



# 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

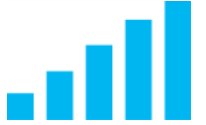
- About ANSYS Inc.
- ANSYS Customer Portal
- ANSYS Workbench Overview

# 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

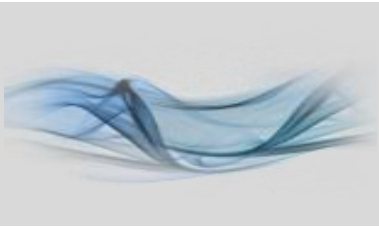
**40**  
countries

**FOCUSED**

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



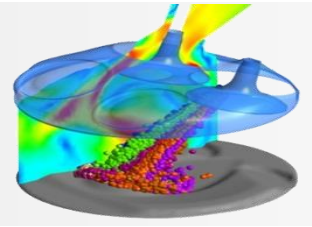
# Breadth of Technologies



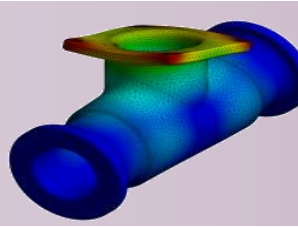
**Fluid Mechanics**  
From Single-Phase Flows...



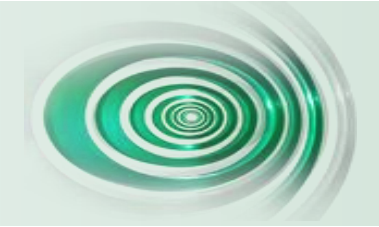
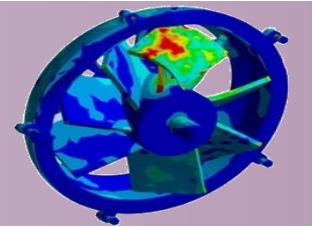
To Multiphase Combustion



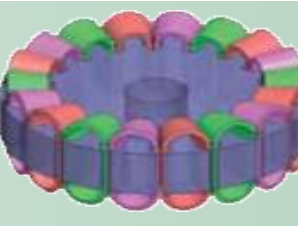
**Structural Mechanics**  
From Linear Statics...



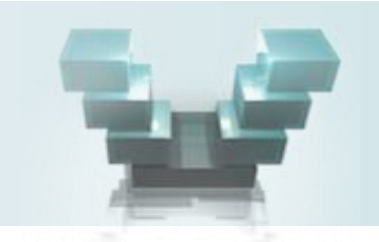
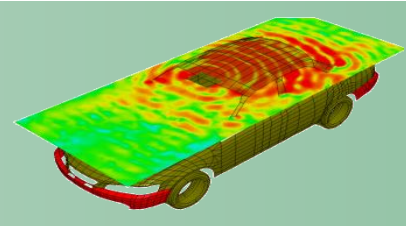
To High-Speed Impact



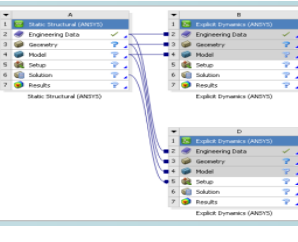
**Electromagnetics**  
From Low-Frequency Windings...



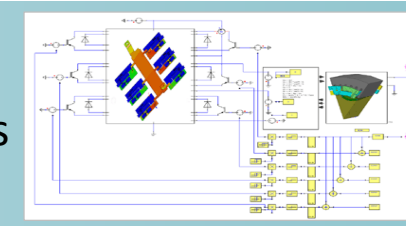
To High-Frequency Field Analysis



**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.

Classroom Training  
Webinars  
Service Requests

Support



Product Assets

Products



Latest Release  
Updates  
Tools  
Previous Release(s)

Downloads



Solutions  
Conference Proceedings  
Class3 Reports  
Documentation  
Training & Tutorials

Knowledge  
Resources





# 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

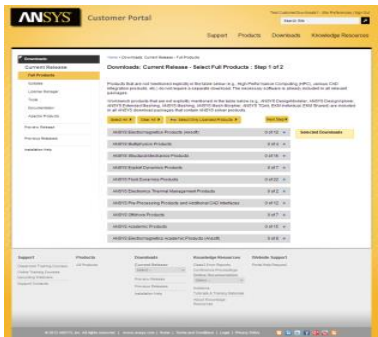
- ANSYS Fluent (5385)
- ANSYS Mechanical APDL (4092)
- ANSYS CFX (2635)
- Other (2494)
- ANSYS ICEM CFD (2028)
- ANSYS Polyflow (1333)
- ANSYS Icepak (1147)
- ANSYS Meshing (1016)
- ANSYS TurboGrid (734)
- ANSYS CFD-Post (287)
- ANSYS Autodyn (253)
- ANSYS Structural Mechanics (140)
- ANSYS Aqwa (102)

# Support, Downloads and Training



## Submit and review service requests

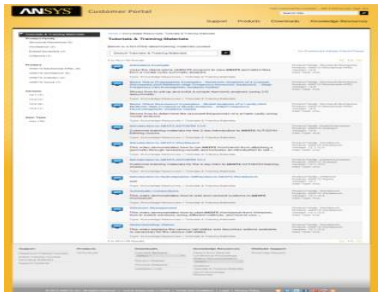
If you cannot find the answer to your question within the ANSYS Customer Portal then you can submit a service request. A member of ANSYS technical support will then get back to you with advice or a solution.



## 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.

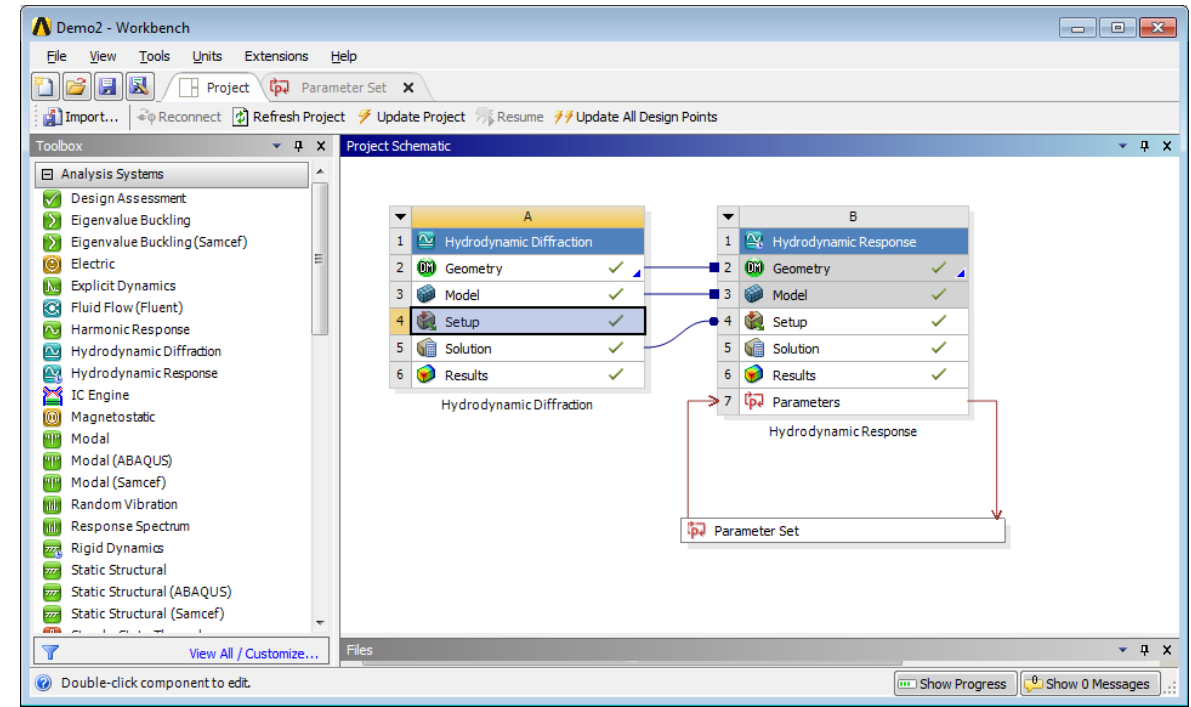
# ANSYS Workbench

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.

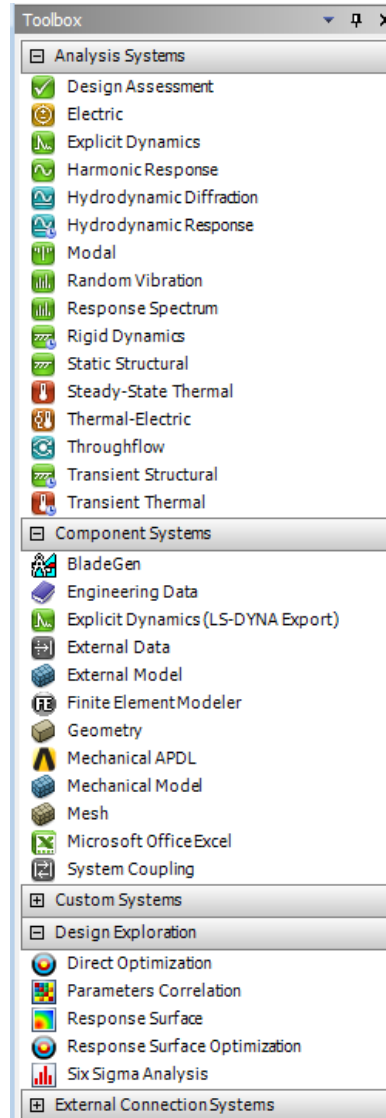


# Workbench Overview

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!*

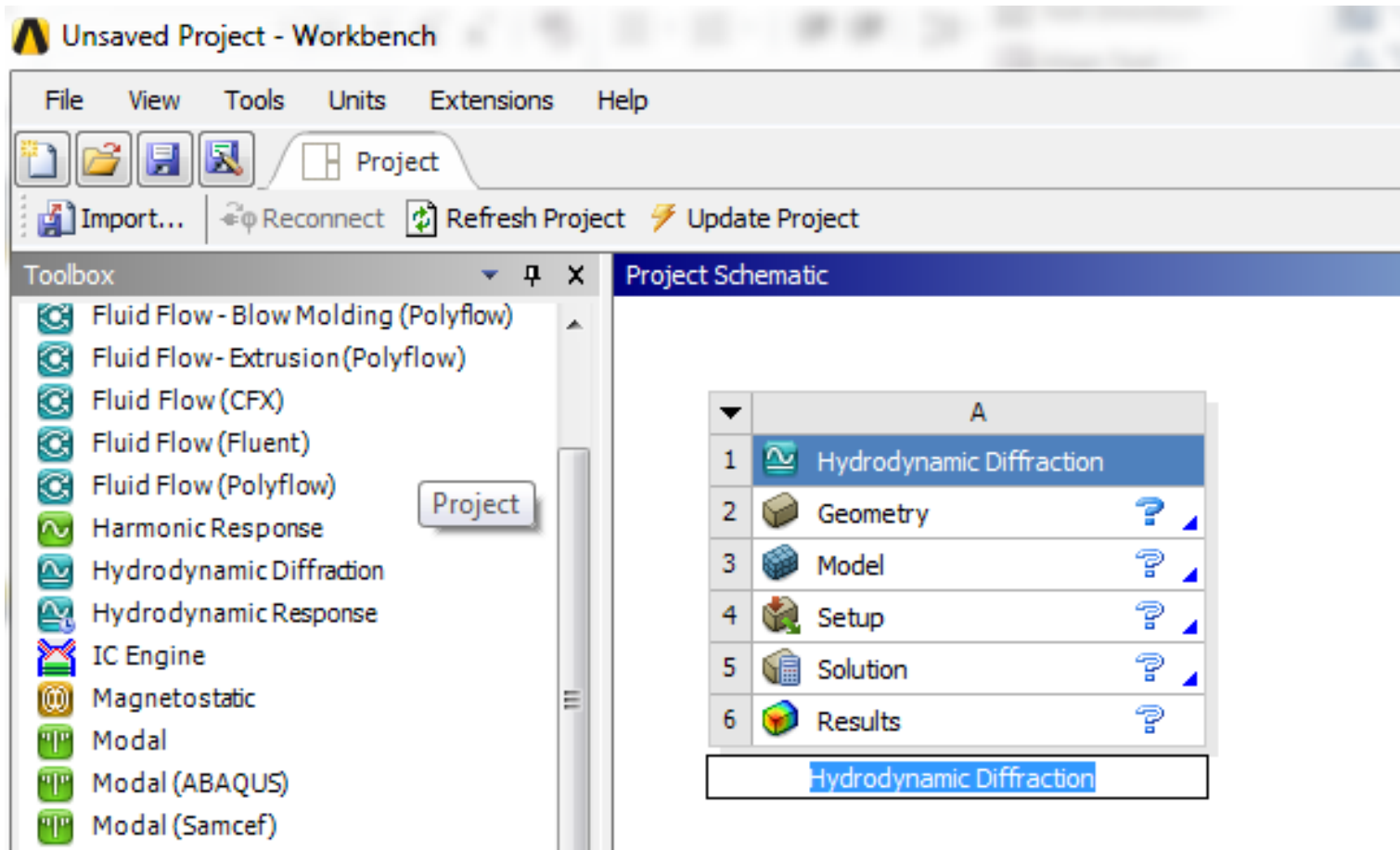
**Design Exploration** provides tools for optimising designs and understanding the parametric response.



**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.

# 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.

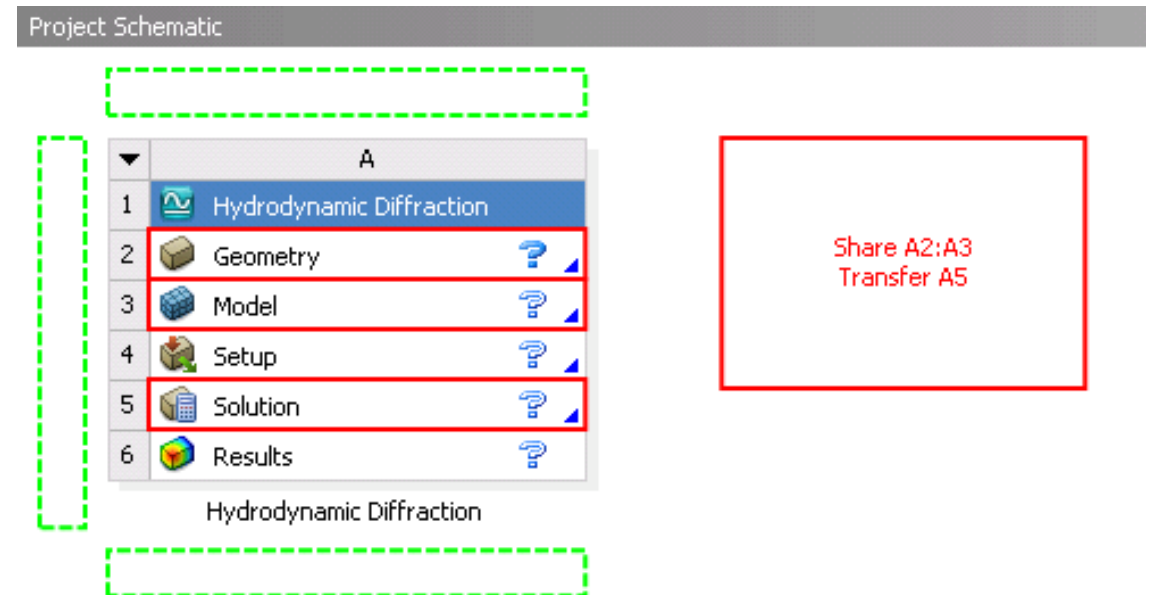
# Basic Workflow

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.

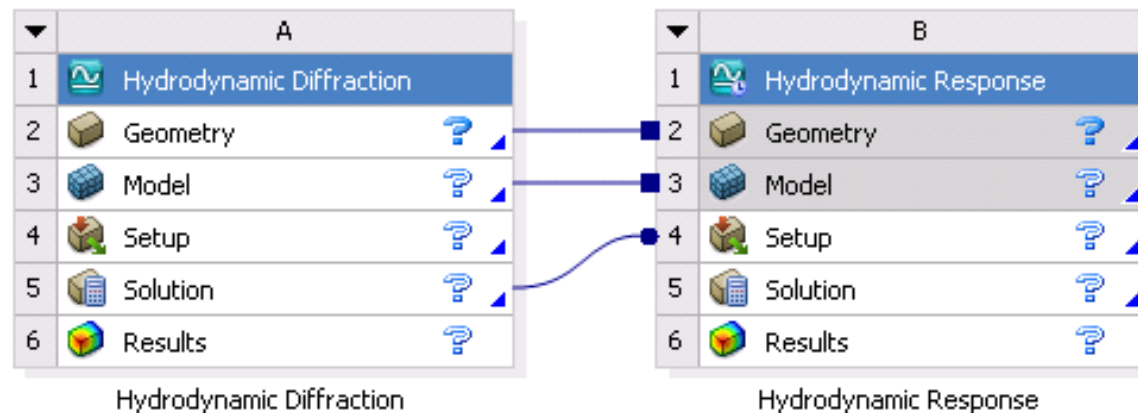


# Basic Workflow

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.

Project Schematic

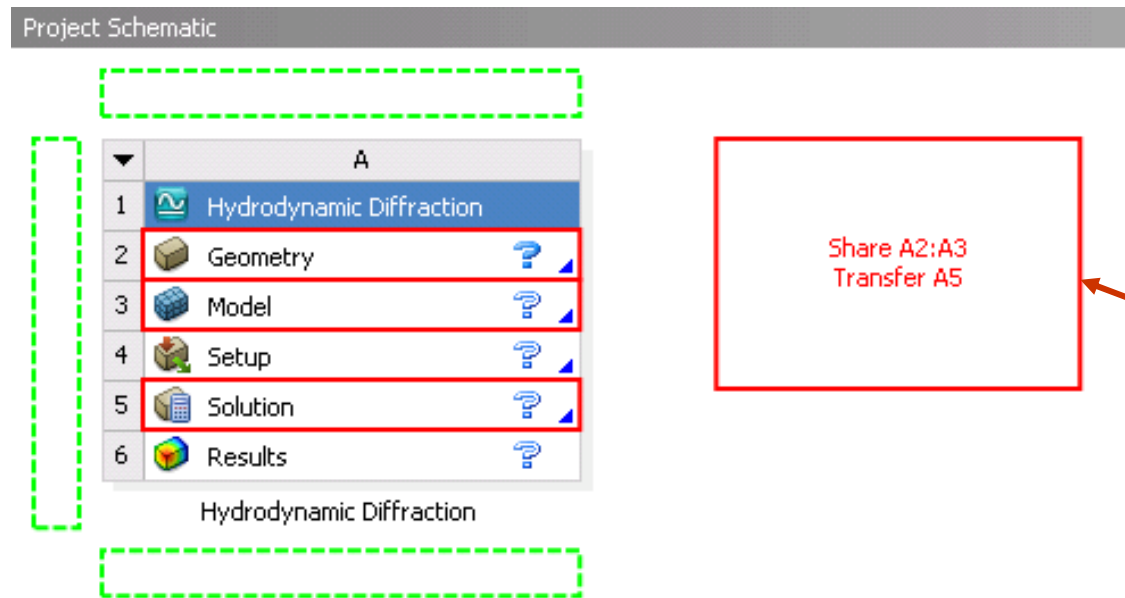


Notice that each system block is given an alphabetic designation (A, B, C), and each cell is numbered for reference.



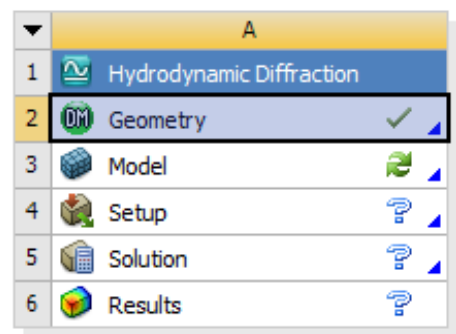
# Basic Workflow

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.



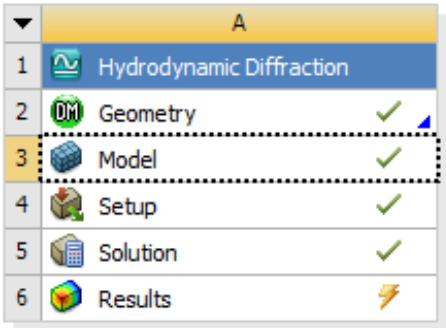
The candidate “drop target” indicates that data will be shared from fields A2 to A3 (Geometry and Model), and transferred from A5 (Solution).

# Cell States



Hydrodynamic Diffraction

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












Hydrodynamic Diffraction

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.

The screenshot displays the ANSYS Workbench interface with a Project Schematic. On the left, the 'Analysis Systems' toolbox lists various solvers. The main area shows two analysis cells: 'A' (Hydrodynamic Diffraction) and 'B' (Static Structural). Cell A contains a sequence of steps: 1. Hydrodynamic Diffraction, 2. Geometry, 3. Model, 4. Setup, 5. Solution, and 6. Results. Cell B contains: 1. Static Structural, 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, and 7. Results. A red arrow points from the 'Geometry' step (A2) in cell A to the 'Geometry' step (B3) in cell B, with a square connector at the destination. Another red arrow points from the 'Results' step (A6) in cell A to the 'Setup' step (B5) in cell B, with a round connector at the destination. Red circles highlight these connectors. Below the schematic, two text boxes provide explanations: 'The square connector shows that the geometry created in cell A2 (HD model) is being shared with cell B3 (FEA model).' and 'The round connector shows that the HD results are being transferred as a Setup (input) condition to be used for FEA stress analysis.'

Analysis Systems

- Design Assessment
- Eigenvalue Buckling
- Electric
- Explicit Dynamics
- Fluid Flow - Blow Molding (Polyflow)
- Fluid Flow - Extrusion (Polyflow)
- Fluid Flow (CFX)
- Fluid Flow (Fluent)
- Fluid Flow (Polyflow)
- Harmonic Acoustics
- Harmonic Response
- Hydrodynamic Diffraction
- Hydrodynamic Response
- IC Engine (Fluent)
- IC Engine (Forte)
- Magnetostatic
- Modal
- Modal Acoustics
- Random Vibration
- Response Spectrum
- Rigid Dynamics

Project Schematic

Cell A: Hydrodynamic Diffraction

- Hydrodynamic Diffraction
- Geometry
- Model
- Setup
- Solution
- Results

Cell B: Static Structural

- Static Structural
- Engineering Data
- Geometry
- Model
- Setup
- Solution
- Results

The square connector shows that the geometry created in cell A2 (HD model) is being shared with cell B3 (FEA model).

The round connector shows that the HD results are being transferred as a Setup (input) condition to be used for FEA 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.

The screenshot shows the ANSYS Workbench interface. The 'View' menu is open, and the 'Files' option is highlighted with a red circle. A red arrow points from this menu item to a table in the 'Files' panel. The table lists project files with columns for Name, Cell, Size, Type, Date Modified, and Location. Two specific cells are highlighted with red boxes: 'A1' in the 'Cell' column and 'dp0\AQW\AQW\AQ\Analysis' in the 'Location' column. Labels 'Filename' and 'Directory' are placed below these respective columns.

	A	B	C	D	E	F
	Name	Cell	Size	Type	Date Modified	Location
1	AQW.agdb	A2,B2	2 MB	Geometry File	3/4/2015 12:09:57 PM	dp0\AQW\DM
2	AQW.aqdb	A3,B3	206 KB	AQWAWB Database	3/4/2015 1:33:07 PM	dp0\AQW\AQW
3	AQW.aqdb.mesh	A3,B3	50 KB	Default File	3/4/2015 1:33:07 PM	dp0\AQW\AQW
4	Demo2.wbpj		240 KB	Workbench Project File	3/4/2015 2:59:54 PM	D:\backedup\Demo
5	Analysis.dat	A1	29 KB	.dat	3/4/2015 12:17:40 PM	dp0\AQW\AQW\AQ\Analysis
6	ANALYSIS.HYD	A1	495 KB	.hyd	3/4/2015 12:17:49 PM	dp0\AQW\AQW\AQ\Analysis
7	ANALYSIS.LIS	A1	507 KB	.lis	3/4/2015 12:18:00 PM	dp0\AQW\AQW\AQ\Analysis
8	ANALYSIS.MQT	A1	86 KB	.mqt	3/4/2015 12:18:00 PM	dp0\AQW\AQW\AQ\Analysis
9	ANALYSIS.PAC	A1	1 MB	.pac	3/4/2015 12:17:49 PM	dp0\AQW\AQW\AQ\Analysis
10	ANALYSIS.PLT	A1	354 KB	.plt	3/4/2015 12:18:00 PM	dp0\AQW\AQW\AQ\Analysis
11		A1	243 KB	.pot	3/4/2015 12:17:49 PM	
12		A1	510 KB	.res	3/4/2015 12:17:49 PM	
13		A1	486 KB	.uss	3/4/2015 12:17:49 PM	
14						

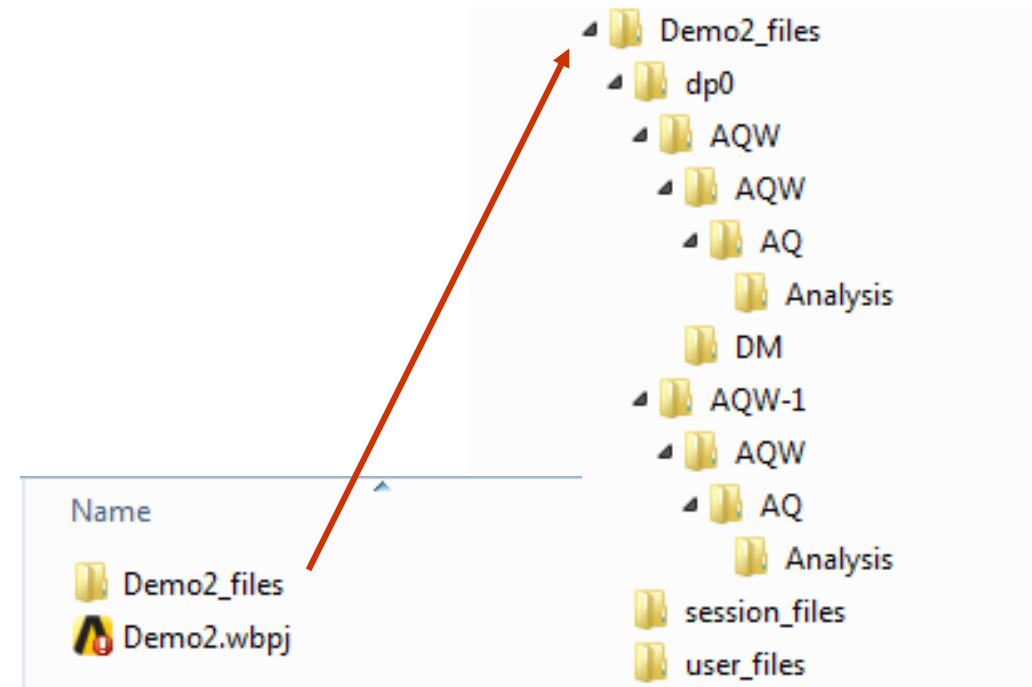
# File Management

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

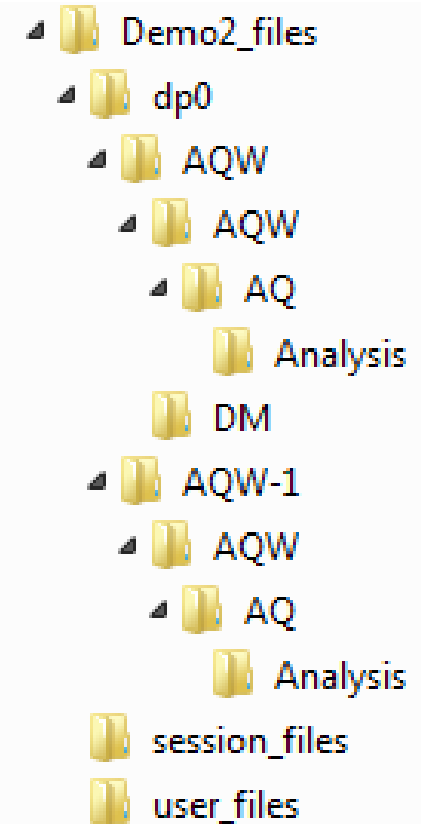
**dpn:** This is the design point directory, which is essentially the state of all parameters for a particular analysis. In the case of a single analysis (no parameterized values) there will be only one “dp0” directory.

**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.

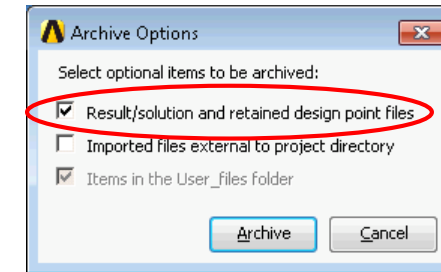
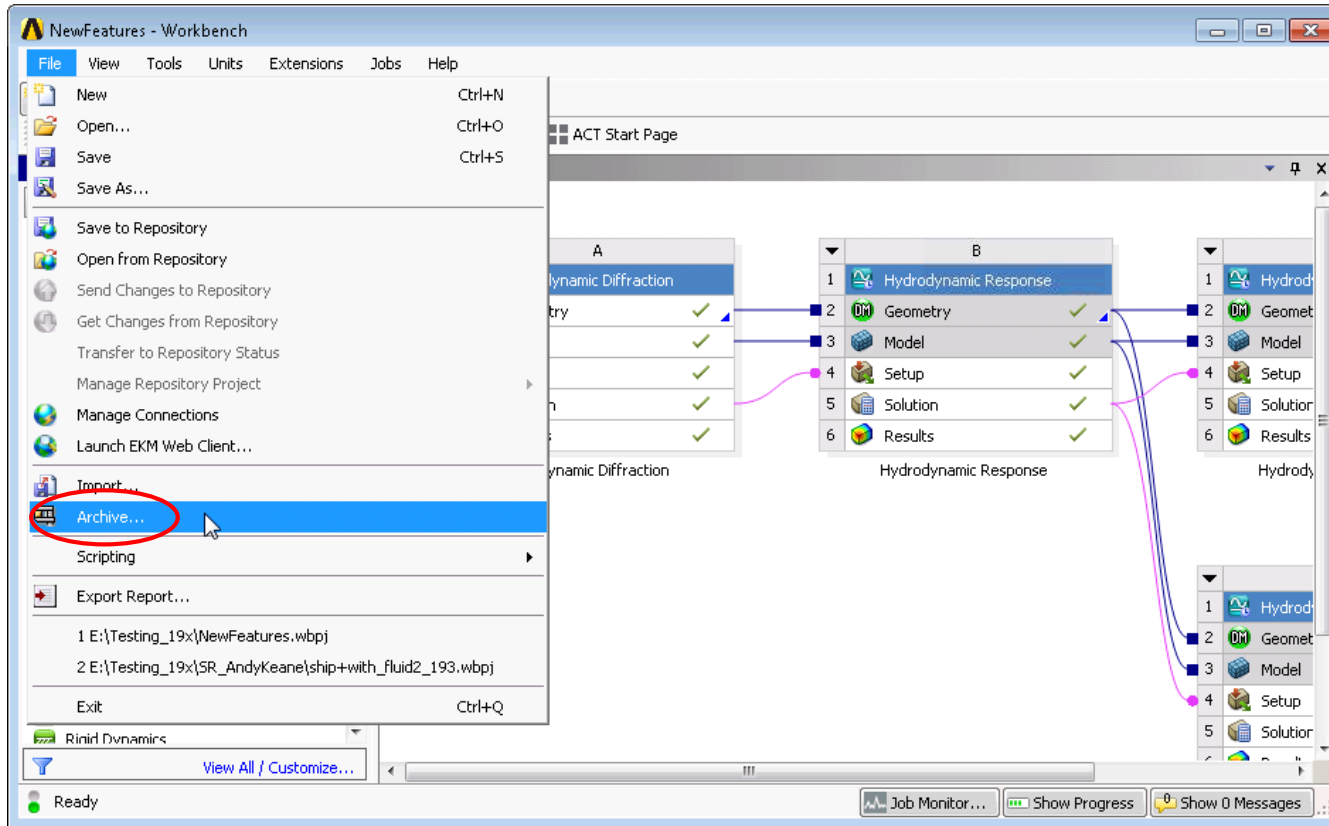
The “AQW-1\AQW\AQ\Analysis” directory will contain the results of the Hydrodynamic Response system.

**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.

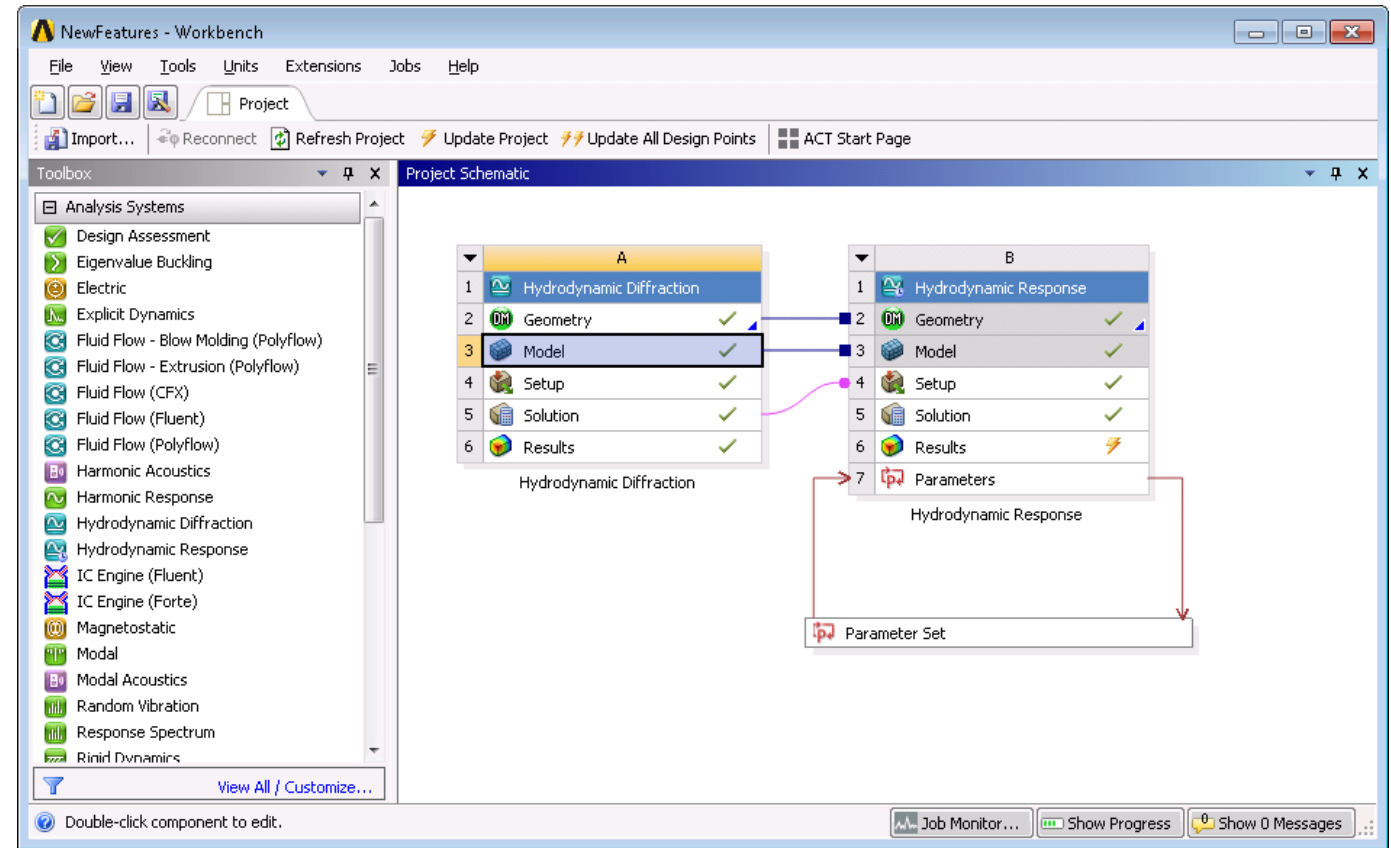


# Working With Parameters

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:

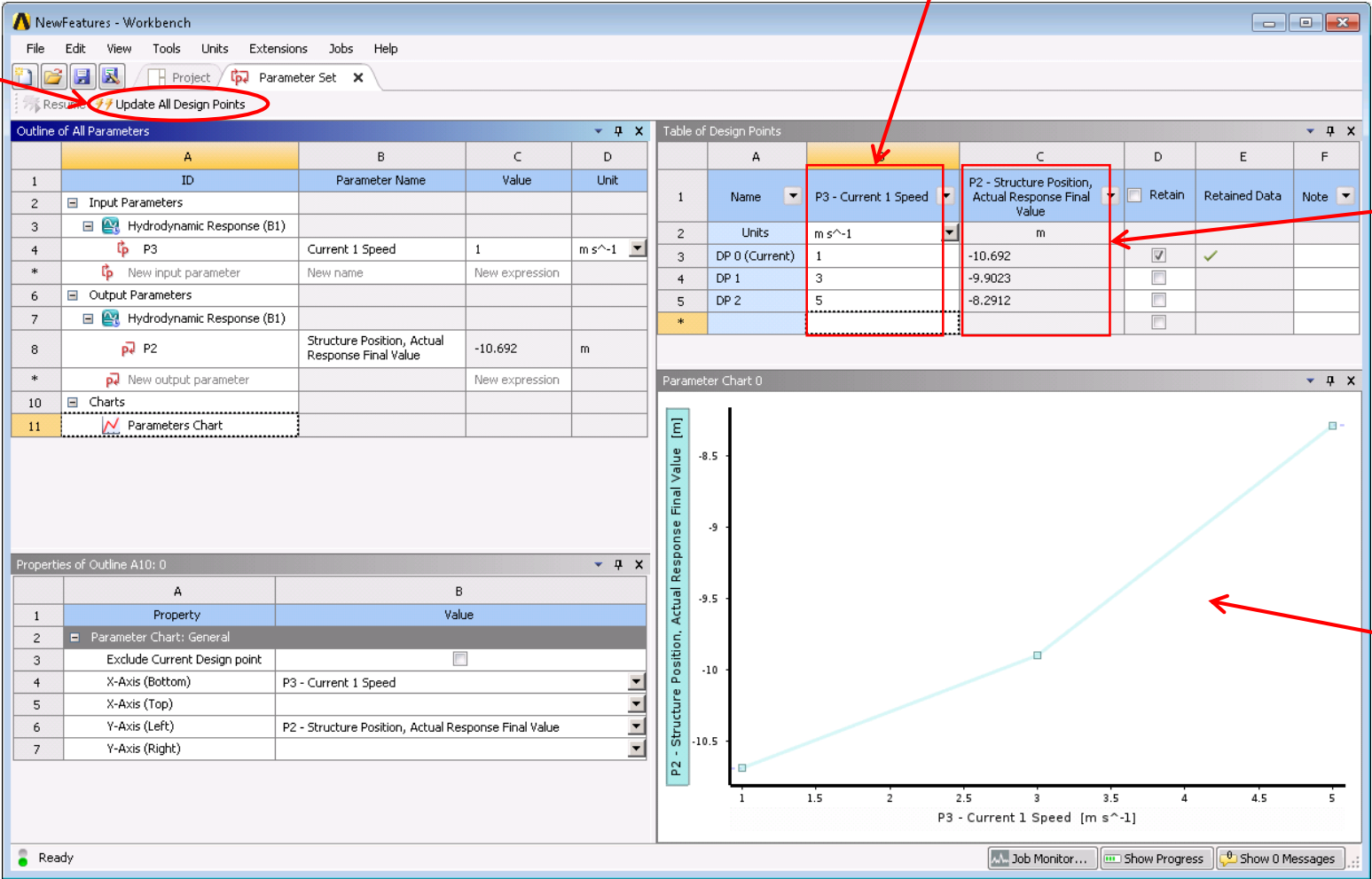
- Update the environmental data (input), based on the values defined in the Parameter Set
- Re-solve the Hydrodynamic Response system and evaluate the required results (output)

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

# Working With Parameters

Click 'Update All Design Points' to compute over all input values

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



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.