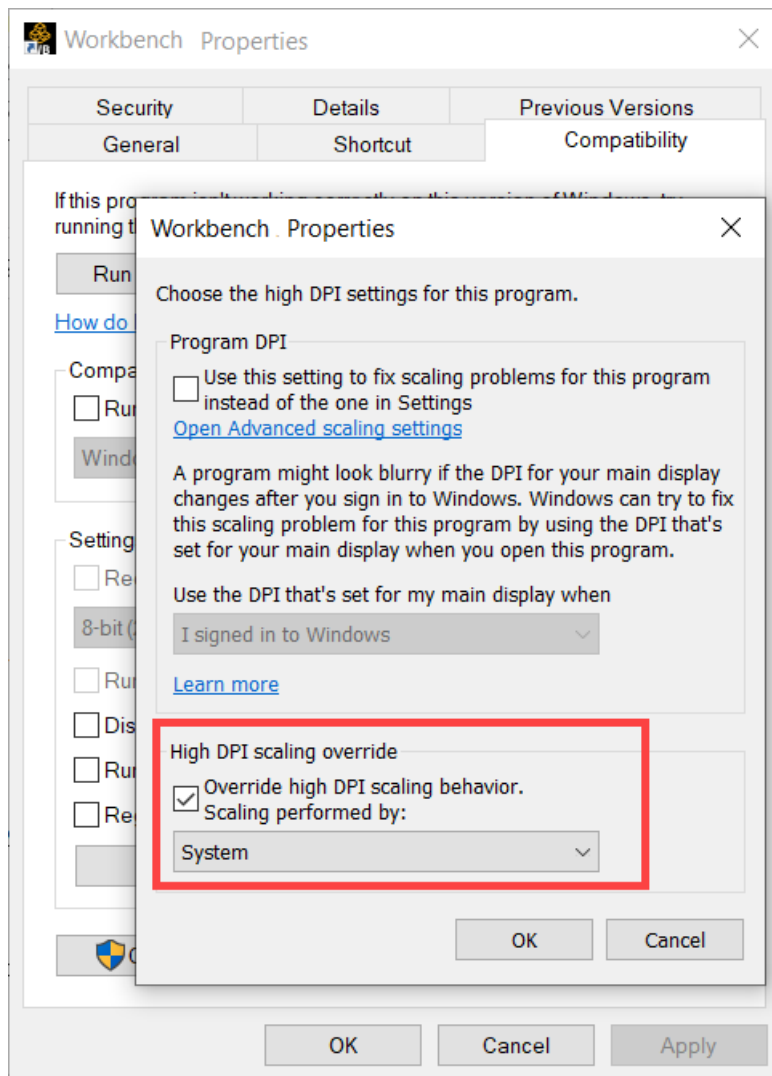
8. To save the change and close the dialog box, click **OK**.

System Coupling Crashes Workbench in Remote Session

If you are loading the System Coupling component system user interface during a remote Workbench session and Workbench crashes, restart the machine.

HDF5 File Locking

The HDF5 version used in 2022 R2 will result in failure when the underlying file system does not support file locking, or where locks have been disabled. To disable all file locking operations, create an environment variable named `HDF5_USE_FILE_LOCKING` and set the value to the five-character string **FALSE**.

Product-Related Issues

This topic lists issues that may occur in products that integrate with Workbench.

Installing the Multilingual User Interface Pack of Microsoft Office for your Language

If you run the 2007 or 2010 English version of Excel and the locale for the current user is configured for a language other than English, Excel tries to locate the language pack for the configured language. If the language pack is not found, then the error is reported and the automation of Excel, as used by the Excel as a calculator feature, cannot be performed. To solve this issue, you must install the Multilingual User Interface Pack of Microsoft Office for your language. As an alternative, you can also configure the locale for the current user to English.

This bug is documented by Microsoft. For more information, see http://support.microsoft.com/kb/320369.

Design point Failure when Updating Project with Multiple Excel Systems

If you have Office Excel 365, Office Excel 2013, or Office Excel 2016 installed on your machine and your Workbench project contains multiple Excel systems and the Excel process window is hidden, updating the project may cause design points to fail and the following exception message to display:

```
We found a problem with this formula. We couldn't find a range reference
or a defined name.
```

Workaround: Before running any update operations, open the Excel process window by right-clicking and selecting **Open File in Excel**.

This issue is documented by Microsoft. For more information, see https://support.microsoft.com/en-us/kb/3083825.

Workbench and optiSLang Crash Dialogs

When you start a Workbench project that contains optiSLang components, you may see some of the following dialogs depending on the situation.

Workbench Crashed and optiSLang was Saving the Project

Workbench encountered a problem and crashed but optiSLang was able to save the `*.opf` project file. In this case, you can recover the project. To see additional information before recovering, click **Show Details**. All files that are modified and will be deleted are displayed. We recommend manually saving a copy of the project files before recovering the project.

To recover the project, click **Yes** in the dialog.
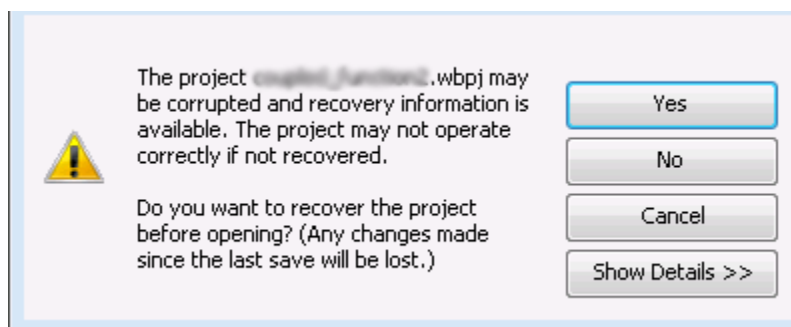
Workbench Crashed and optiSLang Crashed without Saving the Project

Both Workbench and optiSLang crashed and the project could not be saved. To continue with the project, return it to its last saved state. To see additional information before restoring the saved state, click **Show Details**. All files that are modified and will be deleted are displayed. We recommend manually saving a copy of the project files before restoring the saved state.

1. To return the project to its last saved state, click **Yes** in the dialog.

2. Select which project save version to use.

> **Note:**
>
> We strongly recommend selecting the automatic save point.
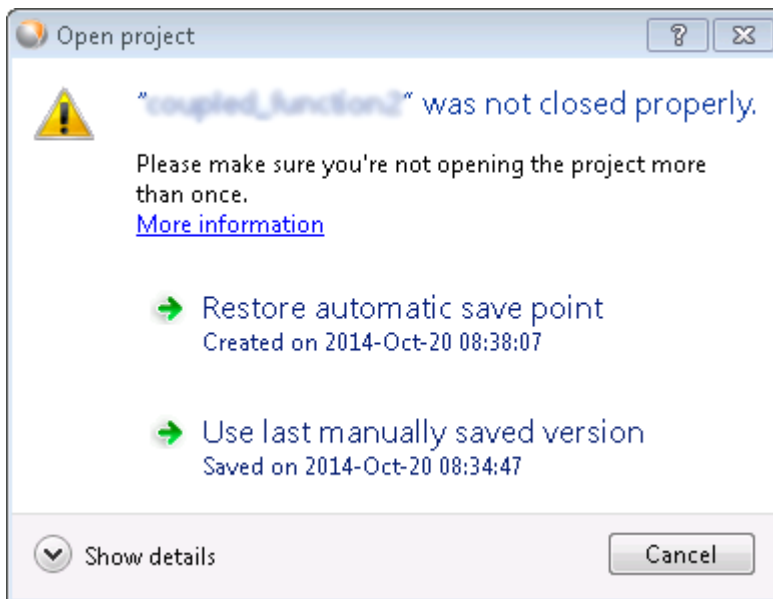
optiSLang Crashed

optiSLang encountered a problem and crashed, and the `optiSLangAddin.exe` file exited. You can now restart the Workbench extension and continue the project.

1. To restart the extension, click **Yes** in the dialog.
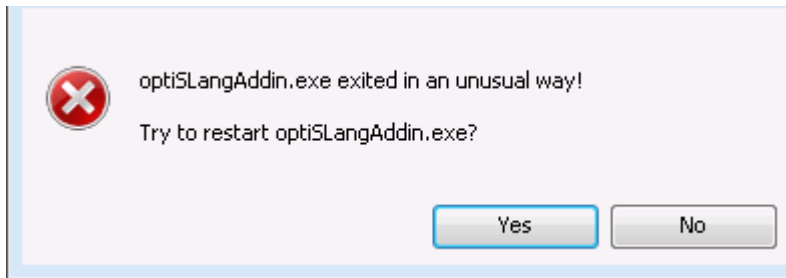
2.   Select which project save version to use.

> **Note:**
>
> We strongly recommend selecting the automatic save point.
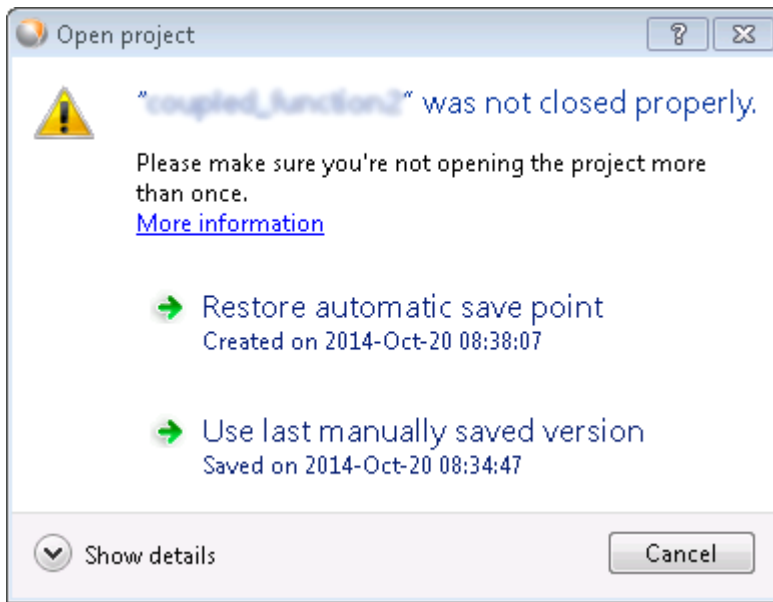
Project Files have been Relocated

Files referenced by the project have been moved and cannot be found. You can automatically relocate these files by selecting the option in the dialog.
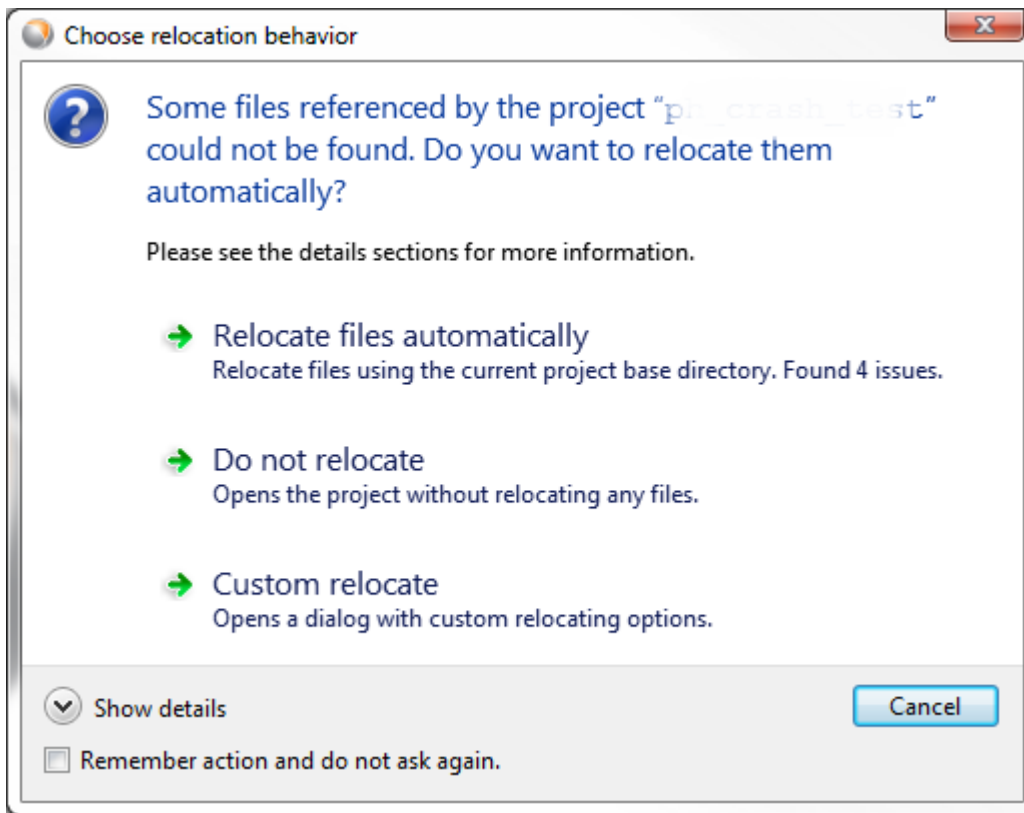
Failed to Start optiSLang Error when using optiSLang Extension in Workbench

When using the optiSLang extension inside Workbench, the following error can occur:

```
'NoneType' object has no attribute 'password' & Exception: Failed to
start optiSLang, no executable
```

To fix this issue, you must edit the `optiSLangWBExtension.ini` file. You can do this by either using the Workbench user interface or editing the file directly in a text editor.

Workbench User Interface

1. Set the value of the `DYNARDO_VERBOSE_LEVEL` environment variable to **1**.

2. Start Workbench.

3. If the optiSLang extension is not loaded, activate it (p. 371).

4. From the menu bar, select **optiSLang** → **Extension Settings**.
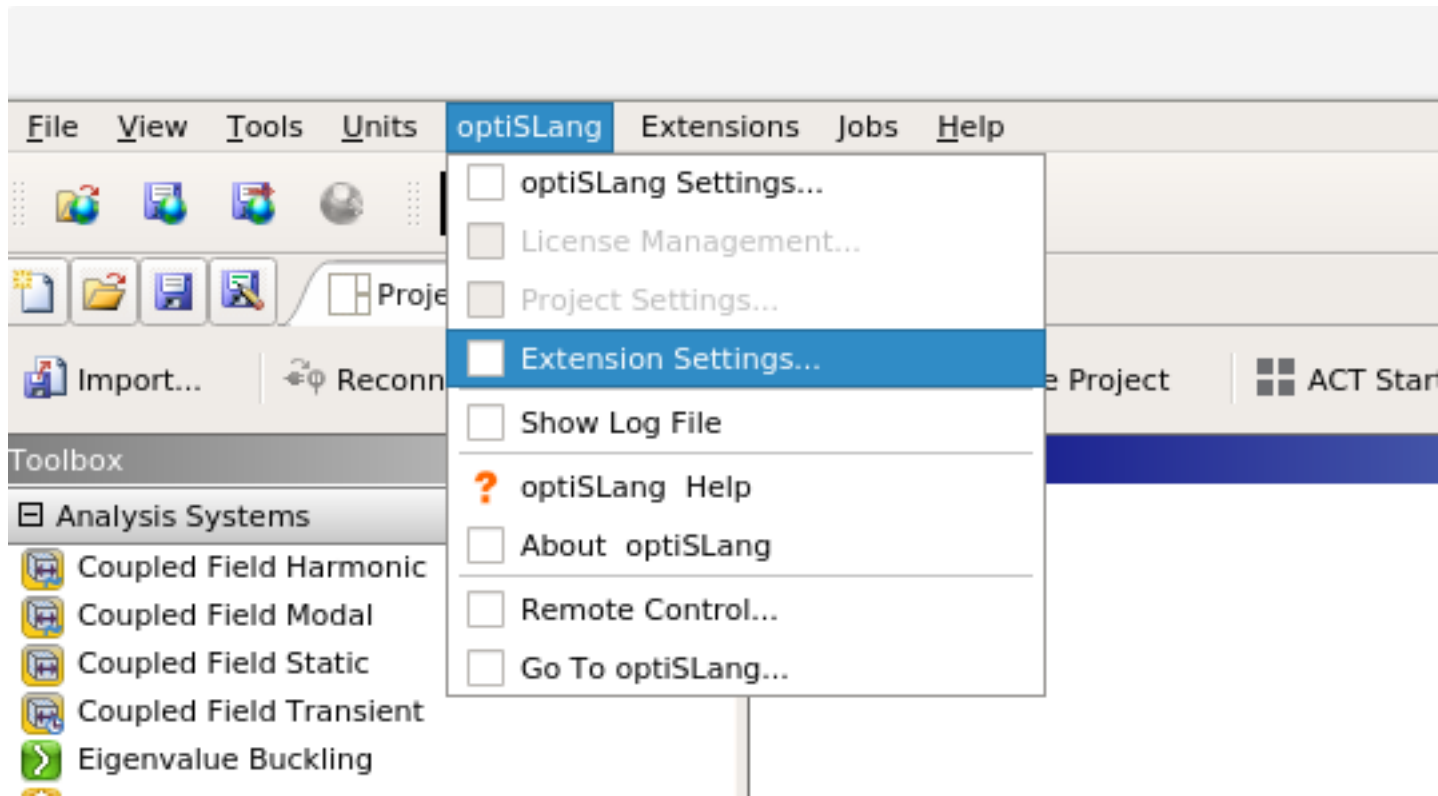
The `optiSLangWBExtension.ini` file opens in an editor.

5. In the file, add one or more paths to optiSLang installations.

The structure of each entry is `OPTISLANG_HOMEXXXX=/path/to/optislang/` where XXXX contains the installation path version number. For example, `OPTISLANG_HOME2210` points to the installation path of optiSLang 2022 R1.

Open in Text Editor

1. Navigate to the location of the `optiSLangWBExtension.ini` file.

    -   Windows: `%APPDATA%\Dynardo`

    -   Linux: `$HOME/.config/Dynardo`

2. Open the file in a text editor.

3. In the file, add one or more paths to optiSLang installations.

The structure of each entry is `OPTISLANG_HOMEXXXX=/path/to/optislang/` where `XXXX` contains the installation path version number. For example, `OPTISLANG_HOME2210` points to the installation path of optiSLang 2022 R1.

Error When Loading Files in TurboGrid Opened Through Workbench

After opening TurboGrid using a TurboGrid component system in Workbench, loading an Init file by selecting **File** → **Load TurboGrid Init File**, and saving the project, the following error message is displayed:

```
Unable to load DLL 'cclapilt.dll': The specified module could not be
found. (Exception from HRESULT: 0x8007007E)
```

Workaround: Install MS VC++ Redistributable for Visual Studio 2012 Update 4.

Startup or Graphics Issues

This topic lists issues that you may encounter with starting Workbench or Workbench graphics.

To minimize graphics problems, always verify that you are running the latest graphics drivers provided by your computer's hardware manufacturer. If you are not, you may see the following message:

```
***An error occurred while setting up the graphics window. Please ensure that you have the latest drivers
from your graphics card manufacturer. If the error persists, you may need to decrease the graphics
acceleration. For more information, please see the Troubleshooting section in the Ansys Workbench help.
```

Linux

If you are running Workbench on Linux and experience problems at startup or with the user interface or graphics displaying correctly, and you are running in accelerated graphics mode, you may need to relaunch Workbench using the `-oglmesa` flag to activate software rendering:

```
runwb2 -oglmesa
```

If Workbench detects that graphics problems are causing crashes, it automatically switches to software rendering. Workbench also uses software rending mode by default when running on a remote display, or on a local display if the hardware does not appear to be accelerated.

To revert to accelerated graphics mode, launch Workbench using the -oglhw flag:

```
runwb2 -oglhw
```

If you are running under Exceed3D, try the following settings if you are having graphics problems:

- Turn off the graphics (hardware) acceleration option in Exceed3D options.

- If graphics acceleration is on, turn on the GLX 1.3 option.

Any version of Exceed that does not have the GLX 1.3 option is unlikely to function correctly with graphics acceleration.

Windows

If you experience graphics issues on Windows systems, you can find the details of your graphics card and the driver that is currently installed by selecting **Start** → **Run** and typing the following into the **Open** field:

```
dxdiag
```

Select the **Display** tab and review your graphics card information. You can then contact the vendor or visit the vendor's website for details of the latest graphics drivers available for your specific graphics card.

You may also have to adjust the hardware acceleration. To adjust the hardware acceleration, Open Control Panel and select **Display** → **Settings** → **Advanced** → **Troubleshoot** or your operating system's equivalent. The hardware acceleration slider should then be visible. You can also select **Display** → **Settings** → **Troubleshoot** and use the troubleshooting tool to guide you to the Hardware Acceleration panel. Try turning hardware acceleration off by dragging the slider to **None**. Try the software again; if the graphics problems are resolved, then gradually increase Hardware Acceleration as far as you can before the software fails again.

Error Messages

This topic lists error messages that you may encounter in Workbench.

Unable to connect to Solver Manager

Another application might be using the Solver Manager port (10002 by default). Try changing the port number by editing the `Ansys.SolverManager.exe.config` file located in the installation directory at `\AISOL\Bin\<platform>`.

If you are having difficulty launching other applications/editors, it is also possible that the Windows hosts file has been corrupted. Make sure that localhost is specified in the Windows `<os drive>:\Windows\system32\drivers\etc\hosts` file.

Warning at File: myxml, line 1, col 40, Encoding (utf-16, from XMLDecl or manually set) contradicts the auto-sensed encoding, ignoring it

The message is sometimes displayed in the Remote Solve Manager (RSM) log; you can ignore it.

System.IO.IOException: Too many open files

Many Linux distributions set a default maximum for the number of files a process can have open and this can be exceeded when solving some cases. You must increase the maximum number of open files allowed by editing `limits.conf`, which on most Linux distributions is found in `/etc/security/limits.conf`.

For example, adding the lines

```
# ANSYS file handle limits modification
*       soft    nofile  2048
*       hard    nofile  2048
```

to `limits.conf` sets the maximum number of open files to 2048 (the default on most Linux distributions is 1024). Increase the maximum number of open files allowed by powers of 2, trying 2048, 4096, 8192.

Further information can be found in the manual page for `limits.conf`, which can be accessed using the command **man limits.conf** on any Linux machine with the **man** command installed.

You can check the current limit by using the command **ulimit -a** in a bash shell session or **limit** in a tcsh shell session. In the output, the maximum number of open files is displayed under **open files** or **descriptors**, respectively.

Known Issues and Limitations

The following are known issues at the time of release, as well as system and software limitations:

• KDE is not supported on SuSE Linux Enterprise Server & Desktop 12.

• When a job is submitted that requires the MAPDL solver, and no MAPDL license is available, the job may appear to be in the 'starting' state in the job monitor for up to 20 minutes before the issue is reported. Additionally, if you check your system processes (with **ps** on Linux or **Task Manager** on Windows), you may see some solver processes still running after 20 minutes. These may run for up to 80 minutes, but are small and do not consume significant resources.

• When running or updating a Workbench project on a Linux machine that generates many files, such as a large number of Design Points running concurrently, the solver can fail with the following exception:

  ```
  System.IO.IOException: Too many open files
  ```

  To prevent this failure, increase the maximum number of files able to be opened at one time by running the ulimit command `ulimit -n [limit]` where *limit* is either a numerical value (for example, 1000000) or, "unlimited".

• When running Workbench on a Linux machine, it crashes when you select **Extensions → ACT Start Page** from the menu bar or click **ACT Start Page** on the **Project** tab toolbar. This crash is caused by the `libtotem-mozplugin.x86_64 version 2.28.6-4.el6` dll package, used for video playback in browsers. To avoid the crash, uninstall the package.

• The **Excel** feature in Workbench requires Microsoft Office Excel 2013 or 2016.

- The **Excel** feature is only available on Windows systems.

- In a project with retained design points, switching to a different up-to-date design point using the **Set as Current** menu option automatically updates the Excel system, which could be time-consuming if the spreadsheet contains macros or extensive calculations.

- Files written by Ansys products cannot be synchronized with Microsoft's OneDrive file hosting service.

- Workbench products do not support running a network user installation as local root administrator.

- On Linux, transferring DesignModeler geometry from the geometry cell to a downstream HFSS Design system fails if the upstream geometry is not open in the DesignModeler editor. In this case an `OnRefreshTransferData failed!` error message is displayed. Open the geometry in DesignModeler before transferring it so that the transfer succeeds.

- When Material Designer is linked to Engineering Data, modifying a material name in Material Designer does not change the corresponding material in Engineering Data but rather creates a new material with the new name. The prior material should be deleted from Engineering Data before updating downstream components.

- Parameterizing a non-integer, non-zero number in a Forte project when your system is set to a European number format (the decimal separator uses a comma) causes the parameter value to be displayed as zero in Workbench. Ensure your system is set to a number format that uses a period as the decimal separator.

- Negative dimension values can invert the direction vector of SpaceClaim operations that they are associated with; this change is applied to the current and subsequent design point updates. As a result, when a Workbench input parameter is used as a driving dimension for a SpaceClaim geometry, negative dimension values may result in incorrect or unexpected geometric changes.

- Workbench can stop working when updating multiple design points in one task through Remote Solve Manager installed on a Linux cluster. The following error message is displayed in the Remote Solve Manager Job Monitoring application:

  `ClusterJobs Command Exit Code: 134`

  `Assertion at class.c:2876, condition 'iid <= 65535' not met`

  To work around this issue, reduce the number of design points that are updated in each task.

- When using Workbench on Linux systems, you cannot use a Mechanical APDL system to launch MAPDL when the graphics mode is set to 3D. You must use the X11 graphics mode.

- Because Workbench and Ansys AEDT use two different unit systems, upstream parameter values are transferred to AEDT with SI unit values. There are no computations errors, but when you graphically display the design in AEDT, you cannot see the units as specified in Workbench. They are displayed as unitless values in SI.

Workbench User Preferences

User preferences are set in the **Options** dialog box.

The preferences you set here are local settings, affecting only you. Some changes made in the **Options** dialog take place immediately, while others take place after you restart Workbench.

Use the **Restore Defaults** button to reset the settings visible on the current page to their default values; settings on other pages remain unchanged.

> **Note:**
>
> An IT manager can a place a site-wide configuration file into the installation directory of each client machine to ensure that the Workbench software preferences are configured as the company requires. To force a preference to be set to a particular value at startup, edit `\Program Files\ANSYS Inc\22.2\Framework\Site\SiteStartup.wbjn`.

You can set user preferences in the following categories:

Project Management

Appearance

Regional and Language Options

Graphics Interaction

Journals and Logs

Project Reporting

Solution Process

Extensions

Mechanical APDL

CFX

Design Exploration

Distributed Compute Services

External Data

Fluent

Mechanical

Meshing

Engineering Data

Ansys Minerva

Microsoft Office Excel

TurboSystem

Project Management

From **Tools** → **Options** → **Project Management**, you can specify the following.

Preference	Description
Default Folder for Permanent Files	Sets the file directory that is selected by default when saving or opening a project.
	When you select a different location for a project by browsing to it in a dialog, Workbench remembers the folder location for subsequent operations for the rest of the session; however, upon starting a new session, the default returns to the location specified here.
	To have Workbench determine the default location, select the **Automatically Determined by Workbench** check box. To set the default location yourself, clear the **Automatically Determined by Workbench** check box and browse to the required folder.
Folder for Temporary Files	Sets the folder where temporary files are written. The directory specified here holds project files that are generated before the project is saved. Once a project is saved, files are written to the project directory (p. 278).
.wbpz Compression Level	Sets the file compression level for archiving projects (p. 270) to the .wbpz format. The default is **3**. Possible values are **0** through **9**, with **0** as no compression and **9** as maximum compression.
Share a single license between applications when possible	Specifies which licensing method to use. You can:
	• Share a single license between applications (selected by default)
	• Use a separate license for each application (clear the check box)
	If you change this setting, you must restart Workbench for this setting to take effect.
	For more details on these licensing methods, see Ansys Workbench Licensing Methods in the *Ansys, Inc. Licensing Guide*.

Appearance

From **Tools** → **Options** → **Appearance**, you can specify the following.

Preference	Description
Graphics Style	
Controls the graphics style of the Ansys Workbench, DesignModeler, Meshing, and Mechanical applications.	
Background Style	Sets a solid graphic background or a gradient background that varies from top to bottom, left to right, or diagonally. The default is the top to bottom gradient.

Preference	Description
Background Color	Sets a graphic background color from the built-in color palette. The default color is blue.
Background Color2	Sets a second graphic background color from the built-in color palette. The second color is used for gradient background displays. For example, if you want a top-bottom gradient that starts out white and ends up black, **Background Color** should be set to white and **Background Color2** should be set to black. The default color is white.
Text Color	Sets the color of all text from the built-in color palette. The default color is black.
Edge Colors	Sets the edge colors from the built-in color palette.
Meshed	Sets the color of all meshed edges from the built-in color palette. The default color is black.
Display	
Controls how Workbench displays information.	
Number of Significant Digits	Sets the number of digits that appear for numbers in Ansys simulation environments. The default is **5** and possible values are **3** through **10**.
	This setting affects only the numbers that are displayed. It does not imply any numerical round-off of internal calculations.
	When a user-entered number has a decimal halfway between two integers, the number is rounded to the nearest even integer. For example, a value of 20.5 is rounded down to 20, while a value of 21.5 is rounded up to 22.
Number of Files in Recently Used Files List	Sets the number of files that appear in the **Recently Used Files** list for both the **File** menu and the context menus. The default is **4** and the maximum number of files that you can display is **20**.
	If the number specified here exceeds the number of recently used files that are available, the list displays the available number. This setting is applied to the current Workbench session.
Beta Options	When selected, beta features are displayed with the word *beta* in parentheses. This check box is cleared by default.
	Beta features remain untested in this release and therefore are neither documented nor supported and may result in unpredictable behavior.
	The beta option on Linux for Mechanical Explicit Dynamics is not passed to the solver and as such has no effect for solver related beta functionality.
Unsupported Features	When selected, displays deprecated features, with the word *unsupported* in parentheses, in the **Toolbox**. This check box is cleared by default.
	While unsupported features may still be used in older projects, we recommend updating projects to currently supported features. Because these features are unsupported, they may not function correctly in newer versions of the product.

Preference	Description
Text on Toolbars	When selected, displays text labels on toolbars. This check box is selected by default. This option applies to the Ansys Workbench interface, the Mechanical application, the Meshing application, and DesignModeler only.
Connections are Bundled at Startup	When selected, displays connections between systems as a single link. The label indicates all bundled connections. For example, linked systems that shared **Engineering Data**, **Geometry**, and **Model** cells display a single link with the notation *2:4*, indicating that cells two (**Engineering Data**) through four (**Model**) are connected. This check box is cleared by default. This setting defines the initial default for all projects and can be overridden within each project by selecting **View** → **Show Connections Bundled**, or by selecting this option from the context menu on the **Project Schematic**. Use of this option affects only newly created projects.
System Coordinates are Shown at Startup	When selected, displays the system label letters and numbers. This check box is selected by default. This setting defines the initial default for all projects and can be overridden within each project by selecting **View** → **Show System Coordinates**, or by selecting this option from the context menu on the **Project Schematic**.
Quick Help Icons in System Cells	When selected, the quick help icon in cells where quick help is available. The icon appears as a small blue triangle in the lower right corner of the cell. This check box is selected by default.
Word-wrap Text in Tables	When selected, allows text in tables to wrap within the cells. This check box is selected by default.

Regional and Language Options

From **Tools** → **Options** → **Regional and Language Options**, you can specify the language used for Workbench text and messages.

You can choose to use English, German, French, or Japanese, if localized files are available. Workbench records the language selected at installation and specified from this option in the `languageset-tings.txt` file. You can also manually edit the `languagesettings.txt` file to specify the language. You must exit Workbench and start a new session for changes to the language option to take effect.

Workbench looks for the `languagesettings.txt` file in the following locations, in order:

1. `%appdata%\Ansys\v222` on Windows or `$HOME/.ansys/v222` on Linux

2. `<install_dir>\Ansys Inc\v222\commonfiles\Language` (Windows) or `<install_dir>/ansys_inc/v222/commonfiles/language` (Linux)

Be aware that not all external applications may be translated; therefore, you may see some text and messages in English even if you have specified a different language.

Graphics Interaction

From **Tools** → **Options** → **Graphics Interaction**, you can specify the following.

Preference	Description
Mouse Wheel	Sets the action that the mouse wheel performs when used.
Middle Button	Sets the action that the mouse middle button performs when used.
Right Button	Sets the action that the mouse right button performs when used.
Shift +Left Button	Sets the action that occurs when pressing **Shift** on the keyboard and clicking the left mouse button simultaneously.
Shift +Middle Button	Sets the action that occurs when pressing **Shift** on the keyboard and clicking the middle mouse button simultaneously.
Shift +Right Button	Sets the action that occurs when pressing **Shift** on the keyboard and clicking the right mouse button simultaneously.
Ctrl +Left Button	Sets the action that occurs when pressing **Ctrl** on the keyboard and clicking the left mouse button simultaneously.
Ctrl +Middle Button	Sets the action that occurs when pressing **Ctrl** on the keyboard and clicking the middle mouse button simultaneously.
Ctrl +Right Button	Sets the action that occurs when pressing **Ctrl** on the keyboard and clicking the right mouse button simultaneously.
Ctrl +Shift +Left Button	Sets the action that occurs when pressing **Ctrl**+**Shift** on the keyboard and clicking the left mouse button simultaneously.
To Zoom In Closer	Sets whether to zoom on a model by moving the mouse in or out.
Dynamic Viewing During Rotation	When selected, a short animation shows the model moving/twisting toward its final pose if you make a standard view change (such as front, back, left, right, bottom, top, isometric, and Look At Face/Plane/Sketch). This check box is selected by default. Clear this check box if you are using an older graphics card.
Use Spaceball	When selected, enables the use of the Spaceball 3D import device (not supported in UNIX). This check box is selected by default.
Extend Selection Angle Limit	Sets a limit in degrees for what kind of face and edge angles the system considers smooth. This affects the Extend to Adjacent and Extend to Limits Extend Selection toolbar buttons in DesignModeler. Extend Selection buttons are also present in the Mechanical application. The default value is **20°** and the range is from **0°** to **90°**.
Angle Increment for Configure Tool	Sets the angular increment of the Configure tool when defining a joint.

Journals and Logs

From **Tools** → **Options** → **Journals and Logs**, you can specify the following.

Preference	Description
Record Journal File	When selected, records a journal file (p. 252) (`.wbjn`) for each Workbench session and enables the other journal options.
Journal File Directory	Selects the directory where journal files are written. On Windows, the directory preference defaults to `%TEMP%\Workbench-Journals`. On Linux, the directory preference defaults to the following, in order: 1. `$TEMP/WorkbenchJournals` 2. `$TMP/WorkbenchJournals` 3. `$HOME/.ansys/WorkbenchJournals` If you record a session for future playback, the same information is written to both the location specified here and to the file specified when you begin recording.
Days to Keep Journal File	Sets the number of days to keep journal files. The default is **7**.
When running a journal file, pause after each command	When selected, adds a pause after each command when running a journal file. Also enables the **Seconds to Pause** field.
Seconds to Pause	Sets the number of seconds to pause between commands. The default is **1**.
Write Workbench Log Files	When selected, writes two log files for each Workbench session and enables the other log file options. The two files are: • `UIEvents`*`procID`*`.log` • `CoreEvents`*`procID`*`.log` where *`procID`* is the process ID of the Workbench session. To identify the log file of the most recent session, it is most convenient to sort the directory by date. The log file contains information that is useful to technical support whenever an error is encountered. Other applications may continue to write log files to their own directories, even if launched from Workbench.
Workbench Log Files Directory	Selects the directory where log files are written. On Windows, the directory preference defaults to `%TEMP%\Workbench-Logs`. On Linux, the directory preference defaults to the following, in order: 1. `$TEMP/WorkbenchLogs` 2. `$TMP/WorkbenchLogs` 3. `$HOME/.ansys/WorkbenchLogs`

Preference	Description
	The log file directory preference takes effect immediately when the **Options** dialog is closed.
Days to Keep Workbench Log File	Sets the number of days to keep log files. The default is **7**.

Project Reporting

From **Tools** → **Options** → **Project Reporting**, you can set if exported reports automatically open in a browser (p. 266) when they are generated.

Solution Process

From **Tools** → **Options** → **Solution Process**, you can specify the defaults for many of the solution process properties of **Solution** cells. For projects with parametric design points, these settings also control the defaults for the design point update process setting for the **Parameter Set** bar. For more information on using these properties to submit a job to Remote Solve Manager (RSM), see Submitting Solutions to Remote Solve Manager (p. 321).

Preference	Description
Default Update Option	Sets how the solution is run by default. Select from the following: • **Run in Foreground**: The solution runs within the current Workbench session. • **Run in Background**: The solution runs in the background on the local machine. This option is available only for **Solution** cells that support background execution. • **Submit to Remote Solve Manager**: The solution runs in the background either on the local computer or, if Remote Solve Manager is configured, on one or more remote computers. If you submit to RSM, the solution continues to run even if you close the current Workbench session. If you have not defined configurations in RSM, or obtained configurations from your RSM administrator, see RSM Configuration in the *Remote Solve Manager User's Guide*.
Default Design Point Update Order	Specifies the starting condition of each design point for design point updates. By default, when each design point is updated, the design point is initialized with the data of the design point designated as Current (except for retained design points with valid retained data, which do not require initialization data). However, in some cases it may be more efficient to update each design point starting from the data of the previously updated design point, rather than restarting from Current each time. • **Update From Current**: Causes each design point to be initialized from the Current design point. This is the default value.

Preference	Description
	• **Update Design Points in Order**: Causes each design point to be initialized from the previous design point. If you selected **One Job for Each Design Point** in the **Default Job Submission** field, the **Default Design Point Update Order** is applicable only if you also selected **Geometry** in the **Pre-RSM Foreground Update** field. This setting takes effect the next time you create a new project. See Changing the Design Point Update Order (p. 310) for more information.
Show Advanced Solver Options	When selected, enables the **Interconnect** and **MPI Type** parallel run settings available for Fluent RSM runs. These settings require that the remote compute servers can accept the Interconnect and MPI Type that you specify; compatibility is not checked automatically.
Retained Design Point	Specifies how an update is performed for retained design points. Select one of the following options: • **Update parameters**: Only parameters are updated for retained design points. If a component is not needed to get the value of an output parameter, it is not updated. This is the default value. • **Update full project**: Full project is updated for retained design points. Use this setting if you want to generate reports, or other content, from components that do not produce output parameters.
Default RSM Queue	Sets the RSM queue that is selected by default when you select the **Submit to Remote Solve Manager** update option. The list of available queues depends on the configurations that are available in the RSM configuration directory. See Sharing and Accessing RSM Configurations in the *Remote Solve Manager User's Guide*.
Default Job Name	Sets the default name that appears in the **Job Name** field. The name must start with an alphanumeric character, must not contain spaces or the characters ! @ $ * ? \ and must not be longer than 15 characters.
Download Progress Information by Default	When selected, enables the solver monitor to periodically query RSM for output files in order to display progress (where applicable).
Default Progress Download Interval (Seconds)	Specifies the time interval, in seconds, between solver queries to RSM in order to display progress. The default is **30**. Setting this value to zero (0) results in continuous queries; that is, as soon as files are downloaded from the compute server, Workbench immediately queries again. This option is available for CFX systems only.
Default Execution Mode	Specifies if you want the solution to run in serial or parallel mode. The default is **Serial**. The parallel option is available only if the selected solver supports parallel execution mode.
Default Number of Processes	Specifies the default number of processes to use if you choose to run in parallel mode. Must be set to **2** or greater.

Preference	Description
Pre-RSM Foreground Update	Specifies how geometry is handled before submitting project updates to RSM. Select one of the following options: • **None**: Submits project updates to RSM without updating the geometry locally. • **Geometry**: Updates your geometry locally before submitting project updates to RSM.

Extensions

From **Tools** → **Options** → **Extensions**, you can configure settings for Ansys ACT extensions.

For more details on these options, see Extension Configuration in the *ACT Developer's Guide*.

Preference	Description
Additional Extension Folders	Defines the additional folders where ACT searches for the extensions that the Extension Manager displays. You can define several folder names, separating them with a semicolon (;).
Save Binary Extensions with Project	Specifies whether to save binary extensions within the project when the project is saved.
Debug Mode	When selected, enables debug mode for IronPython scripts of loaded extensions.

Mechanical APDL

From **Tools** → **Options** → **Mechanical APDL**, you can specify the following.

For more details on these options, see The File Management Tab in the *Operations Guide*.

Preference	Description
Command Line Options	Sets the startup command line options.
Database Memory (MB)	Specifies the portion (in megabytes) of total memory that the database uses for the initial allocation. The default is **1024**.
Workspace Memory (MB)	Specifies the initial allocation of memory (in megabytes) requested for the Mechanical APDL run. The default is **2048**.
Utilized cores	Sets the number of cores used for parallel processing.
Job Name	Defines the base filename used for all files generated by the Mechanical APDL session. The default is **file**. You can change it to any alphanumeric string up to 32 characters long.
License	Sets the default license level.
Graphics Device	Sets the graphics device. The user interface requires a terminal that supports graphics.
Distributed	When selected, activates distributed computing settings.

Preference	Description
Machine List	Specifies machine information for distributed computing.
GPU Accelerator	Provides access to the Graphics Processing Unit (GPU) acceleration capability offered in the **Project Schematic** and inside Mechanical APDL. Three options are available, **None**, **NVIDIA** and **Intel**. The default is **None**. Selecting this option from **Options** → **Mechanical APDL** applies the setting to all newly added systems in the **Project Schematic**. You can override these settings by changing the **GPU Accelerator** selection on individual systems by right-clicking **Analysis** and editing the properties.
Number of GPUs per Machine	Specifies the number of GPUs each machine has. The default is **1**.
Read START.ANS	When selected, reads the `start.ans` file at start-up.
Custom Executable Path	Sets a custom Mechanical APDL executable to start in the launcher.
Download Distributed Files	When selected, downloads all distributed files.

CFX

From **Tools** → **Options** → **CFX**, you can specify the following.

For more details on these options, see Properties Pane in the *CFX Introduction*.

Preference	Description
Keep Latest Solution Data Only	When selected, deletes any associated old solution data automatically after updating the **Solution** cell.
Cache Solution Data	When selected, retains cached solution data for non-current design points (in addition to the current design point) after they are solved.
Initialization Option	Sets how data is gathered to initialize the solution.
Execution Control Conflict Option	Sets the option used to resolve execution control conflicts. For more information, see Troubleshooting in the *CFX Introduction*.

Design Exploration

From **Tools** → **Options** → **Design Exploration**, you can specify the following.

Preference	Description
Show Advanced Options	When selected, displays advanced properties (in italics) for certain DesignXplorer objects in the **Properties** pane.
Preserve Design Points after DX Run	When selected, saves design points created for a design exploration cell to the project's design points table once the solution completes. Also enables the **Retain Data for Each Preserved Design Point** check box.
Retain Data for Each Preserved Design Point	When selected, retains data for each design point saved in the project's design point table.

Preference	Description
Retry All Failed Design Points	When selected, enables additional attempts to solve design points that failed to update during the first run. Also enables **Number of Retries** and **Retry Delay (seconds)**.
Number of Retries	Sets the number of times to try to update failed design points. The default is **5**.
Retry Delay (seconds)	Sets the number of seconds to elapse between tries. The default is **120**.
Chart Resolution	Sets the number of points for a continuous input parameter axis in a 2D or 3D Response Surface chart. The range is from **2** to **100**. The default is **25**. Increasing the number of points enhances the chart resolution.
Significance Level	Sets the relative importance or significance to assume for input variables. The allowable range is from **0.0** to **1.0**, where 0 means that all input variables are assumed to be insignificant and 1.0 means that all input variables are assumed to be significant. The default is **0.025**.
Correlation Coefficient Calculation Type	Sets the calculation method for determining sensitivity correlation coefficients. The default is **Rank Order (Spearman)**.
Display Parameter Full Name	When selected, displays full parameter names rather than short parameter names.
Parameter Naming Convention	Sets the naming style for input parameters within design exploration. The default is **Uncertainty Management Style**.
Confirm if Min-Max Search can take a long time	When selected, displays an alert before performing a Min-Max search operation when there are discrete input parameters to warn that the search could be time-consuming.

Note:

If you expand **Design Exploration** in the tree, you will see three child tabs: **Design of Experiments**, **Response Surface**, and **Sampling and Optimization**. For more information about the options on all four DesignXplorer tabs, see Design Exploration Options in the DesignXplorer User's Guide.

Distributed Compute Services

From **Tools → Options → Distributed Compute Services**, you can specify the following.

Preference	Description
Server URL	Sets the URL for starting the DC server. The default for starting a local server is `https:/localhost/dcs`. The format of a URL for starting a remote server is `https://hostname:port/dcs` or `https://IPad-dress:port/dcs`.
Auto-Start Local Evaluator	When selected, automatically starts an evaluator on the local machine when design point updates are requested. This check box is selected by default. If you clear this check box, you can start the evaluator on the local machine by going to the Windows **Start** menu and selecting **DC Evaluator**. You can also start it from the command line by typing `dclauncher.exe start evaluator`.

External Data

From **Tools** → **Options** → **External Data**, you can specify the following.

Preference	Description
Files Import Type	Sets how the External Data file is handled when importing External Data (p. 109). • **By Copy**: Copies the External Data file into the project folder. • **By Reference** (default): Directly references the External Data file. For large size files, this drastically reduces disk storage requirements and improves performance.

Fluent

From **Tools** → **Options** → **Fluent**, you can specify the following.

General options apply to all Fluent-based systems in the Workbench project, regardless of whether the system was created before, or after, the option is enabled. Note that these options are not saved with the project, and the settings are always applied to the currently loaded project.

Default options for new Fluent system apply to all newly created Fluent-based systems in the Workbench project. They can be overridden by editing the properties specified for the **Setup** and **Solution** cell. The settings specified in the **Setup** and **Solution** cell properties are always respected and saved with the Workbench project. Therefore, for previously saved projects, these saved settings are used.

Preference	Description
General Options	
Show Warning on Editing Setup if Solution Has Current or Initial Data	When selected, displays a warning message when solution data exists and you attempt to open Fluent from the **Setup** cell. When you open Fluent from the **Setup** cell, the mesh and settings file associated with the **Setup** cell are loaded into Fluent which may or may not be what you want. The check box is selected by default.
Automatically Delete Old Solutions On Start Of New Calculation	When selected, old solution data is automatically removed when starting a new computation. The check box is selected by default.
Reuse Fluent Session for Design Points	When selected, reuses the Fluent session for other design point calculations. This option works with foreground parametric updates only. The check box is cleared by default.
Default Options for New Fluent System	

Preference	Description
Show Launcher at Startup	When selected, displays the Fluent Launcher when Fluent starts. The check box is selected by default.
Display Mesh After Reading	When selected, displays the mesh after the mesh or case/data is read into Fluent. The check box is selected by default.
Embed Graphics Windows	When selected, embeds the graphics windows in the Fluent application window. When cleared, unlocks the graphics windows so they are free-standing. The check box is selected by default.
Use Workbench Color Scheme	When selected, sets the graphics window to the Workbench color scheme. When cleared, sets the graphic window to the classic black background color. The check box is selected by default.
Set up Compilation Environment for UDF	When selected, allows you to specify compiler settings for compiling user-defined functions (UDFs) with Fluent. The check box is selected by default.
Precision	Sets either the single-precision or the double-precision solver.
Enable Generation of Setup Output Case File	When selected, bypasses loading the mesh and setting files and reapplying the pre-set mesh operations every time the **Setup** cell is edited, resulting in faster runs. This option is especially beneficial when computing simulations across multiple design points involving Fluent-related parametric changes. The Fluent solver automatically generates the output case file, `name-Setup-Output.cas`, every time you modify the mesh, set mesh operations prior to running the simulation in Fluent, or start a Fluent session with a mesh file only. The generated output case file is used when launching the next Fluent session from the **Setup** cell if the regular case file is out-of-date or not available. The check box is selected by default.
Enable Solution Monitoring	When selected, allows you to be graphically view Fluent solution convergence and monitor data without having Fluent open. Once this option is enabled, you can use the **Show Solution Monitoring** option in the **Solution** cell context menu to display convergence and monitor charts. This check box is selected by default.

Preference	Description
Enable Generation of Interpolation File	When selected, Instructs the Fluent solver to automatically generate an interpolation file, `*.ip`, at the end of the run. Unlike the `*.dat` file, the `*.ip` file can be used to restart the Fluent session, even if the mesh has been modified This check box is cleared by default.

Mechanical

From **Tools** → **Options** → **Mechanical**, you can specify the following.

Preference	Description
Auto Detect Contact On Attach	When selected, enables computed contact detection when geometry is imported into Mechanical. This check box is enabled by default.
Save Mesh Data In Separate File	When selected, enables the default setting for the **Save Mesh Data In Separate File** property in Mechanical systems (such as Static Structural) for subsequently created Mechanical systems. The property setting controls whether mesh data is stored in a separate `.acmo` file for the current system. Saving the mesh data in a separate file reduces the overall file space required and reduces the possibility of disc corruption in very large database files on Linux systems.
Release License for Pending Jobs	Specifies when the Mechanical application releases its license when running in batch mode: • **On Demand**: When the **Solution** cell for a Mechanical system is in the pending state during a batch run, the right-click menu has a **Release License** option. This closes the Mechanical application but does not interfere with the completion of the run. (Default) • **Always**: Causes the Mechanical application to automatically close and release its license during batch runs when the **Solution** cell is in the pending state. • **Design Point Run Only**: Causes the Mechanical application to automatically close and release its license during **Update All Design Points** runs when the **Solution** cell is in the pending state.
Maximize Design Point Performance	When selected, causes Mechanical not to copy design point results data when parameters are changed in Mechanical simulations, as the design points are re-evaluated in those cases. Note that no working directory files (such as macros) are copied, so to use this setting you must move any files that you have put into the working directory (the solver files directory) to the `user_files` directory.

Preference	Description
Remote Job Status Check Interval in Seconds	Sets the number of seconds between each query to Remote Solve Manager for the Mechanical solution update job status. The default is 20. This setting only takes effect when the Mechanical solution is submitted to RSM.
During a design point update, periodically restart the Mechanical application	When selected, directs the Mechanical application to automatically restart after the specified number of design points when running in batch mode. Each restart resets the Mechanical application and slightly lengthens the processing time, but can improve overall system performance (memory and CPU) when the generation steps of each design point (geometry, mesh, solve, post processing) are long. In such cases, specify a low number of design points before restarts. In cases where the generation steps for each design point are short, reduce processing time by increasing the number of design points before restarts or prevent restarts completely by disabling this preference.
Number of design points to update before restarting the Mechanical application	Sets the number of design points to update when the **During a design point update, periodically restart the Mechanical application** check box is selected. The default is **1**.
Limit Number of Cores for Data Mapping and Post-Processing	When selected, sets the number of cores used by the data transfer mapping, interpolation operations and result postprocessing to a user-specified value.
Maximum number of cores	Sets the maximum number of cores used when the **Limit Number of Cores for Data Mapping and Post-Processing** check box is selected. The default is **2**.
Beta features selection	Select one or more check boxes to enable Mechanical beta features in Workbench.

Meshing

From **Tools** → **Options** → **Meshing**, you can specify the following.

Preference	Description
Auto Detect Contact On Attach	When selected, enables contact detection computing when geometry is imported into Meshing. This check box is selected by default.
Save Mesh Data In Separate File	When selected, enables the default setting for the **Save Mesh Data In Separate File** property in Mechanical systems (such as Static Structural) for subsequently created Mechanical systems. The property setting controls whether mesh data is stored in a separate `.acmo` file for the current system. Saving the mesh data in a separate file reduces the overall file space required and reduces the possibility of disc corruption in very large database files on Linux systems.

Preference	Description
During a design point update, periodically restart the Meshing application	When selected, directs the Meshing application to automatically restart after the specified number of design points when running in batch mode. Each restart resets the Meshing application and slightly lengthens the processing time, but can improve overall system performance (memory and CPU) when the meshing of each design point is long. In such cases, specify a low number of design points before restarts. In cases where the meshing step for each design point are short, reduce processing time by increasing the number of design points before restarts or prevent restarts completely by disabling this preference.
Number of design points to update before restarting the Meshing application	Sets the number of design points to update when the **During a design point update, periodically restart the Meshing application** check box is selected. The default is **1**.

Engineering Data

From **Tools** → **Options** → **Engineering Data**, you can specify the following.

Preference	Description
Filter toolbox content for selection	When selected, filters Toolbox content based on your selection in the Properties pane.
Unset the default material when materials are imported with geometry.	When selected, clears the default material used when materials are imported with geometry attached.
Consolidate materials on import	When selected, consolidates the material information when it is imported into Workbench.

Ansys Minerva

From **Tools** → **Options** → **Ansys Minerva**, you can specify the following.

Preference	Description
Authentication Type	Sets the method to used to authenticate with Ansys Minerva. Select the method that matches what is configured for the Ansys Minerva server. Select from the following options: • **Basic**—Uses a username and password to authenticate using the internal mechanism within Ansys Minerva. • **Browser**—Uses the browser to authenticate. If you are logged into Ansys Minerva in the browser, those credentials are used. • **Windows**—Uses Active Directory to pass the credentials of the user currently logged into the Ansys Minerva server.
Server URL	Sets the URL of the Ansys Minerva server that Workbench connects to.

Preference	Description
Database	Sets the name of the Ansys Minerva database to access.

Microsoft Office Excel

From **Tools** → **Options** → **Microsoft Office Excel Options**, you can specify the following for the Microsoft Office Excel add-in.

Preference	Description
Named Ranges Filtering Prefix	Sets a prefix to filter which Excel named ranges are exposed as parameters in a Workbench project. All named ranges defined in the Excel file that include that prefix are displayed as parameters in the project. By default, this setting is blank (no filter). If you specify a prefix, the prefix is used to filter the named ranges for all new design exploration systems. For example, if you set this option to **WB**, the **Named Range Key** property (in the **Setup** section of the **Properties** pane of the Microsoft Office Excel **Analysis** system) is set to **WB** for all new projects you create. If you have specified a different filtering prefix the project level in the **Named Range Key** property, the project-level setting is not affected by changes to this option.

TurboSystem

From **Tools** → **Options** → **TurboSystem**, you can specify the following.

Preference	Description
Include inlet/outlet domains in Turbo Mesh cells by default	When selected, causes subsequently created Turbo Mesh cells to have the **Meshing** → **Inlet Domain** and **Meshing** → **Outlet Domain** properties initially selected.
Data transfer type	Controls the default value of the **Data Transfer** → **Geometry Transfer Type** property of subsequently created Blade Design cells.

Geometry Import

From **Tools** → **Options** → **Geometry Import** → **Geometry Import**, you can specify the following.

Preference	Description
Preferred Geometry Editor	Sets the geometry editor. You can select from DesignModeler or SpaceClaim Direct Modeler (default). **Note:** The SpaceClaim Direct Modeler option is not available when the **Use SpaceClaim Direct Modeler as an External CAD** option is enabled.
CAD Licensing	Sets how the CAD license is kept or released. • **Hold**: Instructs Workbench to keep the license after the import or refresh operation has completed. The option is useful when executing design studies. If the **Hold** option was previously set and subsequently changed to **Release**, the license is released immediately after clicking **OK** in the **Options** dialog box. A plug-in license that is held is always released when exiting the Workbench session. • **Release** (default): Instructs Workbench to free the license for someone else's use after the import or refresh operation has completed. **Note:** The **CAD Licensing** option is not supported for geometry import into SpaceClaim Direct Modeler. Regardless of the setting of **CAD Licensing**, SpaceClaim Direct Modeler always releases its CAD licenses after importing.
Use SpaceClaim Direct Modeler as an External CAD	When selected, SpaceClaim is controlled as an external CAD system, with no direct integration into the **Project Schematic**. When cleared, SpaceClaim is fully integrated into the **Project Schematic** as a geometry editor. This check box is cleared by default. **Note:** • Switching this preference from within an active project could result in project schematic inconsistencies and undesired behavior. We recommend that you set this preference during project creation and ensure that all users interacting with the project do not change it. • If the preference must be changed, the Geometry systems that reference SpaceClaim geometry should be **Reset** and

Preference	Description
	reconfigured to properly release any previously stored parameters and file references. • You cannot use SpaceClaim with meshing tools enabled as an external CAD system. If the meshing options are enabled and this preference is selected, the following error message is displayed when you connect to a downstream system: `Can not use SpaceClaim w/Meshing as an external CAD system. Please change your settings.`
Use Workbench Project Units for Length and Angle	When selected, SpaceClaim Direct Modeler adopts the units of the Workbench project when it is launched from Workbench.
Use Workbench Language Settings	When selected, SpaceClaim Direct Modeler adopts the language used by Workbench when it is launched from Workbench **Project Schematic**.
Geometry editor behavior during update	Controls the visibility of the geometry editor during a design point update. • **Default**: The update is run in batch mode. If the geometry editor was open prior to the update, it is reopened at the completion of the update. • **Run interactively**: Forces the geometry editor to be run interactively for the entire update. If the geometry editor was open prior to the update, it is reopened at the completion of the update. • **Run in batch mode**: Forces the geometry editor to be run in batch mode for the entire update. The geometry editor is left closed at the completion of the update.
Keep the geometry editor running for optimized design point updates	When selected, directs the geometry editor to stay open (in batch or interactively based on the setting of **Geometry editor behavior during update**) and use the same session during a design point update. Allows for quicker processing time as the geometry editor is not restarted after each design point.
Periodically restart the geometry editor	When selected, directs the geometry editor to automatically restart after the specified number of design points (as specified by the setting of **Number of design points to update before restarting the geometry editor**).
Number of design points to update before restarting the geometry editor	Sets the number of design points that are updated before the geometry editor is restarted. The default is **0**, which executes the same as if this preference were disabled. Each restart resets the geometry editor and slightly lengthens processing time, but can improve overall system performance (memory and CPU) when the time needed for the geometry update of each design point

Preference	Description
	is long. In such cases, specify a low number (1 or more) of design points before each restart. In cases where the time needed for the geometry update of each design point is short, reduce processing time by increasing the number of design points before each restart, or prevent restarts completely by disabling this preference.
Basic options	For detailed descriptions of the basic geometry import options, see the **Basic Geometry Options** table in Geometry Preferences.
Advanced options	For detailed descriptions of the advanced geometry import options, see the **Advanced Geometry Options** table in Geometry Preferences.

Material Designer

From **Tools > Options > Material Designer**, you can specify the following:

Preference	Description
Material Designer behavior during update	Controls the visibility of Material Designer during a design point update. • **Default**: The update is run in batch mode. If Material Designer was open prior to the update, it is reopened at the completion of the update. • **Run interactively**: Forces Material Designer to run interactively for the entire update. If Material Designer was open prior to the update, it is reopened at the completion of the update. • **Run in batch mode**: Forces Material Designer to run in batch mode for the entire update. Material Designer is left closed at the completion of the update.
Keep Material Designer running for optimized design point updates	When selected, directs Material Designer to stay open (in batch or interactively, based on the setting of **Material Designer behavior during update**) and use the same session during a design point update. Allows for quicker processing time as Material Designer is not restarted after each design point.
Periodically restart Material Designer	When selected, directs Material Designer to restart automatically after the specified number of design points (as specified by the setting **Number of design points to update before restarting Material Designer**).
Number of design points to update before restarting the Material Designer	Sets the number of design points that are updated before Material Designer is restarted. The default is **0**, which executes as if this preference were disabled. Each restart resets Material Designer and slightly lengthens processing time, but restarts can improve overall system performance (memory and CPU) when the time needed for the

Preference	Description
	update of each design point is long. In such cases, specify a low number of design points (**1** or more) before each restart. In cases where the time needed for the update of each design point is short, you can reduce processing time by *increasing* the number of design points before each restart, or you can prevent restarts by disabling this preference altogether.

Workbench Interface Reference

The following user interface components are discussed in detail:

Tabs in Workbench

Tabs are workspaces that allow you to interact with different parts of your project. Most cells of a system open in separate windows outside of the Workbench application; the following is a list of tabs for systems and cells that open within Workbench.

- 3D ROM

- Analysis (p. 192) (Mechanical APDL)

- Analysis (p. 206) (Microsoft Office Excel)

- System Coupling (p. 219)

- Design of Experiments

- Direct Optimization

- Engineering Data

- External Data (p. 108)

- External Model (p. 129)

- Parameter Set (p. 298)

- Parameters Correlation

- Project (p. 47)

- Response Surface

- Response Surface Optimization

- Six Sigma Analysis

Panes within Tabs

Workbench provides the following categories of panes.

Common Panes

Common panes are ones that are included on multiple tabs, but are configured per tab. changes to the pane on one tab are not reflected on other tabs of the same sort, the pane contains different data on each tab.

The following common panes are available:

- Project Schematic Pane (p. 440)

- Toolbox Pane (p. 441)

- Toolbox Customization Pane (p. 443)

- Files Pane (p. 443)

- Outline Pane (p. 443)

- Properties Pane (p. 444)

- Table Pane (p. 445)

- Chart Pane (p. 445)

- Scene Pane (p. 445)

- Solution Information Pane (p. 445)

Persistent Panes

Persistent panes remain configured as you have specified when you navigate between tabs. For example, if you display a persistent pane in one tab, then switch to a different one, the persistent pane remains visible in the second tab.
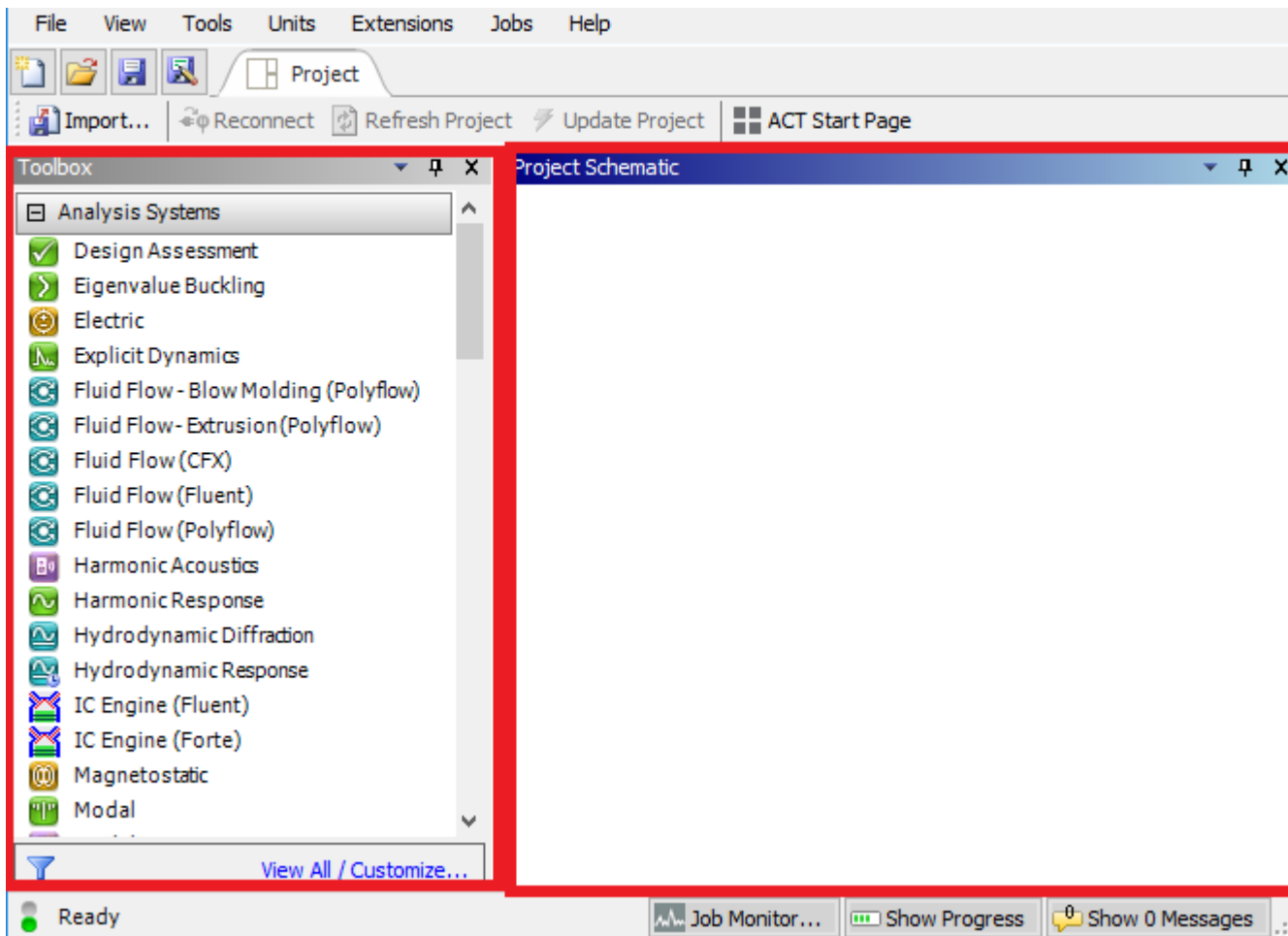
The following persistent panes are available:

- Messages Pane (p. 446)

- Progress Pane (p. 446)

- Sidebar Help (p. 447)

Project Schematic Pane

Located in the **Project** tab, the **Project Schematic** pane is the main workspace for your project.

Opening the **Properties** pane allows you to manage the following properties for the **Project Schematic** pane.

| Property	Description
Notes	Displays all project notes (p. 241) created.

Toolbox Pane

Located in the **Project** tab, the **Toolbox** pane contains the types of data you can add to your project.

To add data, you take the systems located in the **Toolbox** pane and add them to the **Project Schematic** pane.

Systems are divided into categories which can be expanded or collapsed to show or hide the systems in that category. You can select systems from the following categories:

- Analysis Systems (p. 74)

- Component Systems (p. 102)

- Custom Systems (p. 223)

- Design Exploration (p. 229)
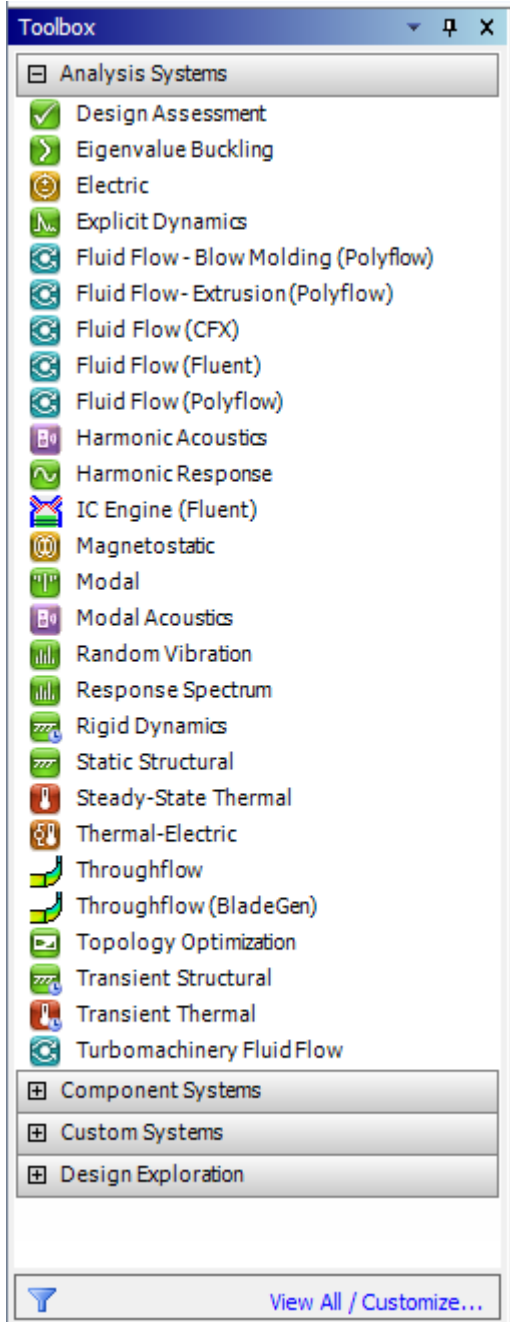
The contents of each category are determined by which products you have installed and what licenses are available. If you do not have a particular product installed or do not have an available license, the individual systems corresponding to that product are not shown in the **Toolbox**.

For detailed information on system categories and individual systems, see Working Through a System (p. 67).

Additionally, you can customize the **Toolbox** further, specifying that only some of the available systems are visible. For customization details, see Configuring the Toolbox (p. 25).

The **Toolbox** pane persists across tabs. It contains systems on the **Project** tab, but on other tabs it contains other sorts of information, such as charts, and engineering materials.

Toolbox Customization Pane

The **Toolbox Customization** pane enables you to specify which systems display in the **Toolbox**. For more information, see Configuring the Toolbox (p. 25).

Files Pane

The **Files** pane shows a list of all files associated with, and added to, the project. It enables you to see the name and type of file, the IDs of the cells the file is associated with, the size of the file, the location of the file, and other information.

For more information on how to manage and work with the files shown in the pane, see Working in the Files Pane (p. 273).
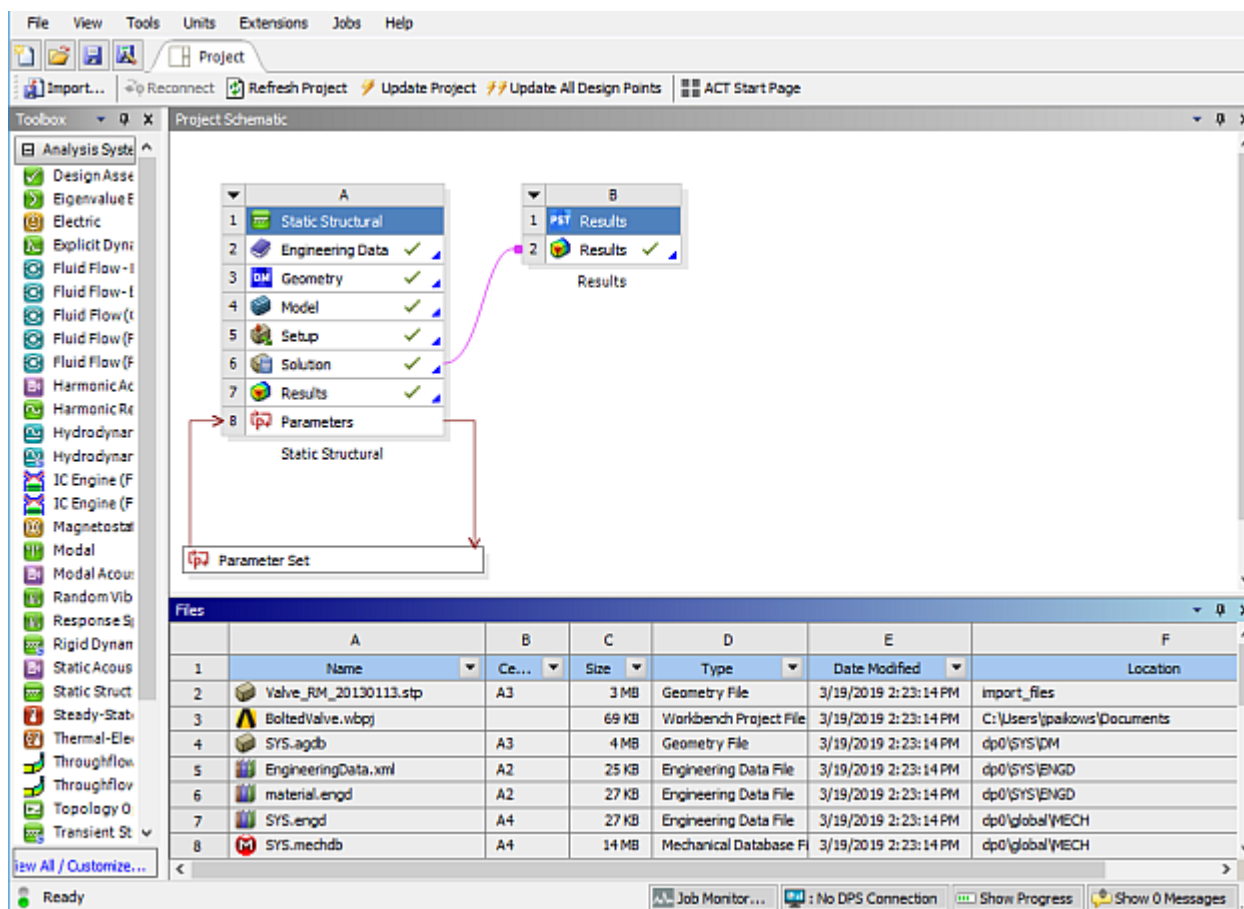
Outline Pane

When data is available for an item, the **Outline** pane presents data in a summary form. You can access the **Outline** pane in the **Project** tab, the **Parameter Set** tab, the **Parameters** tab, or the tab for any cell in a Design Exploration or Engineering Data system.

To open the **Outline** pane, double-click the **Parameter Set** bar or a system cell. Or, right-click the cell and select **Edit** from the context menu.
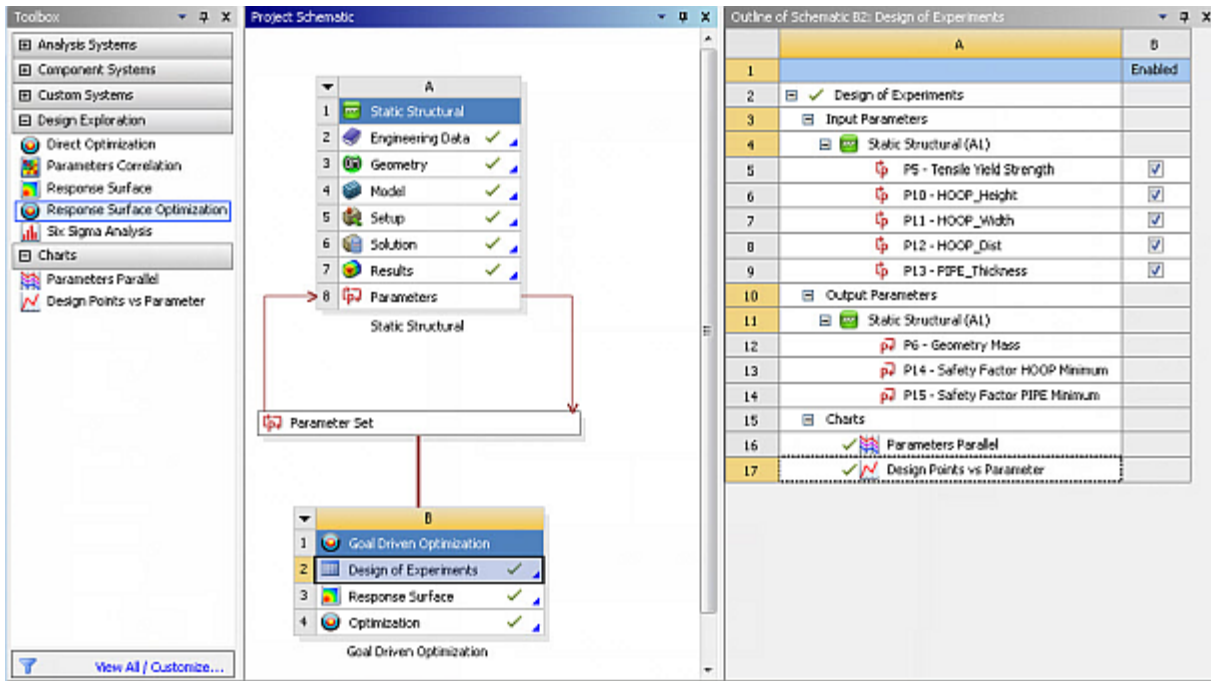

Examples of the **Outline** pane include the **Outline of All Parameters** in the **Parameter Set** tab and the **Outline of Schematic** in an application tab.

For information about using the **Outline** pane for parameters and design points, see Working with Parameters and Design Points (p. 283).

Properties Pane

The **Properties** pane enables you to see properties for the **Project Schematic**, systems, cells, and schematic links.

The specific items shown in the **Properties** pane varies according to what you have selected.
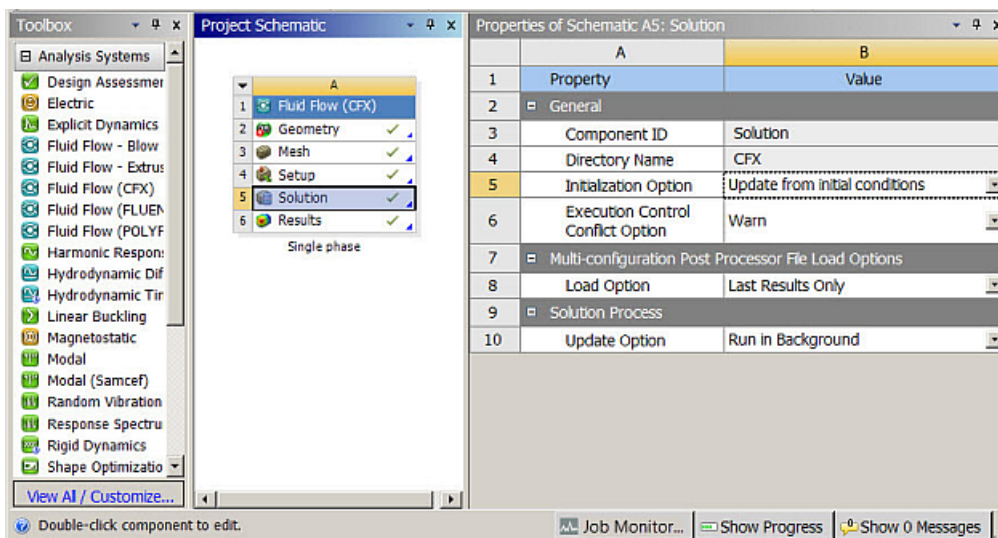
In some cases, it displays basic information about the component that cannot be modified from the **Properties** pane. For example, if you choose to view properties on a link between systems, a list of uneditable connection properties (for example, type and from/to cell identifiers) is displayed.

In other cases, the **Properties** pane displays detailed information that can be edited. For example, if you choose either **Edit** or **Properties** from the Vista TF **Setup** cell, a fully-editable list of properties is displayed.

If more information on a specific **Properties** pane is available, you can view the quick help for the cell for which you are viewing properties.

Table Pane

When the **Table** pane is included in a tab, it allows you to view project data in table format. Examples of the **Table** pane are the **Table of Design Points** in the **Parameter Set** tab and the **Table of Properties** in an **Engineering Data** tab.

Table of Design Points							
	A	B	C	D	E	F	G
1	Name	P2 - WB_X	P3 - WB_Y	P1 - WB_Rosen...	Retain	Retained Data	Note
2	DP 0 (Current)	0	1	101	☑	✓	
*					☐		

Some tables are editable and some are not. In general, if you can add a new row, you an asterisk is displayed (*) in the last row. If a cell is editable, you can double-click in the cell to select the content and change it. In some tables, columns may include a drop-down menu of additional actions that are available for that column. Tables for different applications and different purposes have different features and behavior. See the documentation for the specific application for details on using tables.

Chart Pane

When the **Chart** pane is included on a tab, it enables you to view charts that have been added and generated for the project. Each type of tab has different charts available. For example, the **Parameter Set** tab has a different set of charts than the DesignXplorer **Parameters Correlation** tab has.

For more information about using charts, see Working with the Chart Pane (p. 255).

Scene Pane

The **Scene** pane is available in the Fluent interface. To activate the **Scene** pane, enable **Show Solution Monitoring** from Fluent (right-click the **Solution** cell of a Fluent system to open the **Fluent Solution Monitor**). For more information, see Monitoring Fluent Solutions in Workbench in the *Fluent in Workbench User's Guide*.

Solution Information Pane

The **Solution Information** pane is available in the System Coupling or Fluent system's interface. For more information, see System Coupling User's Guide or Fluent in Workbench User's Guide.

Messages Pane

The **Messages** pane displays any messages you have, such as error and warning messages, solver messages, status messages, and so on.

During any update (cell, system, project, design point, multiple design point), any messages that are generated are displayed in the **Messages** pane. The **Messages** pane opens automatically if an error message occurs during an update, but in other situations you must open it manually.
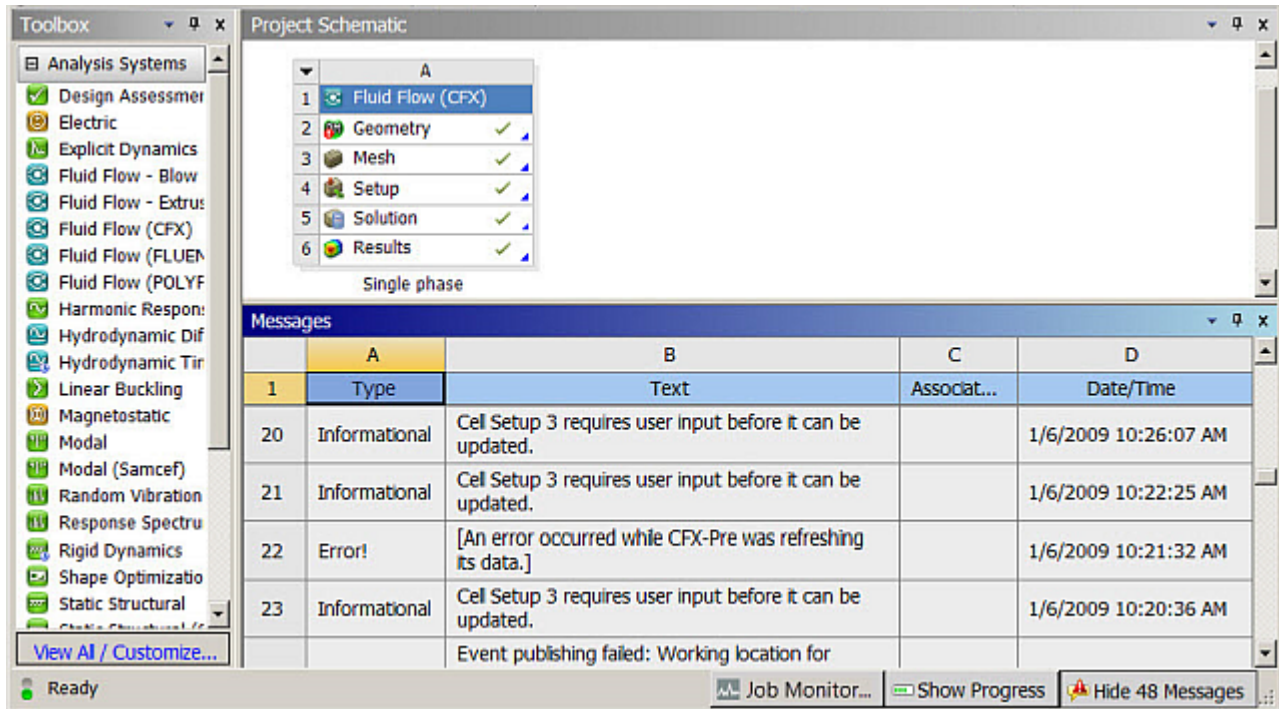
You can display or hide the **Messages** pane by clicking **Show Messages** or **Hide Messages** in the lower right corner of any Workbench tab.

Progress Pane

The **Progress** pane displays a progress bar during an update. You can display or hide the **Progress** pane by clicking **Show Progress** or **Hide Progress** in the lower right corner of the window. You can also interrupt an update process (p. 245).
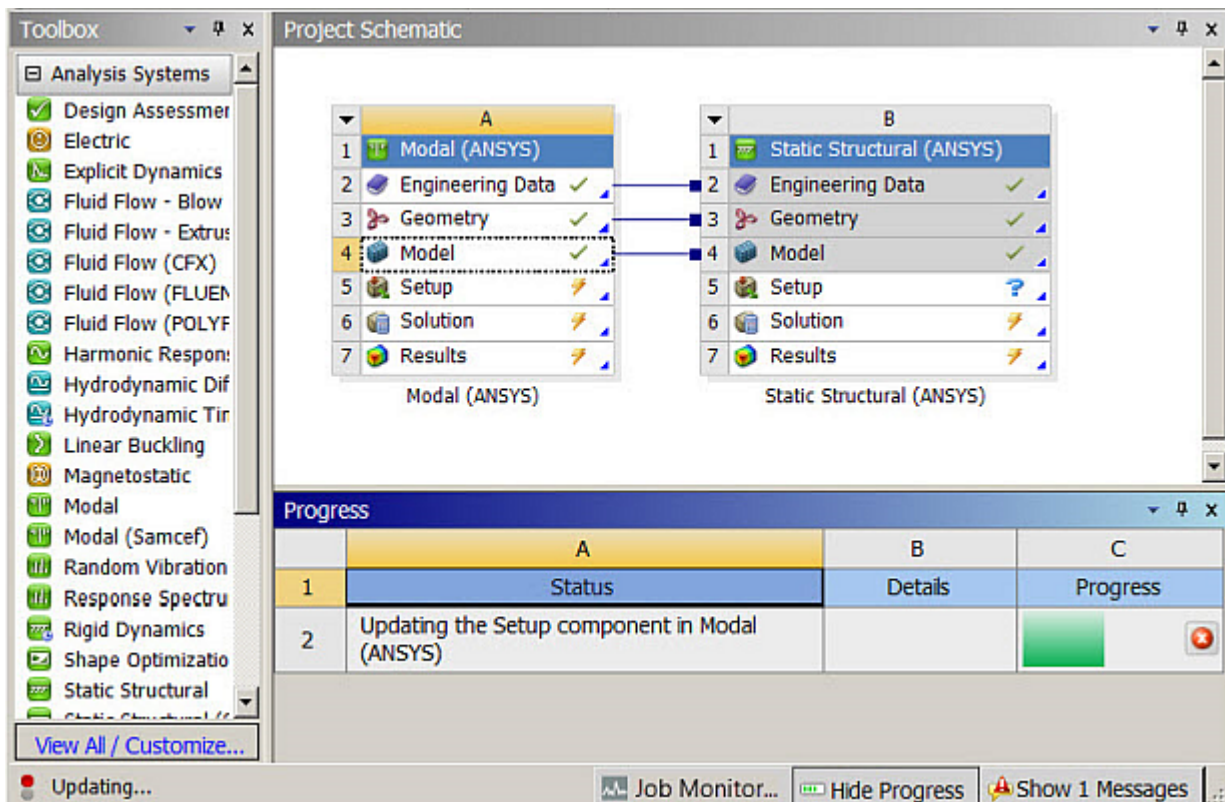
Sidebar Help

The **Sidebar Help** pane displays context sensitive help text and links. For more information on using this pane, see Using Help (p. 22).

Cells in Workbench

The systems that you add to the **Project Schematic** are made up of one or more cells.

Related topics:

Types of Cells

Understanding Cell States

Cell Properties

Types of Cells

The following common types of cells occur in many of the analysis and component systems available in Workbench; Other cell types may be available in certain systems; see the application-specific documentation in Working Through a System (p. 67) for these cell descriptions.

Engineering Data

Use the **Engineering Data** cell with Mechanical systems or the **Engineering Data** component system (p. 107) to define or access material models for use in an analysis. To define material data, open the **Engineering Data** tab by either double-clicking the **Engineering Data** cell or right-clicking the

cell and selecting **Edit** from the context menu. For more information, see Overview in the *Engineering Data User's Guide*.

Geometry

Use the **Geometry** cell to import, create, edit or update the geometry model used for analysis. Right-click the cell to access these functions in the context menu. The right-click options are context sensitive and change as the state of your geometry changes, so not all of the options are available at all times.

> **Note:**
>
> The SpaceClaim Direct Modeler features are available only if you have SpaceClaim Direct Modeler installed and the Ansys SpaceClaim Direct Modeler license available.

Model/Mesh

The **Model** cell in the Mechanical application analysis systems or the Mechanical Model component system is associated with the Model branch in the Mechanical application and affects the definition of the geometry, coordinate systems, connections and mesh branches of the model definition.

When linking two systems, you cannot create a share between the Model cells of two established systems. You can generate a second system that is linked at the **Model** cell of the first system, but you cannot add a share after the second system has been created. Likewise, you cannot delete a link between the **Model** cells of two systems.

The **Mesh** cell in Fluid Flow analysis systems or the Mesh component system (p. 202) is used to create a mesh using the Meshing application. It can also be used to import an existing mesh file.

Setup

Use the **Setup** cell to launch the appropriate application for that system. You define your loads, boundary conditions, and configure your analysis in the application. The data from the application is incorporated into the project in Workbench, including connections between systems.

Solution

From the Solution cell, you can access the Solution branch of your application, and you can share solution data with other downstream systems (for instance, you can specify the solution from one analysis as input conditions to another analysis). If you have an analysis running as a remote process, the Solution cell is in a Pending (p. 449) state until the remote process completes.

Results

The **Results** cell indicates the availability and status of the analysis results (commonly referred to as postprocessing). From the **Results** cell, you cannot share data with any other system.

Understanding Cell States

Workbench integrates multiple applications into a single, seamless project flow, where individual cells can obtain data from and provide data to other cells. As a result of this flow of data, a cell's state can change in response to changes made to the project. Workbench provides visual indications of a cell's state at any given time with icons on the right side of each cell.

Typical Cell States

State	Icon	Description
Unfulfilled		Required upstream data does not exist. Some applications may not allow you to open them with the cell in this state. For example, if you have not yet assigned a geometry to a system, all downstream cells appear as unfulfilled, because they cannot progress until you assign a geometry.
Refresh Required		Upstream data has changed since the last refresh or update. You may or may not need to update output data. When a cell is in this state, you can edit the cell, refresh the data, update upstream components, or update the cell.

The advantage to refreshing rather than updating a cell is that you are alerted to potential effects on downstream cells and make any necessary adjustments before you update it. This option is especially useful if you have a complex system in which an update could take significant time and computer resources. |
| Attention Required | | All of the cell's inputs are current. However, you must take a corrective action to proceed. To complete the corrective action, you may need to interact with this cell or with an upstream cell that provides data to this cell. Cells in this state cannot be updated until the corrective action is taken.

This state can also signify that no upstream data is available, but you can still interact with the cell. For instance, some applications support an "empty" mode of operation in which it is possible to enter the application and perform operations regardless of the consumption of upstream data. |
Update Required		Local data has changed and the output of the cell must be updated.
Up to Date		An update has been performed on the cell and no failures have occurred. It is possible to edit the cell and for the cell to provide up-to-date generated data to other cells.
Input Changes Pending		The cell is locally up-to-date but may change when next updated as a result of changes made to upstream cells.

Solution-Specific States

The **Solution** or **Analysis** cell for certain solvers can display the following solution-specific states.

State	Icon	Description
Interrupted, Update Required		Indicates that you have interrupted the solution during an update, leaving the cell paused in an update required state. This option performs a graceful stop of the solver, which completes its current iteration. Although some calculations may have been performed, output parameters are not updated. A solution file is written containing any results that have been calculated. The solve resumes with the next update command.
Interrupted, Up to Date		Indicates that you have interrupted the solution during an update, leaving the cell in an up-to-date state. This option performs a graceful stop of the solver, which completes its current iteration. Output parameters are updated according to the calculations performed so far, and a solution file is written. You can use the solution for postprocessing. For example, you can look at the intermediate result. Because the cell is already up-to-date, it is not affected by a design point update. To resume the solve, right-click and select **Continue Calculation** from the context menu.
Pending		Signifies that a batch or asynchronous solution is in progress. When a cell enters the pending state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell is not in an up-to-date state when the solution completes.

Note:

When up-to-date cells are connected to cells in a *different type of system*, the state of the up-to-date cells may change to Update Required. This behavior occurs because additional files have to be generated to satisfy the newly added system.

For example, assume that you have built a fluid flow system (Fluid Flow (CFX)) that is up-to-date.

Single phase

If you then connect the **Mesh** cell to a fluid flow system (Fluent), the mesh files for CFX Setup are the only files that will have been generated by the update process. After connecting another system to the up-to-date **Mesh** cell, the **Mesh** cell transitions to the update required state to signify that additional files must be generated for satisfy the Fluent system.

Single phase   FLUENT
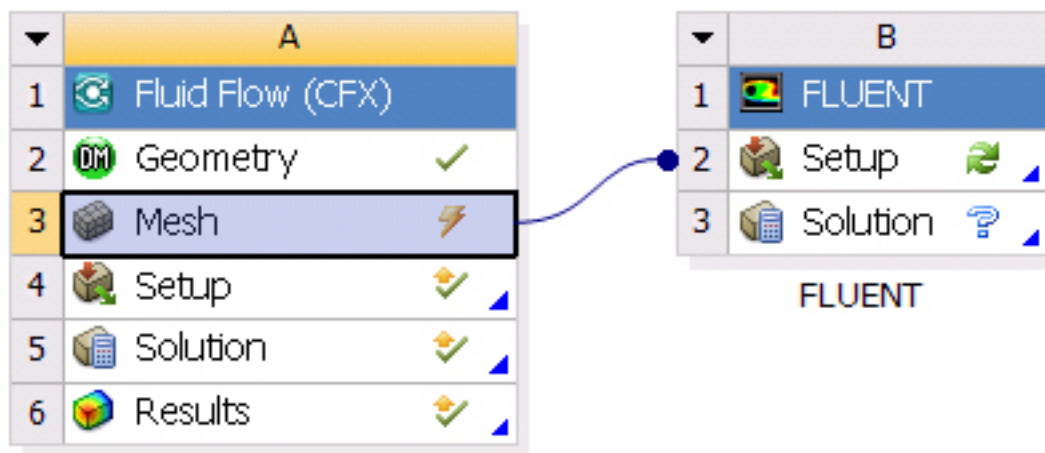
The cells already connected to the **Mesh** cell transition to the refresh required state after the **Mesh** cell is updated. Updating the project unnecessarily re-updates the previously up-to-date cells (CFX **Setup**, **Solution**, and **Results**). We recommended that you establish the connection to the Fluent system before updating components downstream of the Mesh in the Fluid Flow (CFX) system.

Failure States

If a particular action fails, Workbench provides a visual indication as well. You can pane any related error messages in the **Messages** pane by clicking **Show Messages** in the lower right portion of the window.

State	Icon	Description
Refresh Failed, Refresh Required		The last attempt to refresh cell input data failed and the cell remains in a refresh required state.
Update Failed, Update Required		The last attempt to update the cell and calculate output data failed and the cell remains in an update required state.
Update Failed, Attention Required		The last attempt to update the cell and calculate output data failed. The cell remains in an attention required state.

Cell Properties

Every cell in Workbench has various properties associated with it. Most of these properties are specific to the type of system, type of cell, and state of the cell, and are documented in the specific application documentation. However, some cell properties are common across many cells, regardless of type of system or cell state. Those properties are described here.

To pane cell properties, right-click the cell and select **Properties** from the context menu. The **Properties** pane opens, displaying all properties applicable to that cell in its current state. Both the property name and its current value are shown. Some properties are editable, while others cannot be edited because of the specific configuration of that project and/or cell.

Property	Description
Component ID	Displays the directory where any information associated with this component resides. For more information on the general Workbench directory structure, see Project Directories (p. 278).
Notes	Displays all system and cell notes (p. 241) created.
Last Update Used Licenses	Displays the license used by this component during the most recent update. If the component does not require a license during an update, the value displayed is *Not Applicable*.
Always Include in Design Point Update	Specifies that a component should be included in any design point update operation, even if it does not have output parameters. If you enable this property for a component, the component is updated when you update design points, regardless of whether parameter values are affected. Used for **CFD-Post** components.
Save Mesh Data In Separate File	Controls whether the mesh data is stored in a separate `.acmo` file, rather than being stored in the database file, when you save your project. Saving the mesh data in separate file reduces the overall file space required and reduces the possibility of disc corruption in very large database files on Linux systems. Used by the **Model** cell (non-fluid flow analyses such as **Static Structural**) and by the **Mesh** cell (**Fluid Flow** analyses and **Mesh** components). The default setting for this field is controlled by the **Save Mesh Data In Separate File** setting in the Workbench **Mechanical** (p. 430) or **Meshing** (p. 431) options.

Menus in Workbench

The following types of menus are available in Workbench:

Menu Bar

Context Menus

Menu Bar

The menu bar gives you access to the following menus.

File Menu

Menu Item	Description
New	Creates a new project (p. 48).
Open	Opens an existing project (p. 48).
Save	Saves the current project (p. 269).
Save As	Saves the current project (p. 269) under a different name and/or location.
Import	Imports a legacy database (p. 50).
Archive	Generates an archive file that contains all project files (p. 270).
Ansys Minerva	Provides options for connecting to and working with Ansys Minerva repositories (p. 393).
Scripting	Opens a sub-menu containing journalling and scripting options (p. 251).
Export Report	Writes out a report of the current project in .html/.htm format (p. 266).
Recently Viewed Files	Displays the four most recently opened projects.
Exit	Closes Workbench. You are prompted to save any unsaved data.

View Menu

Menu Item	Description
Refresh	Updates the pane.
Reset Workspace	Restores the current workspace layout to its default settings.
Reset Window Layout	Restores the original window layout.
List of panes	Displays the list of Workbench panes. You can select which panes you want to display. Any changes you make to panes are carried forward to subsequent Workbench sessions. For a detailed description of the individual panes listed here, see Panes within Tabs (p. 440).
Show Connections Bundled	Displays multiple links connecting systems as a single link where possible. A bundled connection includes a label indicating the shared cells (for example, *2-4* indicates that cells 2, 3, and 4 are shared between systems).
Show System Coordinates	Shows the alphanumeric column and row headings for each system. This option is selected by default.

Tools Menu

Menu Item	Description
Reconnect	Reconnects to updates that were pending when the project was closed. This option is available only if the project has cells in the Pending (p. 449) state. Upon opening a project, Workbench automatically retrieves the data for any completed background update. After reconnecting to pending solution data, it is important to save the project. For a **Solution** cell update, if you decline to save the project before exiting, the intermediate solver data is discarded and is not accessible in future Workbench sessions. For more information, see Exiting a Project During a Remote Solve Manager Solution Cell Update (p. 347).
Refresh Project	Refreshes all cells in the project (p. 245).
Resume	Resumes design point updates that were pending when the project was closed. This option is available only if the project has design points that are paused due to cells in a Pending (p. 449) state.
Update All Design Points	Updates all design points in a project that are in an Update Required (p. 449) state. This option is available only when the project contains multiple design points.
Update Project	Updates all cells in the project (p. 243) that are in an Update Required (p. 449) state.
Suspend Collecting RSM Results / Resume Collecting RSM Results	Temporarily suspends or resumes the collection of design point data from Remote Solve Manager (RSM) These options are enabled when RSM is processing an Update Project or an Update All Design Points operation. For more information, see Suspending and Resuming Collection of RSM Design Point Results (p. 361).
Abandon Pending Updates	Ignores any results calculated so far and return the project to its normal state. Use this option if you have a remote or background solve process (some component or components in the project were saved in a Pending state) and attempts to reconnect have failed. If you use this option, you may need to manually remove processes that were not stopped or files that were not removed.
Options	Defines your user preferences for Workbench. The preferences you set here are local settings and affect only you. For descriptions, see Workbench User Preferences (p. 417).

Units Menu

Menu Item	Description
Display Values as Defined	Displays the value and unit as defined in Workbench or the original source application. No conversion information is displayed.
Display Values in Project Units	Converts displayed units to correspond to the selected project unit system.
Unit Systems	Opens the **Unit Systems** dialog box, where you can choose which unit systems to display in the **Units** menu and define custom unit systems. For more information, see Configuring Units in Workbench (p. 26).

Extensions Menu

Menu Items	Description
ACT Start Page	Opens the ACT Start Page, which provides you with convenient access to ACT functionality. From this page, you can access multiple tools for both the development and execution of extensions.
Manage Extensions	Opens the Extension Manager, which enables you to specify which extensions you want to load.
Install Extension	Provides for installing a binary extension (WBEX file) into your `Application Data` folder so that this extension is available in the Extension Manager.
Build Binary Extension	Opens the binary extension builder so that you can compile a binary version of an extension from the scripted version.
View ACT Console	Opens the ACT Console, where you can interactively test commands during the development or debugging of an extension.
Open App Builder	Opens the ACT App Builder, where you can create ACT extensions in a visual environment.
View Log File	Opens the Extensions Log File, which provides messages generated by the extension. If present, warnings are shown in orange text, and errors are shown in red text.

Jobs Menu

Menu Item	Description
Open Job Monitor	Displays the **Job Monitor** dialog box, which enables you to see the status of jobs that have been submitted to Remote Solve Manager for the current project. For more information, see Monitoring and Controlling Remote Solve Manager Jobs in Workbench (p. 348).
Open the Design Point Service Web App	Opens a browser window and launches the DPS Web App (p. 367).
Sync with Design Point Service	Synchronizes (p. 369) the Workbench project with the Design Point Service.
Sign out from Distributed Compute Services	Signs you out from Distributed Compute Services (DCS).

Help Menu

Menu Item	Description
Ansys Workbench Help	Opens the online help (p. 24).
Show Context Help	Opens the **Sidebar Help** (p. 23) pane.
Installation Help	Opens the Ansys, Inc. Installation Guides.
Licensing Help	Opens the Licensing Documentation.
Getting Started	Opens the **Sidebar Help** (p. 23) pane to the Getting Started topic.

Menu Item	Description
Ansys Product Improvement Program	Opens the **Ansys Product Improvement Program** (p. 15) dialog box.
About Ansys Workbench	Opens the **About Ansys Workbench** dialog box containing various product details.

Context Menus

Context menu options provide capabilities that enable you to work with your existing systems or to add to and modify projects.

The following types of context menus are available in Workbench:

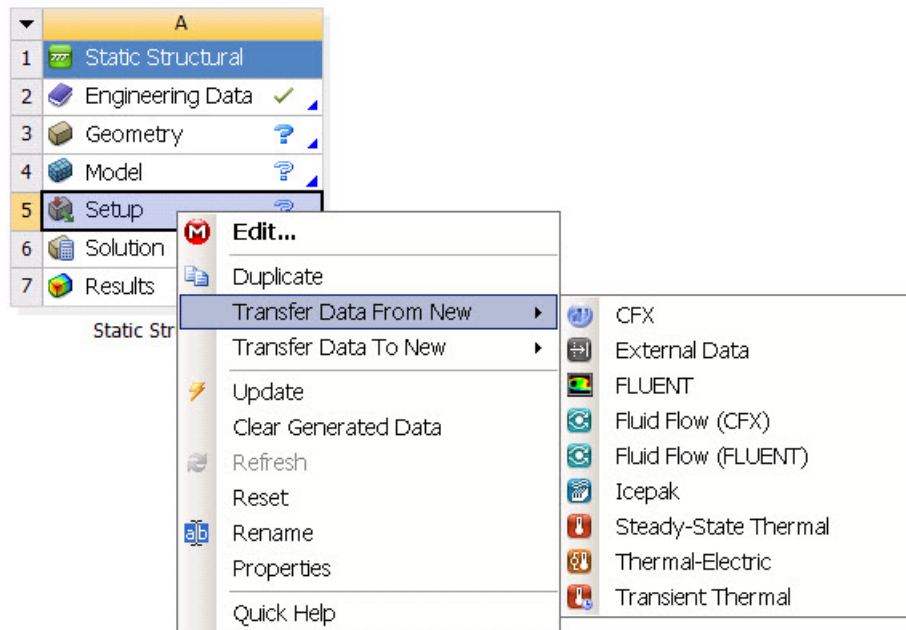## Common Context Menu Options

In addition to the menu items that are unique to each system or cell, some of the options on the context-sensitive menu are available with most systems and cells.

Menu Item	Description
Edit	Opens the cell workspace tab so you can review the data and edit as needed.
Duplicate	Creates a new system that is a duplicate of the selected system (p. 236).
Update	Refreshes input data and generates required output data (p. 242).
Update Upstream Components	Updates the components upstream of the cell without updating the cell itself (p. 244).
Rename	Renames the system or cell.
Refresh	Reads in all modified upstream data but does not necessarily regenerate the outputs of the cell (p. 245).
Quick Help	Displays a quick help panel for the cell, if available (p. 22).
Add/Edit Note	Displays an editable panel where you can enter notes about a system, cell, or project (p. 241).
Properties	Displays applicable properties in the **Properties** pane.
Recently Used	Lists all recently used files.
Clear Generated Data	Removes or erases any data that the cell has generated or is to generate and store if any such data is present (p. 247).
Reset	Removes or erases both input and output data to the cell and sets the cell state back to the default (p. 248).

Transfer Data Context Menu Options

Many cells include transfer data options in their context menu. Transfer data options are available only from root cells. If a cell is derived from an upstream cell (for example, when two systems share a geometry), you can transfer data only to/from the originating cell.

Menu Item	Description
Transfer Data from New	Creates an upstream system that can provide data to the selected cell (p. 64).
Transfer Data to New	Creates a downstream system that can accept data from the selected cell (p. 64).

Tab Context Menu Options

When you click cells in the **Project Schematic**, some of them open in separate tabs. The context menu options available depend on which tabs are open. For example, if only the **Project** tab is open, no context options are available. The following context menu options are available for tabs.

Menu Item	Description
Close Tab	Closes the selected tab. Available for all tabs except for the **Project** tab.
Close Other Tabs	Closes all other tabs if accessed from the **Project** tab. Closes all tabs except for the one selected and the **Project** tab if accessed from another tab. Available for all tabs when multiple tabs are open.
Close All	Closes all tabs (including the selected tab) except for the **Project** tab. Available for all tabs except for the **Project** tab.

System Header Context Menu Options

The following options are available from the context menu for system headers.

Menu Item	Description
Refresh	Refreshes all cells in the selected system (p. 245).
Update	Updates all cells in the selected system, along with any cells in upstream systems that provide data to them (p. 244).

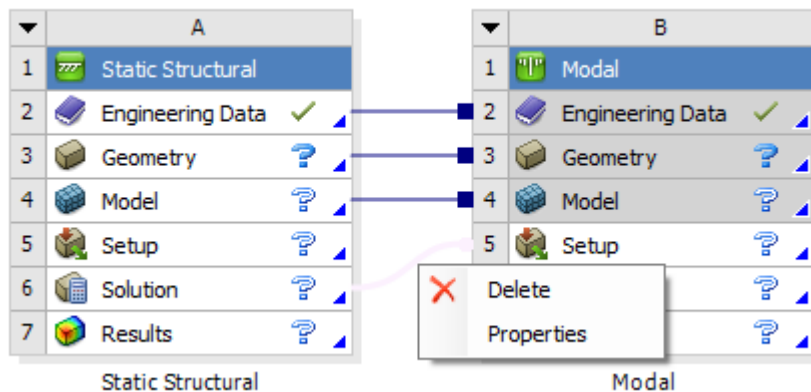| Menu Item	Description
Duplicate	Duplicates all cells (p. 236). No data is shared between the two systems. Equivalent to issuing a duplicate on the first cell of the system.
Replace with	Switches the analysis or solver type for an existing system (p. 241).
Delete	Deletes the system from the schematic (p. 240).
Recreate Deleted Cells	Creates new cells at the points where the previous cells were removed. Some cells can be deleted manually (for example the **Results** cell in a Fluid Flow system). in some cases, transferring data from one system replaces the data that was previously consumed from upstream cells in the same system, and so those upstream cells are deleted.
Rename	Renames the system (p. 235).
Properties	Displays system properties in the **Properties** pane.
Add/Edit Note	Displays an editable panel where you can enter notes about a system, cell, or project (p. 241)

Project Schematic Context Menu Options

Right-click in white space in the **Project Schematic** to select a context menu option for the project. The options that are visible depend on the specific configuration of the project.

Menu Item	Description
Resume	Resumes design point updates that were pending when the project was closed. This option is available only if the project has design points that are paused due to cells in a Pending (p. 449) state.
Refresh Project	Refreshes all cells in the project (p. 245).
Reconnect	Reconnects to updates that were pending when the project was closed. This option is available only if the project has cells in the Pending (p. 449) state. Upon opening a project, Workbench automatically retrieves the data for any completed background update. After reconnecting to pending solution data, it is important to save the project. For a **Solution** cell update, if you decline to save the project before exiting, the intermediate solver data is discarded and is not accessible in future Workbench sessions. For more information, see Exiting a Project During a Remote Solve Manager Solution Cell Update (p. 347).
Update All Design Points	Updates all design points in a project (p. 309) that are in an Update Required (p. 449) state. This option is available only when the project contains multiple design points.
Update Project	Updates all cells in the project (p. 243) that are in an Update Required (p. 449) state.
Add to Custom	Adds systems to custom template (p. 228).
New *system type*...	Select a new type of system to add to the project (p. 58).
Show Connections Bundled	Displays multiple links connecting systems as a single link where possible. A bundled connection includes a label indicating the shared cells (for example, *2-4* indicates that cells 2, 3, and 4 are shared between systems).

Menu Item	Description
Show System Coordinates	Shows the alphanumeric column and row headings for each system. This option is selected by default.
Fit	Resizes the systems in the **Project Schematic** to better fit in the window as it is currently sized.
Add/Edit Note	Displays an editable panel where you can enter notes about a system, cell, or project (p. 241).

Link Context Menu Options

You can right-click a link to see additional context menu options.

Menu Item	Description
Delete	Deletes the selected link (p. 65).
Properties	Opens the **Properties** pane for the link, where you can see the link type, origination cell, and destination cell. **Note:** To properly update the state of a cell for a linked system, either open the Mechanical application or click **Update Project** on the toolbar.

Geometry Context Menu Options

The following options are available from the context menu for a **Geometry** cell.

Menu Item	Description
New Geometry (or New DesignModeler Geometry/New SpaceClaim Direct Modeler Geometry)	Launches DesignModeler or SpaceClaim Direct Modeler, where you can build a new geometry.

Menu Item	Description
Import Geometry	Opens a dialog box where you can navigate to an existing geometry file or select a file from the list of recently viewed files.
Edit (or Edit Geometry in DesignModeler/Edit Geometry in SpaceClaim Direct Modeler)	Opens the model in DesignModeler or SpaceClaim Direct Modeler to modify it.
Replace Geometry	Opens a dialog box where you can navigate to an existing geometry file or select a file from the list of recently viewed files to replace the currently specified file.
Update from CAD	Generates an existing CAD geometry using the parameter values as defined in the CAD system.
Refresh	Reads in all modified upstream data but does not regenerate the geometry. Enabled when the Geometry cell is in the Refresh Required state.
Properties	Displays a **Properties** pane where you can select basic and advanced geometry properties. For property descriptions, see Geometry Preferences.

Setup Context Menu Options

The following options are available from the context menu for a **Setup** cell.

Menu Item	Description
Edit	Launches one of the following options: • Mechanical application with the geometry loaded and with cells mapped to their respective tree locations in the Mechanical application. • CFX-Pre.
Import Case	Imports either an existing Mechanical case file containing physics data, region, and mesh information for your analysis, or an existing Fluent case file.

Solution Context Menu Options

The following options are available from the context menu for a **Solution** cell.

Menu Item	Description
Edit	Launches one of the following options: • Mechanical application open to the solution branch. • CFX-Pre. • Fluent.

Menu Item	Description
Delete	Deletes the **Solution** and **Results** cell. This changes the system to a setup-only system, meaning the system generates only an input file. It does not solve or post results. The solution object and objects below it are removed from the Mechanical application tree.
Import Solution	Displays the most recent CFX-Solver results files imported (if any) and specify the CFX-Solver results file to load in the **Open** dialog box. When the results file is loaded, the system displays only the **Solution** cell and the **Results** cell.
Display Monitors	Opens the Ansys CFX-Solver Manager and shows the results of the previous run.
Import Final Data	Selects an existing Fluent data set (for example, one solved on an external cluster) into a **Solution** cell in a Fluent system and immediately start postprocessing in CFD-Post, without the need to run the minimum of one more solver iteration. This option becomes available after importing case file into the **Setup** cell.

Glossary

analysis system
A template that has all the cells required to complete an analysis for a particular type of physics, such as static structural.

archive
Save all project files and data into a single package that can be shared, stored, and reused.

cell
A part of a system that represents a discrete task in the process of completing the overall analysis. Typical cells include **Engineering Data**, **Geometry**, **Model** or **Mesh**, **Setup**, **Solution**, and **Results**.

component system
A system template that has the cells necessary to complete only a portion of a complete analysis. Often these systems are task-oriented (for example, a system to create a geometry or to produce a mesh) or are associated with a particular application.

component update
An update of a single component (or cell) within a system in an Ansys Workbench project. For example, an update of the **Analysis** cell within a project is a component update.

A **Solution** cell update is the only component update currently supported by Remote Solve Manager.

context menu
A context-sensitive list of options available from a cell or other component, accessed by a right-mouse click.

custom system
A system template that has all of systems necessary to complete a coupled analysis, such as FSI or thermal-stress.

data-integrated application
An application that has a separate interface from the Ansys Workbench project window but that still communicates with Ansys Workbench.

Design Exploration
Systems in the Design Exploration group are used to access DesignXplorer functionality. These systems connect to the Parameter Set bar in order to drive parametric studies by varying project parameters.

design point
A set of input parameter values and corresponding output parameter values that make up a single solved instance of a project.

design point update
An update of one or more design points in an Ansys Workbench project. You can update a single design point, a selected set of design points, or all the design points in a project. During a design point update, solution data is updated only where output parameters have been defined.

Design point updates can be submitted to Remote Solve Manager.

downstream
In the Project Schematic, data flows from top-to-bottom within systems and from left-to-right between systems. A cell is said to be

---

downstream if it is below a cell in the same system, or to the right of a cell in a separate system. Downstream cells use data from upstream cells as input.

links
Systems in the Project Schematic that are dependent on each other in some manner are connected with links. Links with a square terminator indicate that data is shared between the two cells connected by the link, and links with a round terminator indicate that data is transferred from the upstream to the downstream cell.

parameter
A property of a model that can be varied (input parameter) or the result of varying such a property (output parameter). Custom input or custom output parameters can be defined by a constant value; derived parameters are defined by an expression of other parameters (for example, "P2+3*P3").

project
The project is the full collection of systems, components, data, and their connections that you create to achieve an overall CAE goal.

Project Schematic
A region of the Ansys Workbench project window where you will construct and interact with your project. Projects are represented as connected systems displayed in a flowchart form that allows engineering intent, data relationships, and the state of the analysis project to be understood at a glance.

project update
An update of an entire Workbench project. All the systems, components, and design points in the project are updated.

refresh
An action that reads in all modified upstream data but does not necessarily regenerate the outputs of the cell.

systems
A collection of cells that together perform a dedicated task. Types of systems include analysis systems, component systems, and custom systems.

Toolbox
A region of the Ansys Workbench interface, located on the left side of the interface, from which you can choose systems or other components to add to the project. The Toolbox is context-sensitive, meaning the options that appear will change based on what is selected elsewhere in the interface.

update
An action that updates data in a particular system or cell with any new information that has been added to the project since the last update and regenerates the outputs.

upstream
In the **Project Schematic**, data flows from top-to-bottom within systems and from left-to-right between systems. A cell is said to be upstream if it is above a cell in the same system or to the left of a cell in a separate system. Upstream cells provide their output data to downstream cells.