Lecture 1: Introduction

Introduction to Hydrodynamic Analysis with ANSYS Aqwa

ANSYS Release 19.2

Welcome

Welcome to the *ANSYS Hydrodynamic Analysis* introductory training course!

This training course covers the basics of using ANSYS Aqwa for performing hydrodynamic analyses.

It is intended for all new or occasional ANSYS Aqwa users, regardless of the CAD software used.

Agenda (Day 1)

Morning	Lecture 1 – Introduction
	Lecture 2 – Aqwa Basics – Hydrodynamic Diffraction
	Workshop 2.1 – Ship Hydrodynamic Diffraction
Afternoon	Lecture 3 – Aqwa Basics – Hydrodynamic Response
	Workshop 3.1 – Ship Hydrodynamic Response
	Lecture 4 – Articulations and Fenders
	Workshop 4.1 – Aqwa Articulation – FPSO and Turret

Agenda (Day 2)

Morning	Lecture 5 – Fixed Structures and Multi-Body Interaction
	Workshop 5.1 – Ship and Pier Hydrodynamic Interaction
	Lecture 6 – Slender Body Modelling and Drag Linearization
	Workshop 6.1 – Truss Spar Including Drag Linearization
Afternoon	Lecture 7 – Aqwa/Mechanical Load Mapping
	Workshop 7.1 – Load Mapping

Lecture 1: Contents

- About ANSYS Inc.
- ANSYS Customer Portal
- ANSYS Workbench Overview

About ANSYS Inc.

PROVEN 40+ years

Validating our solutions on the most advanced product applications

MARKET LEADER

Long-term growth, financial stability and CAD agnostic

DEDICATED

2,900+ employees

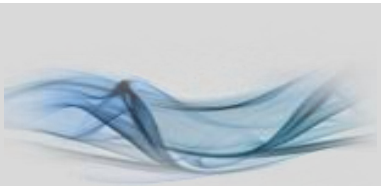
75+ locations

40 countries

FOCUSED

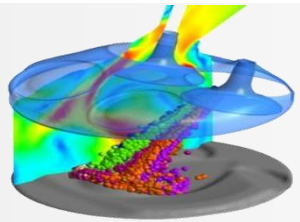
This is all we do.
Leading product technologies in all physics areas
Largest development team focused on simulation

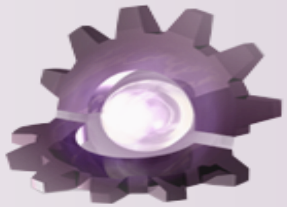
Breadth of Technologies



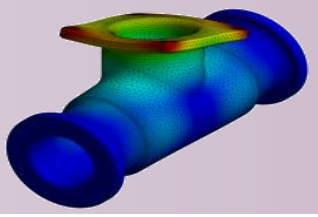
Fluid Mechanics
From Single-Phase Flows…

To Multiphase Combustion

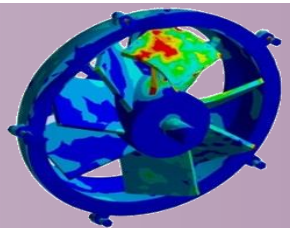


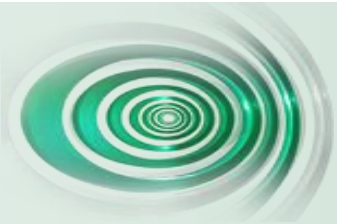


Structural Mechanics
From Linear Statics...

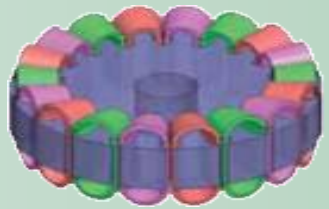


To High-Speed Impact

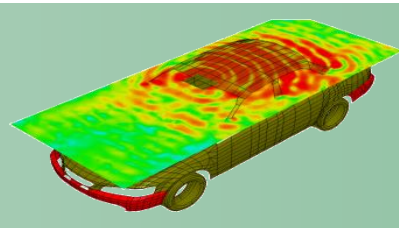




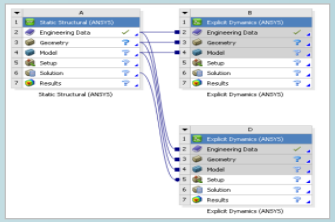
Electromagnetics
From Low-Frequency Windings…



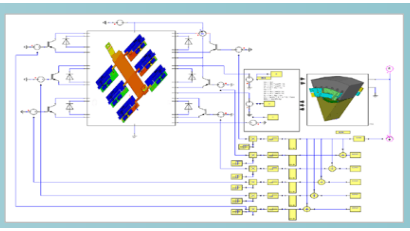
To High-Frequency Field Analysis



Systems
From Data Sharing…



To Multi-Domain System Analysis



The ANSYS Customer Portal

support.ansys.com

Contains over 50,000 support assets powered by a modern web user interface and powerful search engine.

Support	Products	Downloads	Knowledge Resources
Classroom Training Webinars Service Requests	Product Assets	Latest Release Updates Tools Previous Release(s)	Solutions Conference Proceedings Class3 Reports Documentation Training & Tutorials

About Search

The ANSYS Customer Portal's search is powered by dedicated Google® hardware.

Mesh = Meshed = Meshing

Export = Exported = Exporting

Example:
You want a meshing tutorial for ANSYS Meshing and your search has results for other products that are not of interest to you; by selecting the product facet "ANSYS Meshing" you can narrow down your results further.

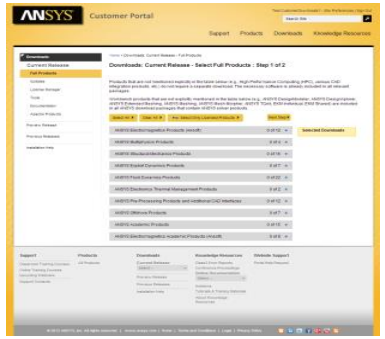
Product

- ANSYS Fluent (5385)
- ANSYS Mechanical APDL (4092)
- ANSYS CFX (2635)
- Other (2494)
- ANSYS ICEM CFD (2028)
- ANSYS Polyflow (1333)
- ANSYS Icepak (1147)
- ANSYS Meshing (1016)
- ANSYS TurboGrid (734)
- ANSYS CFD-Post (287)
- ANSYS Autodyn (253)
- ANSYS Structural Mechanics (140)
- ANSYS Aqwa (102)

Support, Downloads and Training

Submit and review service requests

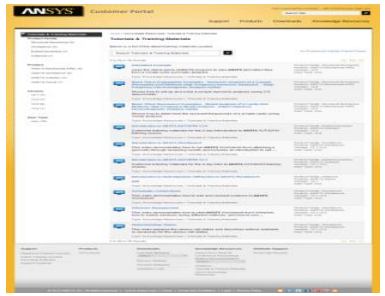
If you cannot find the answer to your question within the ANSYS Customer Portal then you can submit a service request. A member of ANSYS technical support will then get back to you with advice or a solution.



Download the latest software and updates

Download ISO images if you wish to create a DVD which is recommended for installations on multiple computers and allows you to keep an archive of the installation for later re-use.

Package downloads can also be selected if you want to install files directly.



Download classroom and video training material

Training and tutorial material are available for both a broad range of ANSYS products and user's experience. Search the hundreds of courses available and improve your knowledge of ANSYS software.

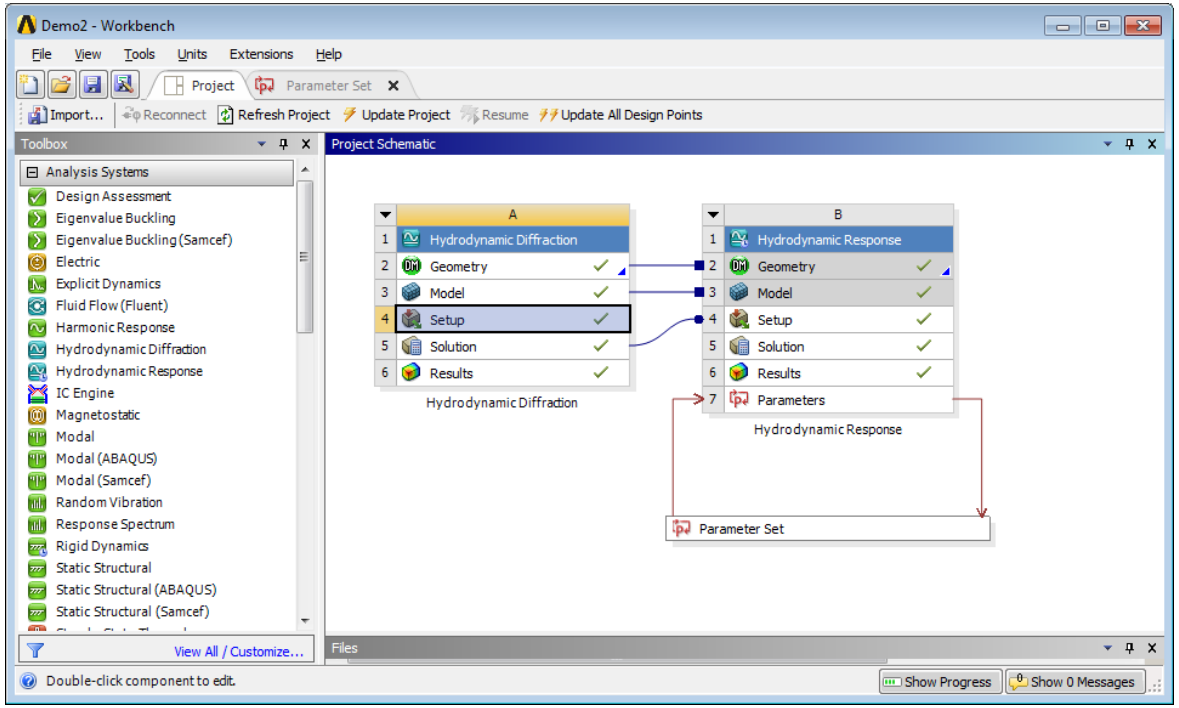
ANSYS Workbench

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

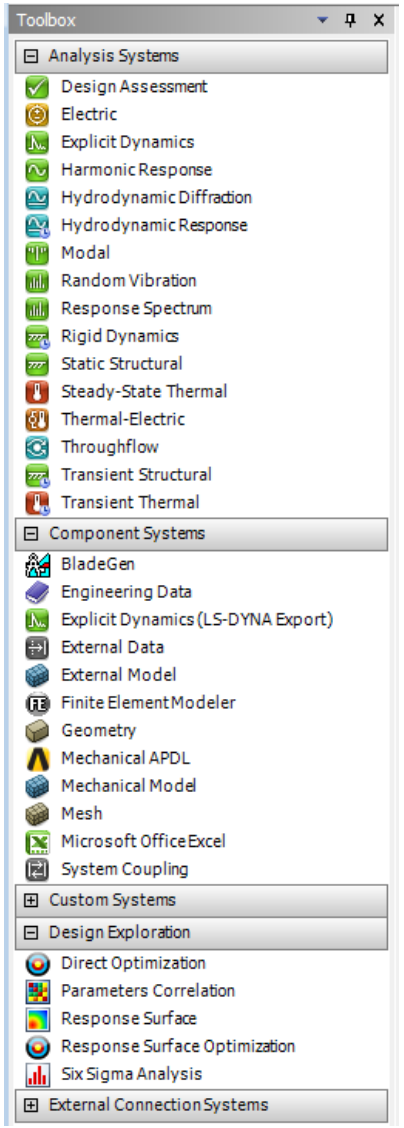
Because Workbench can manage the individual applications AND pass data between them, it is easy to automatically perform design studies (parametric analyses) for design optimisation.



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!

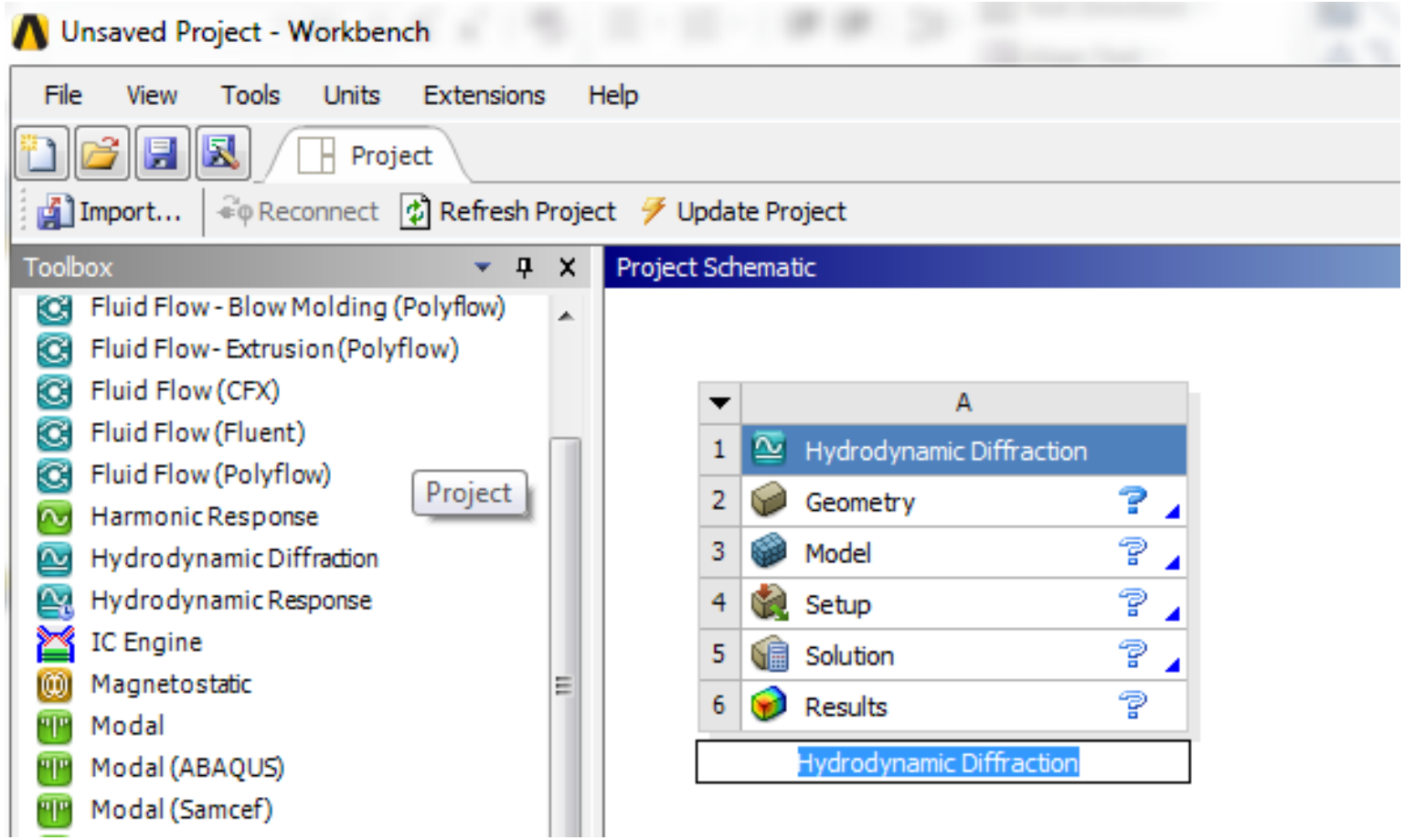


Analysis Systems are ready-made stencils that include all the individual systems (applications) needed for common analyses (for example *Geometry + Mesh + Solver + Post-Processor).*

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.

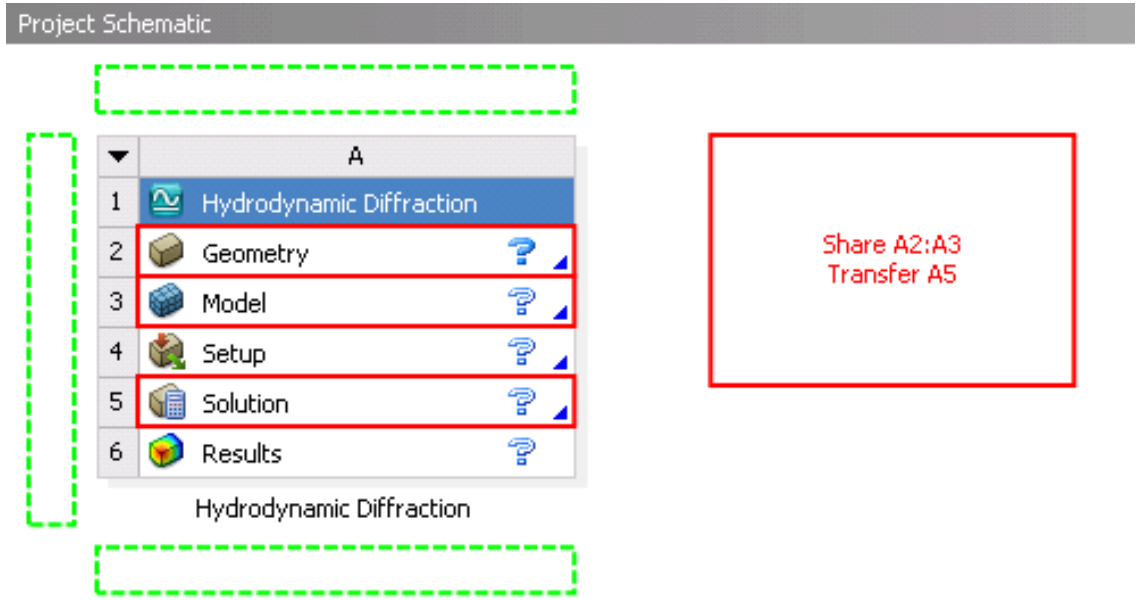
Basic Workflow

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).

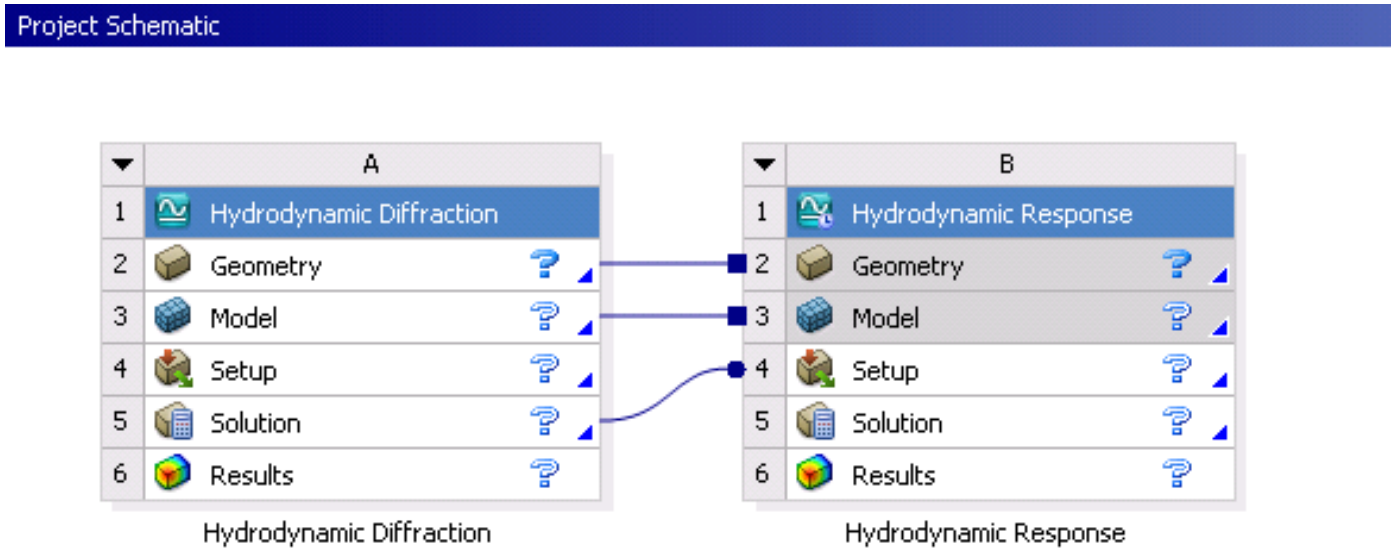
Before completing the operation notice there are a number of optional “drop targets” that will provide various types of linkage between systems.



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.

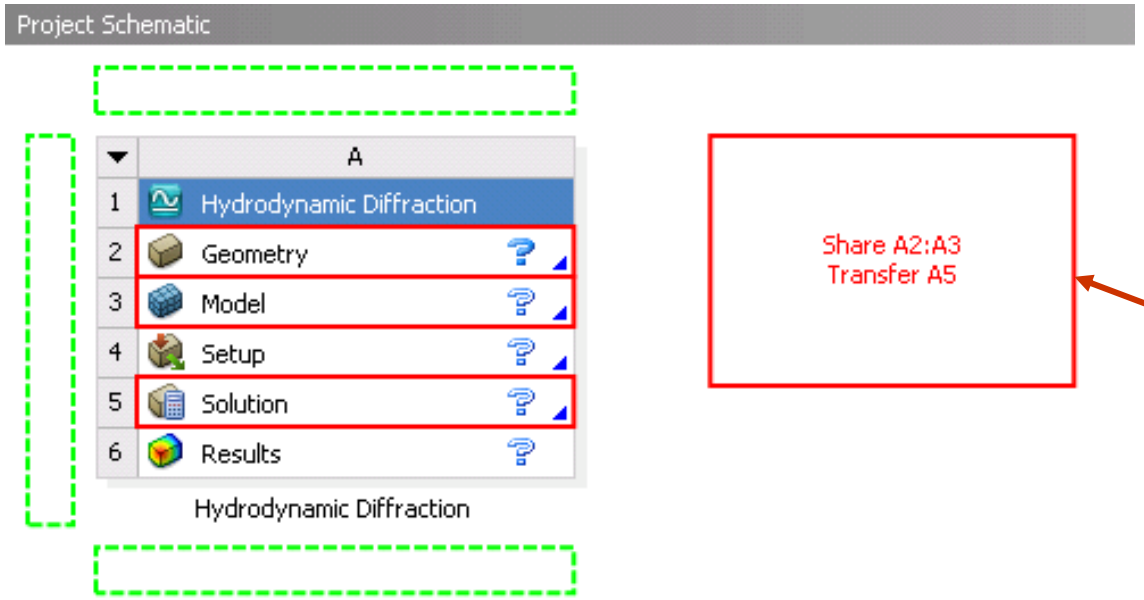
In this way we have coupled the hydrodynamic database so that it can be used for a subsequent frequency or time domain analysis.



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

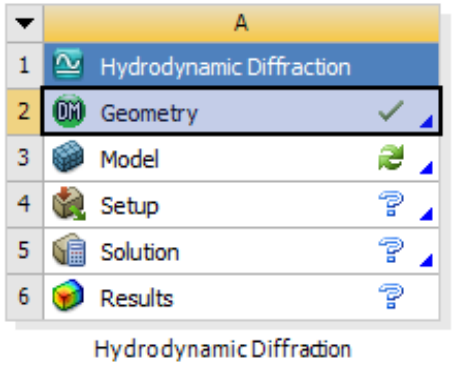
Basic Workflow

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.

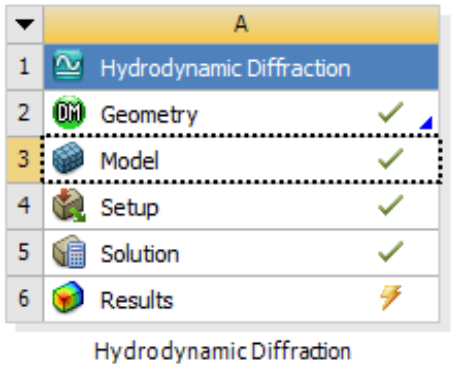


The candidate “drop target” indicates that data will be shared from fields A2 to A3 (Geometry and Model), and transferred from A5 (Solution).

Cell States



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



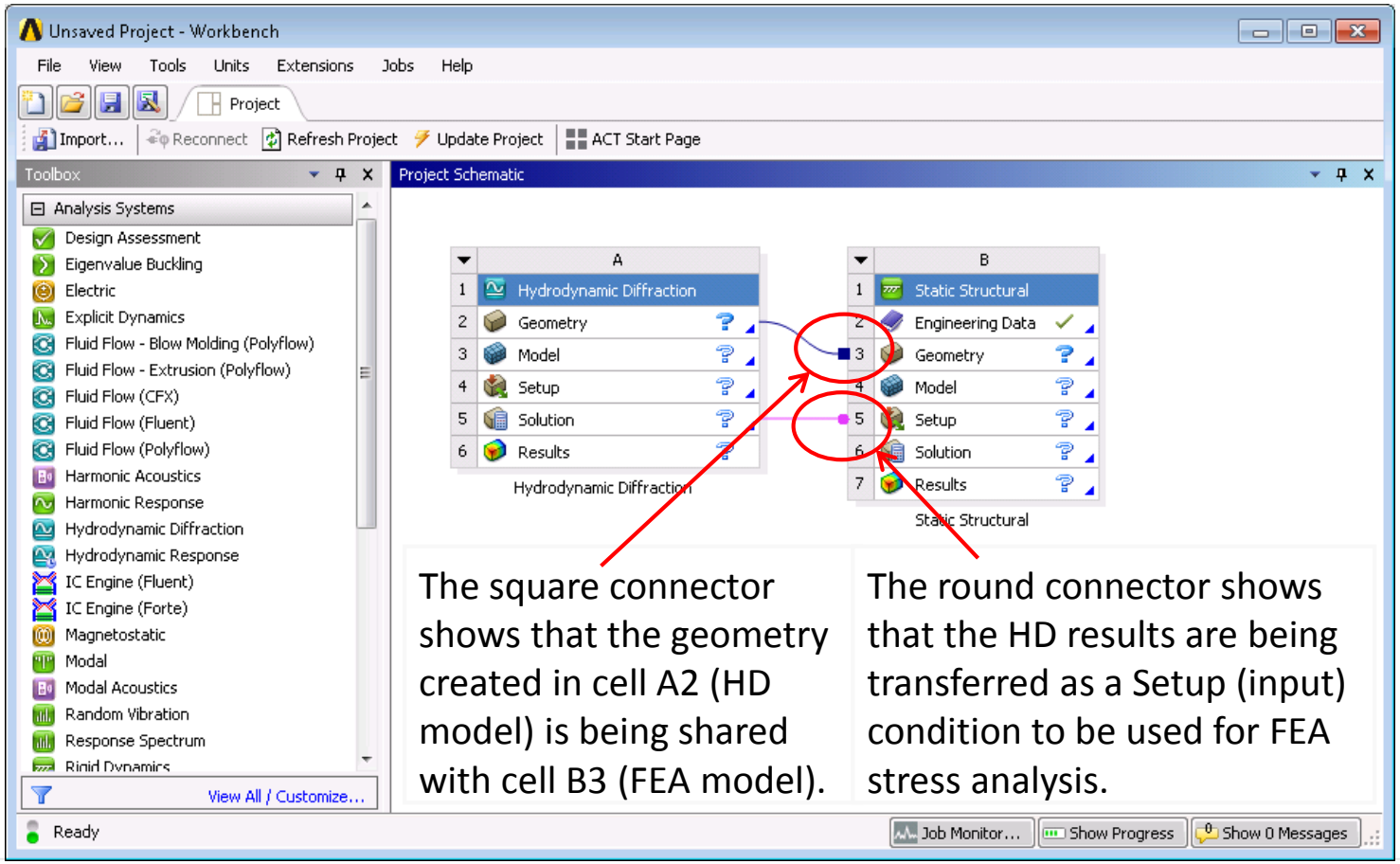
Status after model has been solved, waiting for post-processing

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)

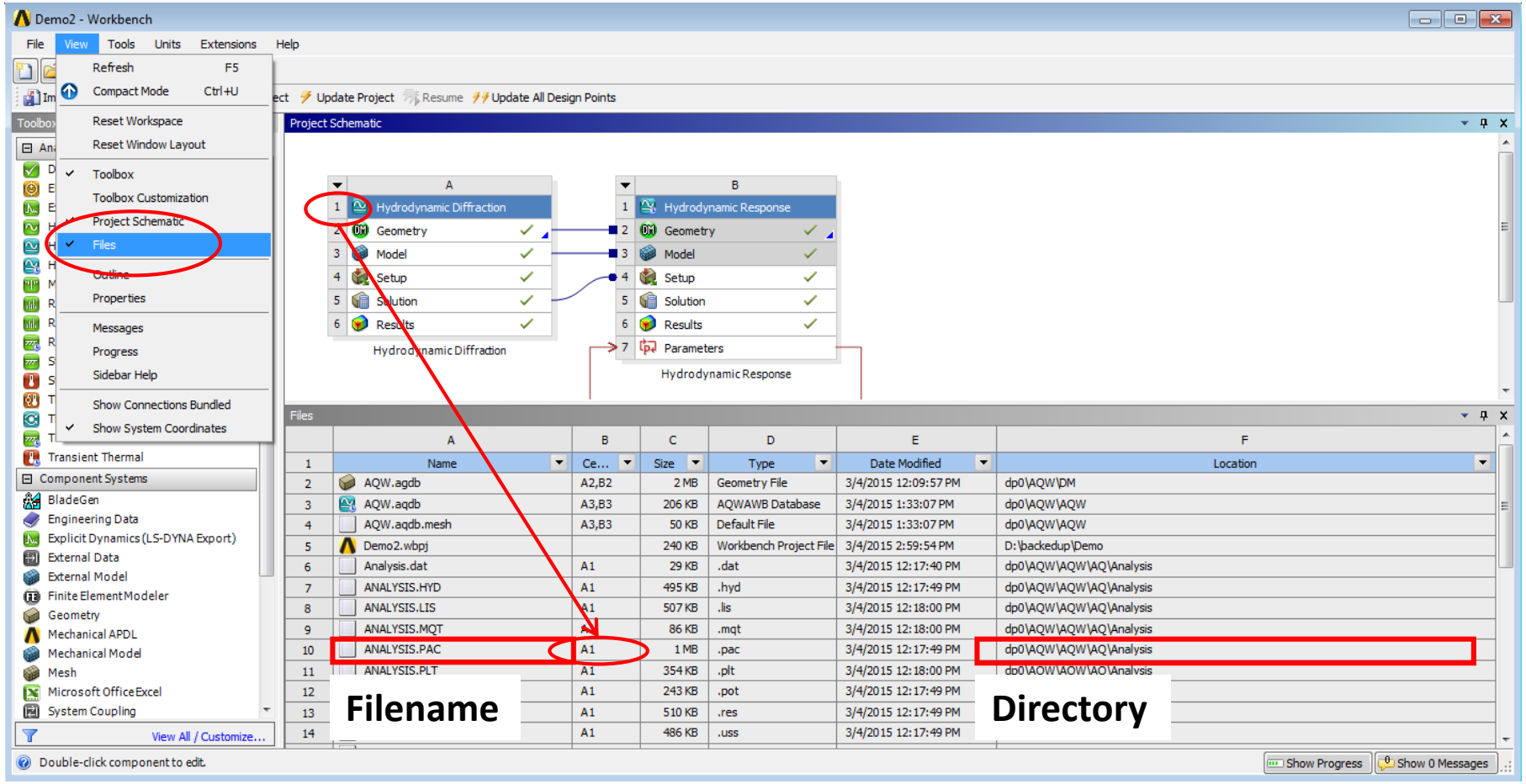
Sharing Data between Different Solvers

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

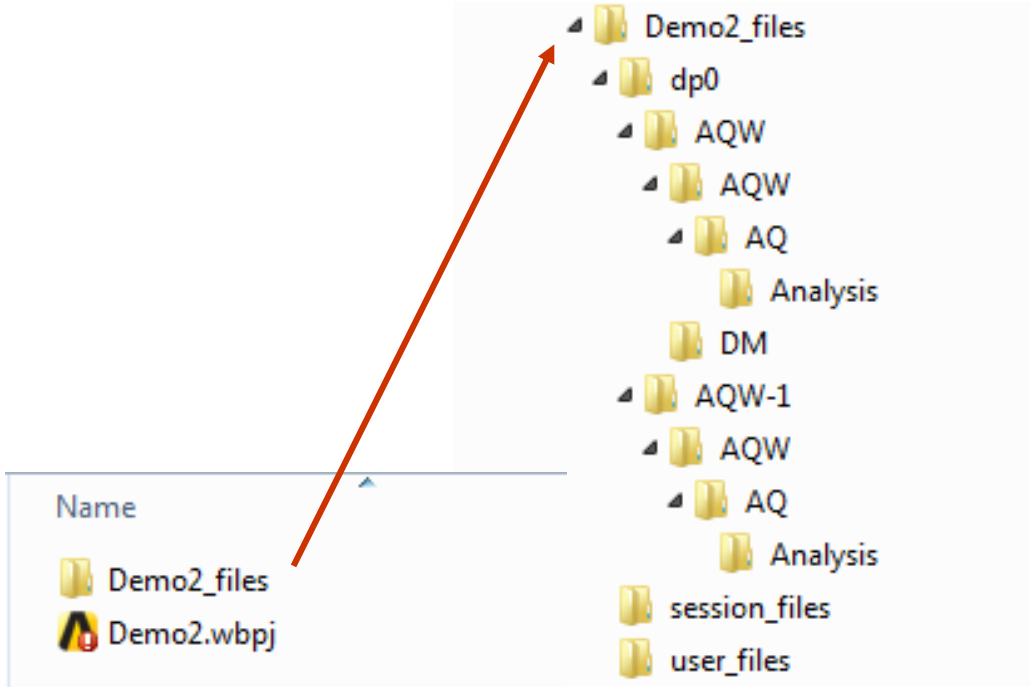
Workbench creates a project file and a series of subdirectories to manage all associated files.

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

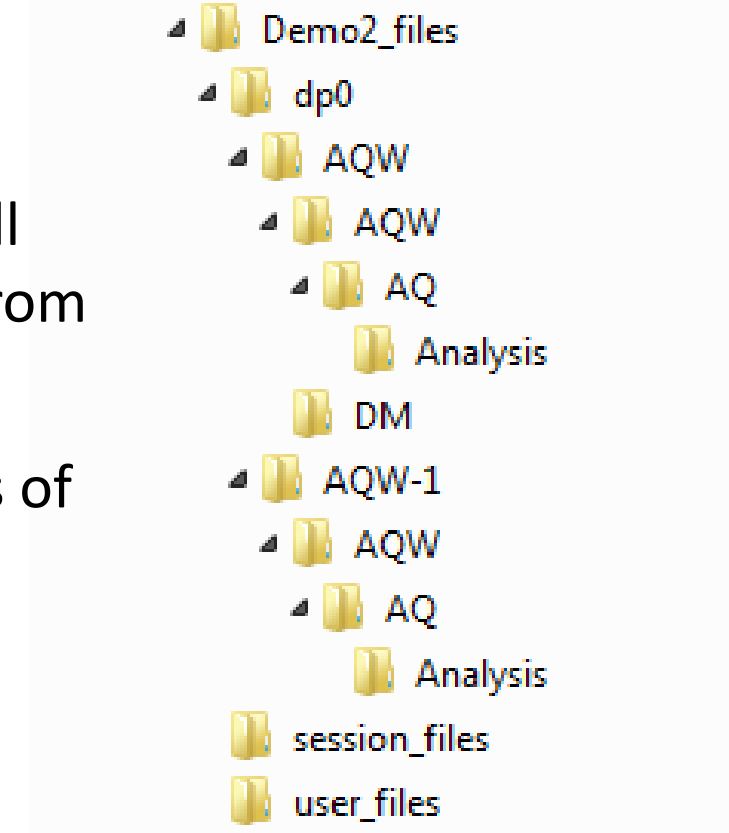
dp*n*: This is the design point directory, which is essentially the state of all parameters for a particular analysis. In the case of a single analysis (no parameterized values) there will be only one “dp0” directory.

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.

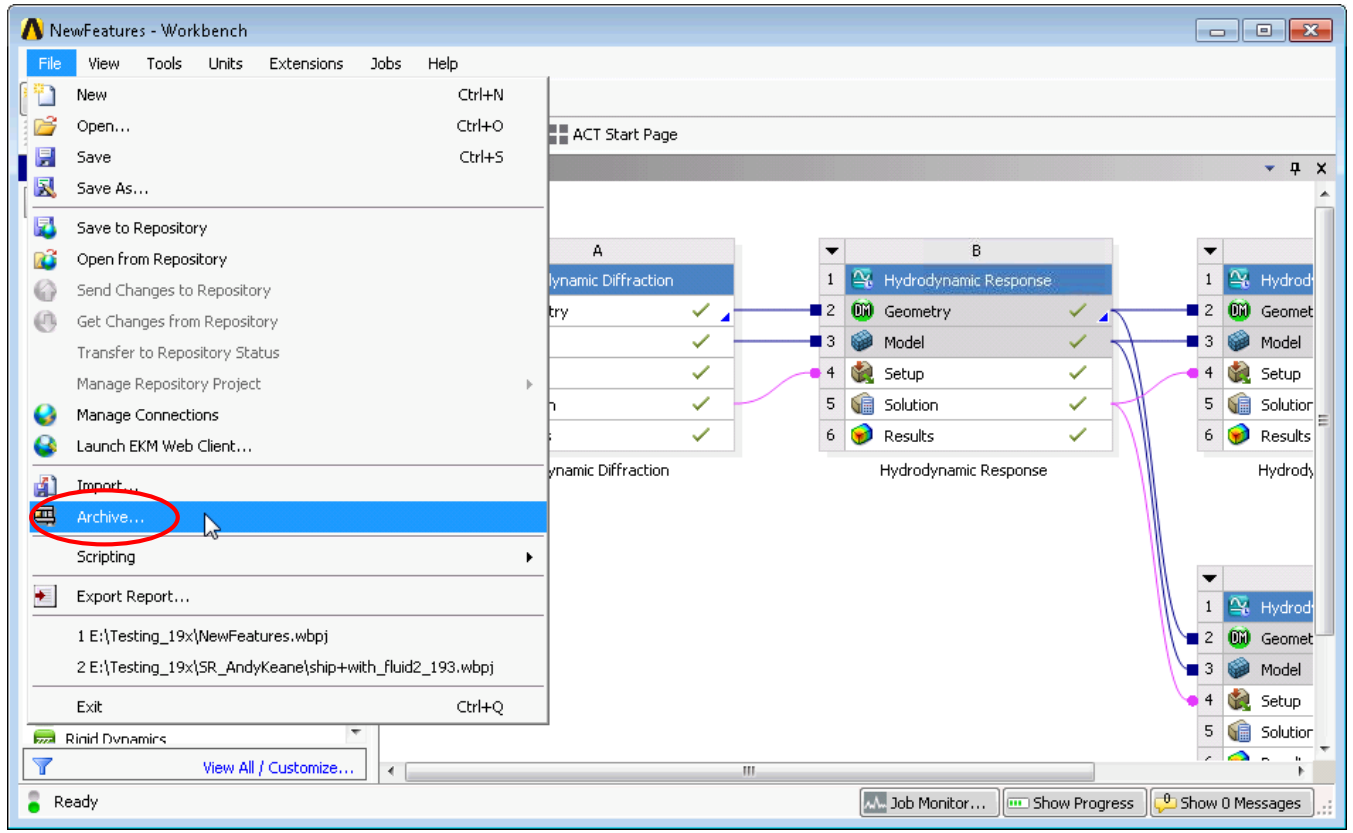
The “AQW-1\AQW\AQ\Analysis” directory will contain the results of the Hydrodynamic Response system.

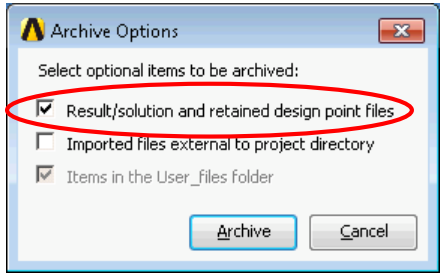
user_files: Contains external user-defined files that may be associated with a project. The user is free to use this directory as desired.



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).

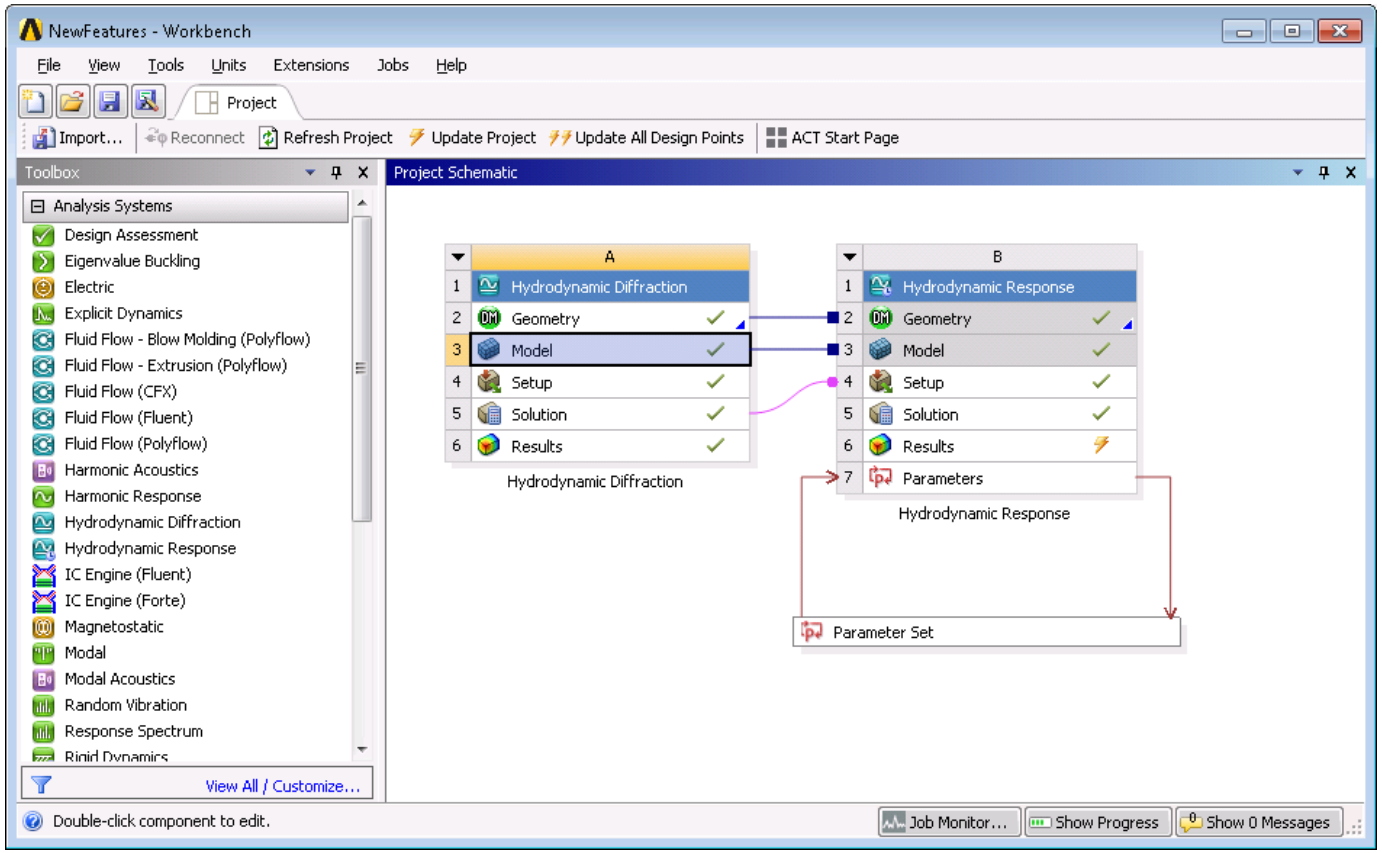
An existing Workbench archive can be restored via File > Open.

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:

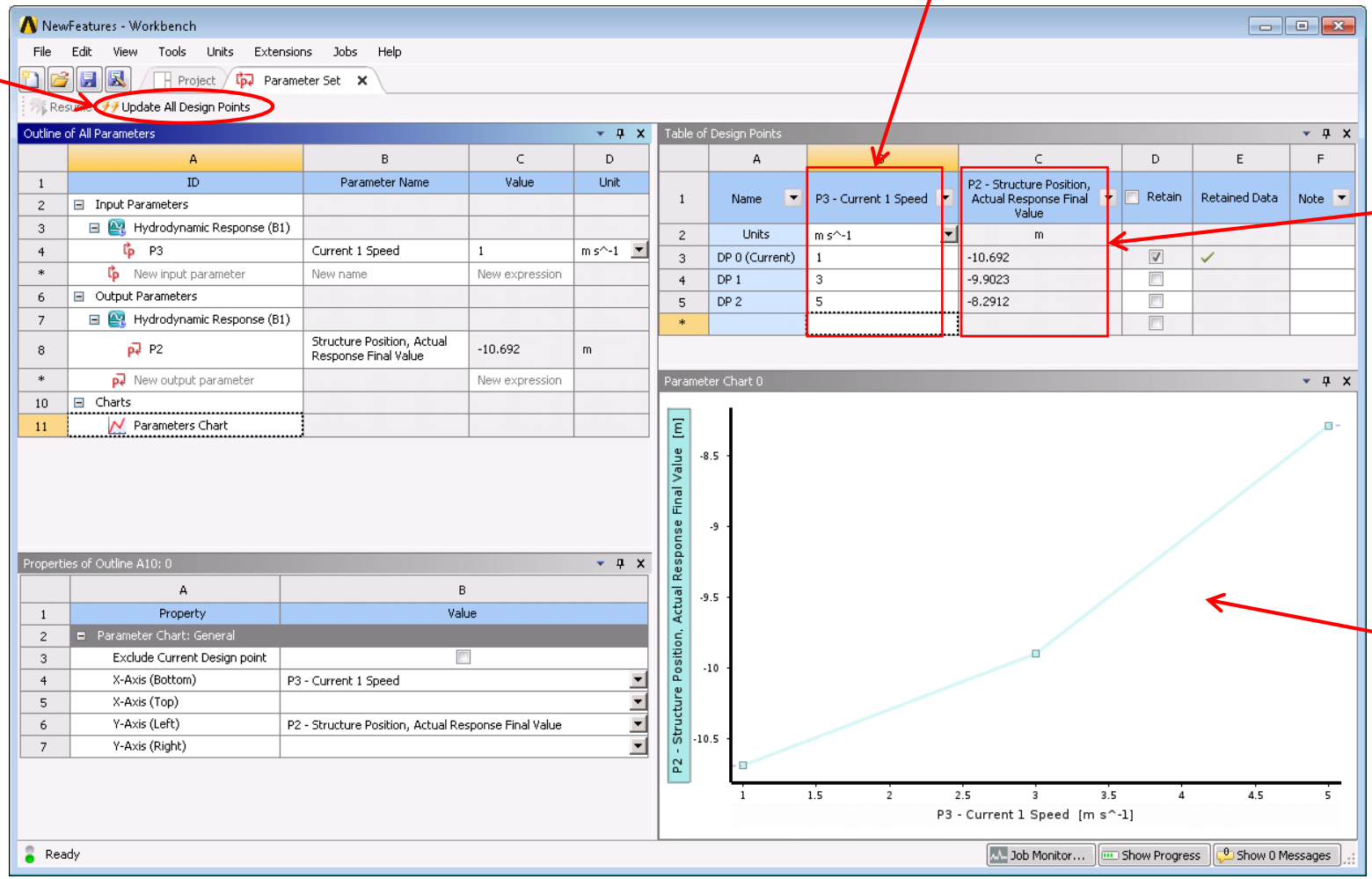
- Update the environmental data (input), based on the values defined in the Parameter Set
- Re-solve the Hydrodynamic Response system and evaluate the required results (output)

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.