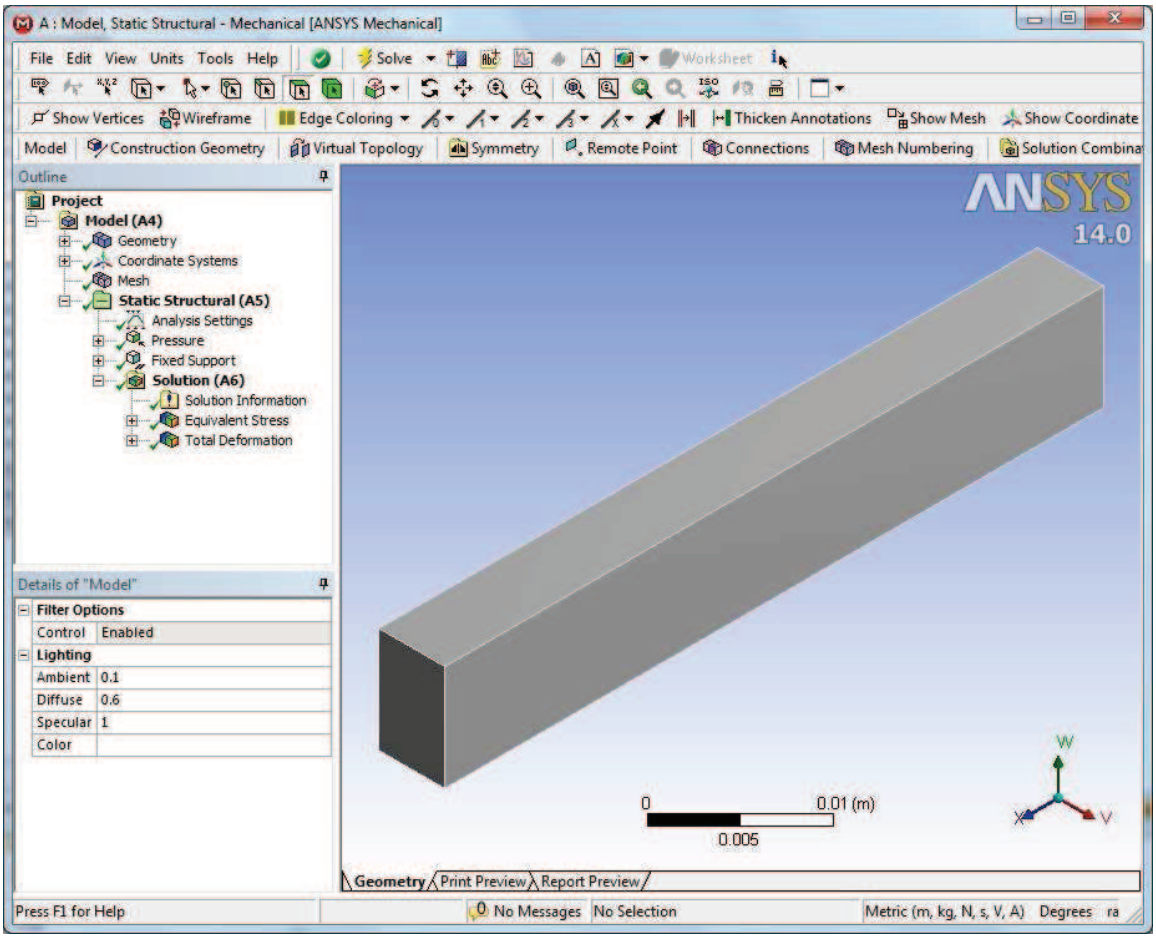
- Restore Workbench Project From Sample Archive
- Prepare DOES ReferenceRun
- Prepare DOES ApplicationModel
- Prepare DOES Task
- Prepare DOES DesignSpace
- Save DOES ExpertDesign

Virtual Engines Example

- Prepare Virtual Engines Batch Run
- Submit Virtual Engines Batch Run
- Prepare DOES ReferenceRun
- Prepare DOES ApplicationModel
- Prepare DOES Task
- Prepare DOES DesignSpace
- Save DOES ExpertDesign

## ANSYS Example

This example shows how to optimize an ANSYS Workbench project with DOES. DOES interfaces with the ANSYS Parameter Manager to change and observe values. The example finds the dimensions Example files for this example may be found in the common application data folder (usually C:\ProgramData) at subfolder OPTIMUM Power Technology\DOES\Samples\ANSYS14.

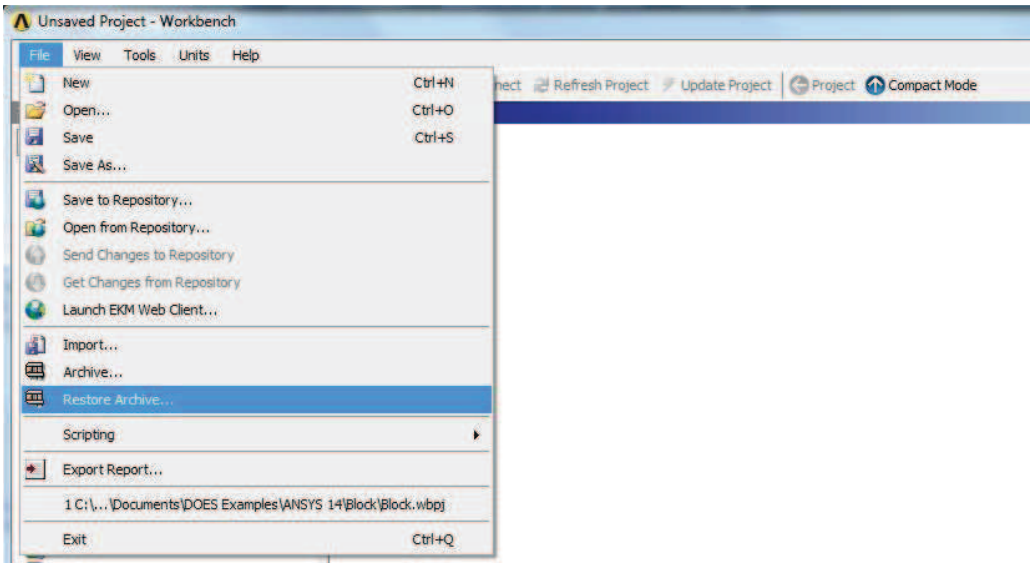



Optimizing the sample ANSYS Workbench project with DOES consists of the following steps:

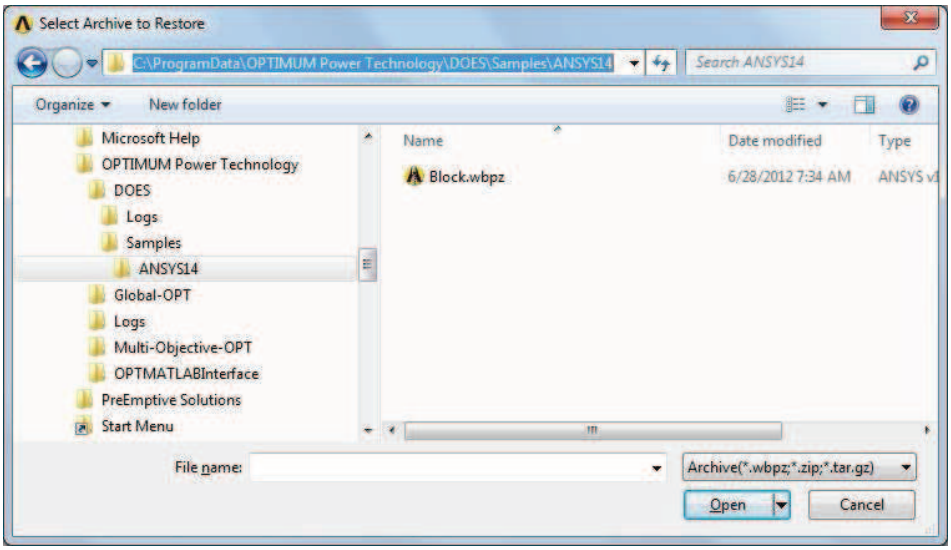
- Restore Workbench Project From Sample Archive
- Prepare DOES ReferenceRun
- Prepare DOES ApplicationModel
- Prepare DOES Task
- Prepare DOES DesignSpace
- Save DOES ExpertDesign

## Restore Workbench Project From Sample Archive

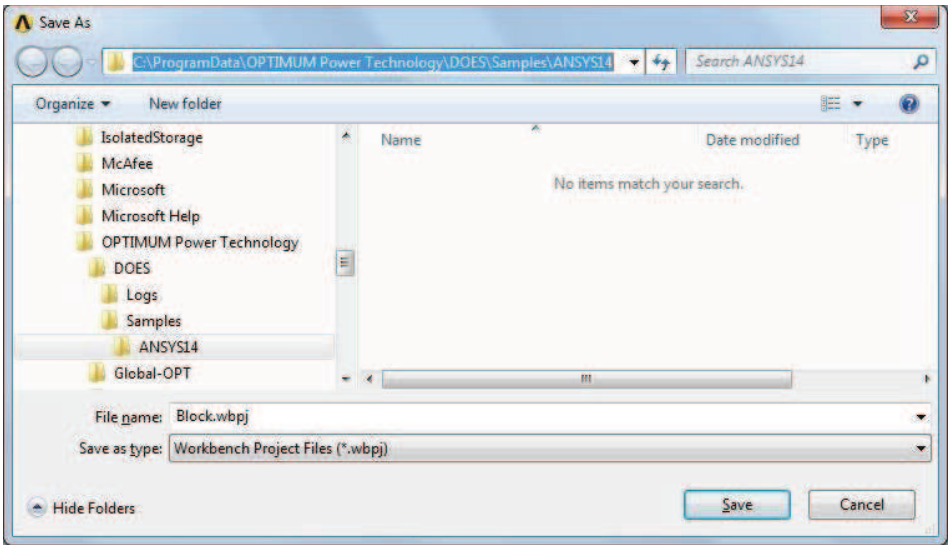
The first step is to restore the Block Workbench project from the DOES Samples. Start ANSYS Workbench and select File then Restore Archive... from the Workbench menu.



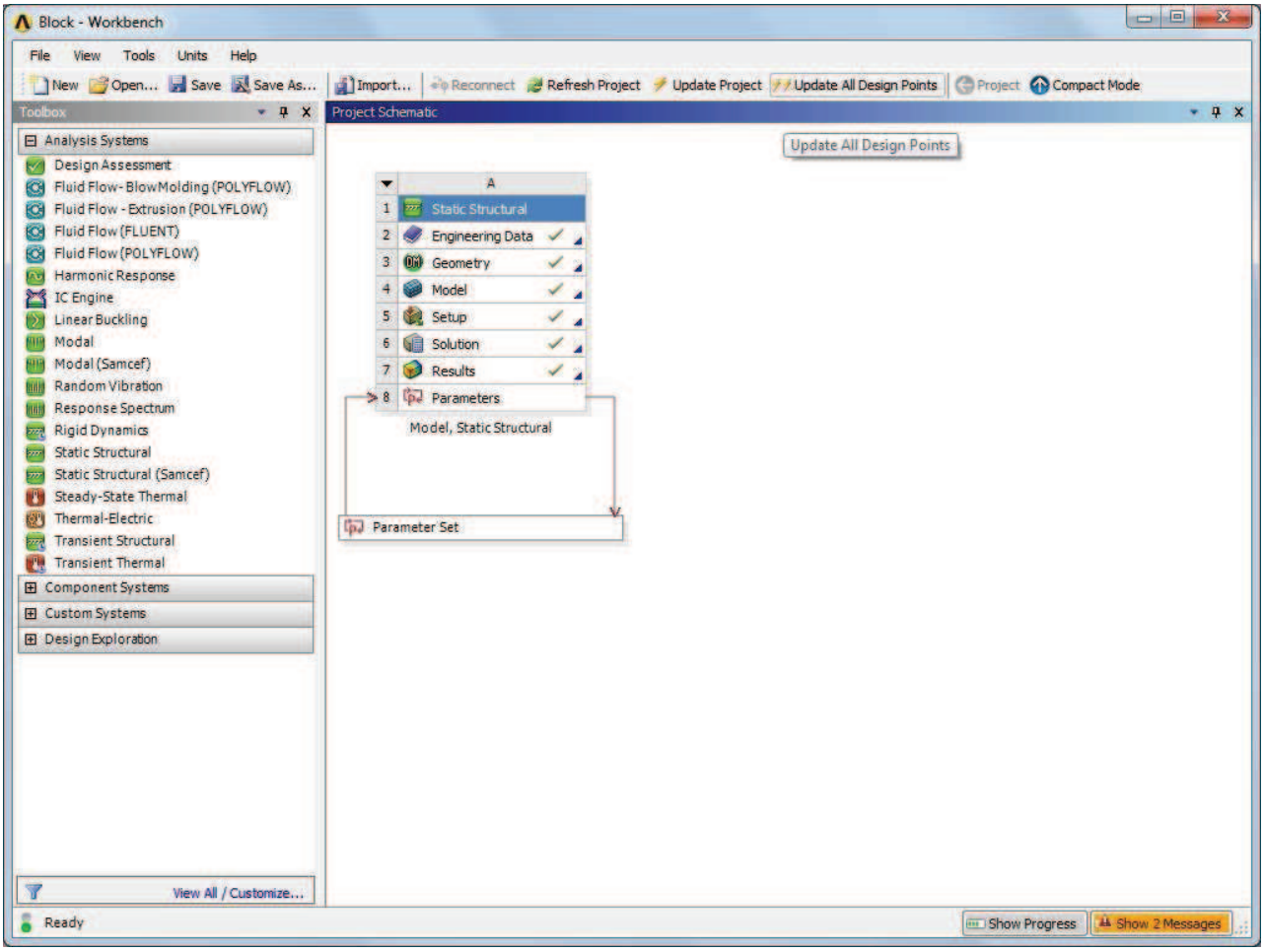

ANSYS Workbench responds with the Select Archive To Restore dialog. Navigate to the DOES sample folder at C:\ProgramData\OPTIMUM Power Technology\DOES\Samples\ANSYS14 and click Block.wbpz. Click the "Open" button.




ANSYS Workbench responds with a Save As dialog. Navigate back to the DOES sample folder C:\ProgramData\OPTIMUM Power Technology\DOES\Samples\ANSYS14 if necessary and click the Save button.



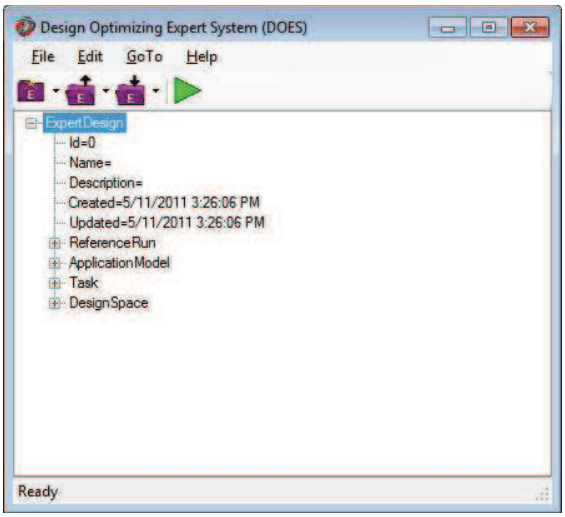

ANSYS Workbench restores the Archive to Block.wpbj and displays the Project Schematic when complete.



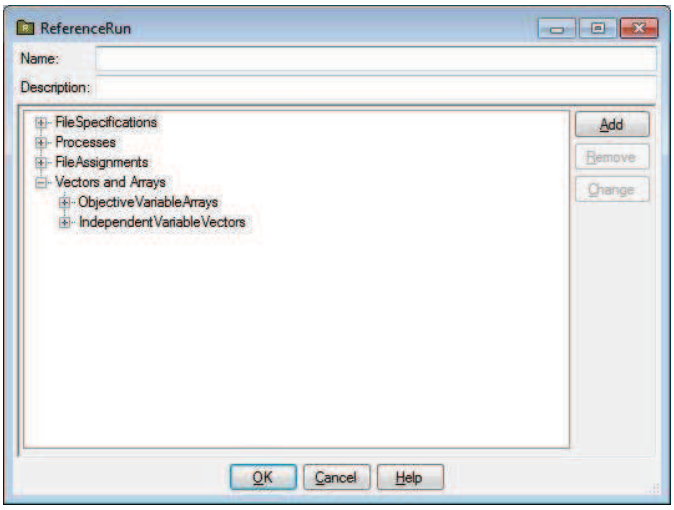

Close the ANSYS Workbench.

## Prepare DOES ReferenceRun

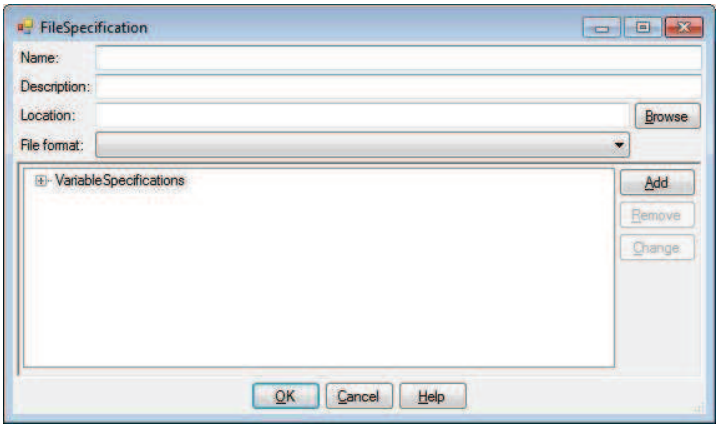
Start Expert Design.



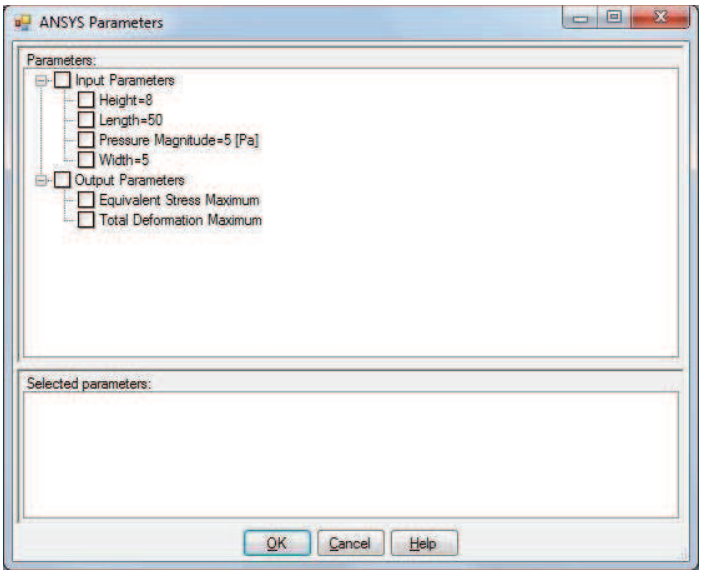

The first step in creating an Expert Design model is to create a ReferenceRun. The ReferenceRun is a definition of all the files, variables within each file, and processes that make up your model. Creation of a ReferenceRun is a Knowledge Engineer task and requires knowledge of the file layouts and processes that make up your model. Right-click on the ReferenceRun node and select Change from the menu. The ReferenceRun dialog is displayed.



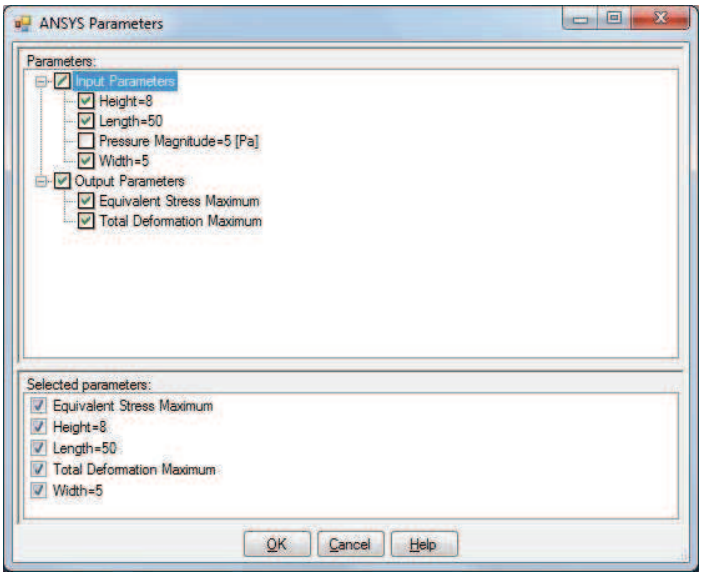

Right-click on the FileSpecifications node in the ReferenceRun dialog and select Add from the menu. The FileSpecification dialog is displayed.



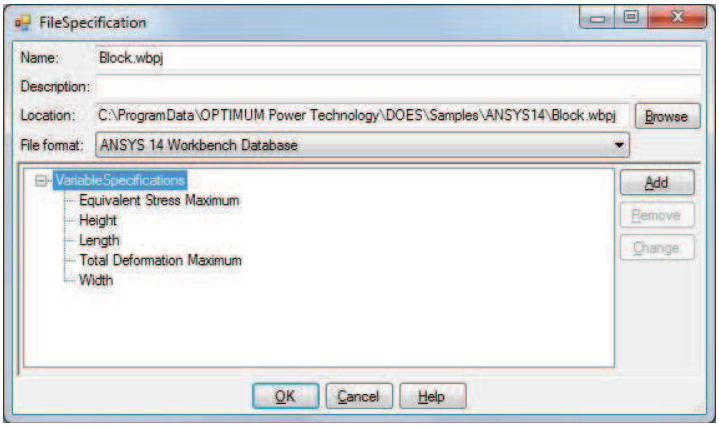

Click the Browse button, navigate to the DOES sample folder C:\ProgramData\OPTIMUM Power Technology\DOES\Samples\ANSYS14 and select Block.wbpj as the location. The FileSpecification dialog will fill in the FileSpecification name, location and format for you. Right click on the VariableSpecifications node in the FileSpecification dialog and select Add from the menu. The ANSYS Parameters dialog is displayed after a short interval 30-60 seconds. This wait occurs on the first DOES access to an ANSYS Workbench project because DOES needs to run ANSYS to extract and cache the ANSYS Parameter definitions from the project. Subsequent accesses to the ANSYS parameters are from the DOES cache and occur instantaneously.



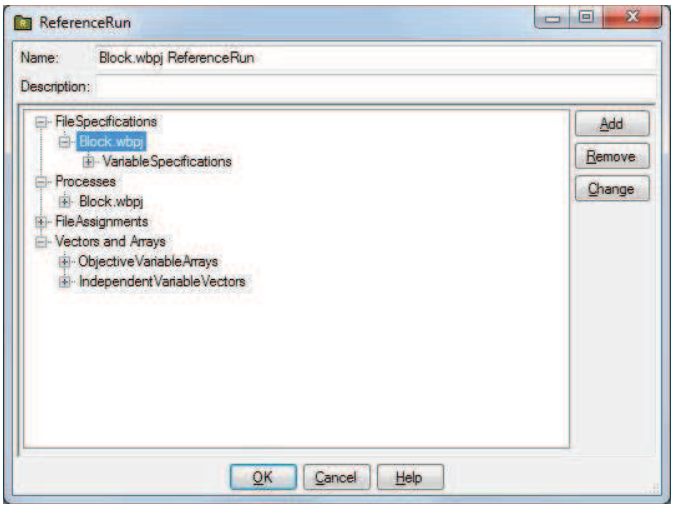

The ANSYS Parameters dialog displays a tree view of Input and Output parameters. Click the checkboxes on the Height, Length and Width Input Parameter nodes. Click the checkbox on the Output Parameters node to select all the Output Parameters. Your Variable Selection dialog should look like this:




Click the OK button to return to the FileSpecification dialog which should look like this:



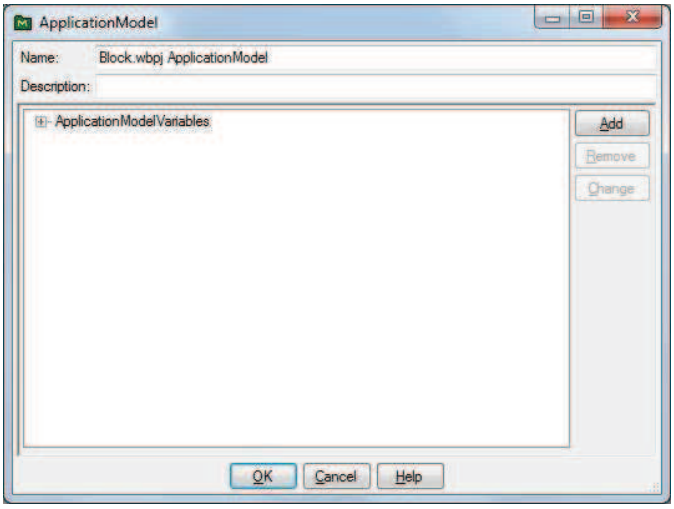

Click the OK button to return to the ReferenceRun dialog which should look like this:



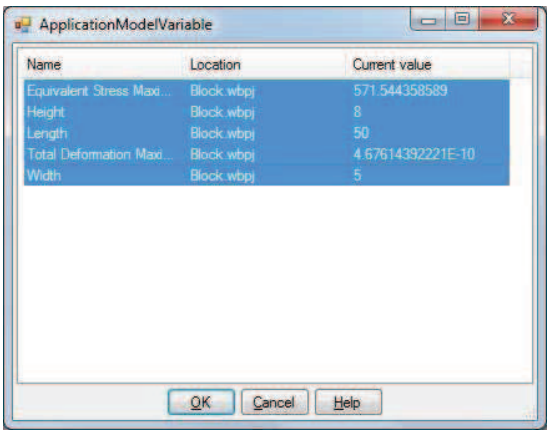

The DOES ANSYS Parameter Manager interface has filled in all the ReferenceRun specifications with a few mouse clicks! Note that, in addition to the FileSpecification, the DOES ANSYS Parameter Manager interface has filled in the Name and Process. Feel free to explore the ReferenceModel nodes by double clicking them. When you are ready to continue, dismiss all ReferenceRun dialogs to return to the main Design Window.

## Prepare DOES ApplicationModel

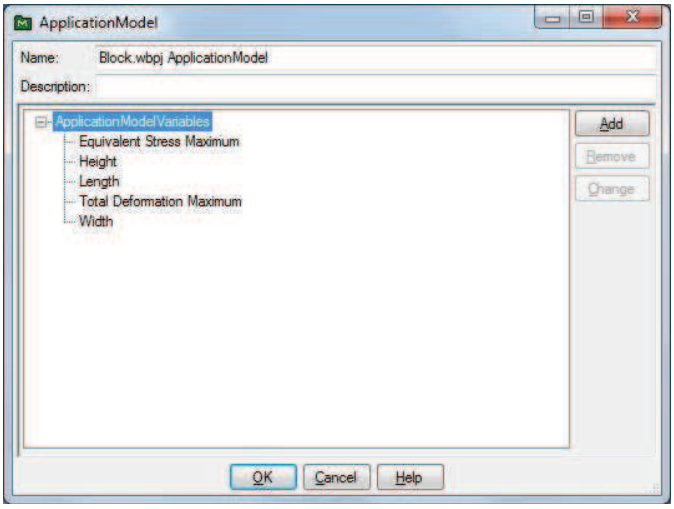
The next step is to create the ApplicationModel. An ApplicationModel is a set of Variables used for a specific application of DOES. Creation of an ApplicationModel is a Knowledge Engineer task. Right-click on the ApplicationModel node and select Change from the menu to display the ApplicationModel dialog.




Note that the DOES ANSYS Parameter Interface interface has supplied a name for the ApplicationModel. Right-click the ApplicationModelVariables node on the ApplicationModel dialog and select Add from the menu to display the ApplicationModelVariable dialog. Scroll to the bottom of the list of VariableSpecifications. Click Exp1.S(4).EntranceDia. Shift-click Exp1.S(4).Len and BMEP. The dialog will look like this:



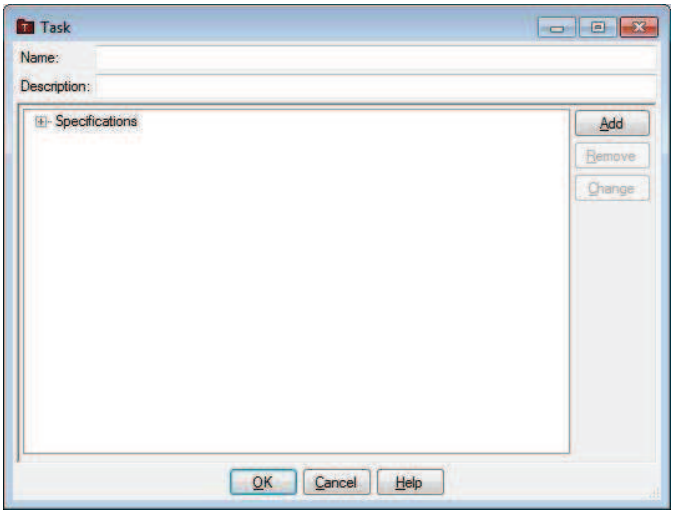
Click OK to return to the ApplicationModel dialog.



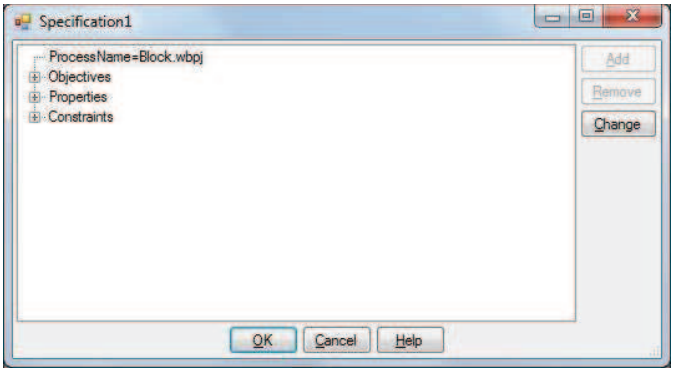

Click OK to close the ApplicationModel dialog.

## Prepare DOES Task

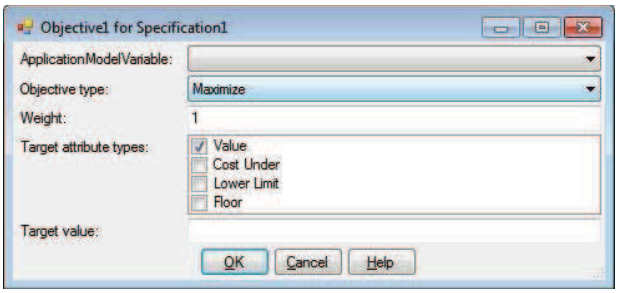
Now it is time to create the Task for this ExpertDesign. Creation of a Task is an Application Engineer task. The Task is where you define the Objectives of an ExpertDesign. Right-click on the Task node and select Change from the menu to display the Task dialog.



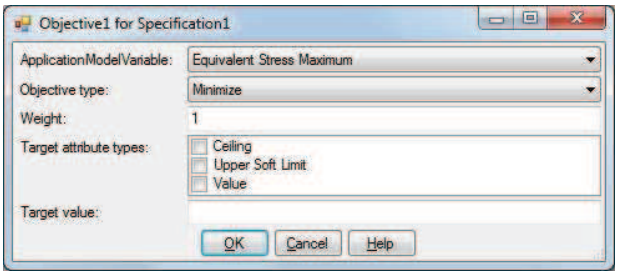

Type Minimize Deformation and Stress as the Task name. Click the Specifications node and click the Add button to display the Specification dialog.



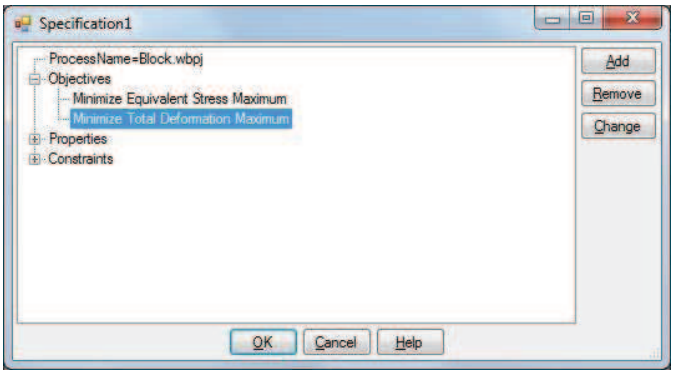

Note that the Process has been filled in already because the ReferenceRun contains only one Process. If the ReferenceRun had contained multiple Processes, you would have been required to select the Process node, click the Change button and select the Process for this Specification. Click the Objectives node and click the Add button. The Objective Dialog is displayed.



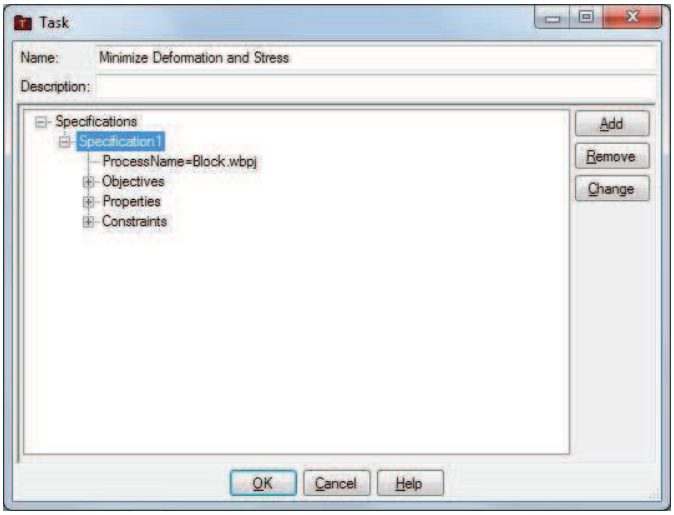

Select Equivalent Stress Maximum as the ApplicationModel variable and select Minimize as the Objective type. Your dialog will look like this:




Click OK to return to the Specification dialog, then create another Objective to Minimize Total Deformation Maximum. Your Specification dialog will look like the following:




Click OK to return to the Task dialog. Your Task dialog will look like the following:

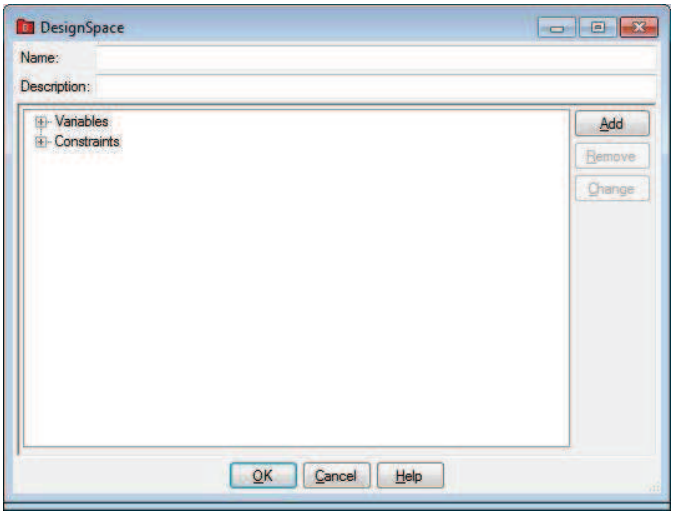



Click OK to close the Task dialog.

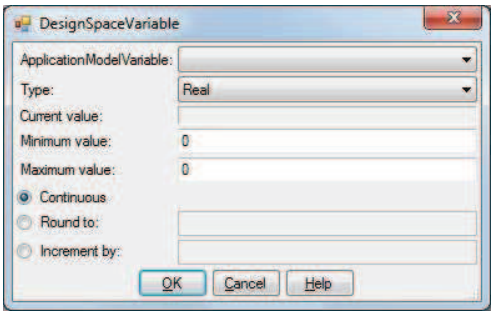
## Prepare DOES DesignSpace

Now it is time to create the DesignSpace for this ExpertDesign. Creation of a DesignSpace is an Application Engineer task. The DesignSpace is where you define the variables to

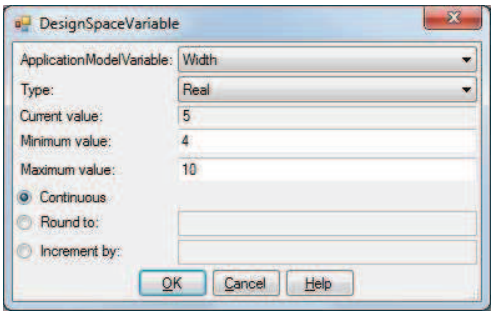
vary to achieve the Objectives that you defined in the Task. Right-click on the DesignSpace node and select Change from the menu to display the DesignSpace dialog.



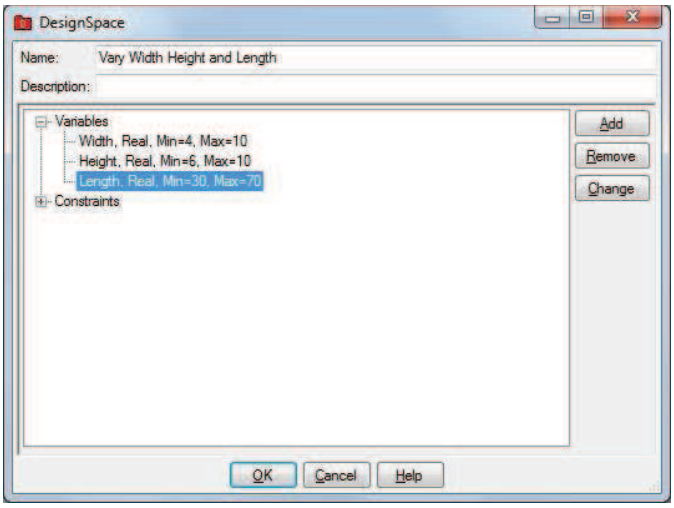

Type Vary Width Height and Length as the DesignSpace name. Click the Variables node and click the Add button to display the DesignSpaceVariable dialog.




Select Width as the ApplicationModel variable and type 4 as the Minimum value and 10 as the Maximum value.



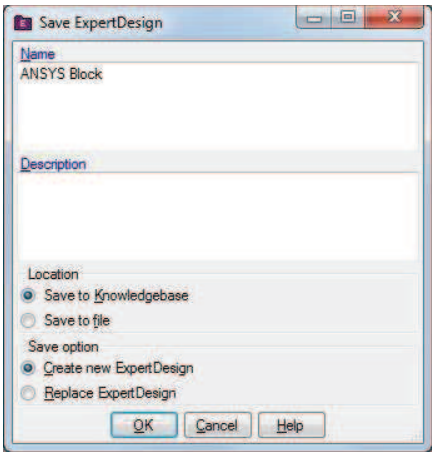

Click OK to return to the DesignSpace dialog and repeat selecting Height as the ApplicationModel variable, 6 as the Minimum value and 10 as the Maximum value, then finally with Length as the ApplicationModel variable, 30 as the Minimum value and 70 as the Maximum value. The DesignSpace dialog will look like this:



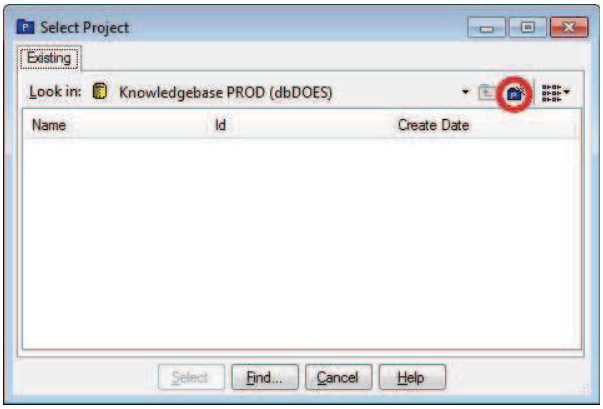

Click OK to close the DesignSpace dialog.

## Save DOES ExpertDesign

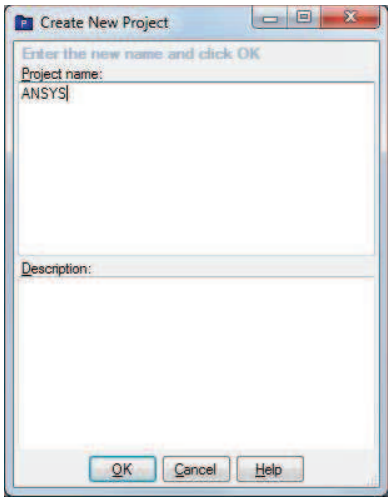
The ExpertDesign must be saved to the Knowledgebase before it can be run. Right-click the ExpertDesign node and click Save from the menu. The Save ExpertDesign dialog will be displayed. Type ANSYS Block as the Name.




Click the OK button. The Select Project dialog will be displayed.




Click the New Project button (circled). The Create New Project dialog will be displayed. Type ANSYS as the Project name.




Click the OK button. The Select Project dialog will be updated. Click the Select button on the Select Project dialog. Click the OK button on the Save ReferenceRun, Save ApplicationModel, Save Task and Save DesignSpace dialogs. The save will take a short interval (30-60 seconds) as DOES invokes ANSYS while it validates the ExpertDesign. When complete, the DOES Design window updates ExpertDesign Id and Name.

## Virtual Engines Example

This example shows how to prepare a Virtual Engines model, a YZ426 engine, to run under DOES. We will attempt to maximize BMEP over a sweep of RPMs by varying the length and diameter of a section of the exhaust pipe. Example files for this example may be found in the common application data folder (usually C:\ProgramData) at subfolder OPTIMUM Power Technology\DOES\Samples\VirtualEngines.
**You should follow the instructions in this tutorial to re-create these files as the files contain information that may be specific to your particular Automated Design or Virtual Engines setup**.

Preparing a Virtual Engines model to run under DOES consists of the following steps:

- Prepare Virtual Engines Batch Run
- Submit Virtual Engines Batch Run
- Prepare DOES ReferenceRun
- Prepare DOES ApplicationModel
- Prepare DOES Task
- Prepare DOES DesignSpace
- Save DOES ExpertDesign

## Prepare Virtual Engines Batch Run

You must first prepare a Virtual Engines batch run. This example assumes that you have imported the Virtual Engines YZ426 sample. Load this sample and its TestProcedure into Virtual Engines or Automated Design.

Click Tools then Submission options from the menu to display the Submission Options dialog.