



Course 3

Running Simulation in GUI mode

Table of Contents

Creating workspace and design	3
Creating workspace using wizard	3
Creating/Adding files to the design	8
Compiling files	8
Initializing the Simulation	9
Running the Simulation	11

Creating workspace and design

Active-HDL allows collecting individual designs along with their resources (source files, output files with simulation results, etc.) into a group of projects referred to as a **workspace**. A workspace allows working with several projects simultaneously because it acts as a container for Active-HDL designs.

Creating workspace using wizard

- Go to menu **File** | **New** and select **Workspace**
- Type the name of the workspace and select the location for where you would like your projects to be saved
- Check the **Add New Design to Workspace** option, and click OK.

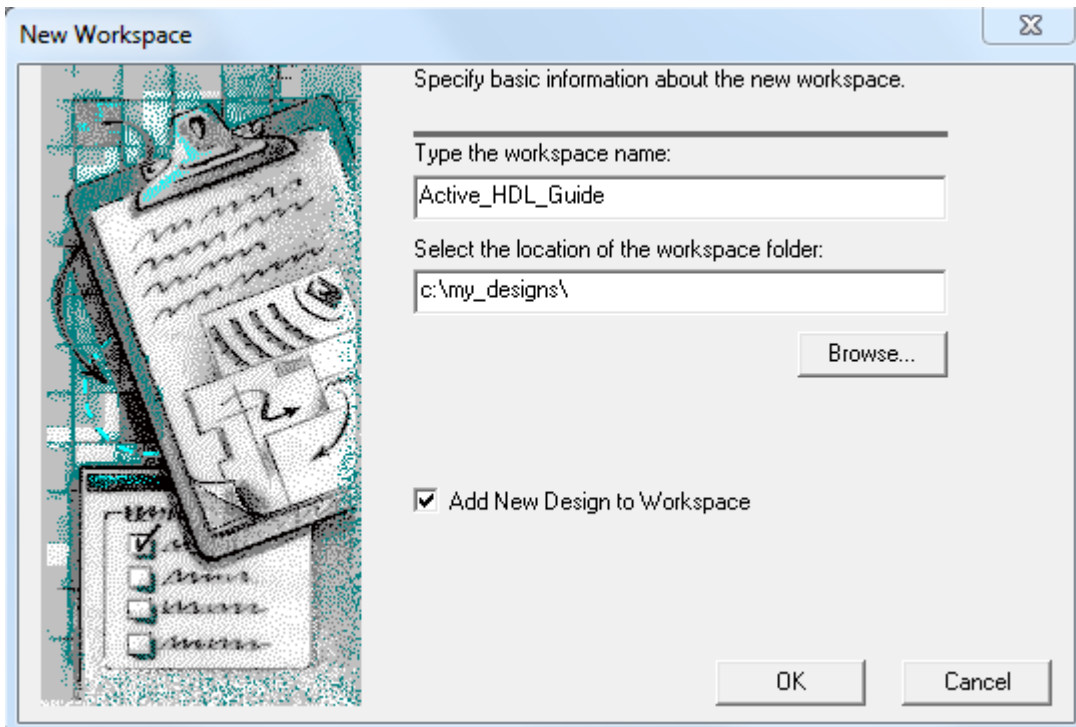


Figure 1 Create new workspace wizard: workspace window

- The **New Design Wizard** window will pop up and give you four options of creating your design:

Create an Empty Design

Creates an empty design with no synthesis or implementation tool set and disables the Design Flow Manager. It allows you to specify the Design Language and Target Technology of the new design.

Create an Empty Design with Design Flow

Creates an empty design and enables the Design Flow Manager. It allows you to specify the Synthesis tool, Implementation tool, Block Diagram Configuration, and Default HDL Language.

Add existing Resource Files

Creates an empty design and enables the Design Flow Manager. It allows you to specify the sources to be added prior to creating the design, as well as specify the Synthesis tool, Implementation tool, Block Diagram Configuration, and Default HDL Language.

Import a Design from Active-CAD

Creates an empty design, imports an Active-CAD project, and enables the Design Flow Manager. It allows you to specify the sources to be added prior to creating the design, as well as specify the Synthesis tool, Implementation tool, Block Diagram Configuration, and Default HDL Language.

Select an option and continue to the next window.

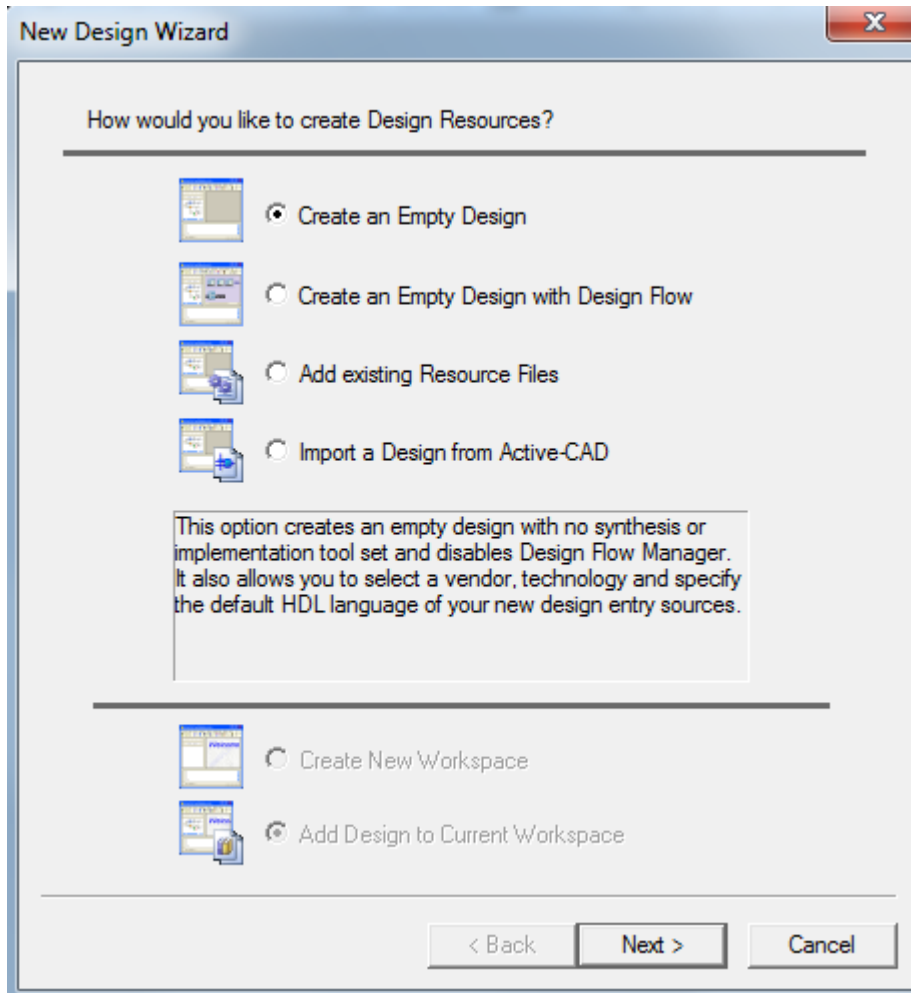
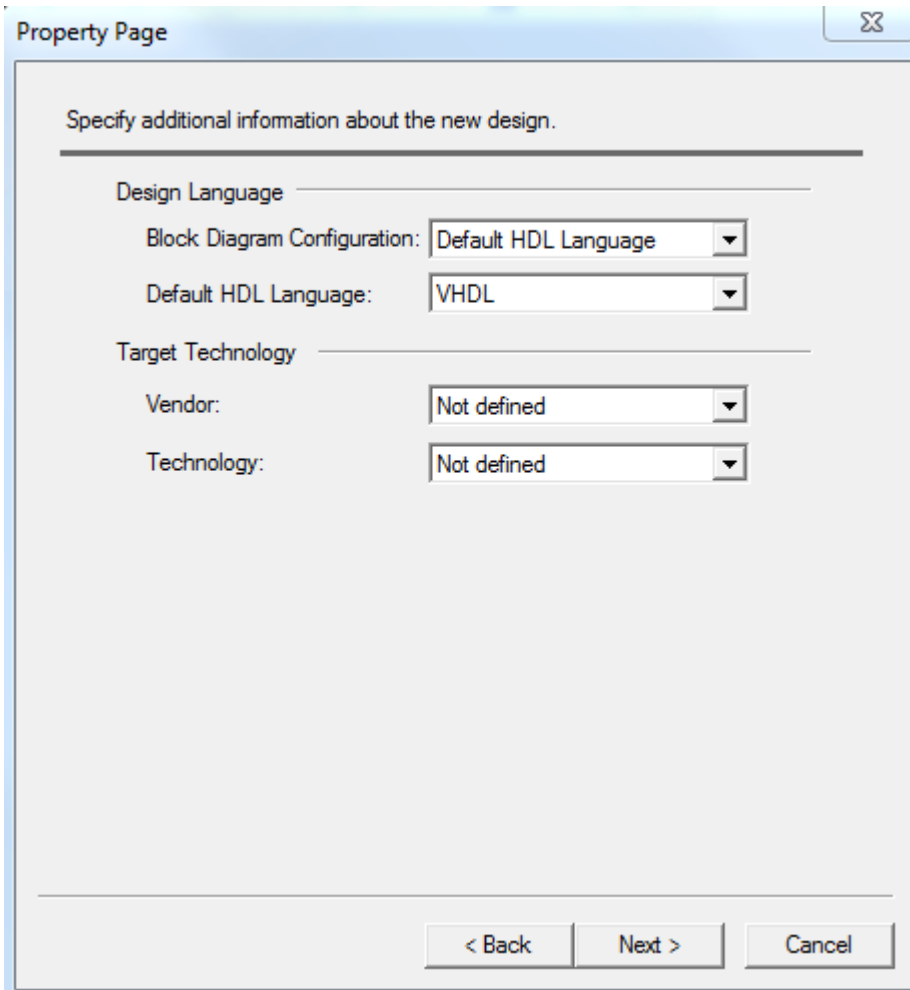


Figure 2 create new workspace wizard: new design

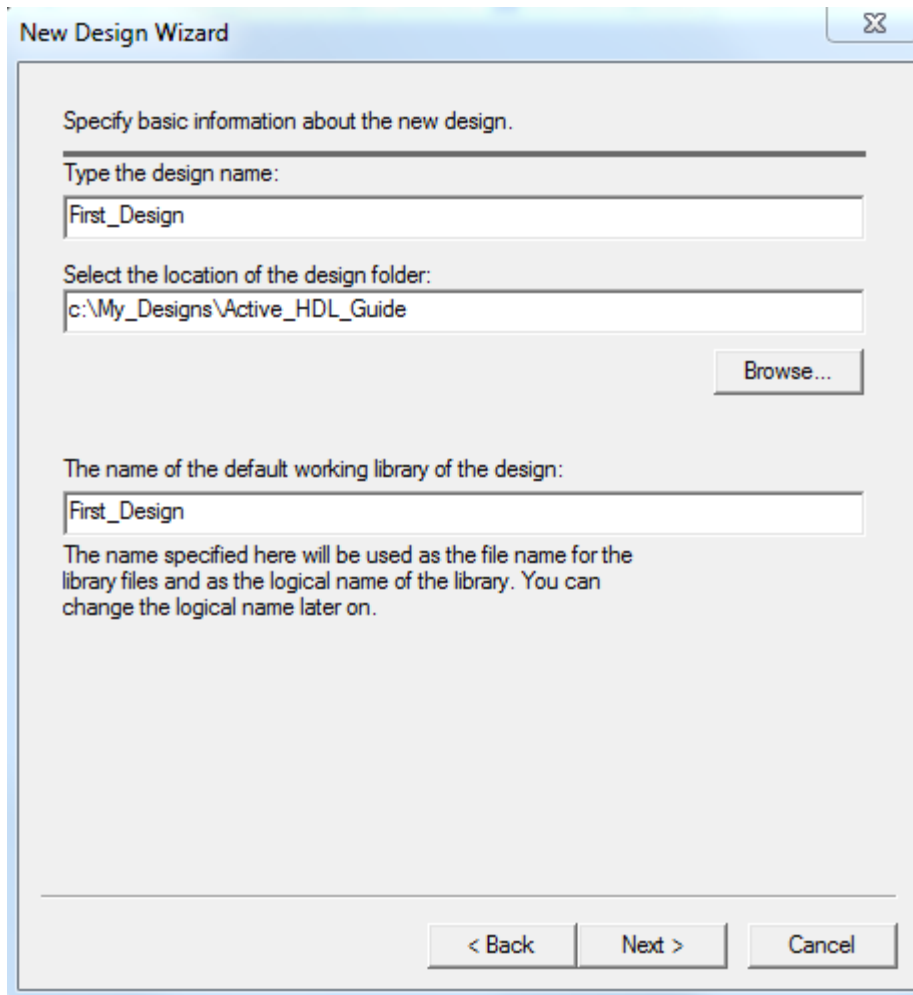
The next window is the **Property Page** which allows you to choose the **Design Language: Block Diagram Configuration** and **Default HDL Language** (Verilog or VHDL). You can then specify the **Target Technology: Vendor** (Lattice, Altera, etc.) and **Technology**. Otherwise, accept the default settings and continue to the next window.



The screenshot shows a dialog box titled "Property Page" with a close button in the top right corner. Inside the dialog, there is a section titled "Specify additional information about the new design." followed by a horizontal line. Below this line, there are two main sections: "Design Language" and "Target Technology", each followed by a horizontal line. Under "Design Language", there are two dropdown menus: "Block Diagram Configuration:" with the value "Default HDL Language" and "Default HDL Language:" with the value "VHDL". Under "Target Technology", there are two dropdown menus: "Vendor:" with the value "Not defined" and "Technology:" with the value "Not defined". At the bottom of the dialog, there are three buttons: "< Back", "Next >", and "Cancel".

Figure 3 create new workspace wizard: property page

The next window asks to specify the name of the design and the name of the default working library. By default, the name is the same for both.



The image shows a 'New Design Wizard' dialog box with a title bar and a close button. The main area contains three sections for specifying design information. The first section, 'Specify basic information about the new design.', has a label 'Type the design name:' followed by a text box containing 'First_Design'. The second section, 'Select the location of the design folder:', has a label followed by a text box containing 'c:\My_Designs\Active_HDL_Guide' and a 'Browse...' button to its right. The third section, 'The name of the default working library of the design:', has a label followed by a text box containing 'First_Design'. Below this text box is a paragraph explaining that the name will be used as the file name for library files and as the logical name of the library, and that it can be changed later. At the bottom of the dialog are three buttons: '< Back', 'Next >', and 'Cancel'.

Specify basic information about the new design.

Type the design name:

First_Design

Select the location of the design folder:

c:\My_Designs\Active_HDL_Guide

Browse...

The name of the default working library of the design:

First_Design

The name specified here will be used as the file name for the library files and as the logical name of the library. You can change the logical name later on.

< Back Next > Cancel

Figure 4 create new workspace wizard: design name

The last window will show you a summary of the design specifications. If you need to change something, click the back button. Otherwise click Finish.

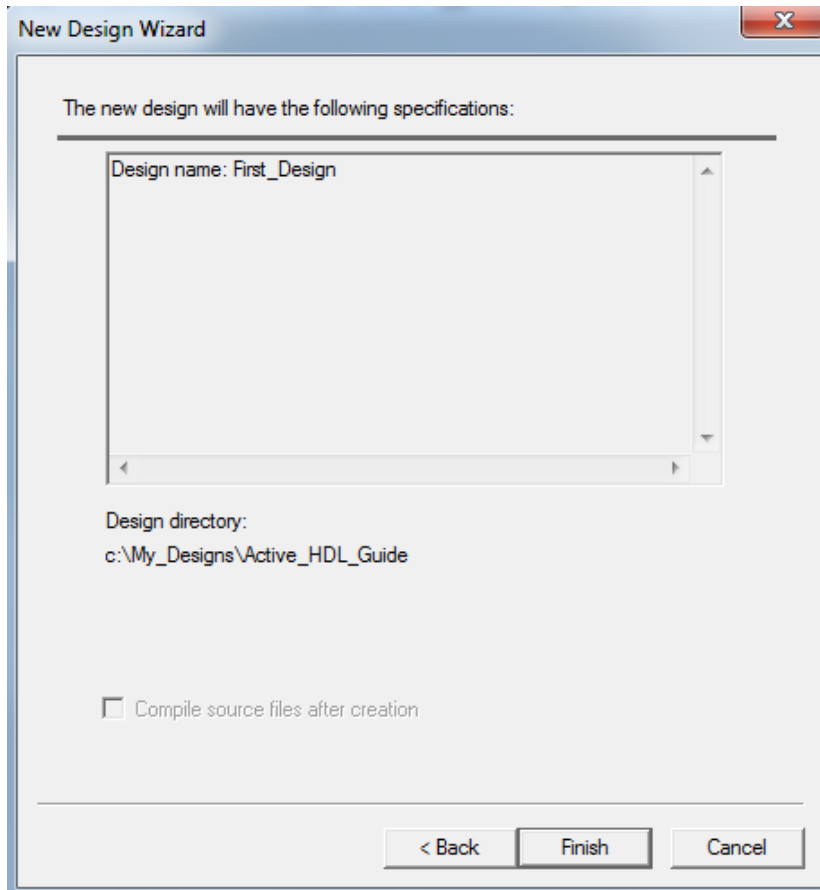


Figure 5 create new workspace wizard: design specifications

- The **Design Browser** now shows a workspace name and new design attached with it

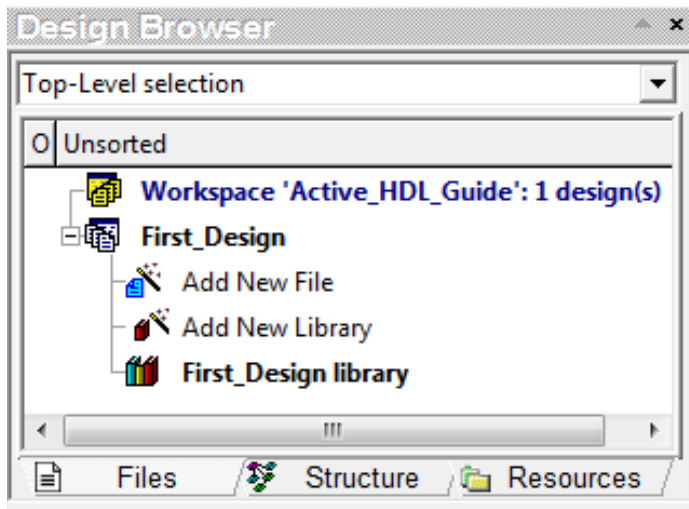


Figure 6 Design manager after workspace and design created

Creating/Adding files to the design

- To create a new file, right click on the “Add New File” option, click on “New” and then select the desired operation.
- To add an existing file, right click on “Add New File” and select “Add Files to Design” option
- You can also use **File | New** menu to open new files and save it to design directory.

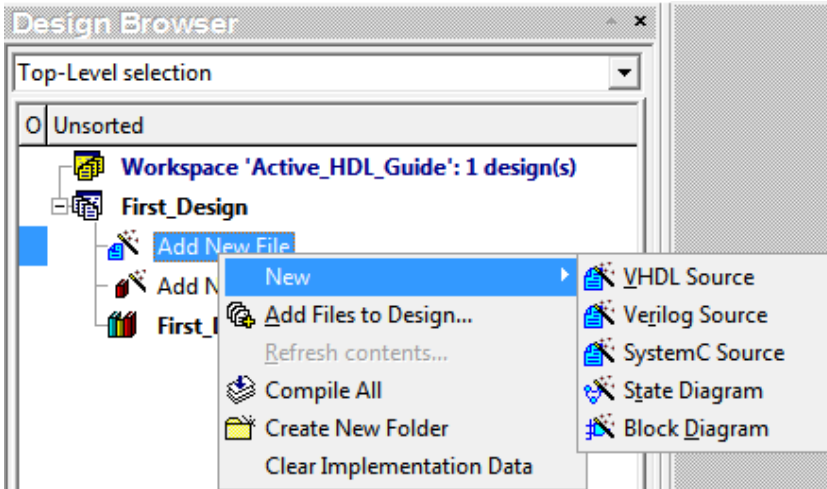


Figure 7 Add new file to design

Compiling files

If you want to compile just one file go the Files tab in the Design Browser, select the file, right click and choose **Compile** from the shortcut menu for a source file.

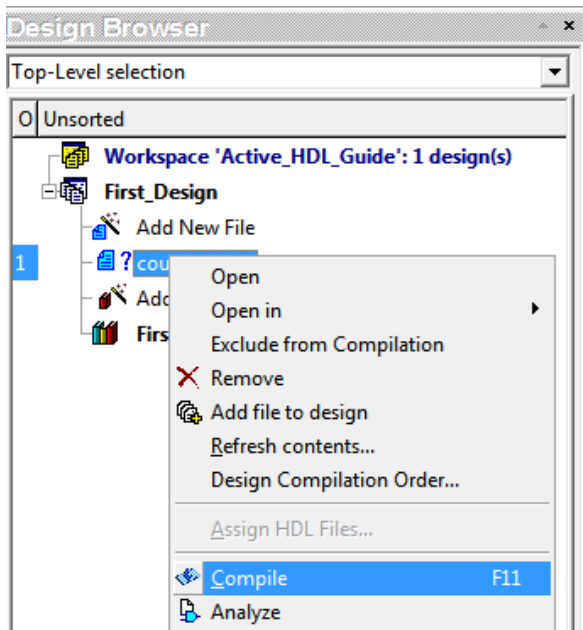


Figure 8 Compile file

If you choose **Compile All** from the **Design** menu for a given design all the files from the design are compiled. Please keep in mind that user is required to order the file manually in design browser. If you select **Compile All With File Reorder**, the compiler automatically reorders source files to ensure proper sequence in which design units are compiled. (Refer Figure 9)

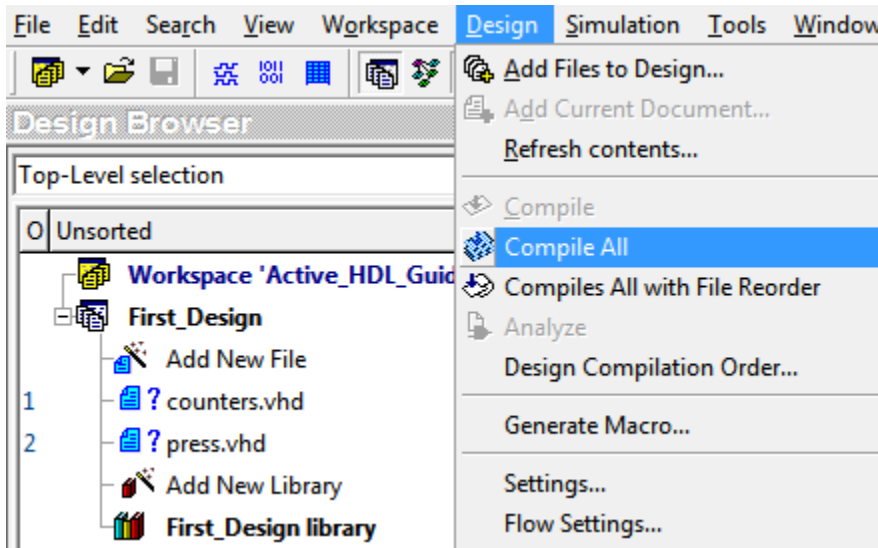


Figure 9 Compile All file option

Initializing the Simulation

Once all needed design units have been successfully compiled, you can initialize the simulation. Before you initialize the simulation, make sure that:

- You have selected the top-level design unit. You can select the top level in three ways:
 1. Select the desired VHDL design entity or configuration, Verilog module, EDIF cell, or SystemC module from the drop-down list located at the very top of the **Design Browser** window

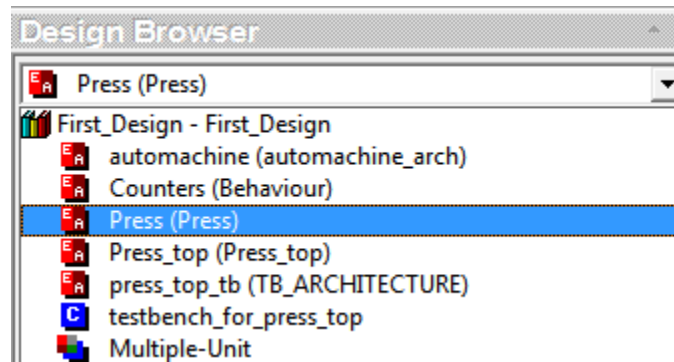


Figure 10 Selecting top level – Drop Down window

- Expand a structure of a source file (containing a top-level unit) or current working library in the **Files** tab, right-click on the desired design unit and then choose **Set as Top-Level** from the shortcut menu.

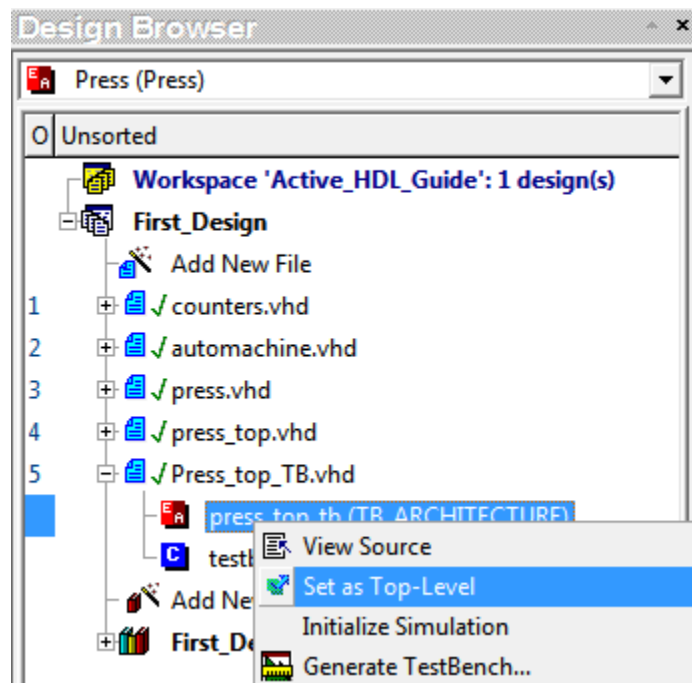


Figure 11 Selecting top level – Expanding the source file

- Open the **Design Setting Window**. By default, the **General** category is displayed. Go to the Top-level category and select a top-level unit from the list of design units.

To begin simulation, you must choose **Initialize Simulation** from the **Simulation** menu. This will launch the elaboration and initialization of the simulation model. During elaboration, the simulator loads design units, and builds the simulation model in the computer memory. During the initialization, all objects in the model (signals, variables, etc.) acquire their initial values (either default or explicitly specified) and all concurrent processes are executed once until their suspension.

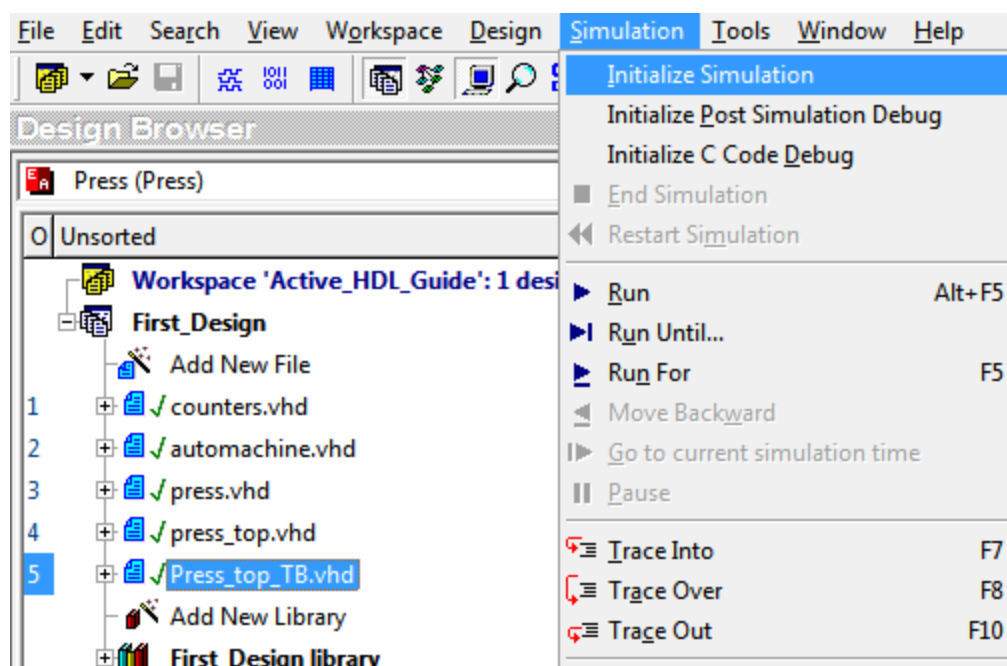


Figure 12 Initializing the simulation

Running the Simulation

After simulation has been initialized, design can be run using **Run** from the **Simulation** menu.

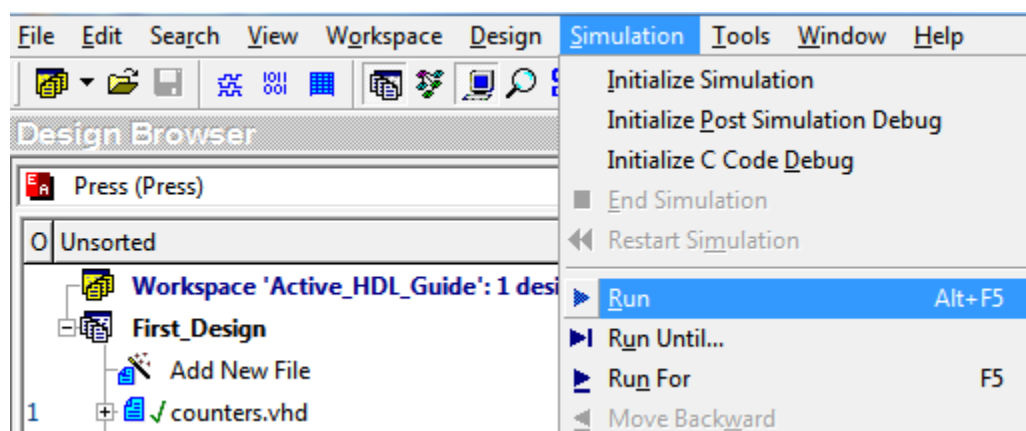


Figure 13 Running Simulation