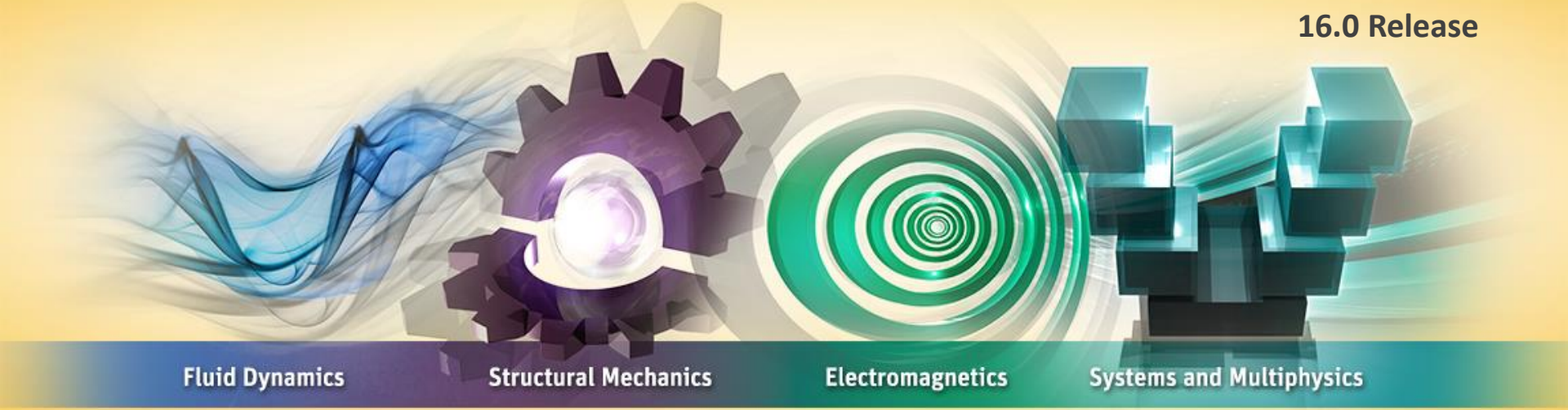
Lecture 1
Introduction to ANSYS Workbench

16.0 Release



Introduction to ANSYS Mechanical

Realize Your Product Promise®

Welcome!

Welcome to the *ANSYS Mechanical application* introductory training course.

This training course covers the basics of using ANSYS Mechanical in performing structural and thermal analyses.

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

Course Objectives:

- **General understanding of the user interface, as related to geometry import, meshing, application of loads and supports, and postprocessing**
- **Procedure for performing FEA simulations, including linear static, modal, and harmonic structural analyses and nonlinear steady-state thermal analyses**
- **Utilizing parameters for 'what-if' scenarios**

- **Training Courses are also available covering the use of other Workbench modules (e.g. *DesignModeler, Design Exploration, etc.)* .**

Agenda (Day 1)

Morning	Lecture – Introduction
	Lecture – Chapter 2: Mechanical Basics
	Workshop 2.1
	Lecture – Chapter 3: General Preprocessing
	Workshop 3.1
	Lecture – Chapter 3, continued
Afternoon	Workshop 3.2
	Lecture – Chapter 3, continued
	Workshop 3.3 or Workshop 3.4
	Lecture – Chapter 4: Meshing in Mechanical
	Workshop 4.1
	Lecture – Chapter 4 (continued)
	Workshop 4.2
	Lecture – Chapter 5: Modeling Connections
	Workshop 5.1

Agenda (Day 2)

Morning	Lecture – Chapter 5 (continued)
	Workshop 5.2
	Lecture – Chapter 6: Remote Boundary Conditions
	Workshop 6.1
	Lecture – Chapter 6 (continued)
	Workshop 6.2
	Lecture – Chapter 7: Static Structural Analysis
	Workshop 7.1
Afternoon	Lecture – Chapter 7 (continued)
	Workshop 7.2
	Lecture – Chapter 8: Modal Analysis
	Workshop 8.1
	Lecture – Chapter 9: Thermal Analysis
	Workshop 9.1

Agenda (Day 3)

Morning	Lecture – Chapter 10: Multistep Analysis
	Workshop 10.1
	Lecture – Chapter 11: Results and Postprocessing
	Workshop 11.1
Afternoon	Lecture – Chapter 12: CAD and Parameters
	Workshop 12.1
	Choice of Appendix Chapters or discussion of user issues*

* This course has been designed to run to just over 2.5 days to allow the last afternoon to be more informal. There is a choice of Appendix Chapters that explore different analysis types or go into some subjects in more depth. If time allows user problems can also be discussed depending on the number of attendees.

Lecture 1: Contents

A: 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

2,700+ employees

75 locations

40 countries

FOCUSED

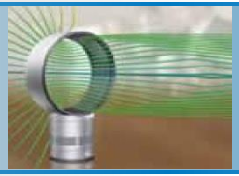
This is all we do.

Leading product technologies in all physics areas

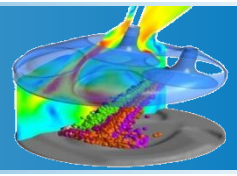
Largest development team focused on simulation

Breadth of Technologies

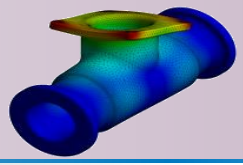
Fluid Mechanics:
From Single-Phase Flows



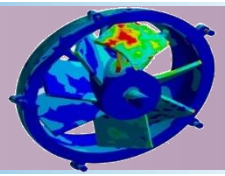
To Multiphase
Combustion



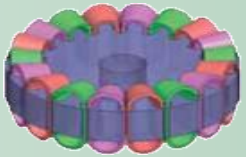
Structural Mechanics:
From Linear Statics



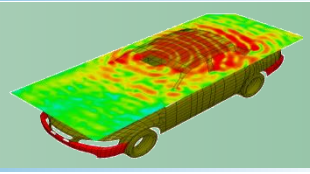
To High-Speed Impact



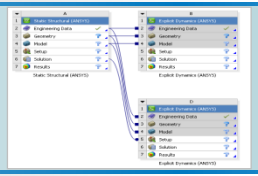
Electromagnetics: From
Low-Frequency Windings



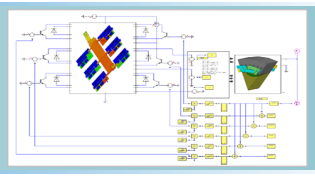
To High-Frequency
Field Analysis



Systems:
From Data Sharing



To Multi-Domain
System Analysis



The ANSYS Customer Portal

https://support.ansys.com

Contains over 85,000 support assets powered by a modern web user interface and powerful search engine.

Over 5.2 million page views in 2014

Support
- Classroom Training
- Webinars
- Service Requests

Products
- Product Assets

Downloads
- Latest Release
- Updates
- Tools
- Previous Release(s)
- Extension Library (App Store)

Knowledge Resources
- Solutions
- Conference Proceedings
- Class3 Reports
- Documentation
- Training & Tutorials

About search

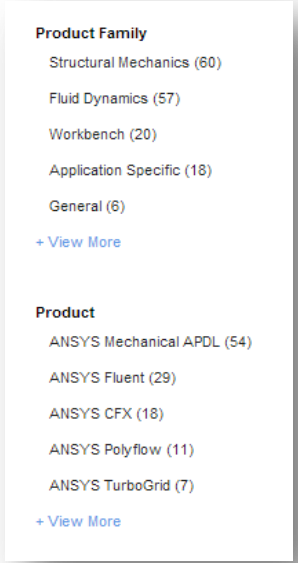
The ANSYS Customer Portal's search is powered by dedicated Google® hardware.

Mesh = Meshed = Meshing

Export = Exported = Exporting

XXXXX = YYYYY = ZZZZZ

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.

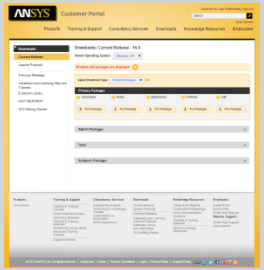


Search Facets

Support / downloads / 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 recommend 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.

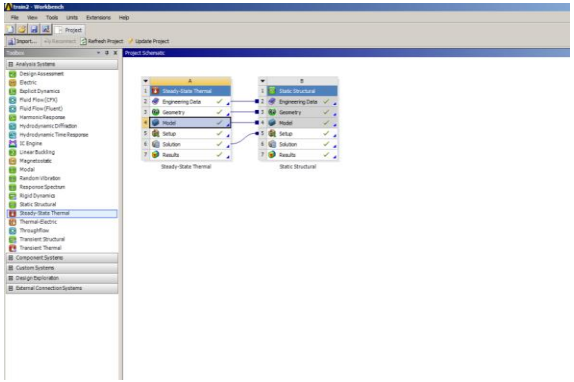
C. ANSYS Workbench

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

Workbench handles the passing of data between ANSYS Geometry / Mesh / Solver / Postprocessing 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.

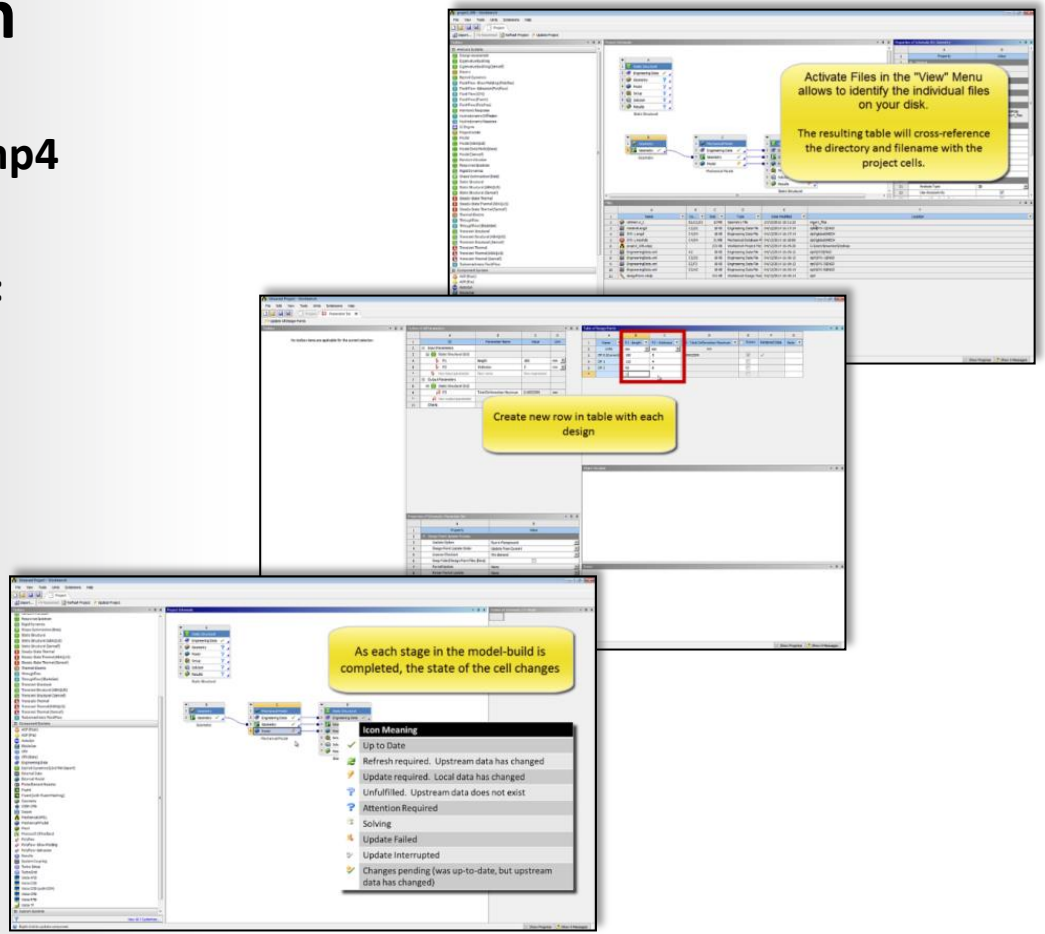
Because Workbench can manage the individual applications AND pass data between them, it is easy to automatically perform design studies (parametric analyses) for design optimisation.



D. Demonstration

Demonstration : 01_WB_Presentation.mp4

- **What you can learn in this demonstration :**
  - **How to use Workbench project page**
  - **How to save, open, archive a project**
  - **How to create different analysis**
  - **Understand each stage of an analysis**
  - **Edit units, properties and files options**
  - **Working with parameters**



E: Summary

ANSYS Workbench is a convenient way of managing your simulation projects.

Workbench is used to launch the individual software components, and used to transfer data between them.

It is easy to see at-a-glace how a model has been built, and 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.

F. Mechanical Overview

Analysis types available in Workbench - *Mechanical*:†

- **Structural (static and transient):**
  - **Linear and nonlinear structural analyses.**
- **Dynamics:**
  - **modal, harmonic, response spectrum, random vibration, flexible and rigid dynamics.**
- **Heat Transfer (steady state and transient):**
  - **Solve for temperature field and heat flux. Temperature-dependent conductivity, convection, radiation and materials allowed.**
- **Magnetostatic:**
  - **Perform various magnetic field analyses.**
- **Electrical:**
  - **Simulate electrical devices such as motors, solenoids, etc..**

† Note, the active ANSYS license dictates what functionality is available to the user. Not all features listed are covered in this Introductory course.

… Mechanical Overview

Add-on licenses for Mechanical:

- **Rigid Dynamics**
- **Fatigue Module**
- **ACP**

16.0 Workbench products are available for Windows and Linux operating systems.

- **Check the ANSYS web site or online documentation for the latest compatibilities.**

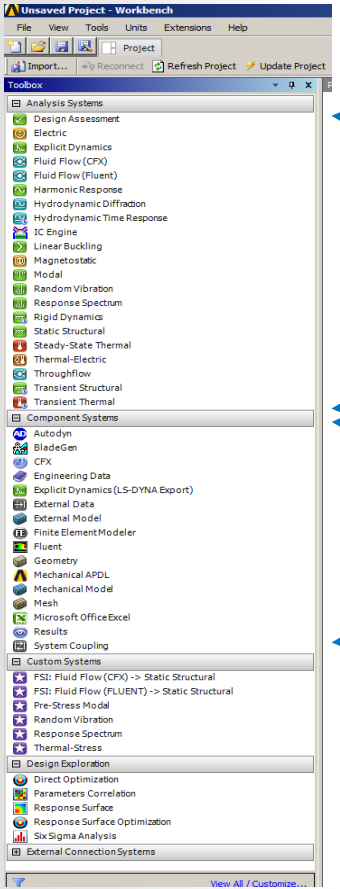
Network licensing capabilities are used for all ANSYS and ANSYS Workbench products.

G. APPENDIX

. . . Workbench Overview

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

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

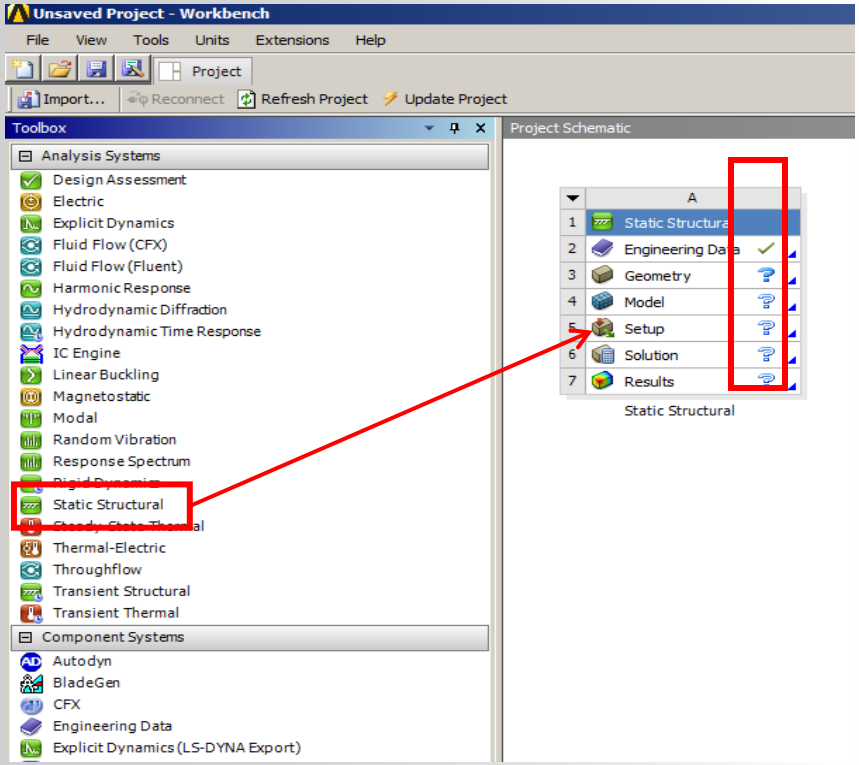


"**Analysis Systems**" are ready-made stencils that include all the individual systems (applications) needed for common analyses (*for example Geometry + Mesh + Solver + Post-Processor)*

"**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.

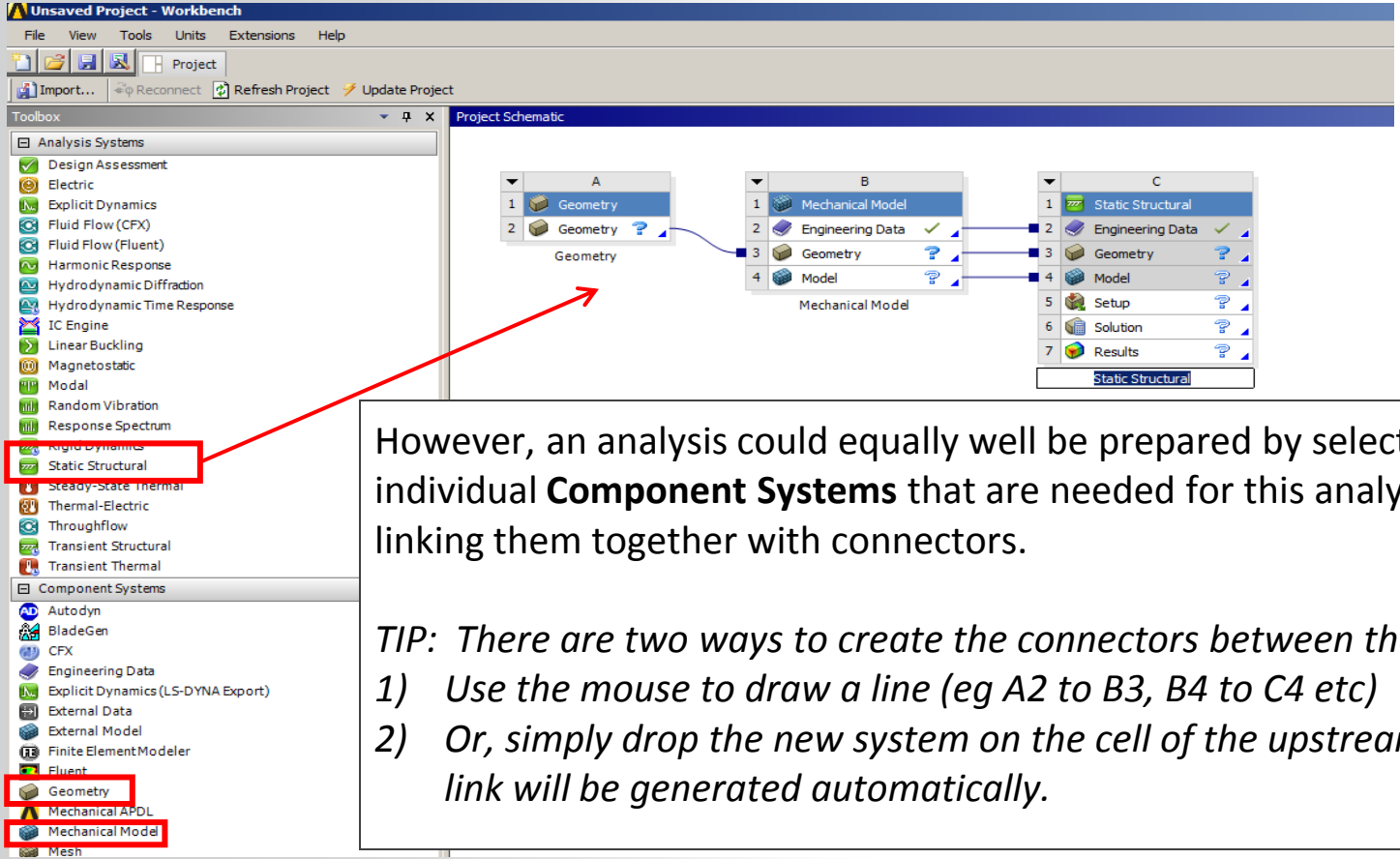
. . . Basic Workflow



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 changes

. . . Alternative Workflow

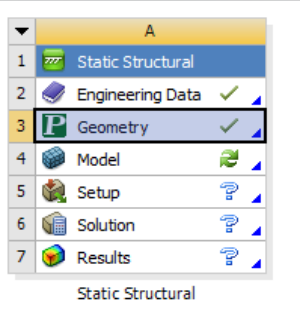


However, an analysis could equally well be prepared by selecting the individual **Component Systems** that are needed for this analysis, and then linking them together with connectors.

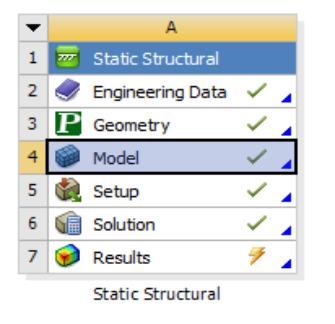
TIP: There are two ways to create the connectors between the systems:

1) *Use the mouse to draw a line (eg A2 to B3, B4 to C4 etc)*
2) *Or, simply drop the new system on the cell of the upstream one, and the link will be generated automatically.*

. . . Cell States



Status after creating Geometry **in A3, not yet opened Mechanical in A4**



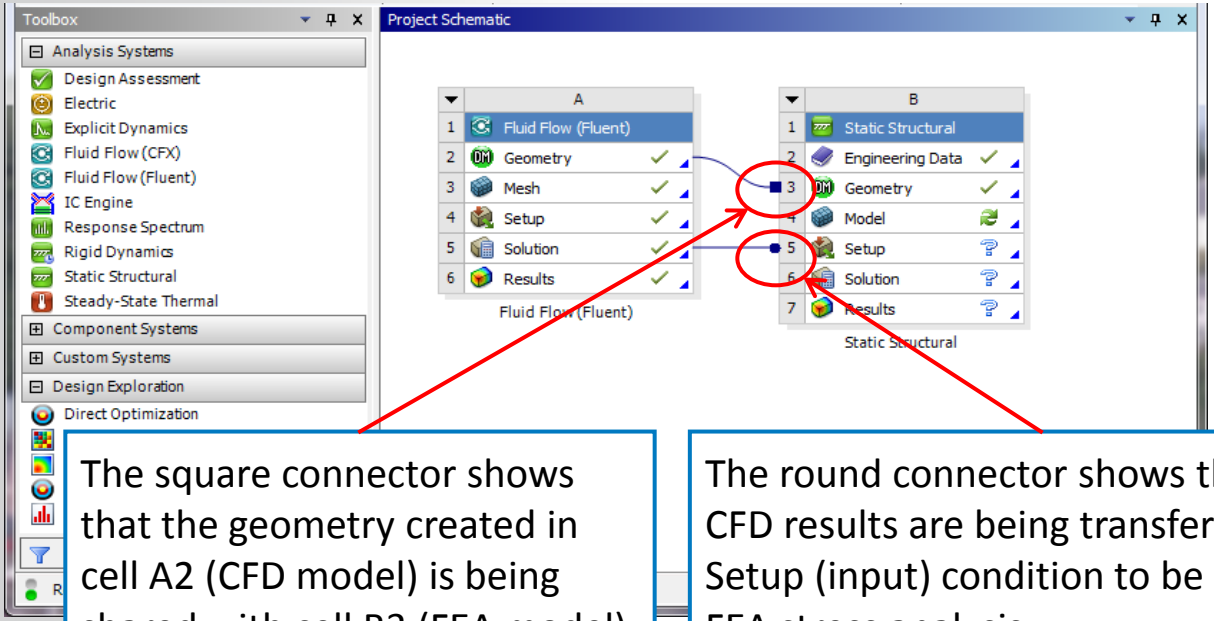
Status after model has solved waiting for post-processing

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 Fluent CFD simulation over to a Mechanical system to perform a stress analysis

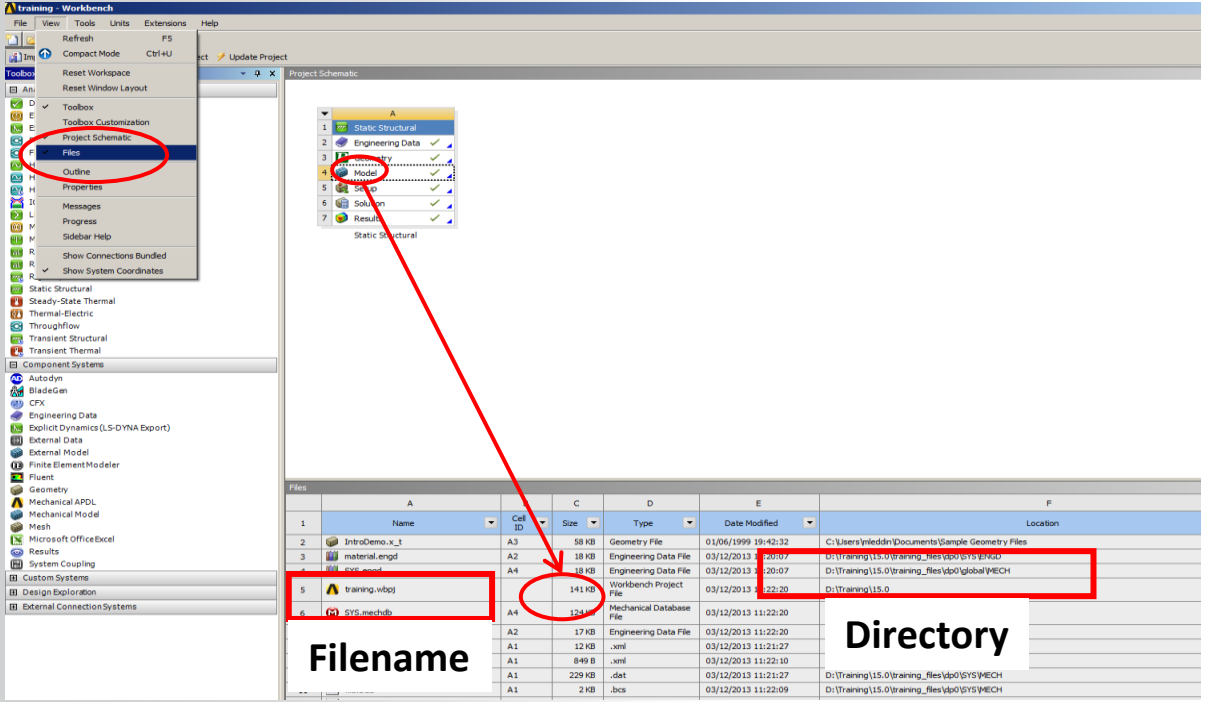


The square connector shows that the geometry created in cell A2 (CFD model) is being shared with cell B3 (FEA model).

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

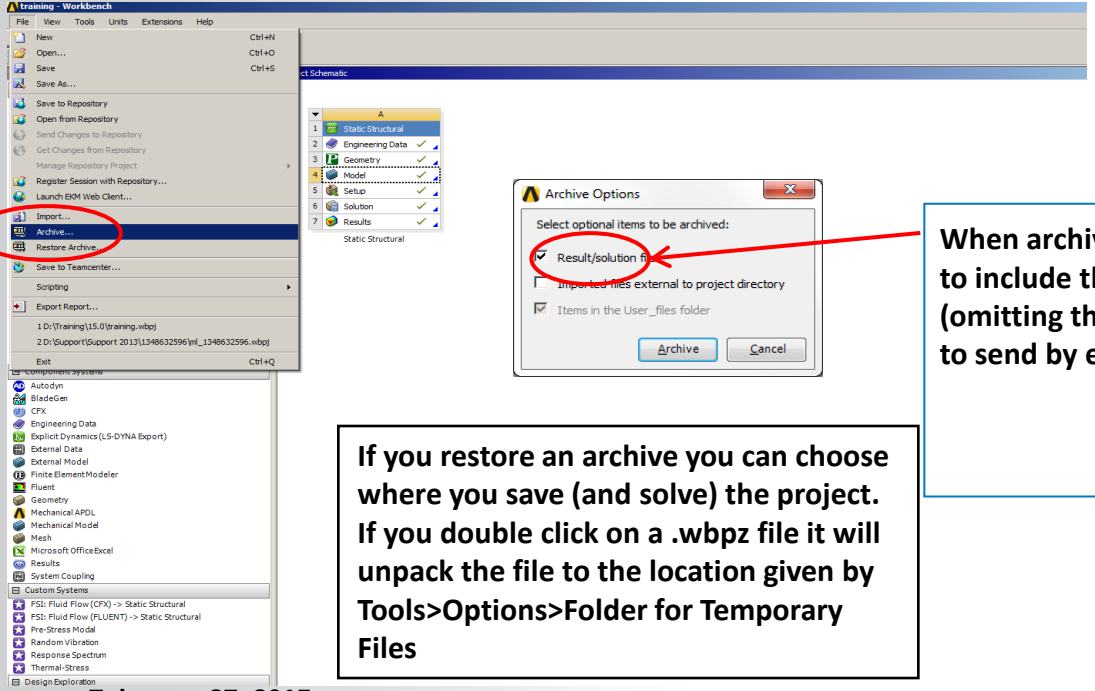
. . . File Location on Disk

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.



. . . Use of Archive / Restore

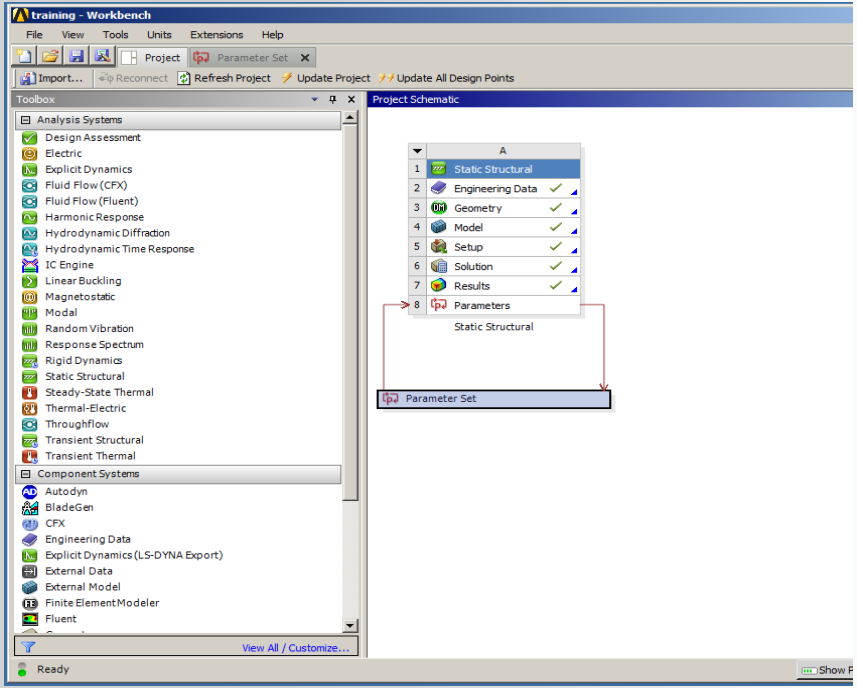
The workbench project comprises many files and directories. If you need to either archive the project, or bundle it to send to us for a Technical Support query, use the '**Archive**' tool. This generates a single zipfile of the entire project.



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)

If you restore an archive you can choose where you save (and solve) the project. If you double click on a .wbpz file it will unpack the file to the location given by Tools>Options>Folder for Temporary Files

. . . Working With Parameters [1]



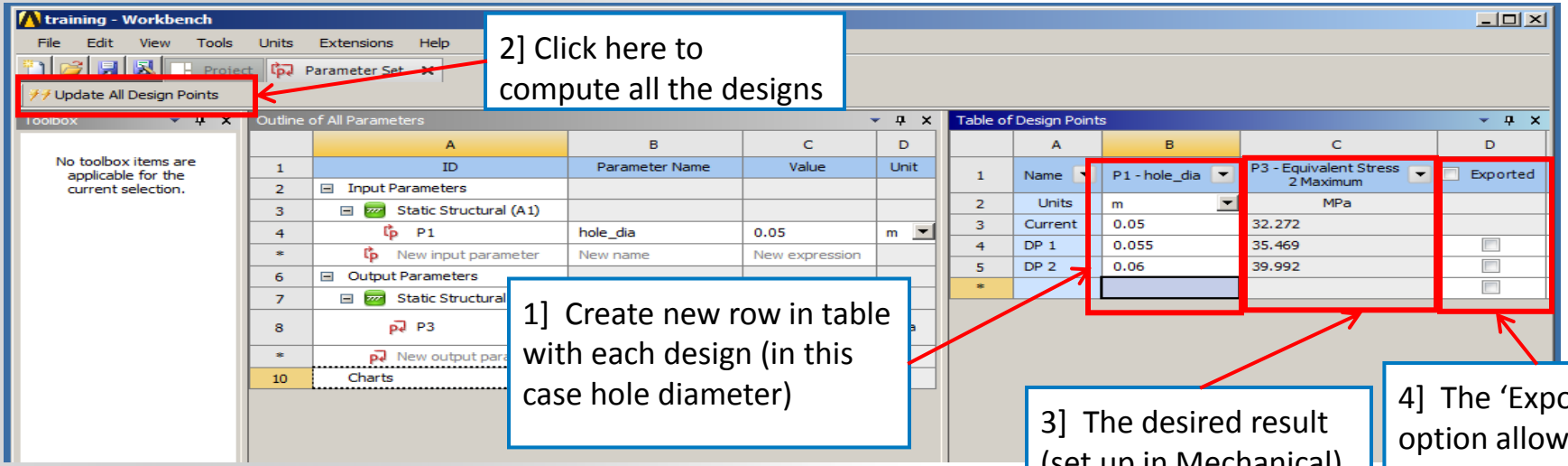
Most Workbench applications will let you specify key quantities as a **parameter** (rather than a constant). This will be covered later.

In this example:

- When creating the geometry in DesignModeler, hole diameter is set to be an *input parameter*.
- When reviewing the results, the maximum stress is set as an *output parameter*

We could just have easily set up a CFD analysis, looking at different loading conditions and reporting the pressure drops.

. . . Working With Parameters [2]

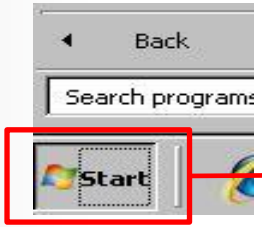


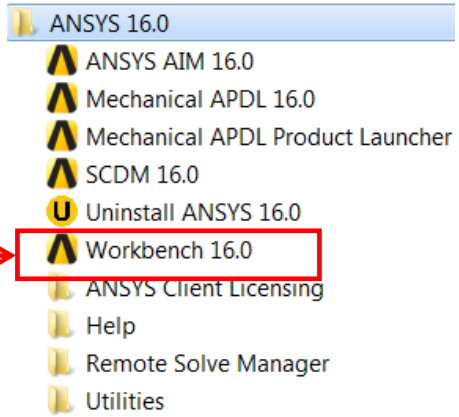
- Clicking on 'Parameter Set' lets us vary these parameters.
- Four different geometric designs are being tested.
- The whole process is automated. Workbench will recursively:
  i. Create the geometry, based on the parameters in the table
  ii. Take this into Mechanical and remesh and solve and then the postprocessor
- The user just needs to sit back and wait, and the matrix of experiments (each requiring several different applications to be launched in turn) is computed automatically.

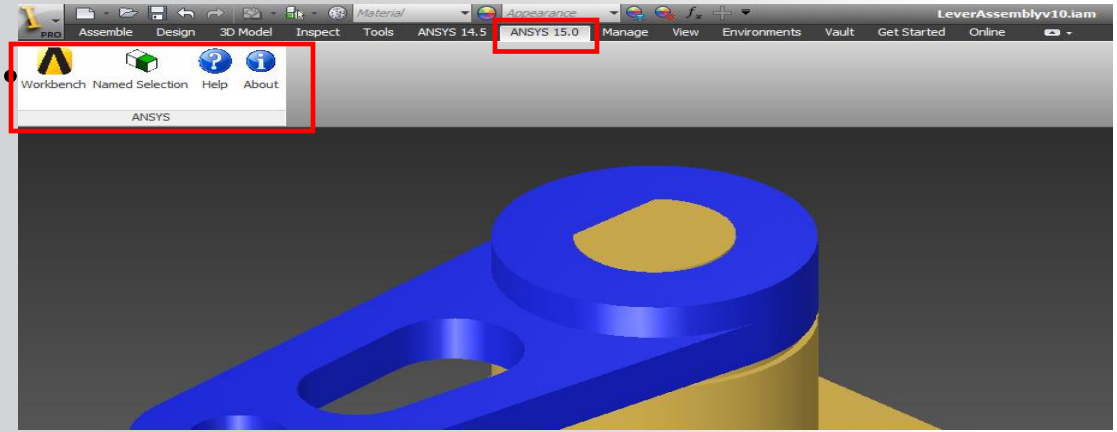
… Starting Mechanical

There are two methods of launching Workbench:

- **From the Windows start menu:**







. . . Working With Units

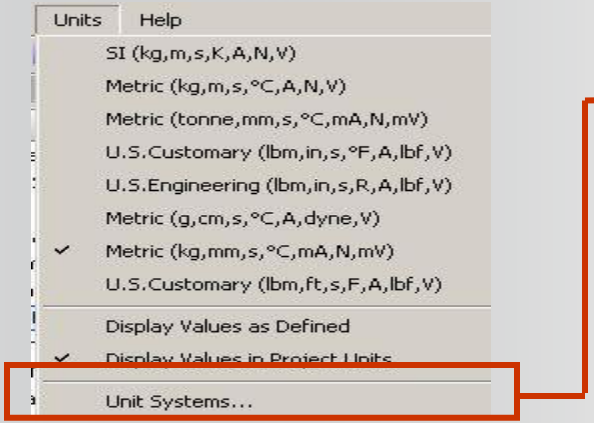
The Units menu in Workbench:

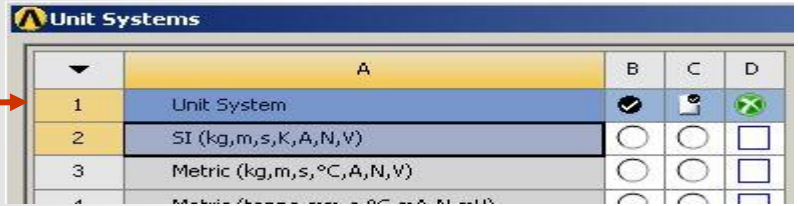
- **Select from predefined unit systems.**
- **Create custom unit systems.**
- **Controls unit display for Engineering Data, Parameters and Charts.**
- **Activate the Units System dialog to unit display preferences.**

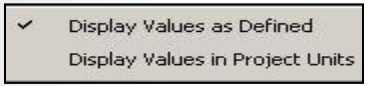
Active Project

Default Unit System

Suppress Unit Display





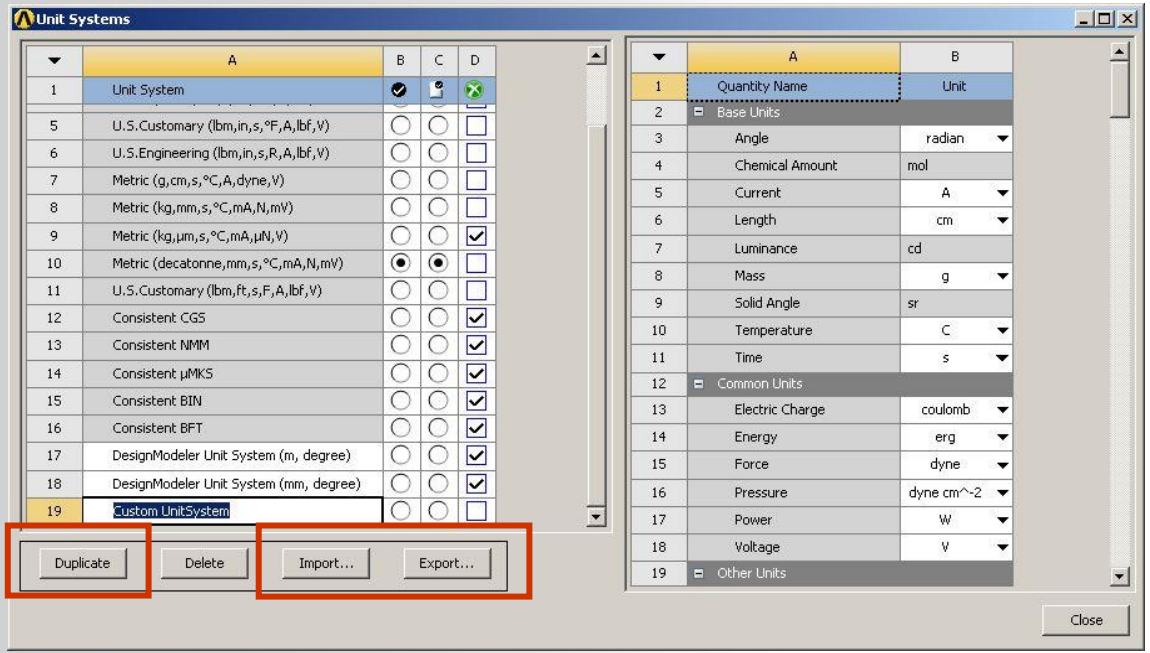


Units can be displayed in the active Project system or as they were defined in their source (e.g. CAD system).

. . . Working With Units

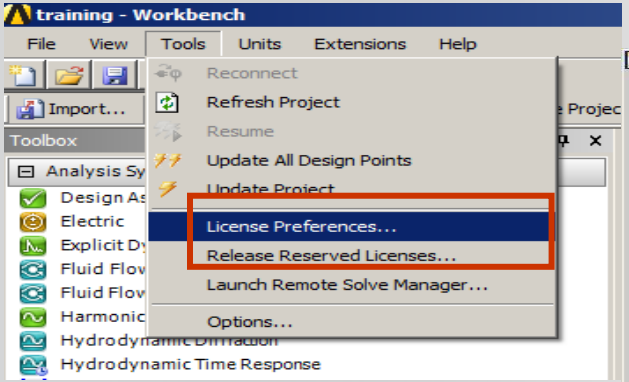
Create custom unit systems by duplicating existing systems then modifying.

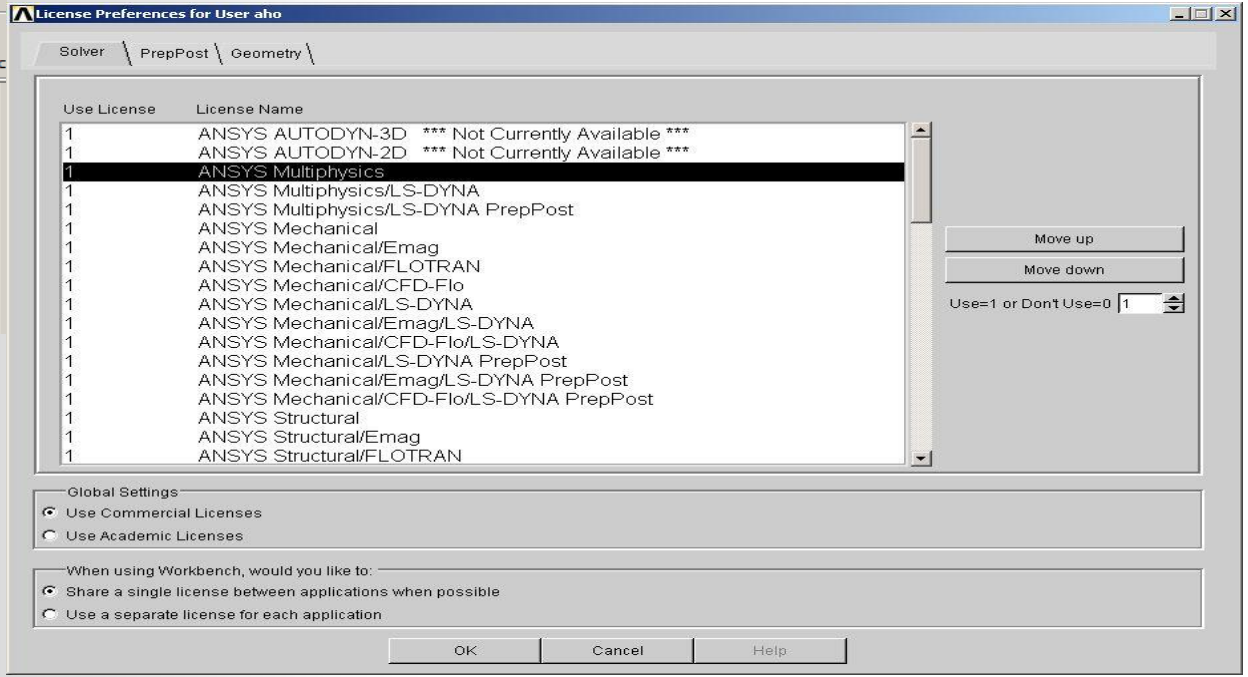
Custom unit systems can be exported and imported.



. . . License Preferences

Workbench license control is handled through the user interface shown below, activated from the Workbench project page:"Tools > License Preferences . . . "

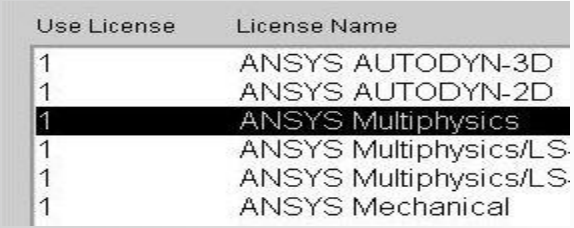


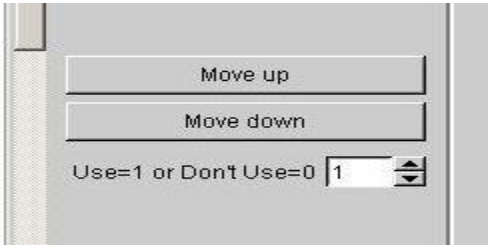


. . . License Preferences

The order of license preference is specified using the up/down arrows (first available is used).

- **The "Use License" column indicates desired licenses to use (0 = off, 1 = on).**





Workbench users can specify whether a single license is shared when multiple applications are open, or if each application accesses their own license.