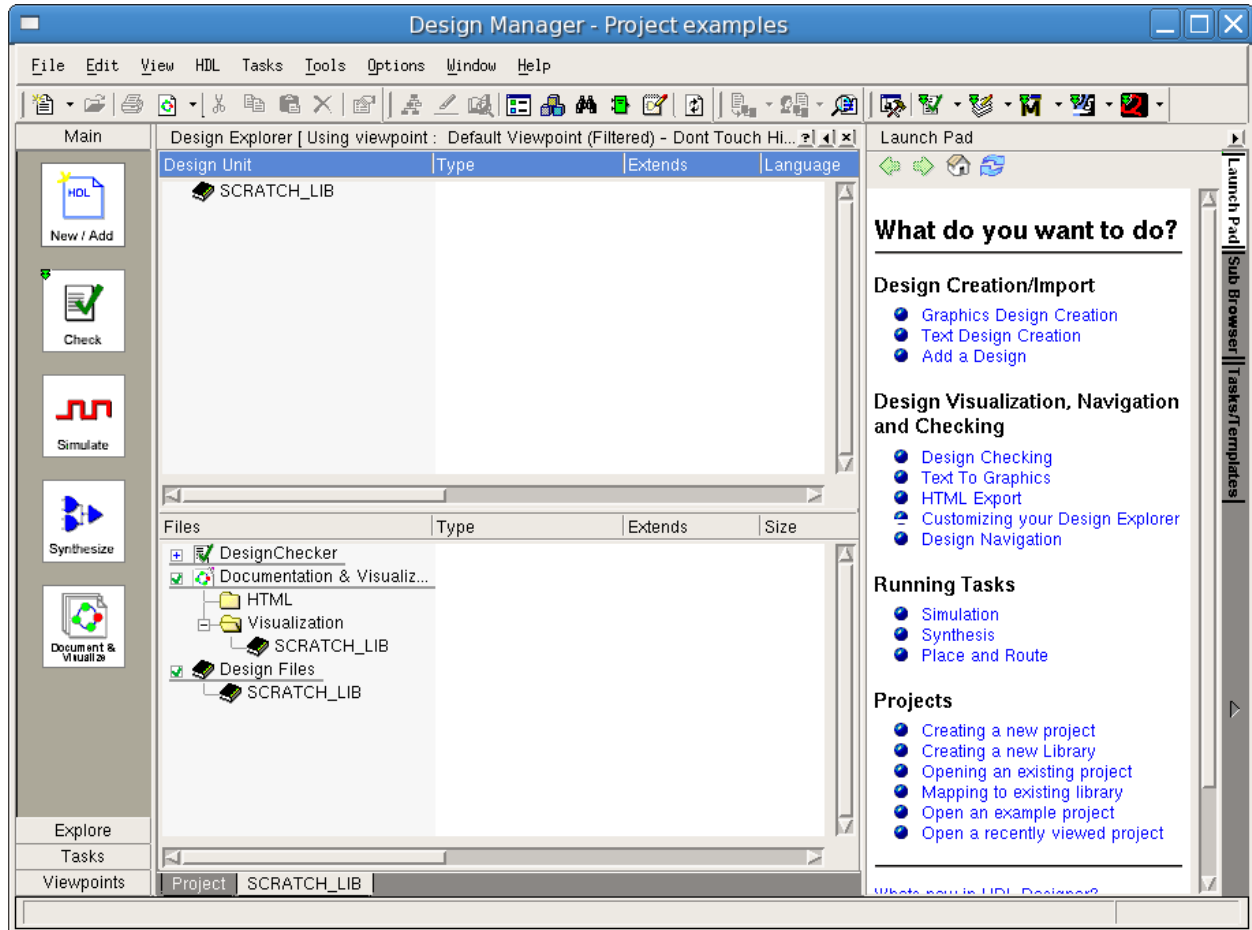
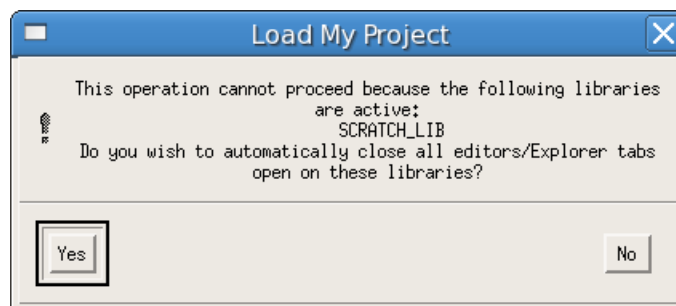


The **Design Manager** is your tool for creating and managing libraries. A library is a collection of designs, or components, which may be used within other designs (in other libraries).

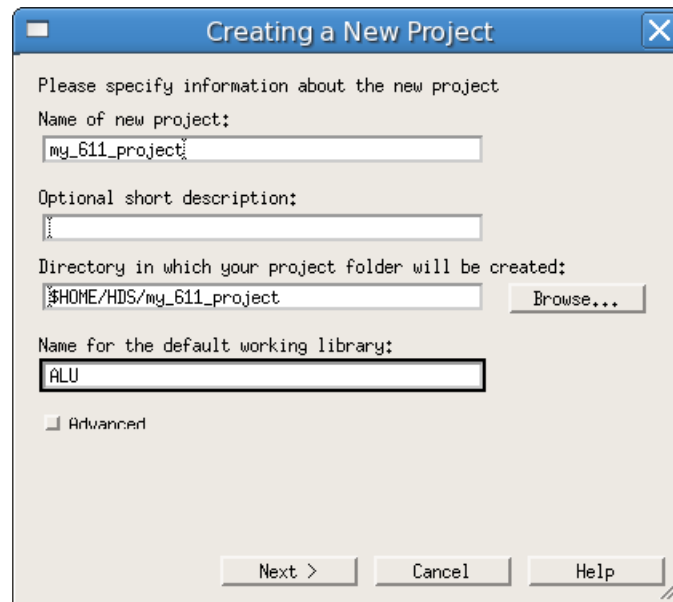
The first time you open HDL Designer, the Design Manager will display the default *SCRATCH\_LIB* library. The *SCRATCH\_LIB* library is created in a default location in your home directory.



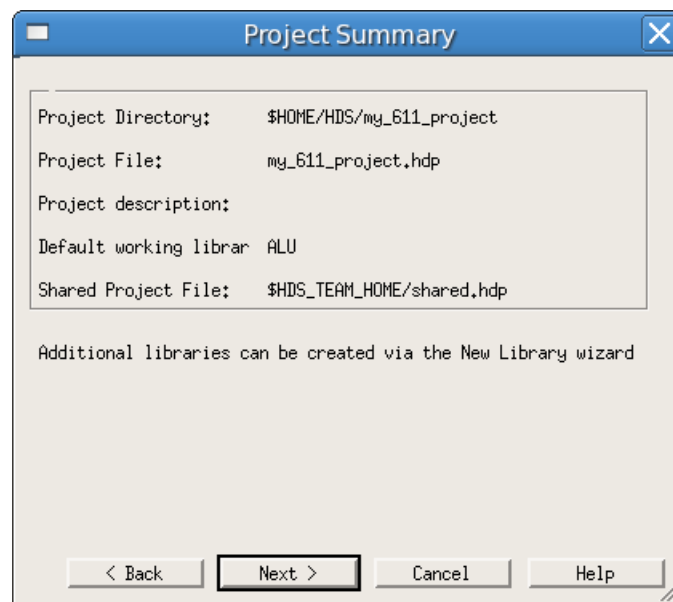
The project file contains information about the libraries that you are using. Firstly, let's create our project. Choose **File > New > Project**. Note that if the default *SCRATCH\_LIB* library is active, the following dialog box is prompted. Click the **Yes** button to close the default library when.



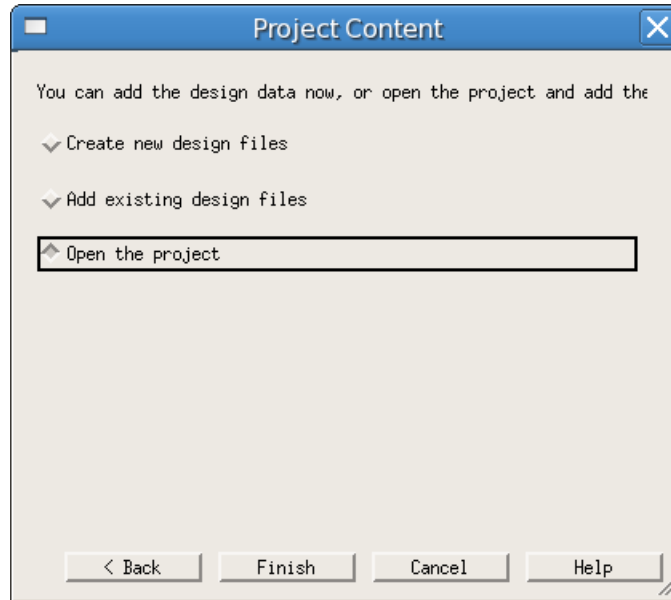
In the **Create a New Project** window, type **my\_611\_project** as the project name. You can enter a short description if you wish. The default directory in which the project folder is created is ok. If you wish to alter the path for organizational purposes, please make sure it is located somewhere inside your **PERSONAL DIRECTORY**, for example: **/acct/sl/username/**. Upon selection of your file storage location, type **ALU** as the name of the default working library which will be. Click the **Next** button.



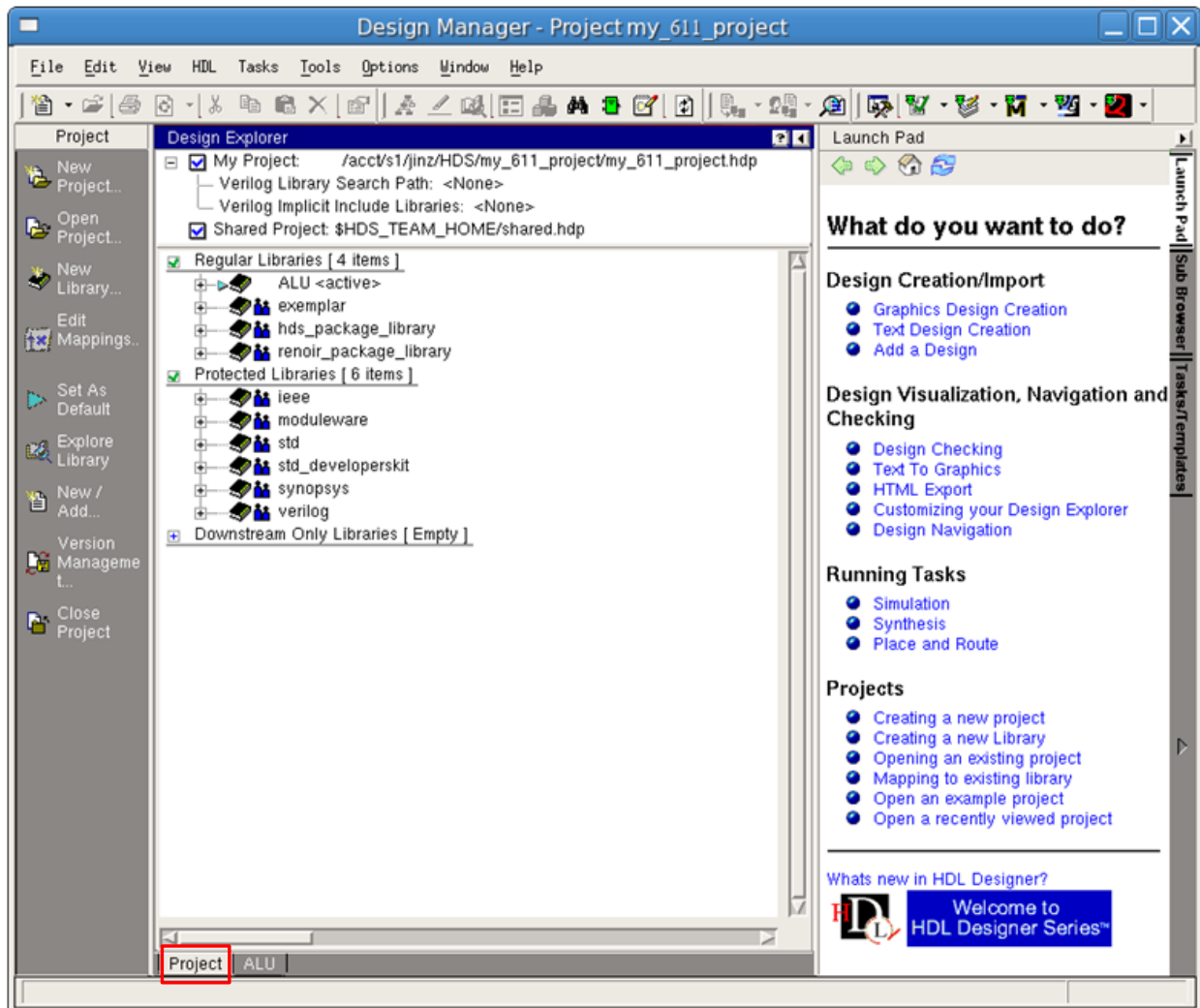
Click the **Next** button again in the Project Summary window.



Choose **Open the Project** and then click the **Finish** button.

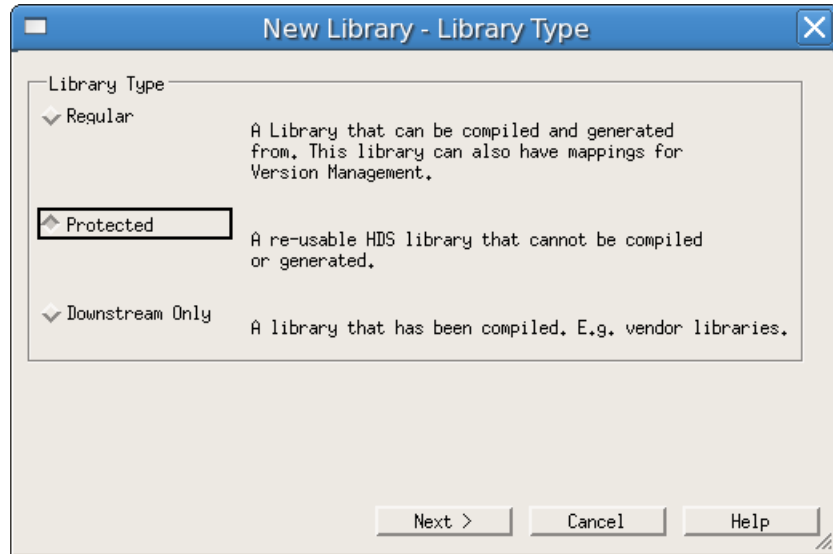


In the bottom left corner of the Design Manager window, click the **Project** tab to see your library mappings.

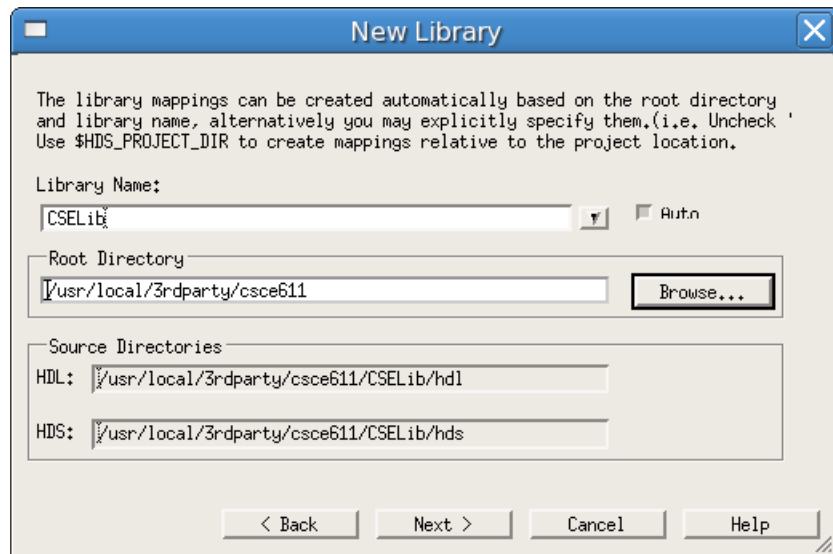


In creating our project, we have already created a new library called ALU. This is the library in which we will be designing the ALU. After completing the ALU design, you will use a component from this library in the design of your MIPS processor. We need to create another library, *CSELib*, which will contain pre-made designs that you will use in the design of your ALU. When creating a new library in the Design Manager, we're actually only creating the path mappings to the design data. Design Manager has already created the ALU library for us, so we just need to create the CSELib library. Let's get started.

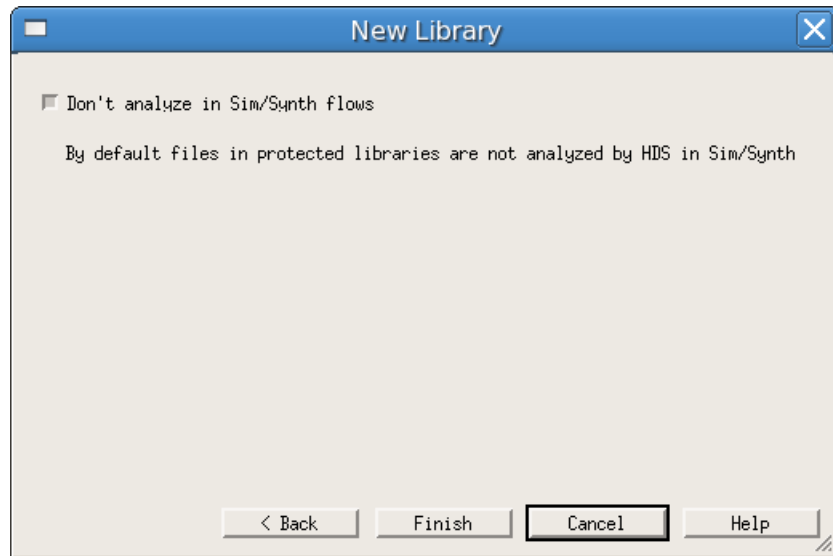
Go to **File > New > Library**. From this dialog box, choose **Protected** and click the **Next** button.



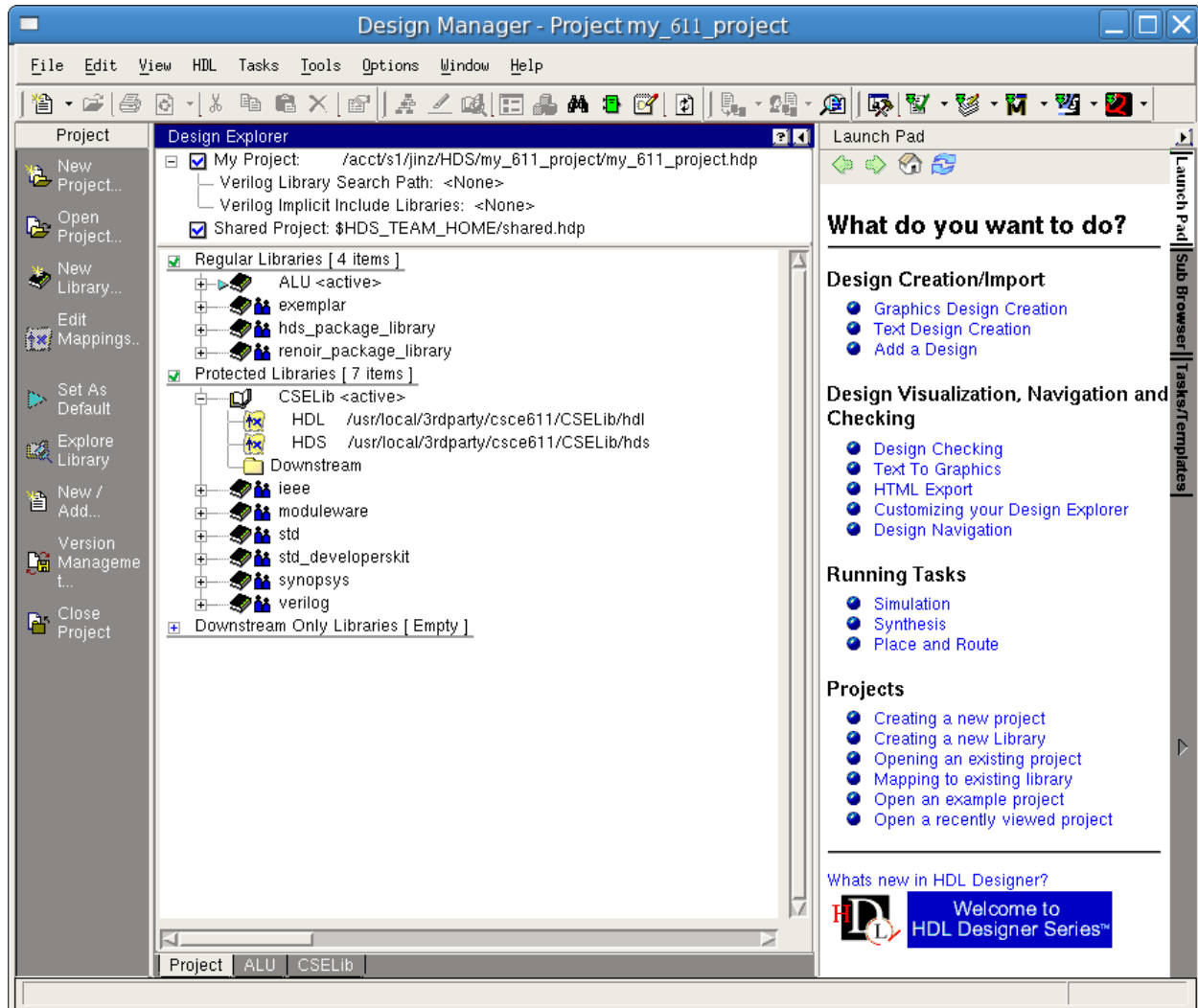
The library name is **"CSELib"** and the root directory for this library is `/usr/local/3rdparty/csce611`. Click the **Next** button.



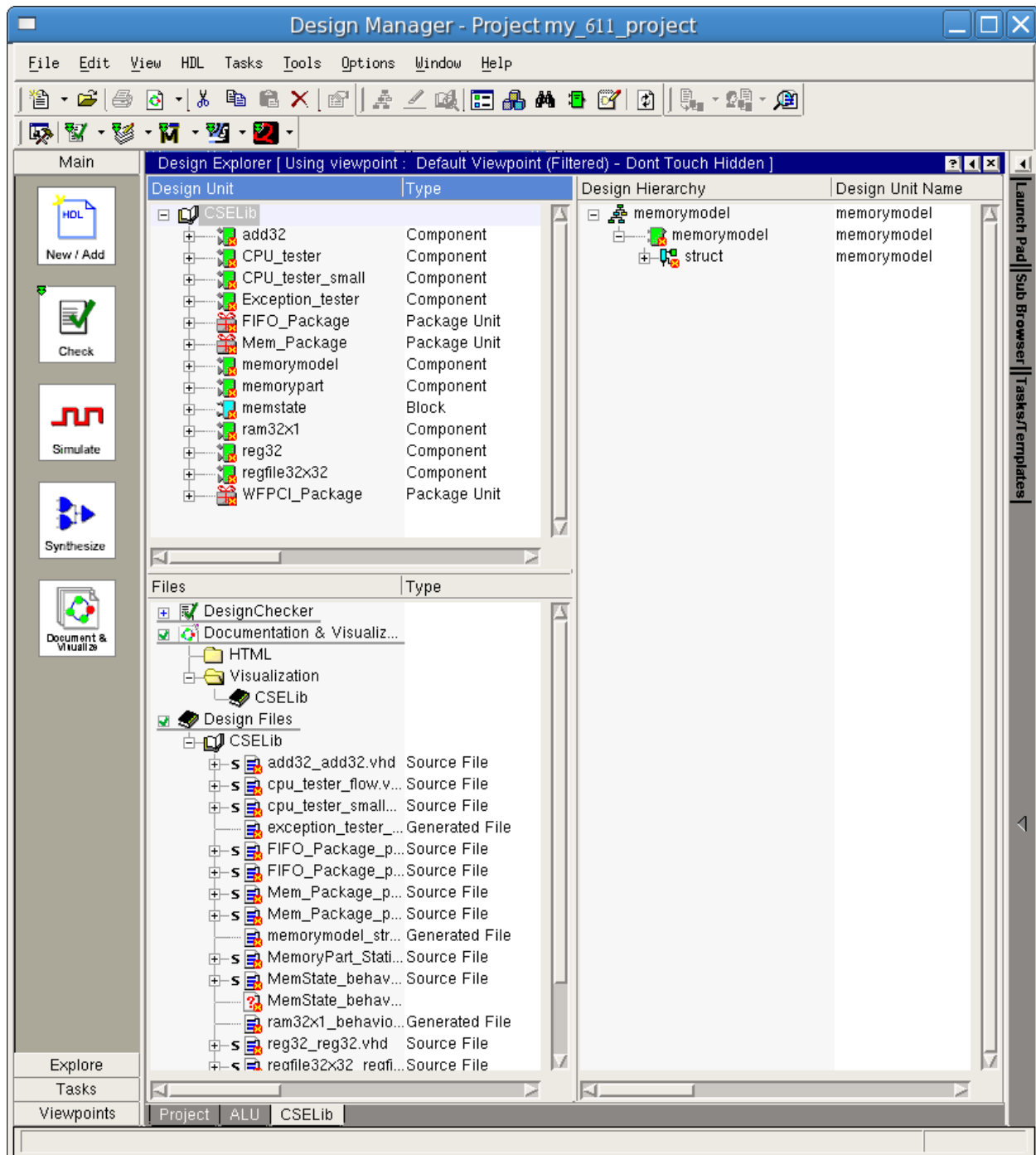
Finally, click the **Finish** button.



Now we have two libraries. The ALU library is the library where you will be storing your designs for this tutorial. The CSELib library is the library where several design components are provided so that you can use them later on. Click the tabs at the bottom of the Design Manager to switch between the Project View and the Library View. Go to the Project View by clicking on the **Project** tab.



Here is where you'll see the new libraries you've added, plus several others that are there by default. If you double-click the CSELib library, a new or existing "Design Explorer" tab will open that will allow you to look at the contents of the CSELib library.



Do not worry that there are red x's over the components. This means that they are read-only.

In the next tutorial, we will start building our ALU.