

## **Launching ANSYS Workbench 12.0**

- 1) From Start menu
  - All programs > ANSYS 12.0 > ANSYS Workbench
- 2) Changing License Preferences (Only needs to be done once on first start-up)
  - Tool > License Preferences
  - Move: 'Academic Research License' to top of list (Stress Analysis)
  - Move: 'Academic Research License' CFD to top of list (Computational Fluid Analysis)
  - Apply Settings
- 3) Changing Toolbox layout (ANSYS Workbench)

Add or remove the following components by following the corresponding links

You will need the toolbox and project schematic

- View > Toolbox
    - Allows you begin specific new analysis; i.e. Static Structural
  - View > Toolbox Customization
    - Allows you to add or remove specific analysis options from your toolbox. Will probably only be required once.
  - View > Project Schematic
    - Window for outlining current project analysis
  - View > Files
    - Lists active files in current analysis
- 4) Beginning a new analysis
    - Double click option under 'Analysis Systems' to begin a preformatted analysis
    - Double click option under 'Custom Systems' to begin a custom analysis
  
    - New window will appear in 'Project Schematic Window'
      - Double click Geometry to import Geometry file into analysis
      - Double click Setup to begin analysis