Introduction to Hydrodynamic Analysis with Ansys Aqwa

Module 02: Hydrodynamic Analysis in Ansys Workbench

Release 2021 R2

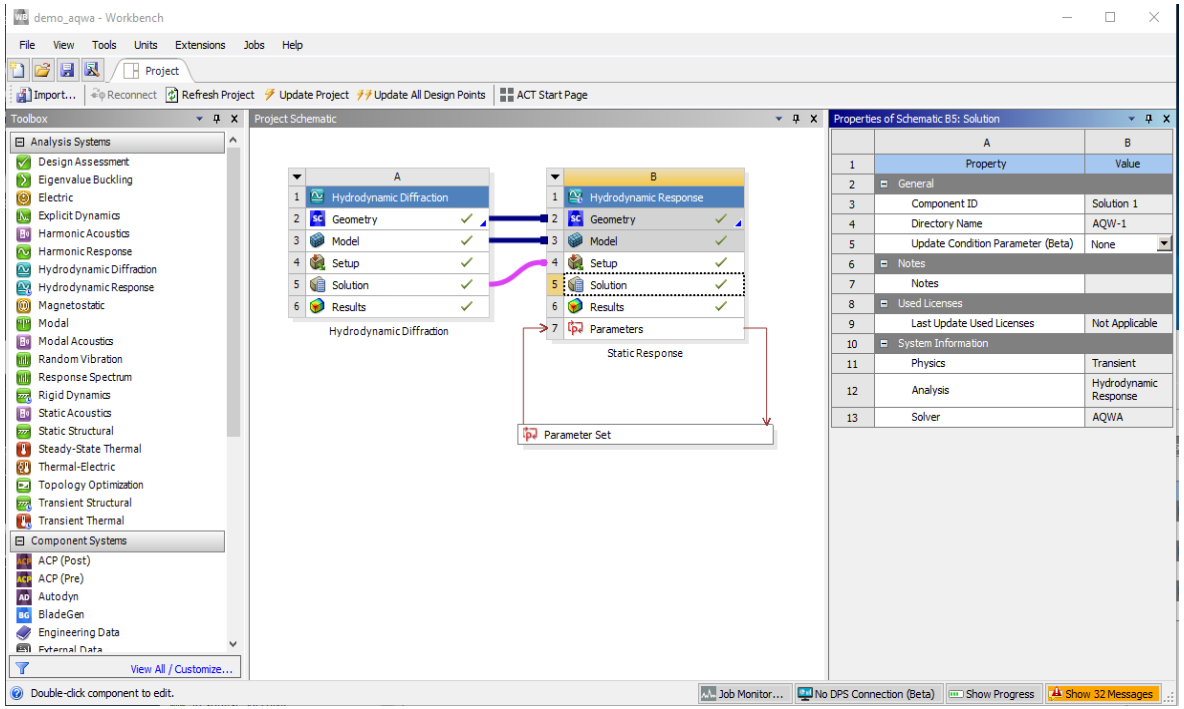
Ansys Workbench

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

Workbench handles the passing of data between Ansys Geometry/Mesh/Solver/Post-processing tools.

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

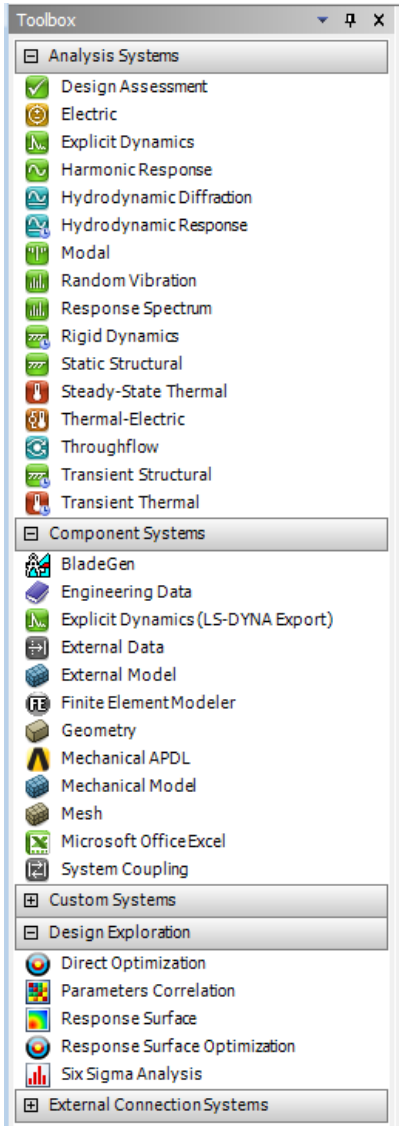
Because Workbench can manage the individual applications AND pass data between them, it is easy to automatically perform design studies (parametric analyses) for design optimization.



Workbench Overview

The options visible in the left-hand column show all of the products (systems) that you have licenses for.

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

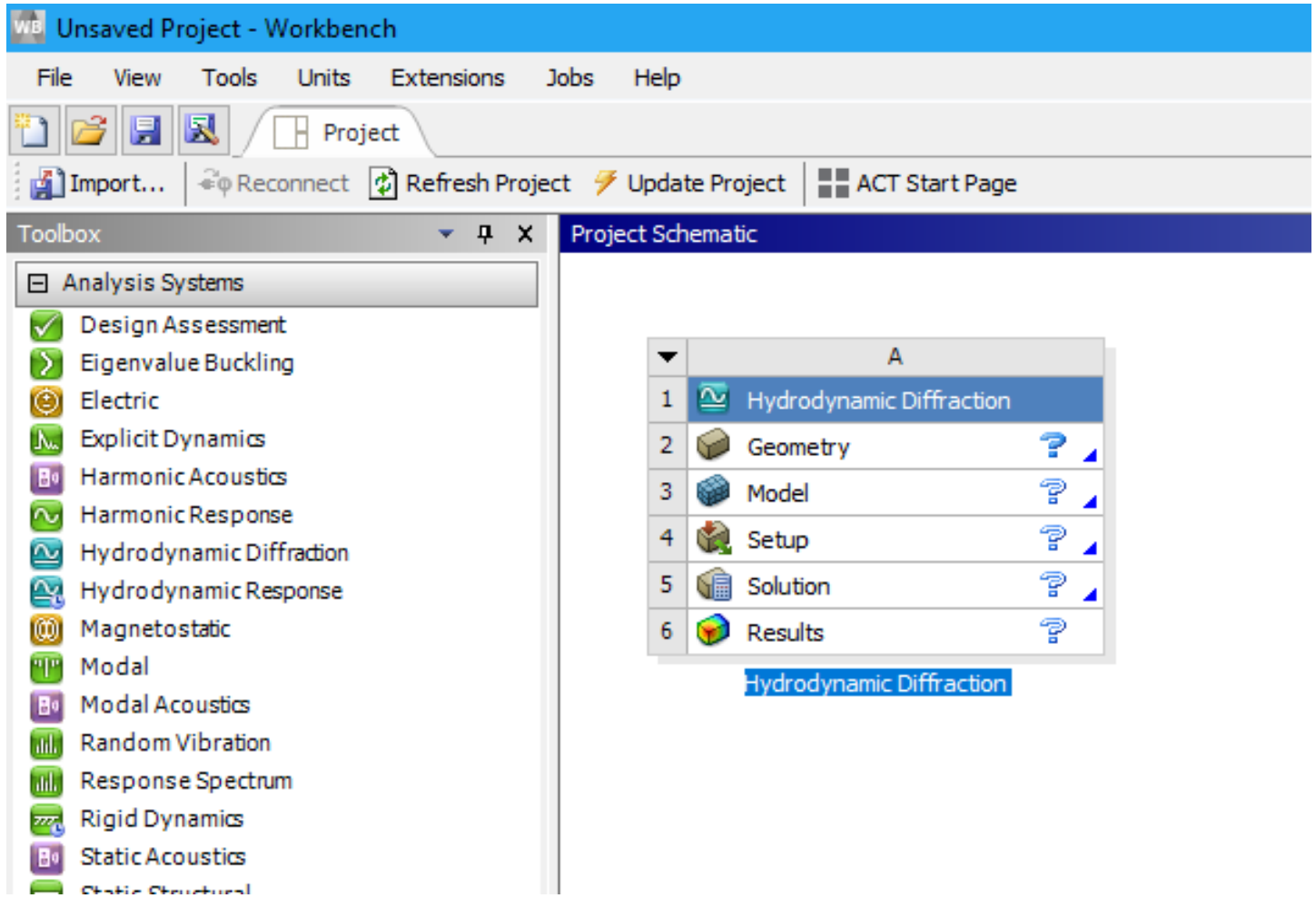


Analysis Systems are ready-made stencils that include all the individual systems (applications) needed for common analyses (for example *Geometry + Mesh + Solver + Post-Processor).*

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

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

Basic Workflow



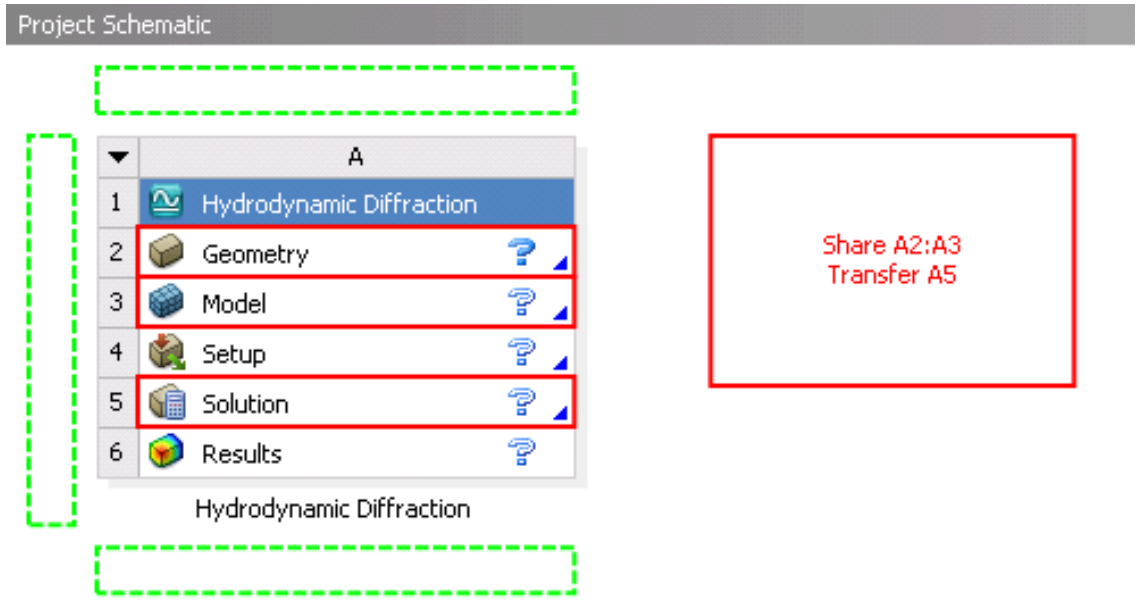
Dragging an Analysis System onto the project desktop lays out a workflow, comprising all the steps needed for a typical analysis.

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

Basic Workflow

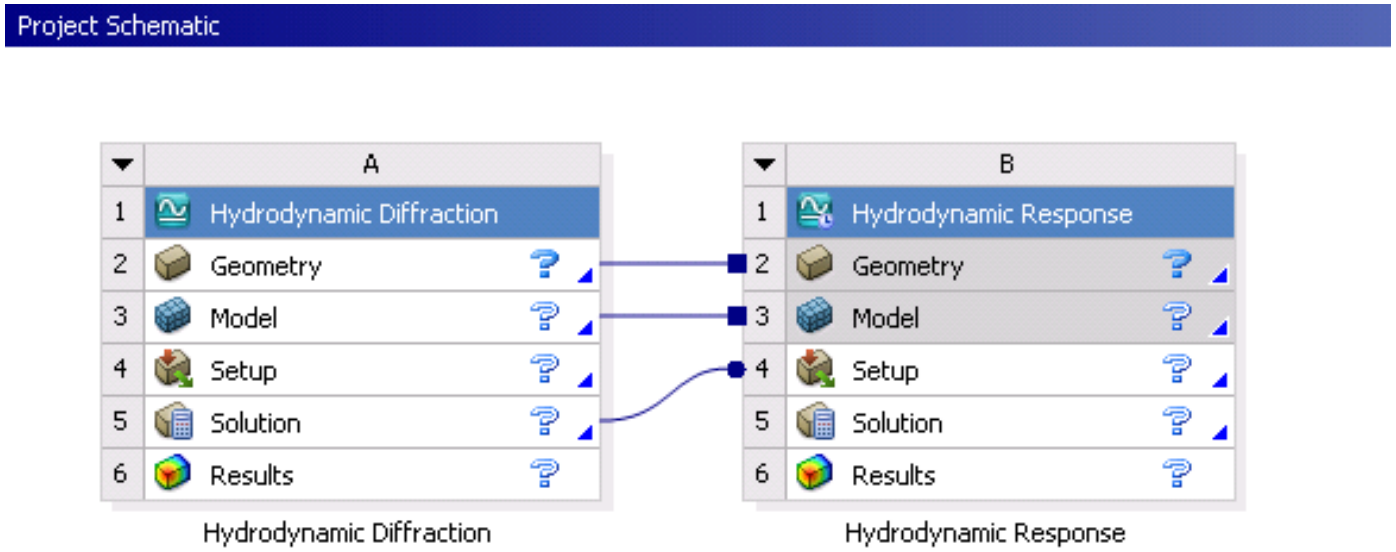
- By dropping applications and/or systems into various locations in the schematic, an overall analysis project is defined.
- “Connectors” indicate the level of collaboration between systems.
- In the example below a hydrodynamic response system is dragged and dropped onto a Hydrodynamic Diffraction system at the Solution cell (A5).

- Before completing the operation, notice there are a number of optional “drop targets” that will provide various types of linkage between systems.



Basic Workflow

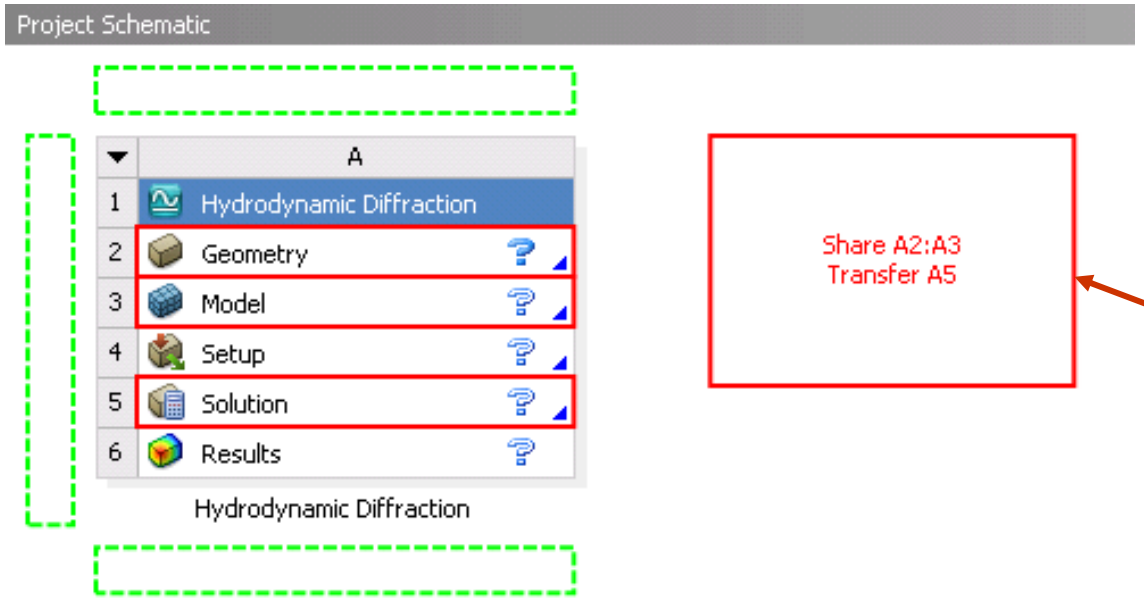
- By completing the operation from the previous page, we have linked the Solution of a Hydrodynamic Diffraction system to the Setup of a Hydrodynamic Response system.
- In this way we have coupled the hydrodynamic database so that it can be used for a subsequent frequency or time domain analysis.



- Each system block is given an alphabetic designation (A, B, C), and each cell is numbered for reference.

Basic Workflow

- Make sure to drop the Hydrodynamic Response system on to the correct target: without the linkage between the Hydrodynamic Diffraction Solution and Hydrodynamic Response Setup cells, there would be no hydrodynamic database coupling.

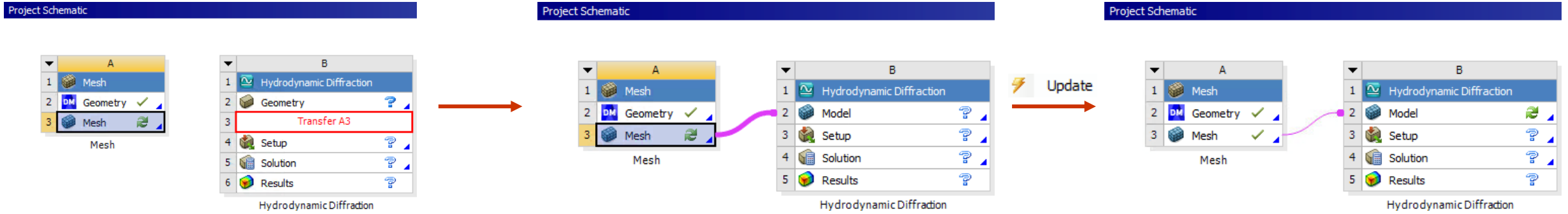


The candidate “drop target” indicates that data will be shared from fields A2 to A3 (Geometry and Model) and transferred from A5 (Solution).

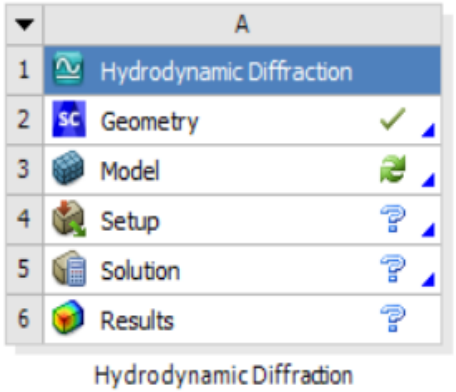
Basic Workflow with Ansys Mesh

- The Hydrodynamic Diffraction system includes a simple Meshing capability.
- For greater flexibility, link a Mesh component (from the Component Systems toolbox) to the Model cell of a Hydrodynamic Diffraction system.
- Set the Mesh object **Physics Preference** to *Hydrodynamics* for the best results.

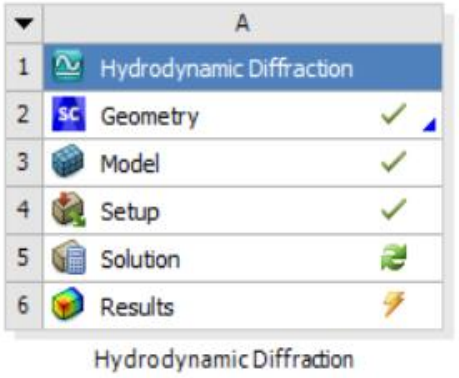
- **Important:**
  - Do not share the Geometry between the Mesh and Hydrodynamic Diffraction systems.
  - Drag-and-drop from the Toolbox does not work in this context: create the systems separately, then drag the Mesh cell on to the Hydrodynamic Diffraction Model cell.
  - Once you have set up the mesh properties in the Meshing app, right-click > Update on the Mesh cell to ensure that the mesh data is transferred to the Hydrodynamic Diffraction system.



Cell States



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



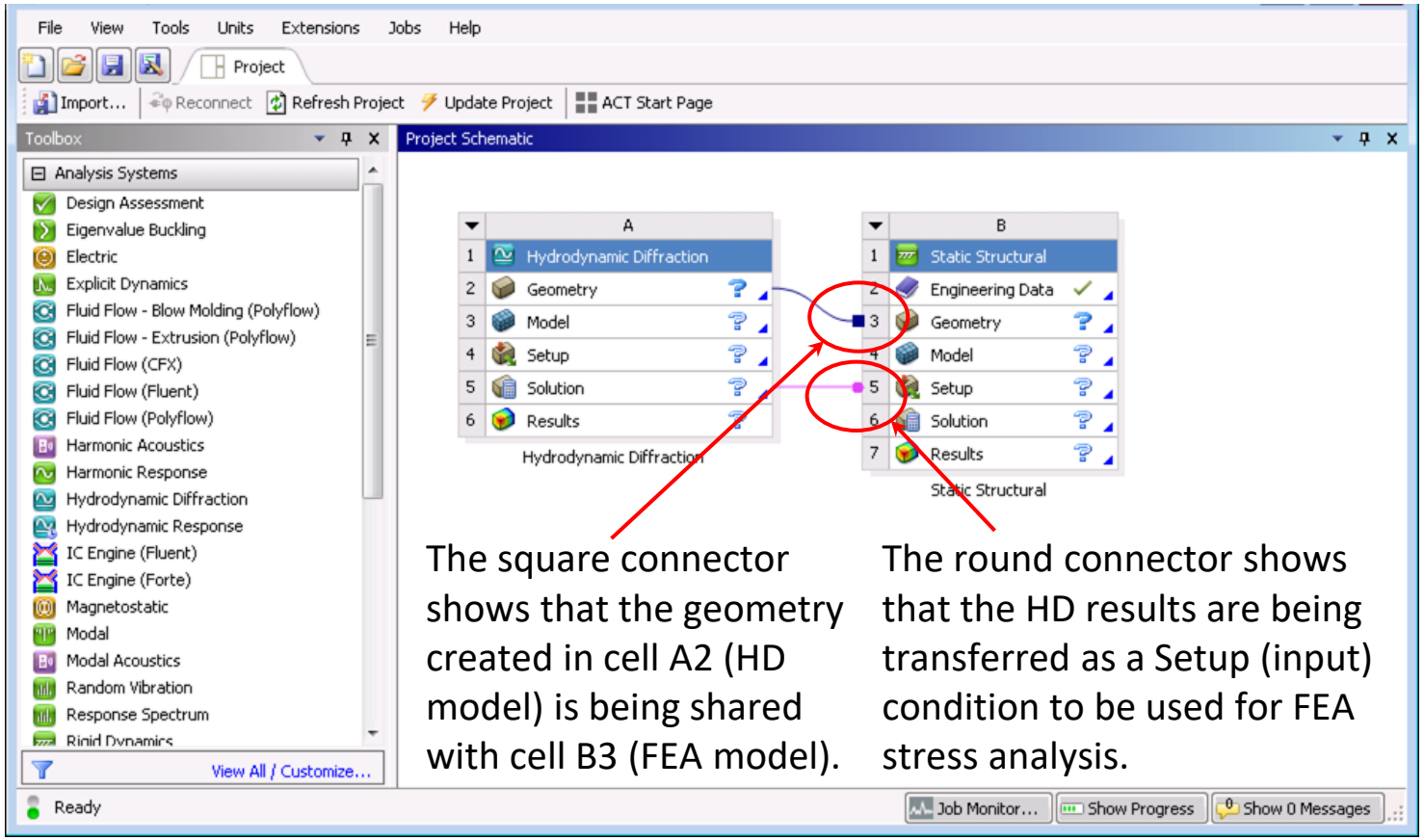
Status after model has been set-up, waiting for solving

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

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

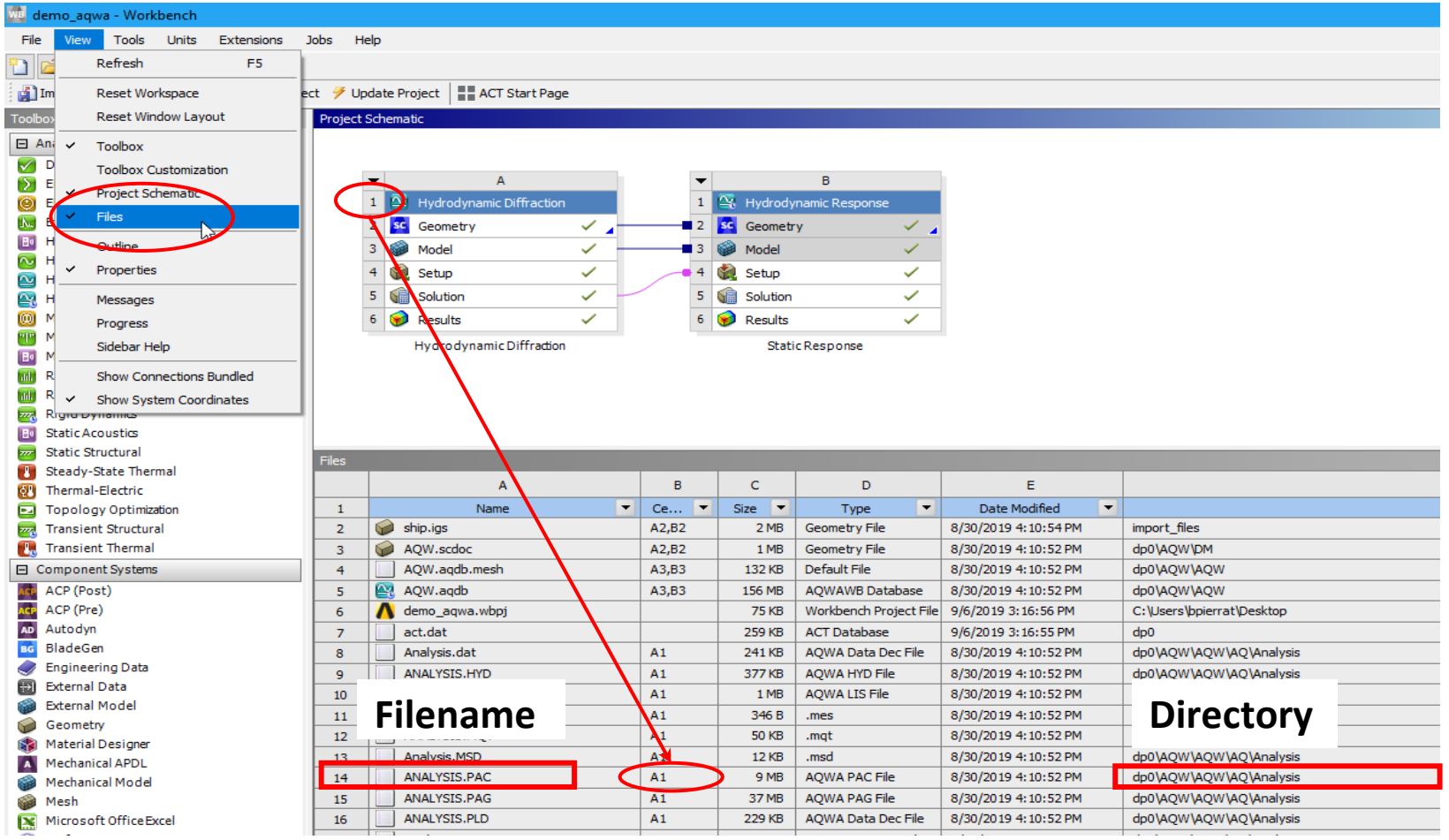
Sharing Data between Different Solvers

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



File Location on Disk

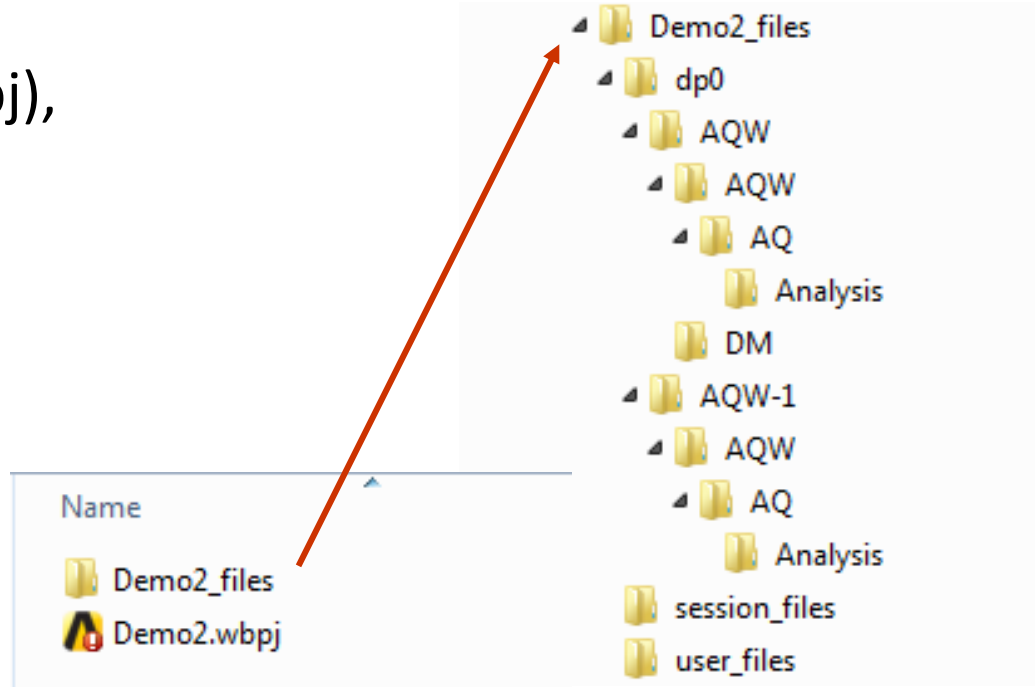
- Should you need to identify the individual files on your disk for each stage of the project, these can be found by enabling View > Files. The resulting table will cross-reference the directory and filename with the project cells.



File Management

- Workbench creates a project file and a series of subdirectories to manage all associated files.
- Users should allow Workbench to manage the content of these directories. Please do not manually modify the content or structure of the project directories!

- When a project is saved a project file is created (.wbpj), using the user specified file name (e.g. Demo2.wbpj).
- A project directory will be created using the project name. In the above example the directory would be Demo2_files.
- A number of subdirectories will be created in the project directory, as shown.



Workbench File Management

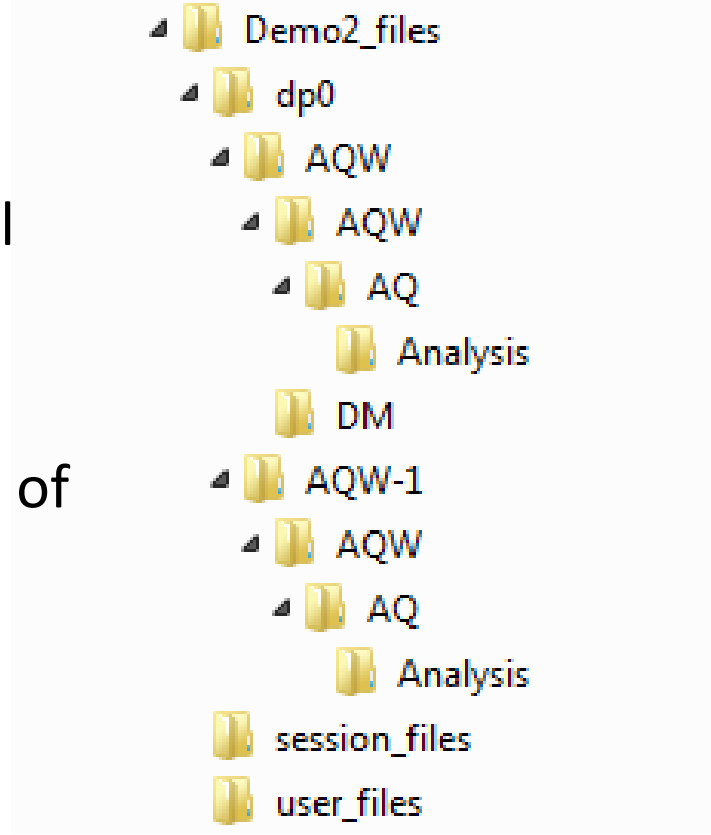
dp*n*: This is the design point directory, which is essentially the state of all parameters for a particular analysis. In the case of a single analysis (no parameterized values) there will be only one “dp0” directory.

AQW-n: Contains subdirectories for each system in the project.

In the example below the “AQW\AQW\AQ\Analysis” directory will contain the hydrodynamic database, and other associated files, from the Aqwa Hydrodynamic Diffraction system.

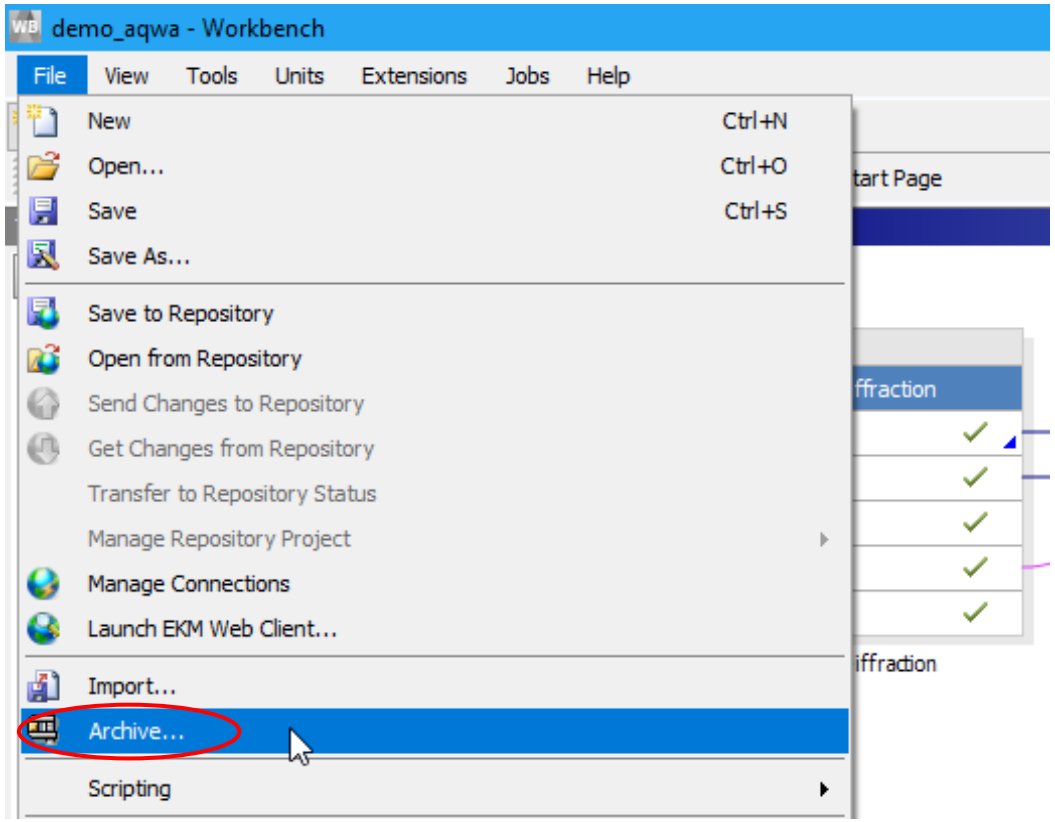
The “AQW-1\AQW\AQ\Analysis” directory will contain the results of the Hydrodynamic Response system.

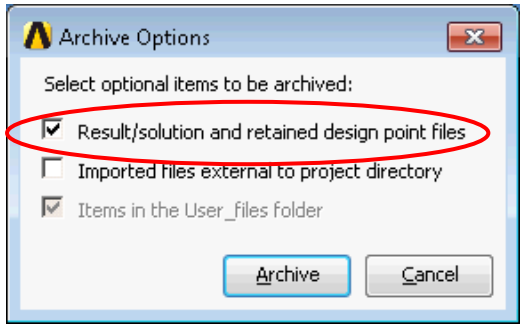
user_files: Contains external user-defined files that may be associated with a project. The user is free to use this directory as desired.



Project Archives

The Workbench project comprises many files and directories. If you need to either archive the project or bundle it to send to us for a Technical Support query, use the 'Archive' tool. This generates a single zipped file of the entire project, saved in a .wbpz format.





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

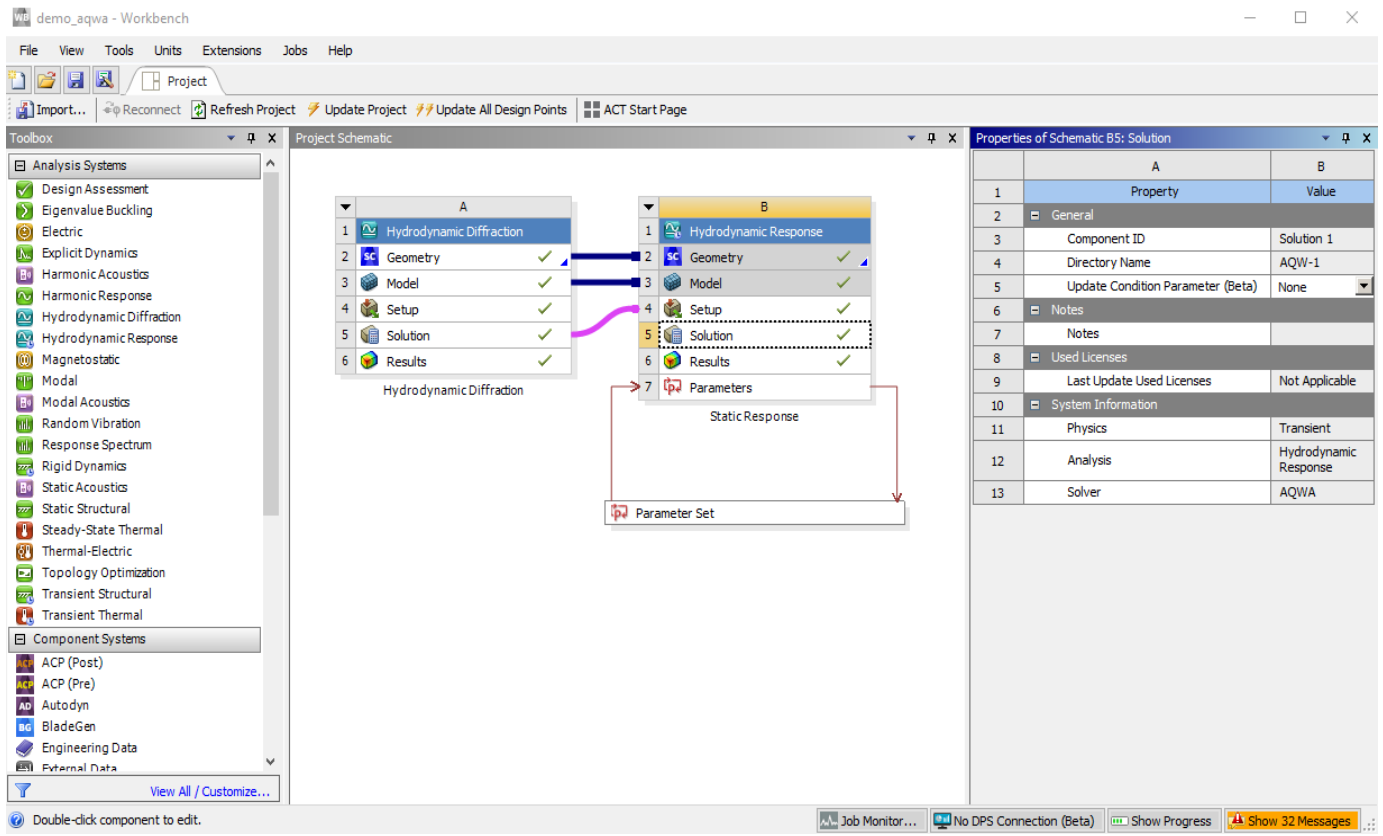
An existing Workbench archive can be restored via File > Open.

Working With Parameters

Most Workbench applications will let you specify key quantities as a parameter (rather than a constant).

In this example:

- When defining the environmental loading for an Aqwa stability analysis, the ocean current speed is set to be an *input parameter*.
- When reviewing the results, the final structure X position is set as an *output parameter*.



Clicking on “Parameter Set” allows us to set up the input parameter values.

The whole process is automated; Workbench will recursively:

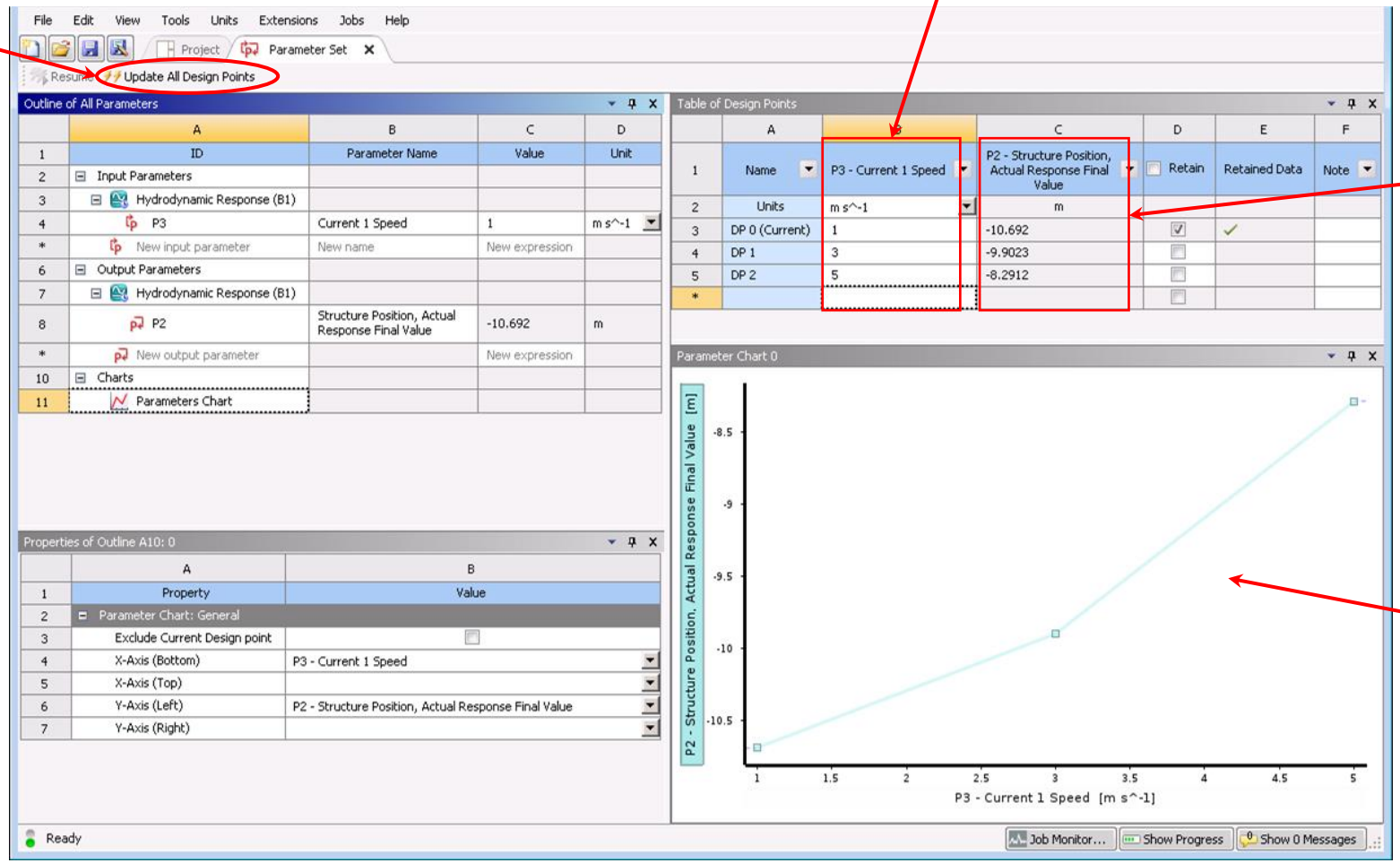
- Update the environmental data (input), based on the values defined in the Parameter Set
- Re-solve the Hydrodynamic Response system and evaluate the required results (output)

The user just needs to sit back and wait (or go home for the evening).

Working With Parameters

Create new rows in the Table of Design Points for each case (in this example, 3 values of Current Speed)

Click ‘Update All Design Points’ to compute over all input values



Requested output values are shown here (structure X position)

The relationship between inputs and outputs can be displayed graphically

Summary

- Ansys Workbench is a convenient way of managing your simulation projects.
- Workbench is used to launch the individual software components, and to transfer data between them.
- It is easy to see at-a-glance how a model has been built, and to determine which files were used for a particular simulation (pairing geometry files to solver runs).
- Workbench also makes it straightforward to perform parametric analyses (without the user needing to manually launch each application in turn) and makes it easy to simulate multi-physics scenarios like fluid-structure interaction.