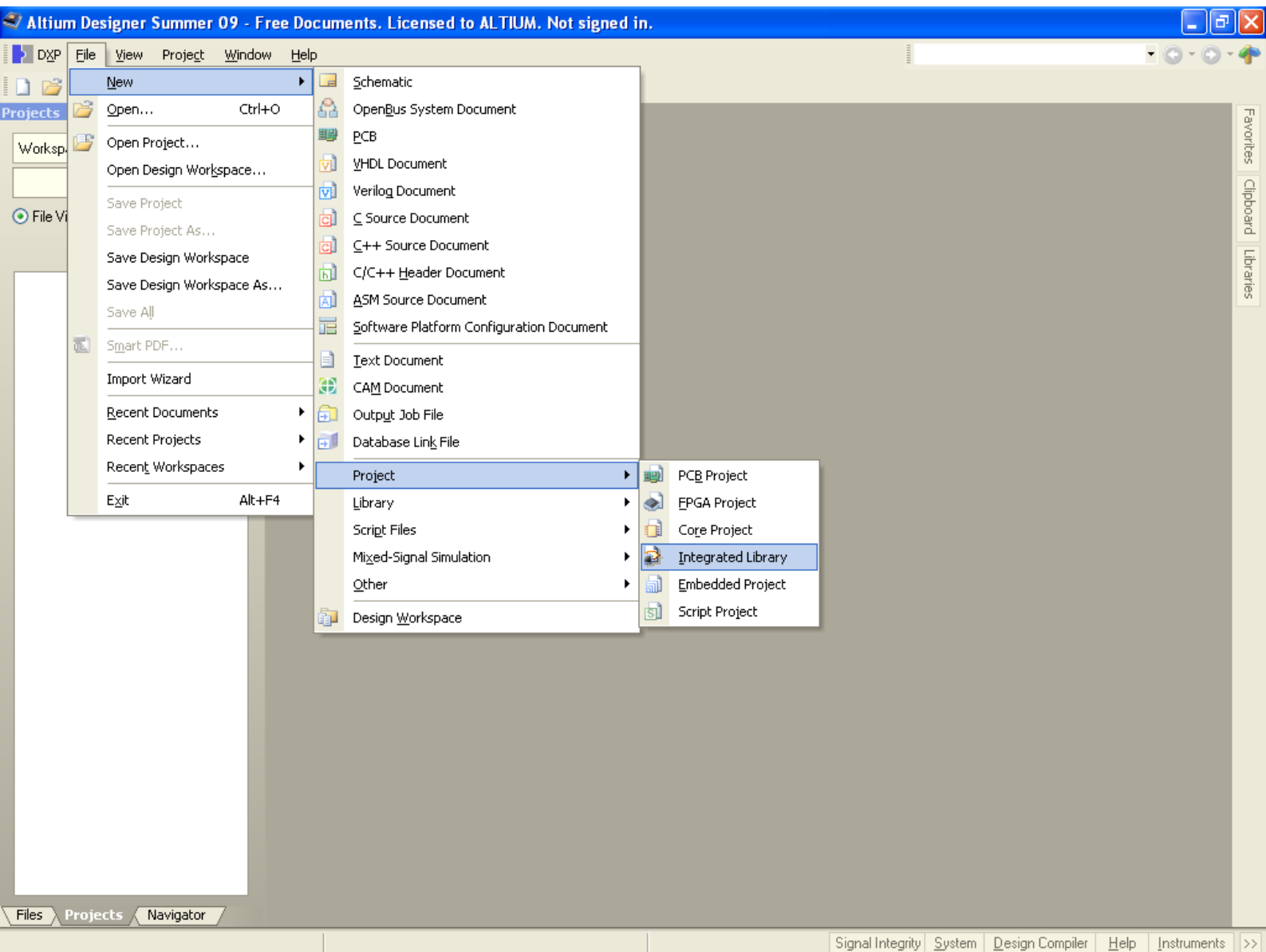


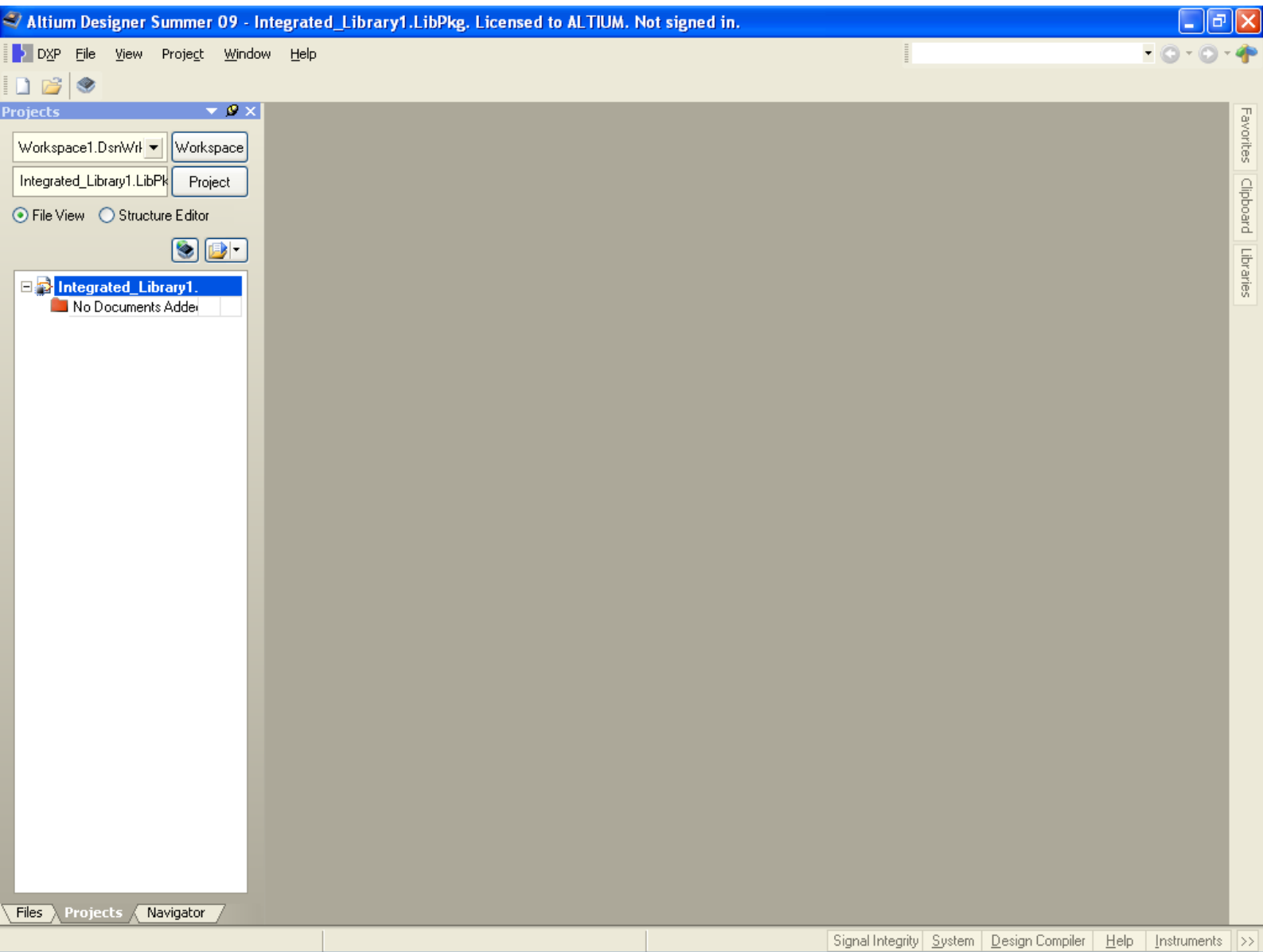
Before we start this session these are the two prerequisites :

- 1)Construction of Schematic component
- 2)Construction of PCB components (Footprints)

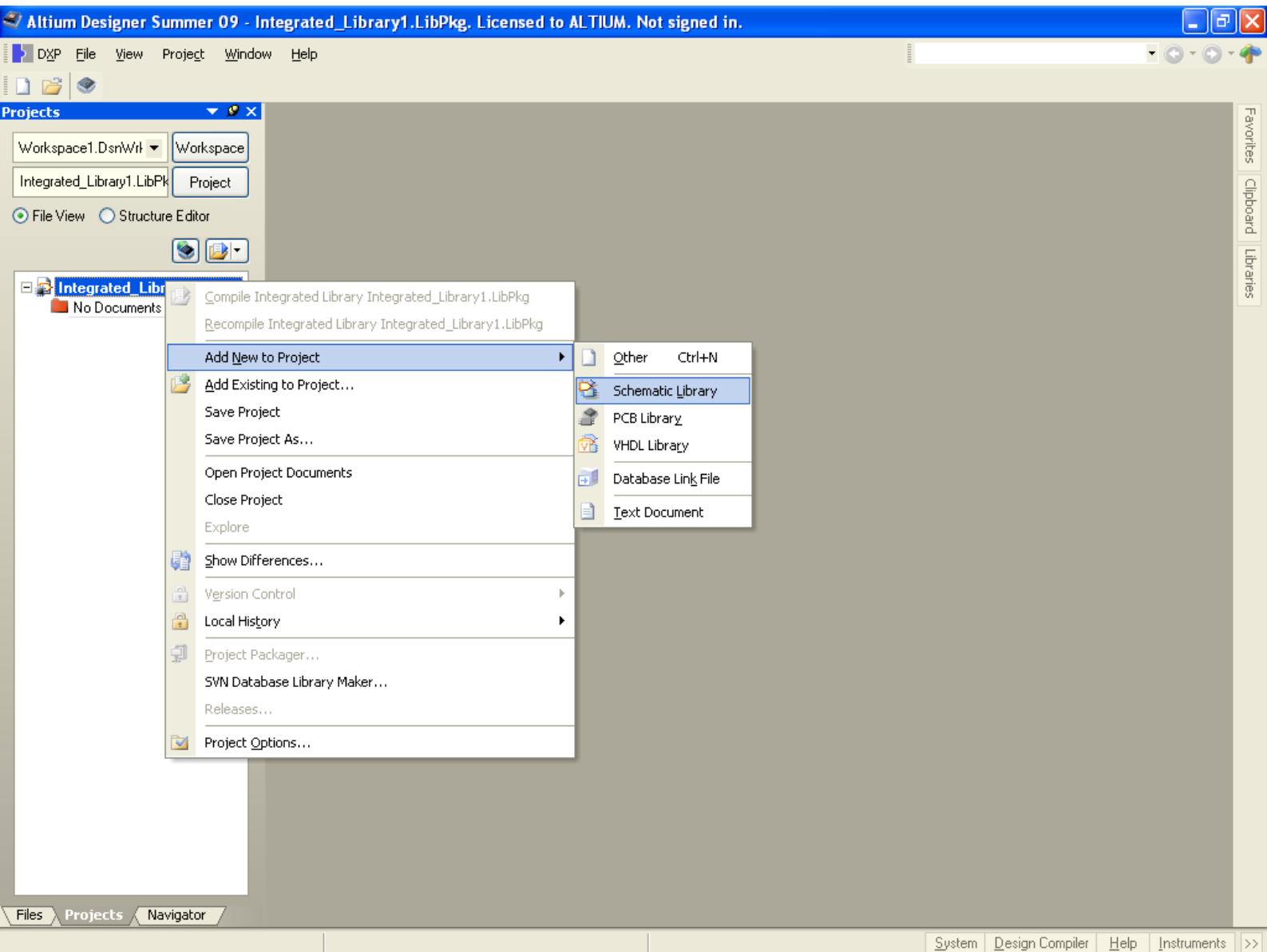
First of all to create your own library in ALTIUM you need to go to :
Go to File → New → Project → Integrated Library



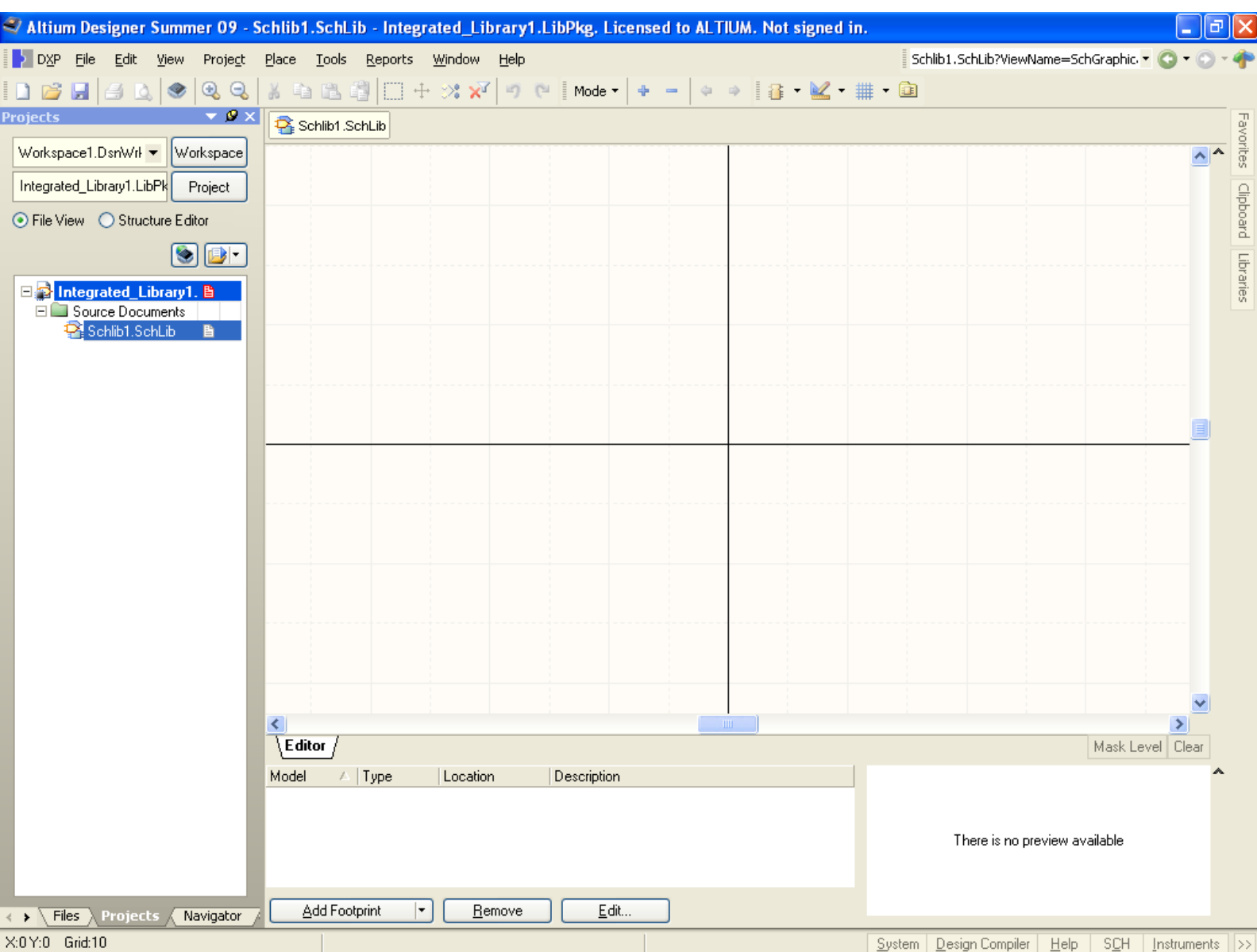
Then you will observe the window like this :



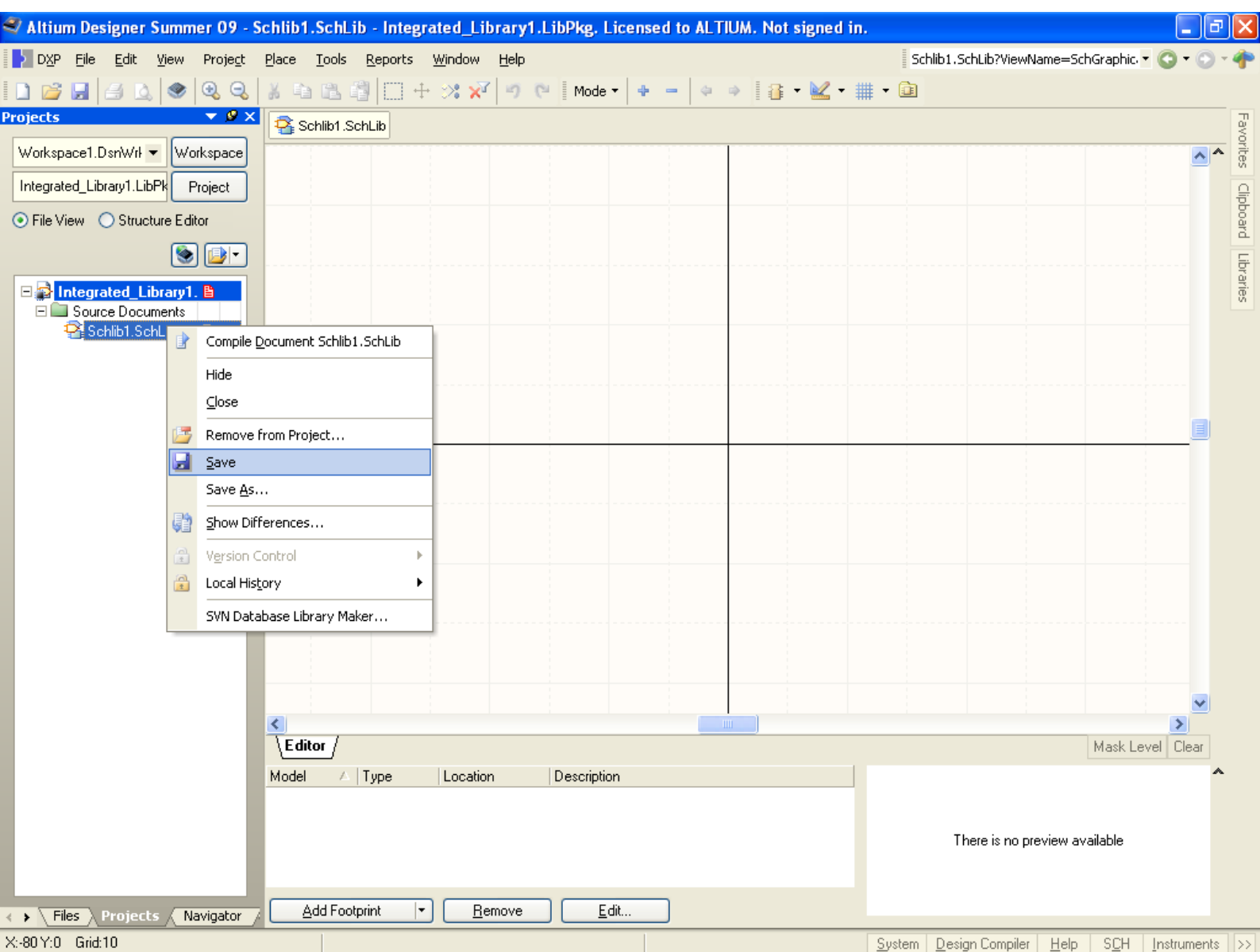
Now add the schematic library to the integrated library as shown below :



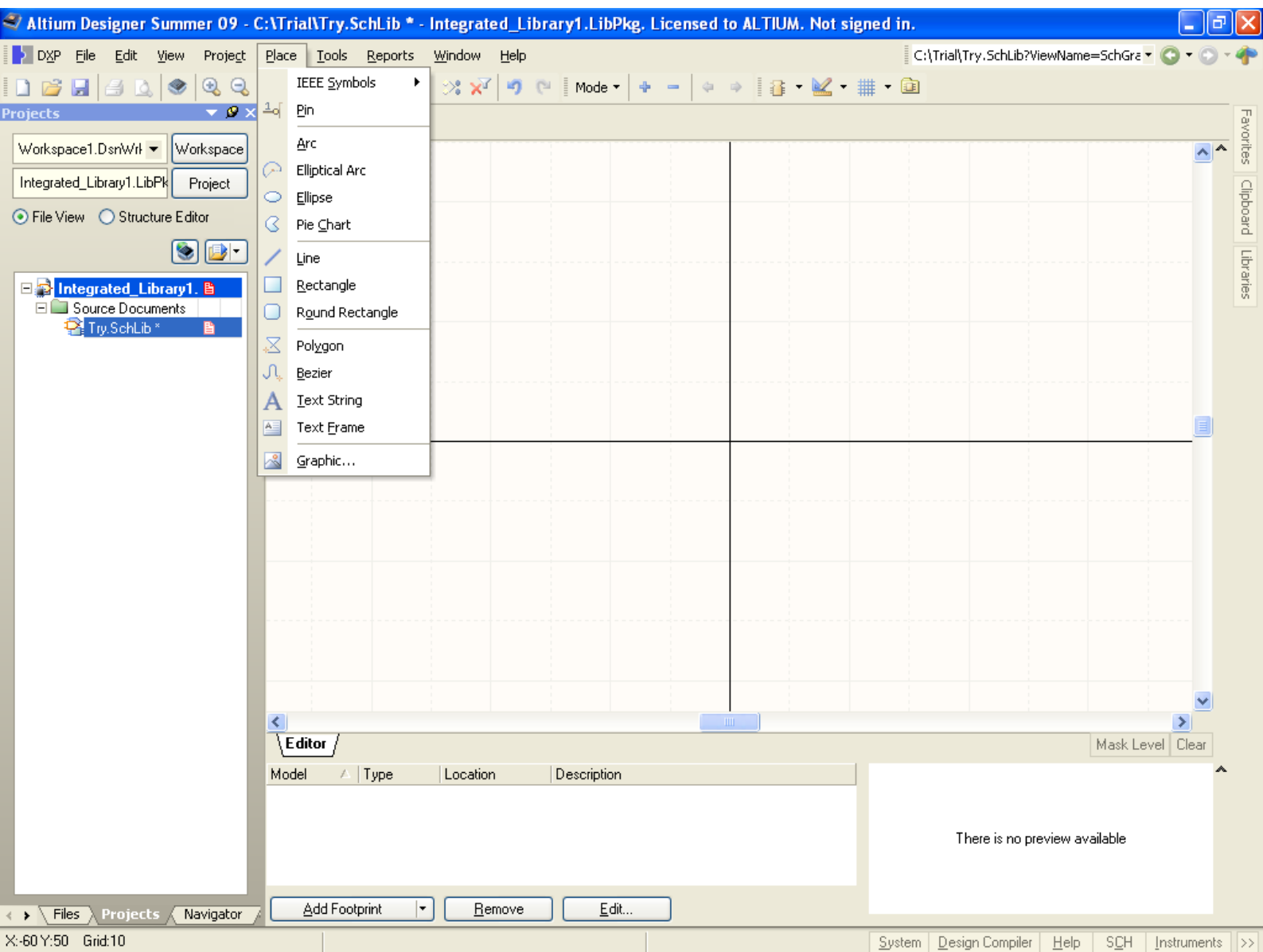
The schematic library will open as below :



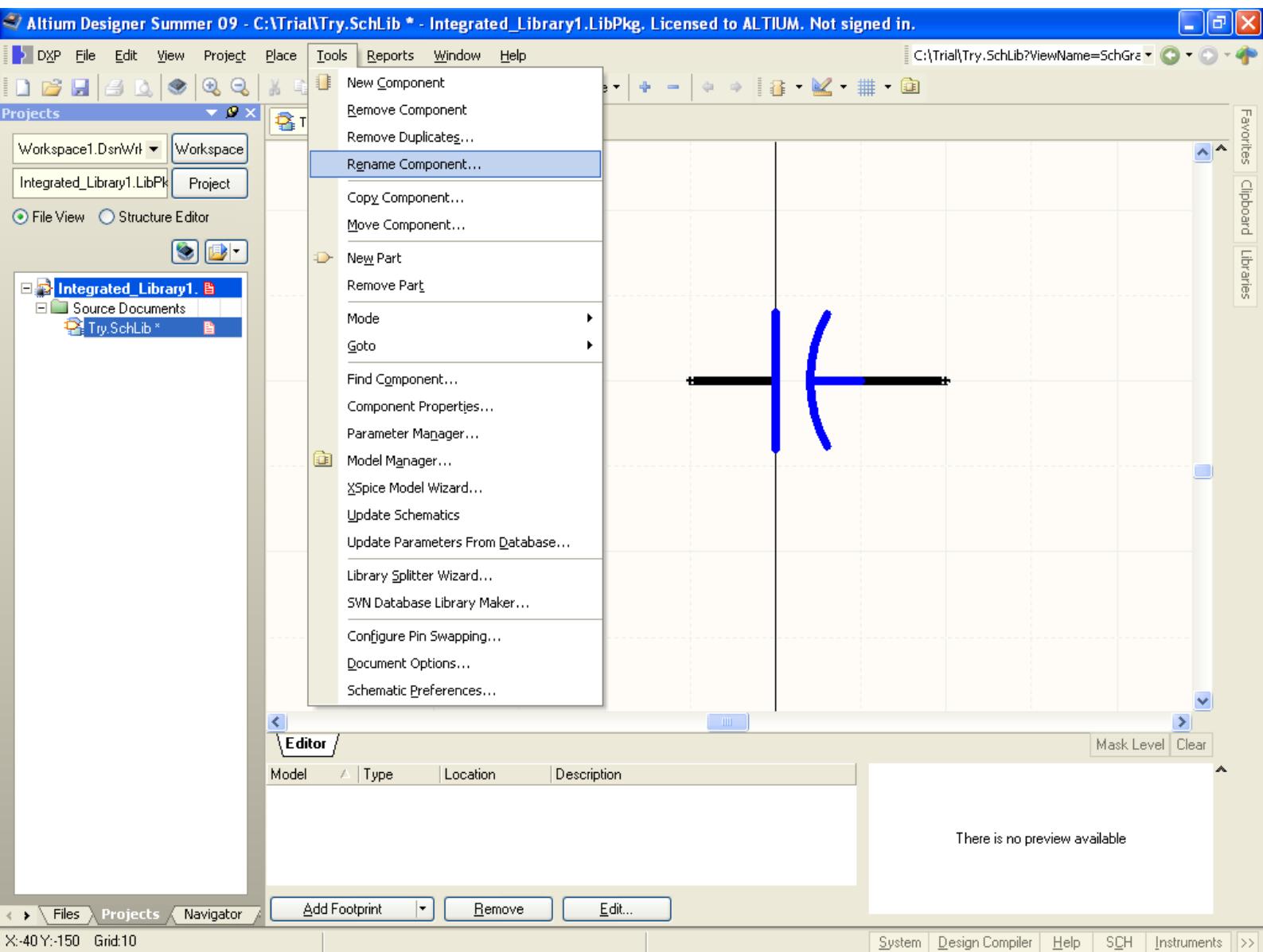
Save the schematic library in one folder and give specific name to the library :



Now we need to make the schematic diagram of the component using the following option in Place :

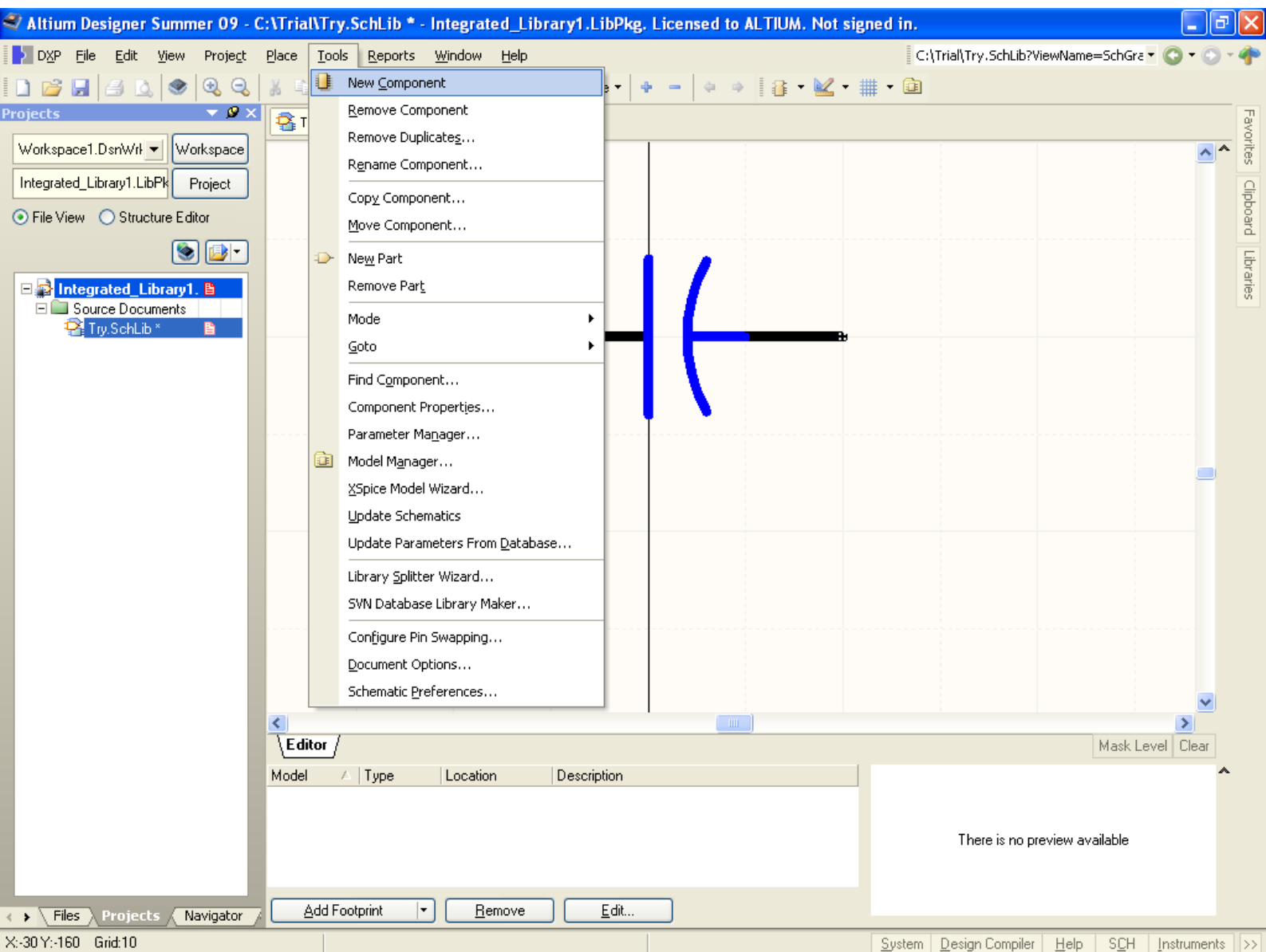


After making the schematic diagram of the component you need to give the component its name from :



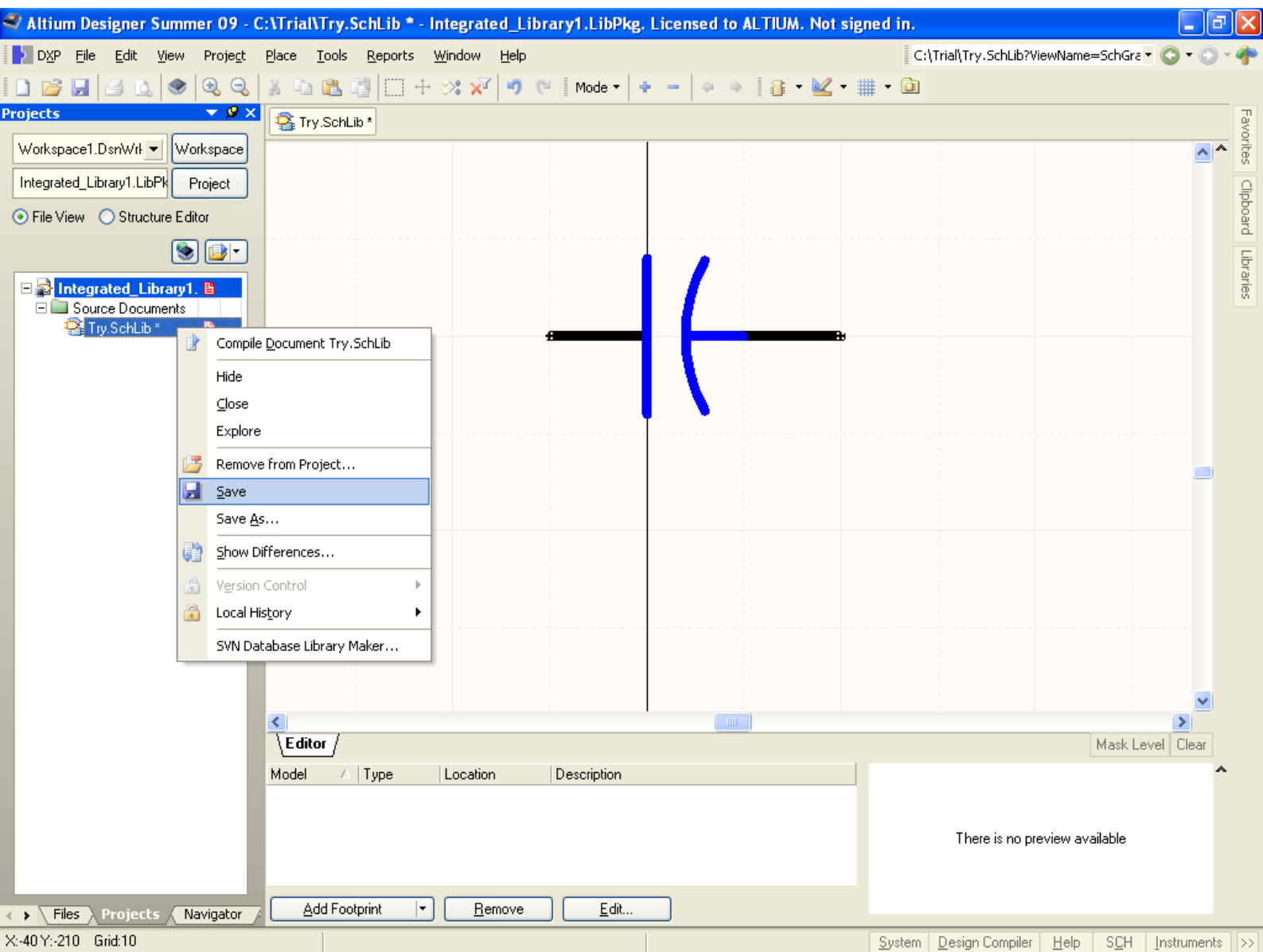
Give appropriate name to the component and click ok.

Now we need to add another component to the same library so that can be done as shown in below screen shot :

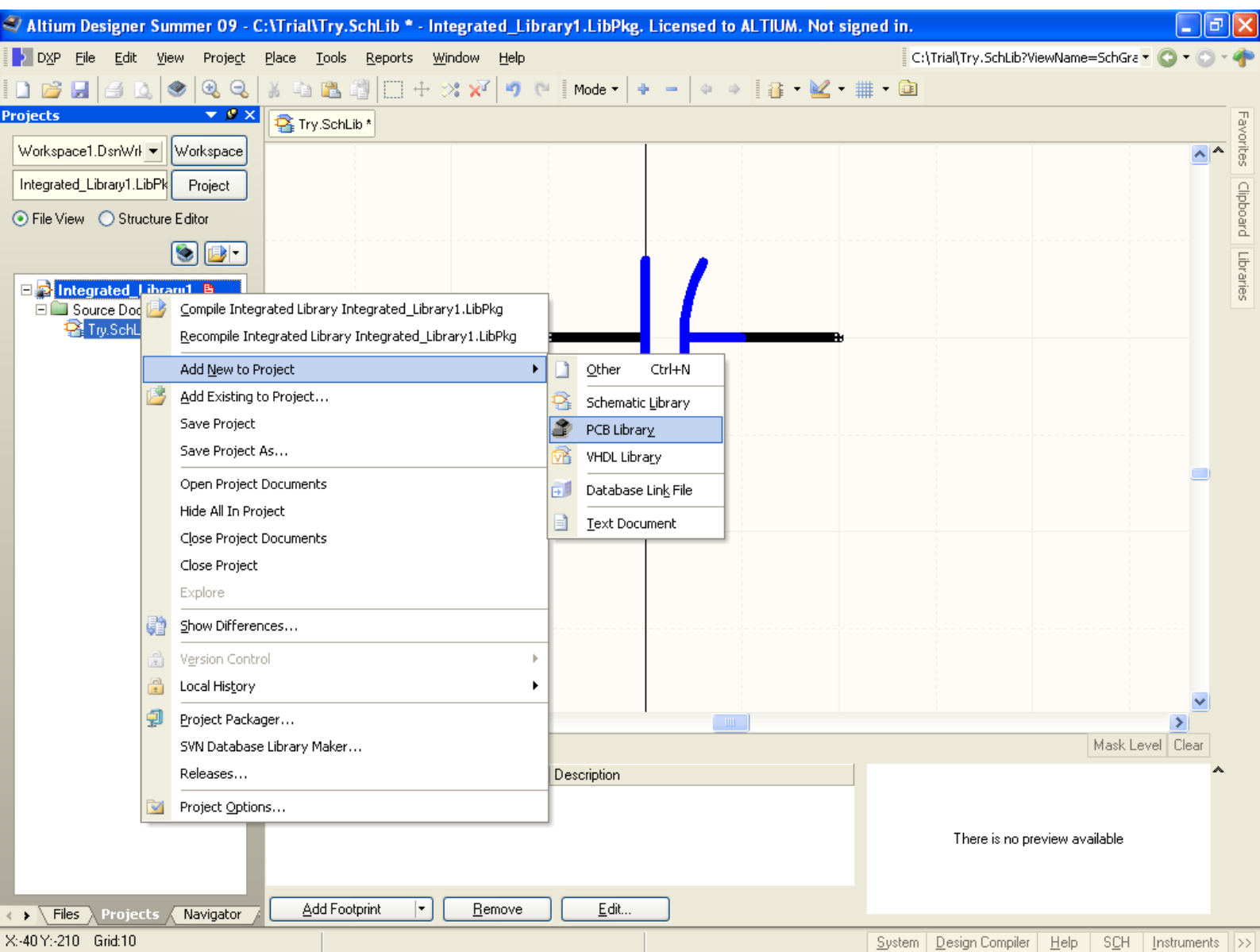


Now another blank space will be created for schematic diagram now draw the schematic diagram for that component and follow the same procedure{ go to Place and then draw schematic diagram and again give the component the name from rename component option } and draw components schematics you need as described above.

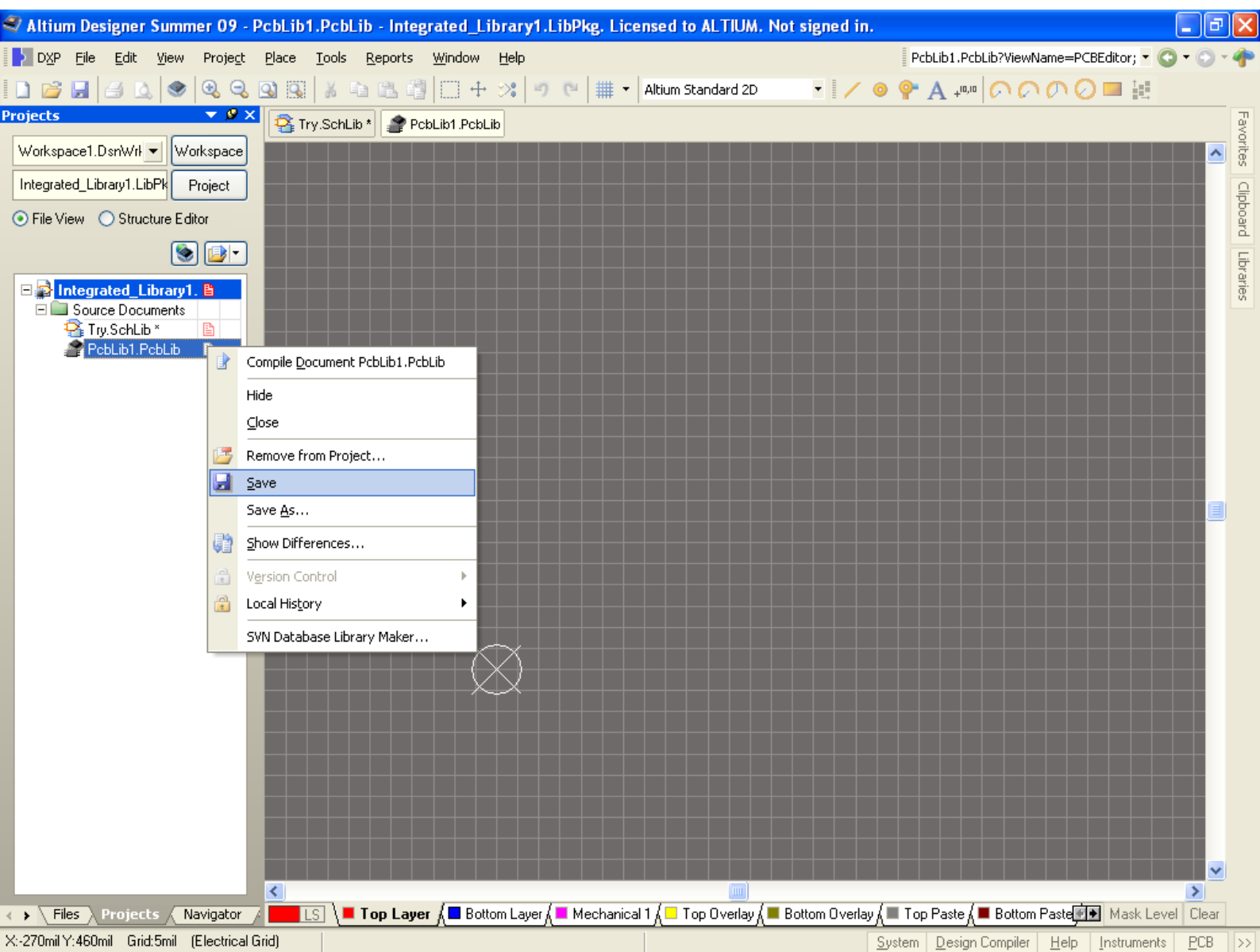
Now we have to save the updated schematic library as shown below :



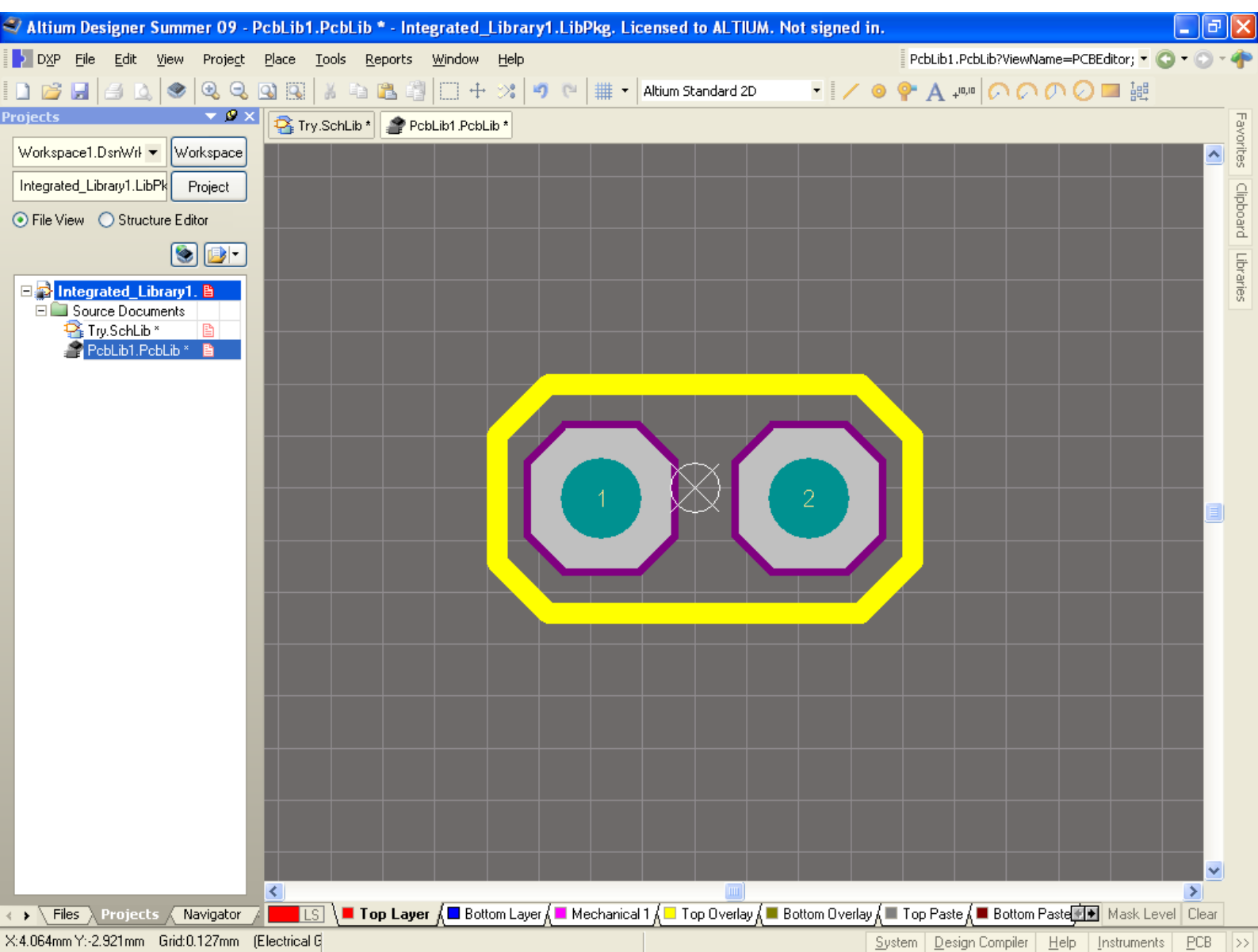
Then add the PCB library to the Integrated library as given below :



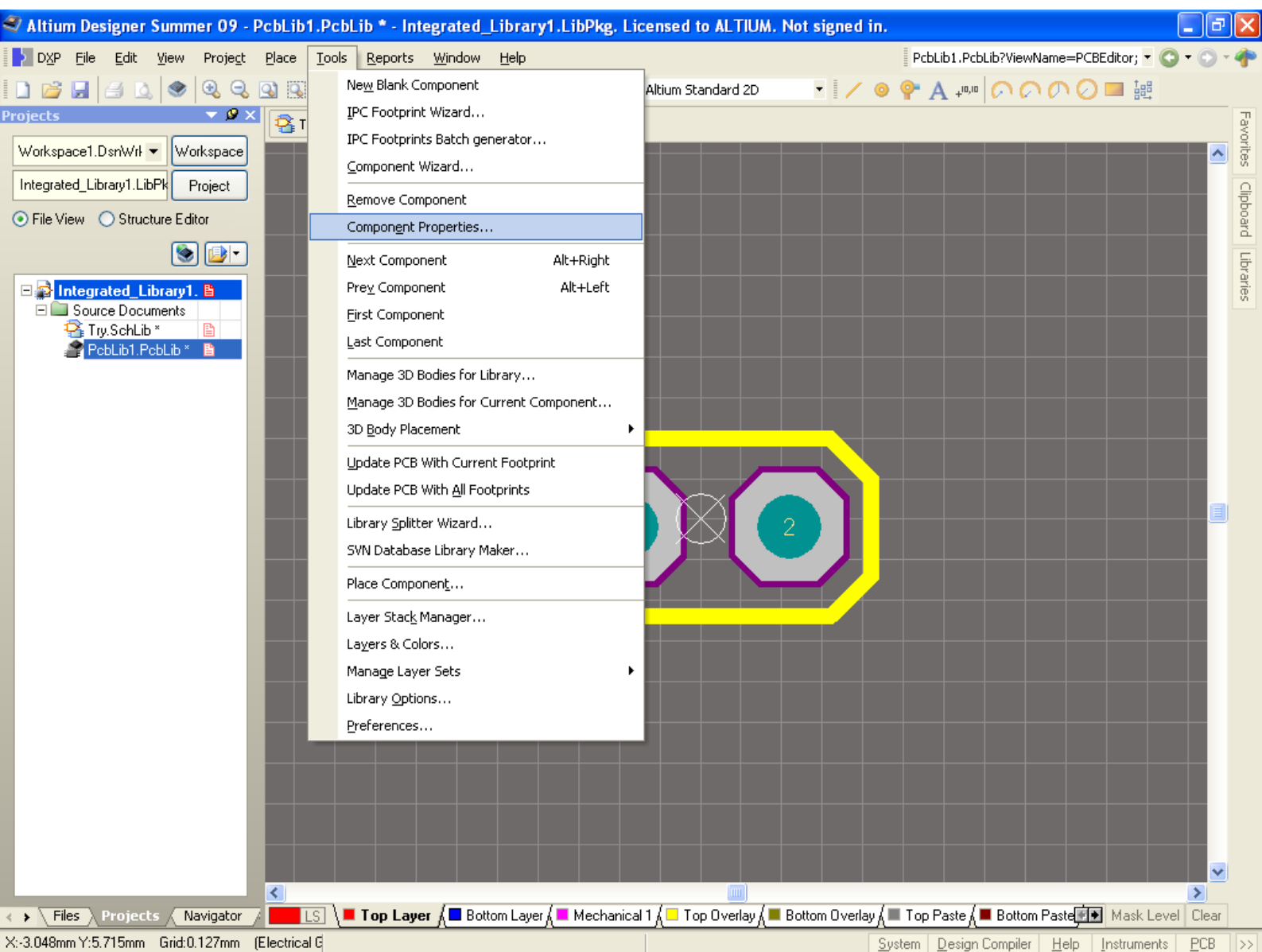
Save the PCB library in the same folder as the schematic library was saved :



Now make the footprints of all the schematics you created in the schematic library as shown below :

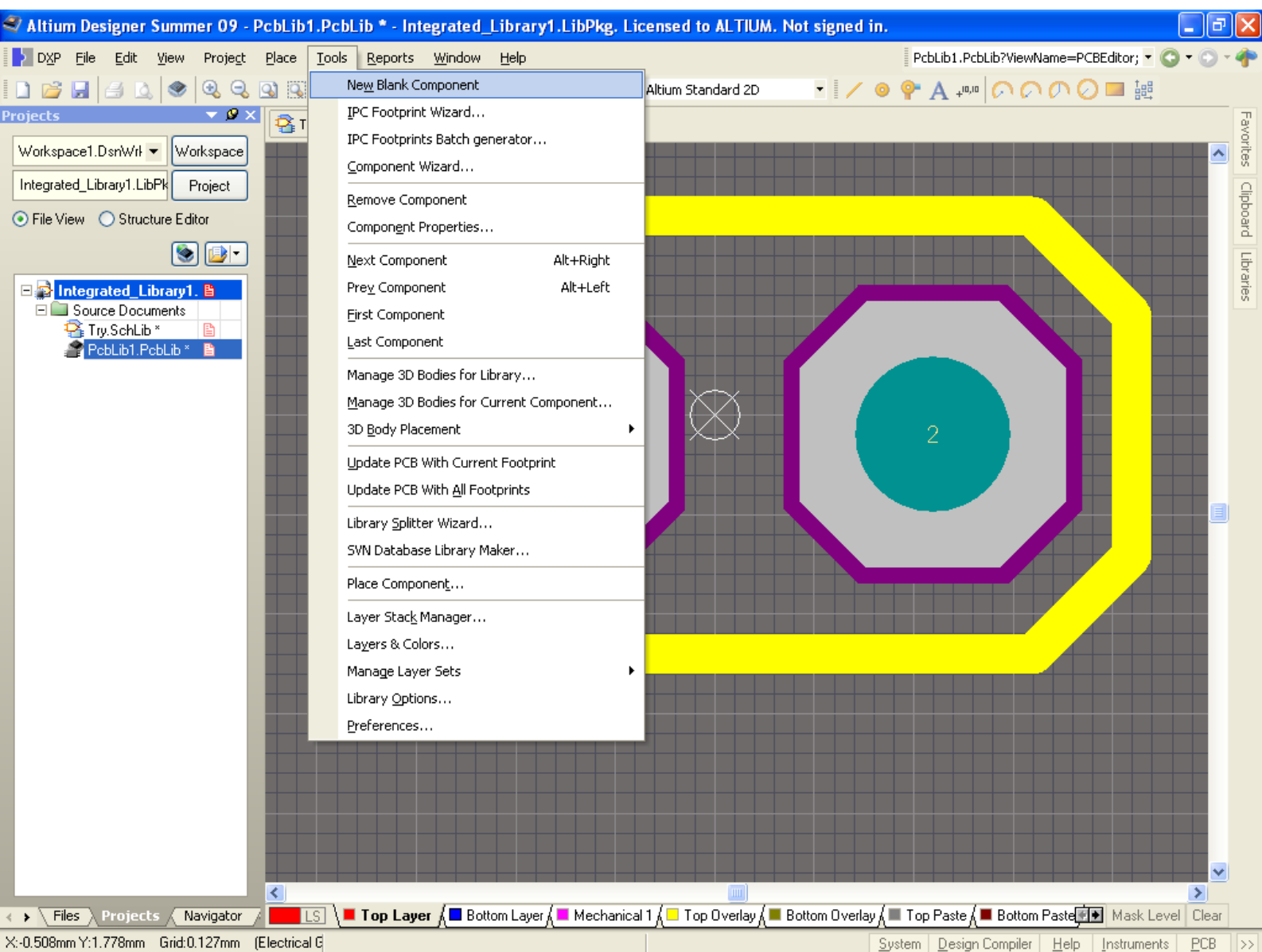


Now here also give the footprint its name as shown below :



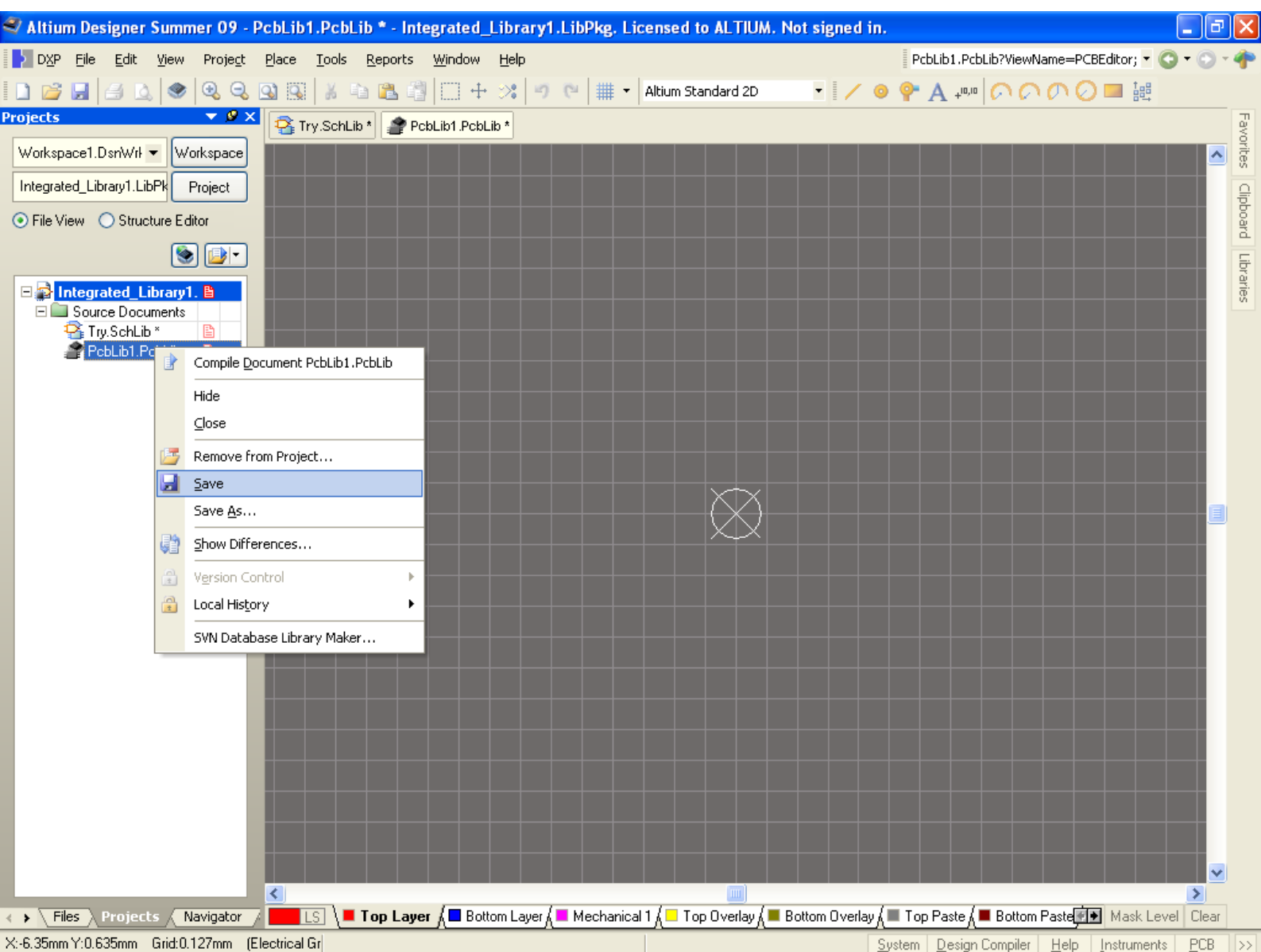
Give appropriate name (give the same name as that of schematic component) and then click ok .

Then now we need to add new blank component to the PCB library as shown below :

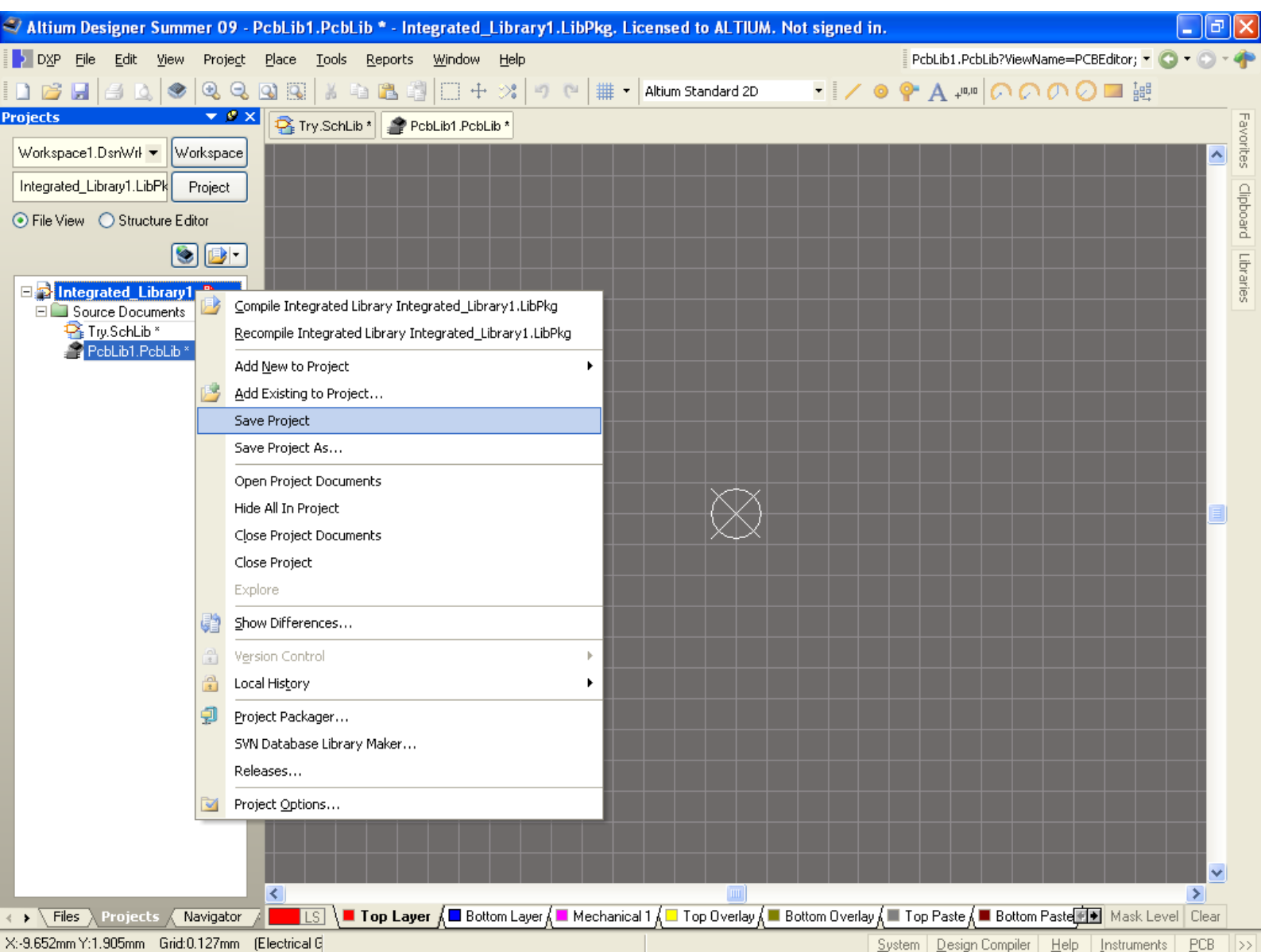


Similarly add all the footprints to the PCB library one by one and follow the same procedure for other footprints.

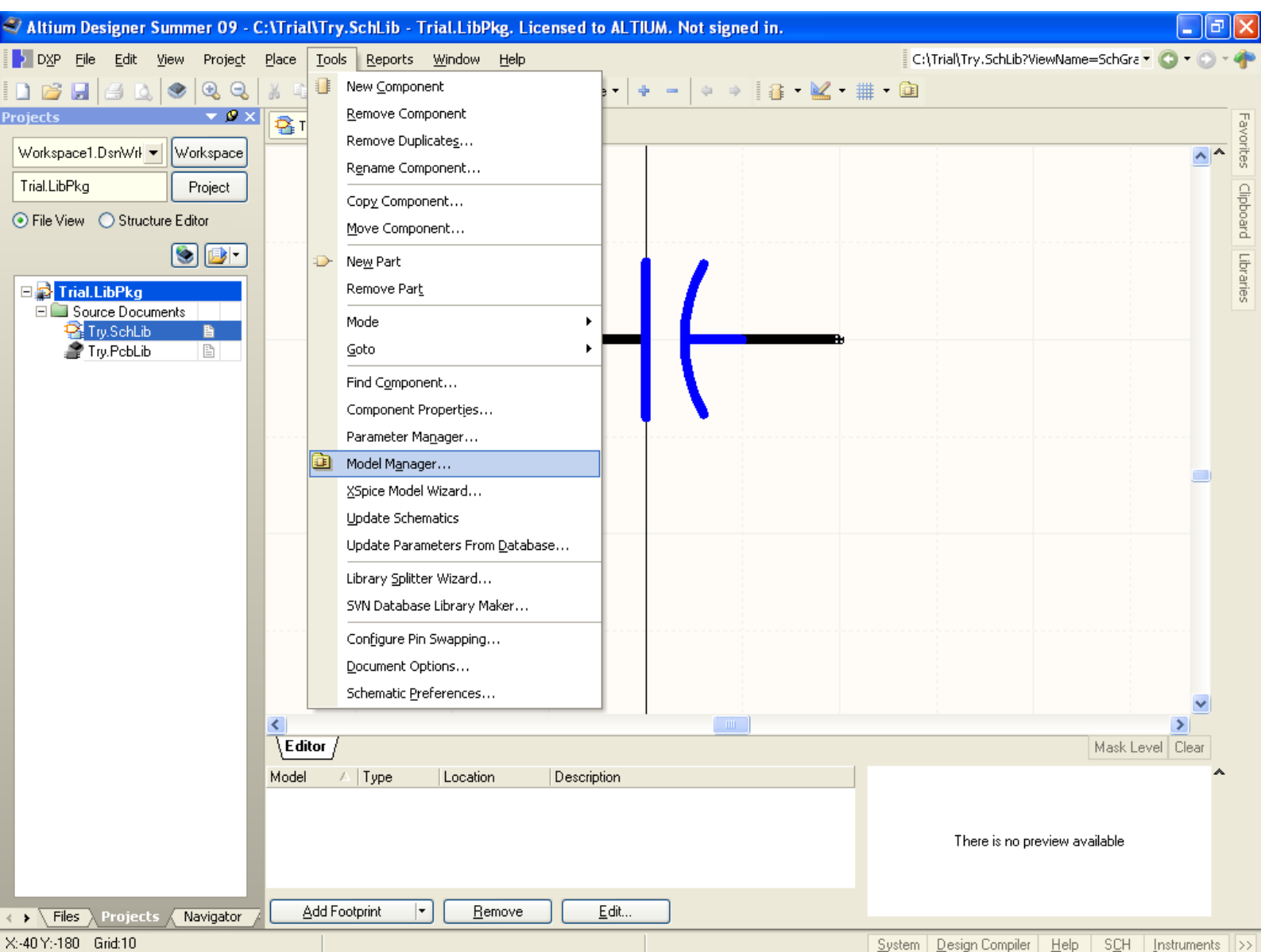
Then we need to save the updated PCB library and then save the Integrated Library as shown in below screen shots :



Save the project in the same folder as that of schematic as shown in the screen shot given below :

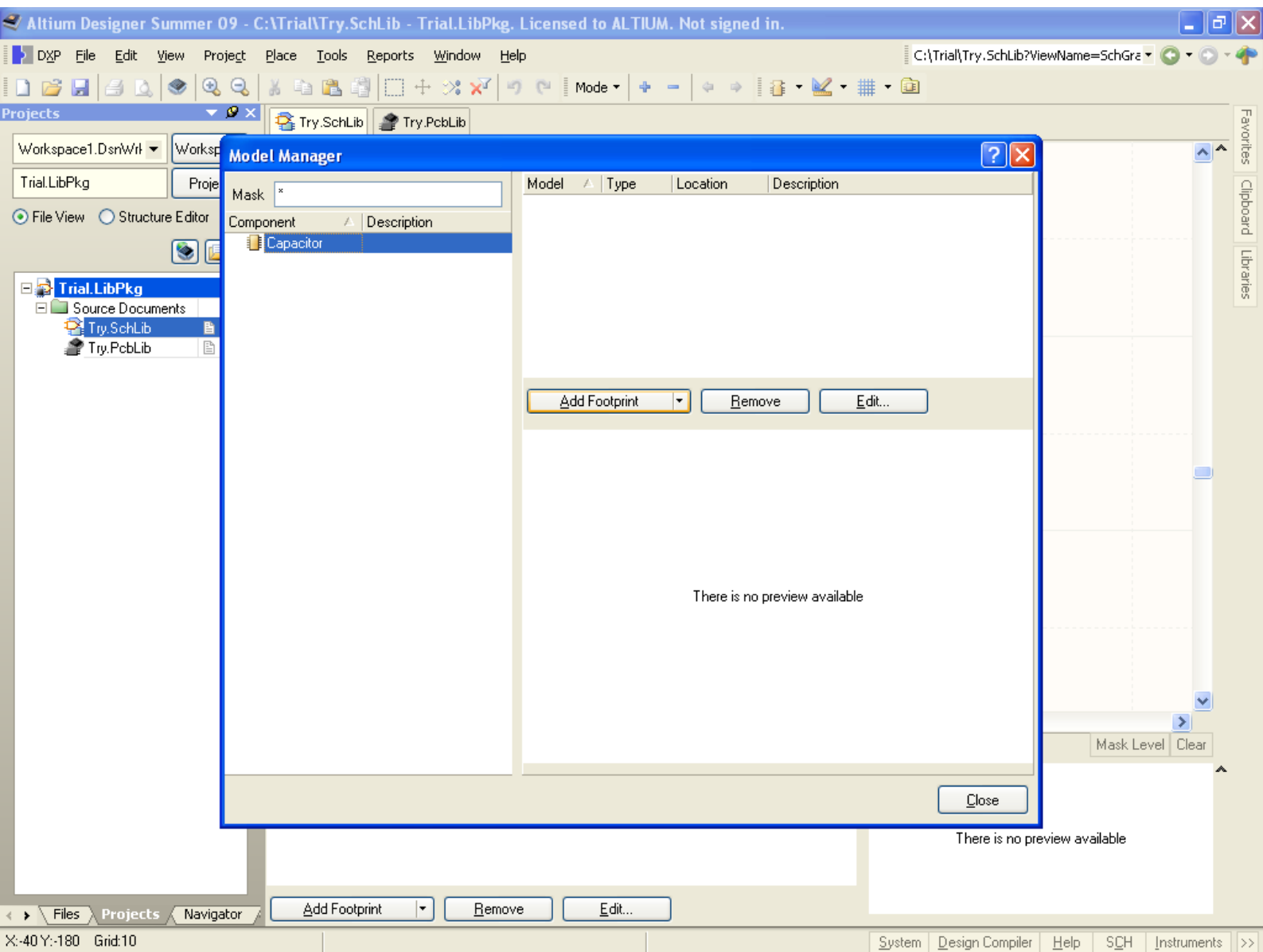


Then we have to add all the footprints created , to their respective schematics as shown below go to model manager :

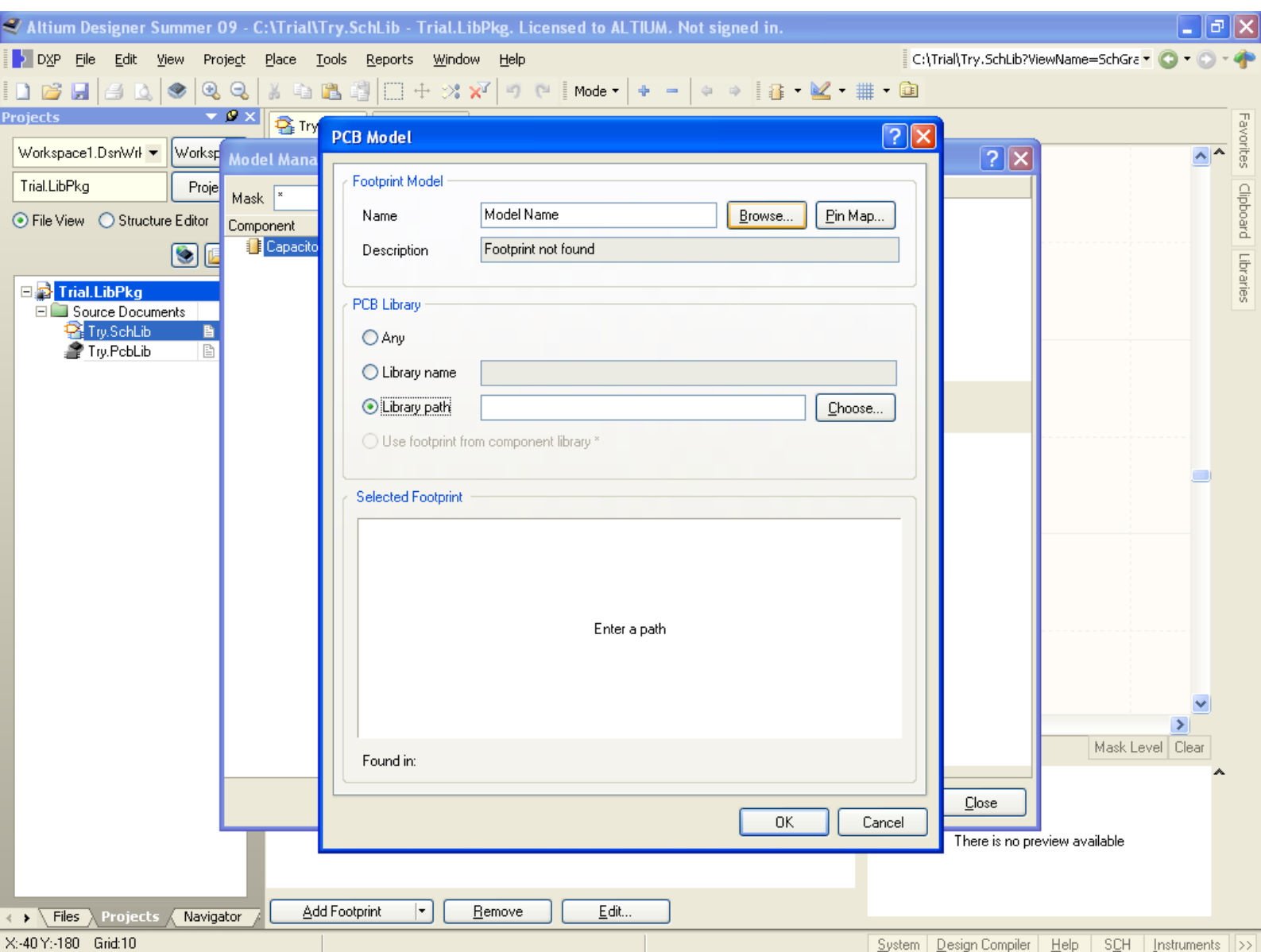


Here in model manager do as shown in the below screen shots one by one .

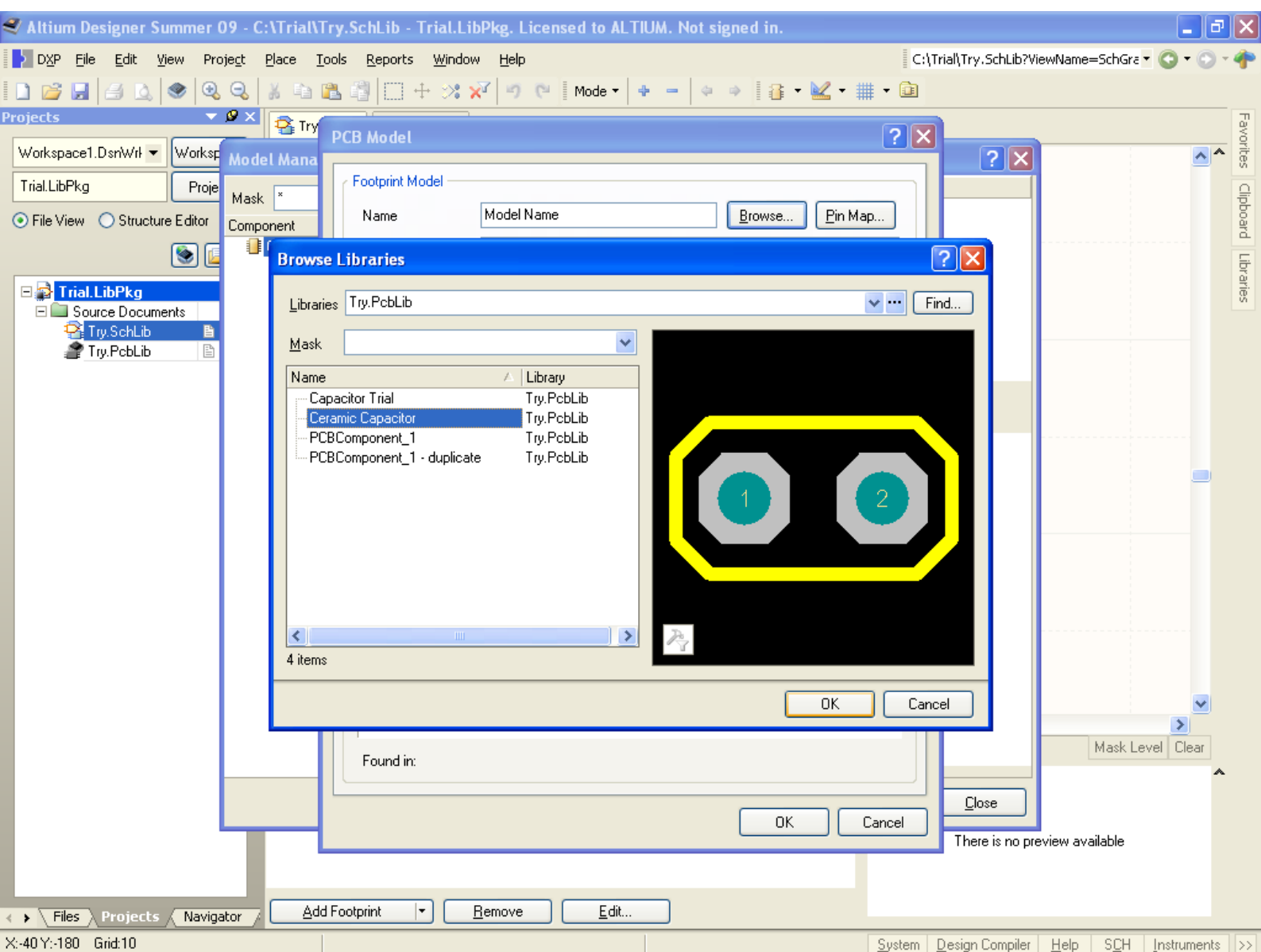
Select the component whose footprint is to be added and then click on add footprint :



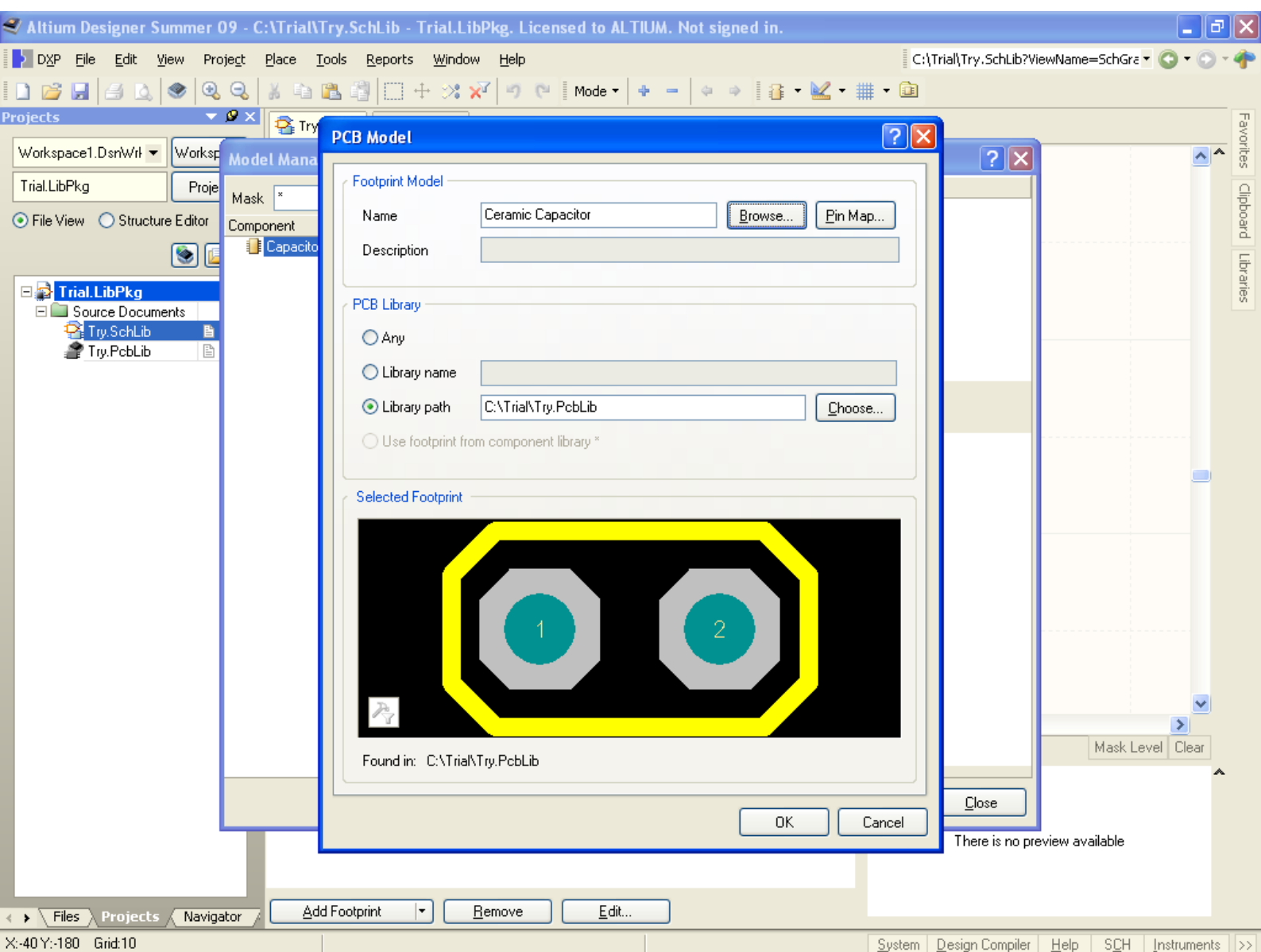
In PCB library option click on Library path then in footprint model option click on browse as shown below :



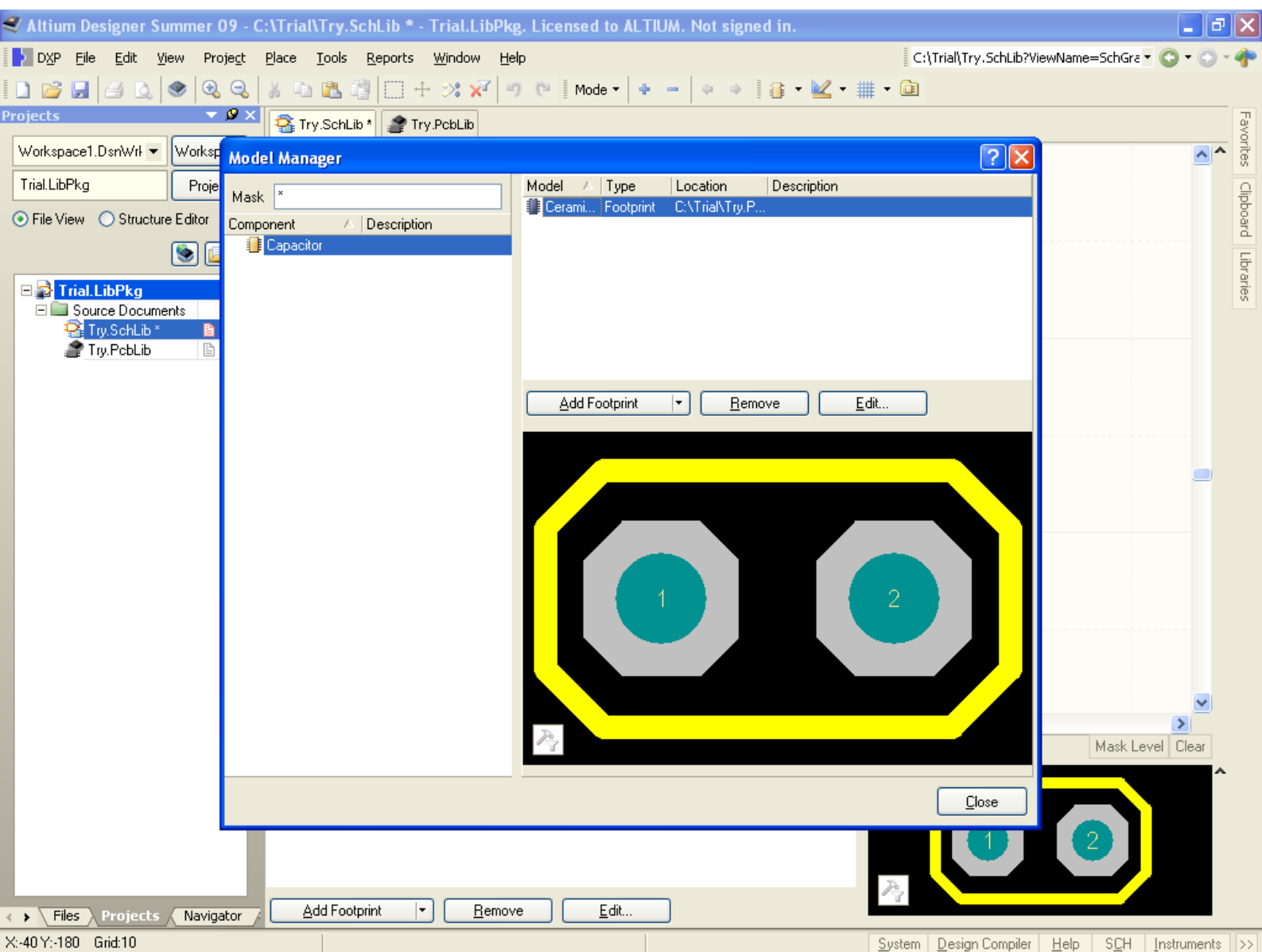
After clicking on browse option one new window will open in which we need to select the respective footprint for the schematic selected as shown and then click ok :



After that please check the name of the footprint & the library path in appropriate boxes :

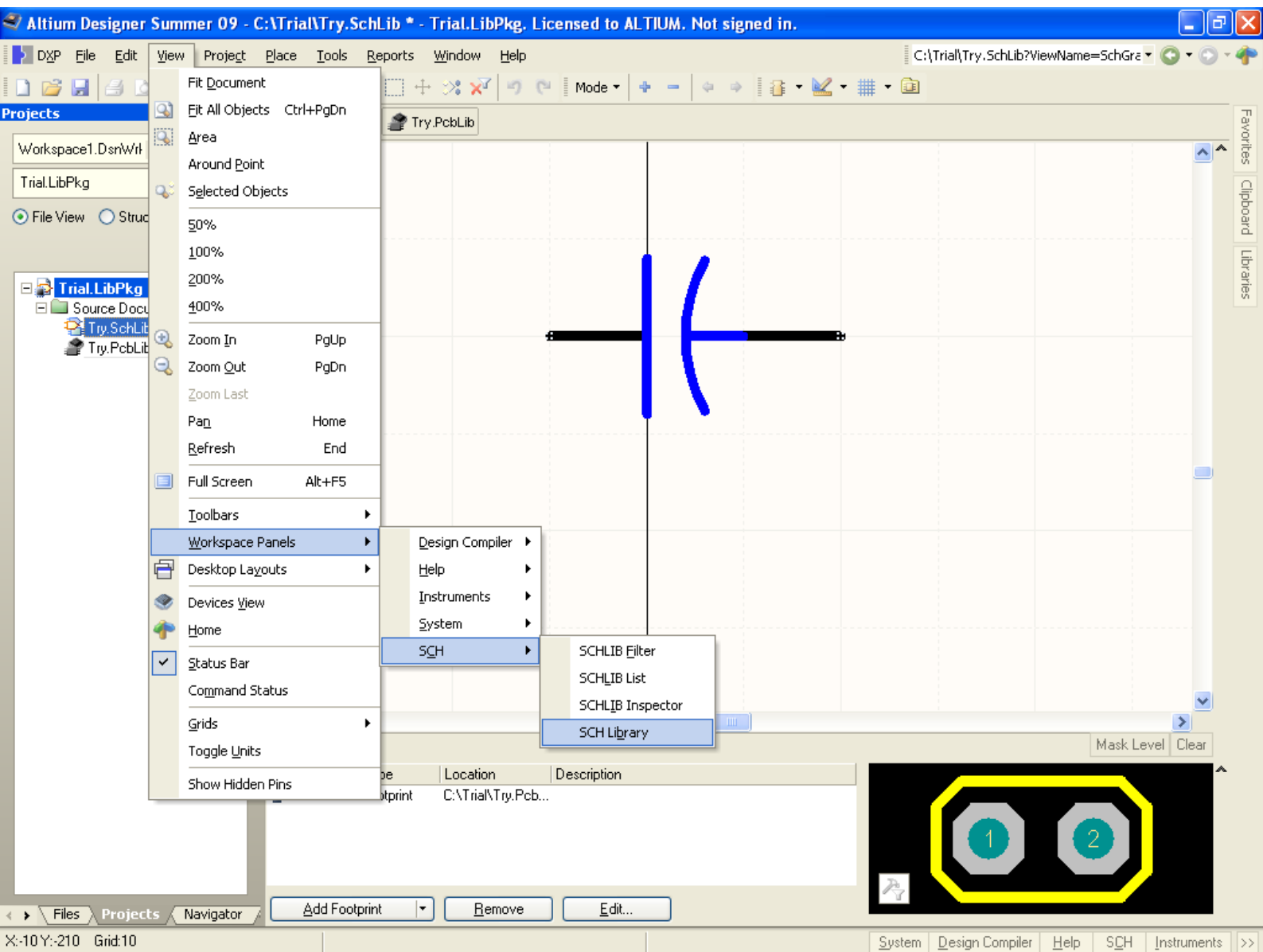


Then click on ok then see the window of model manager which will contain the footprint diagram with the associated component. Do follow the same procedure for adding the footprints to all of the components in the model manager :

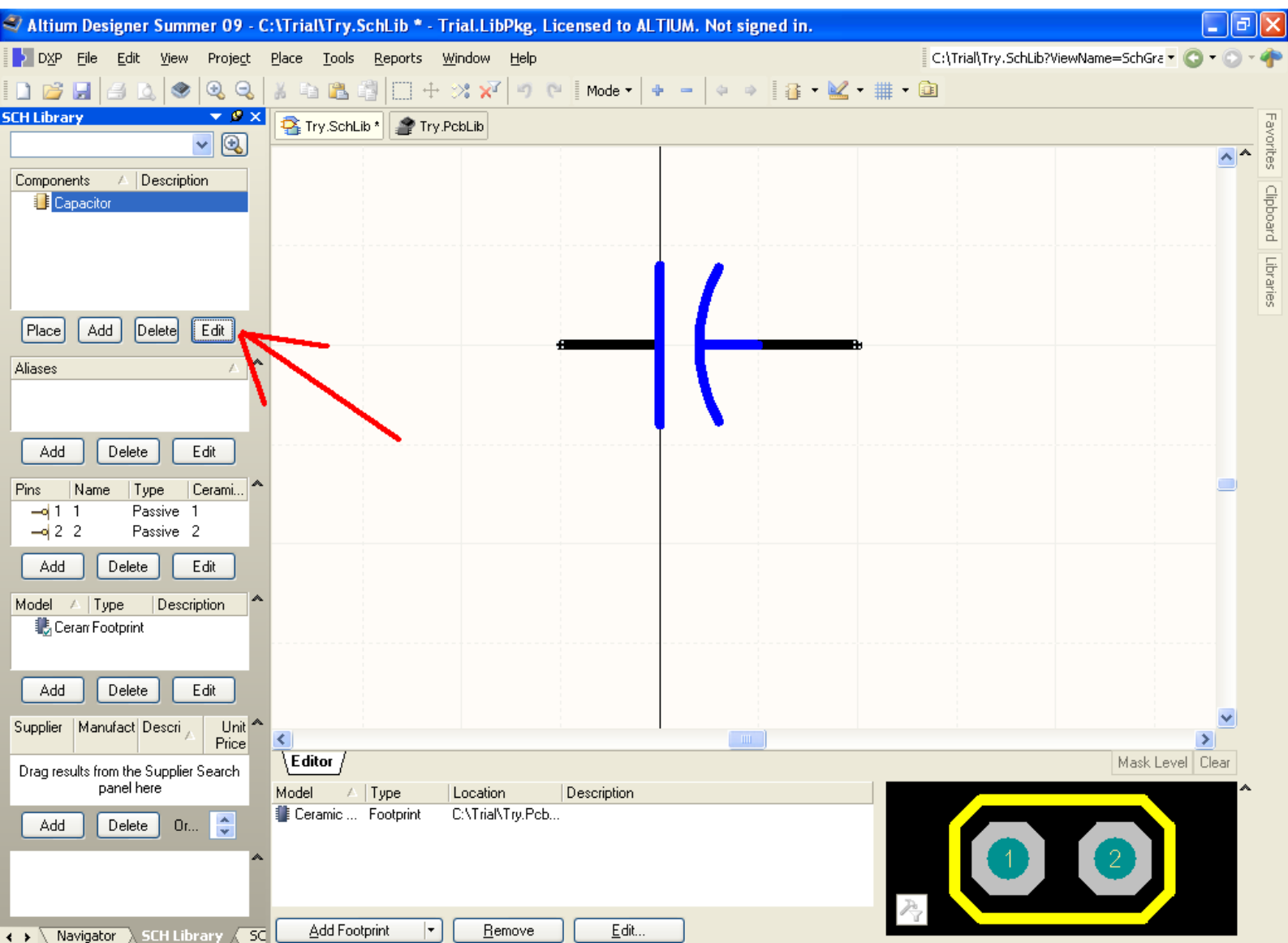


After adding the footprints to all of the schematic components close the window of model manager.

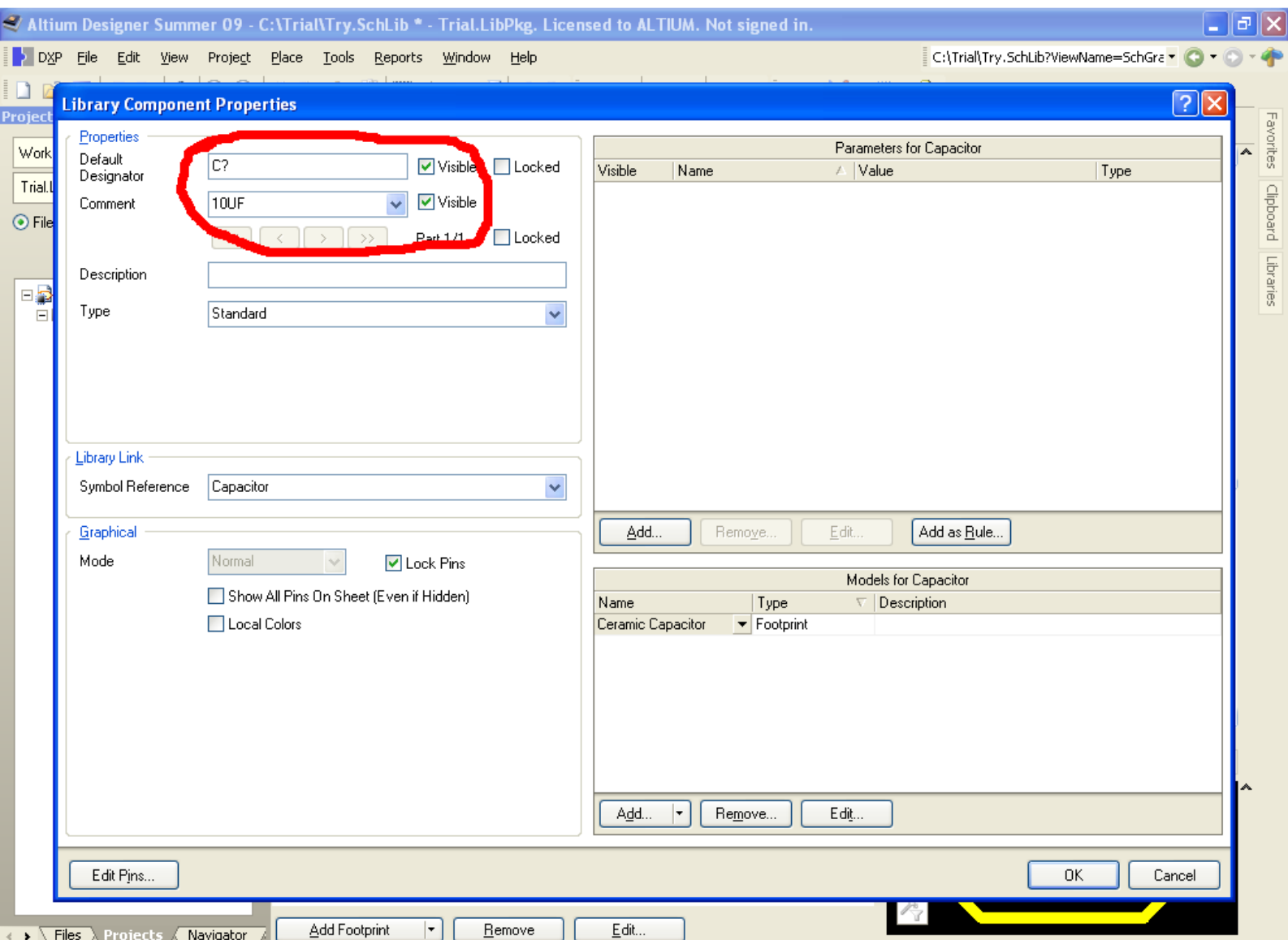
Now for editing all the components do as following :



Then an another tab will open on your left side as shown below :

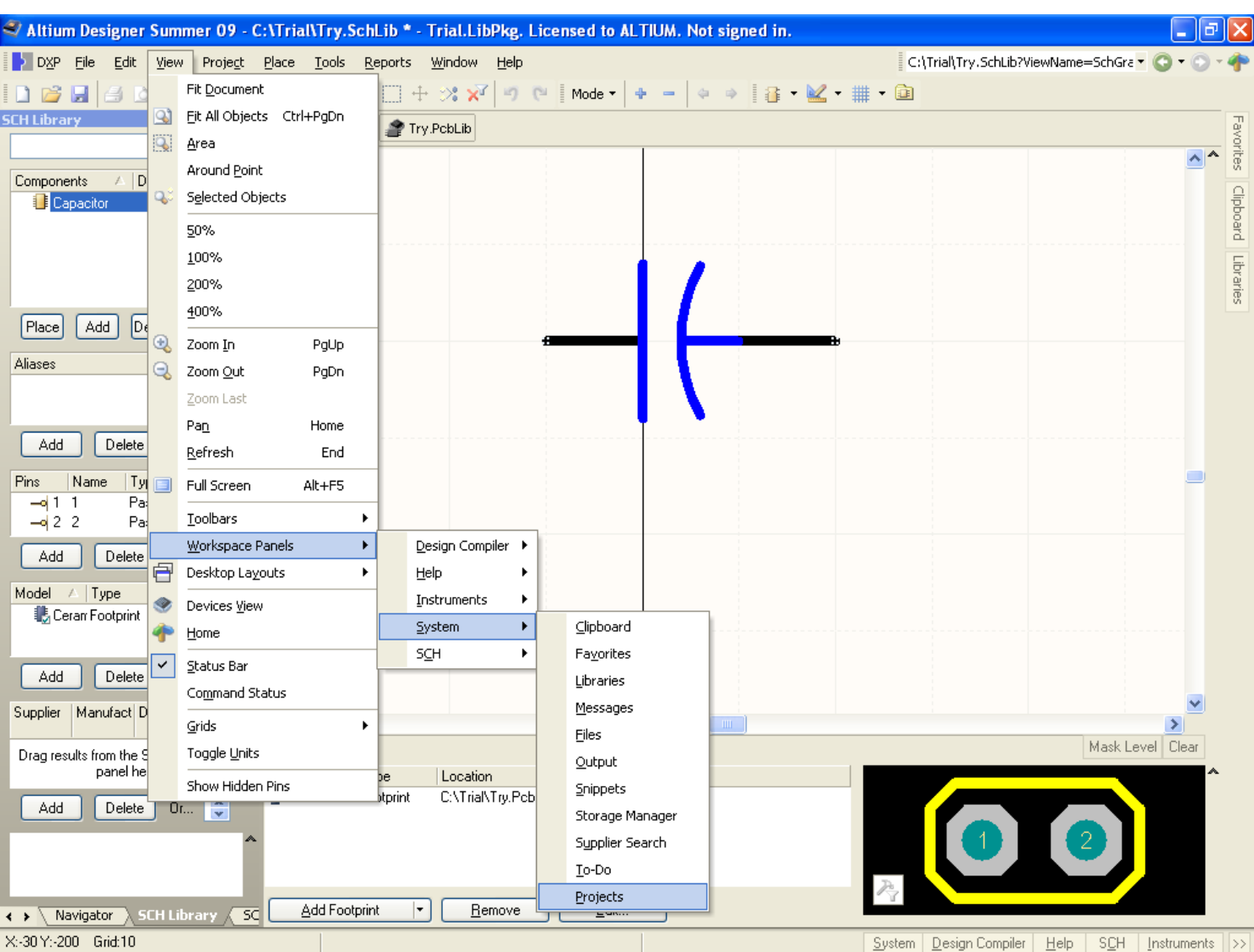


Here in edit option give proper name in the Default Designator & Comment do tick on visible option and then click on ok :



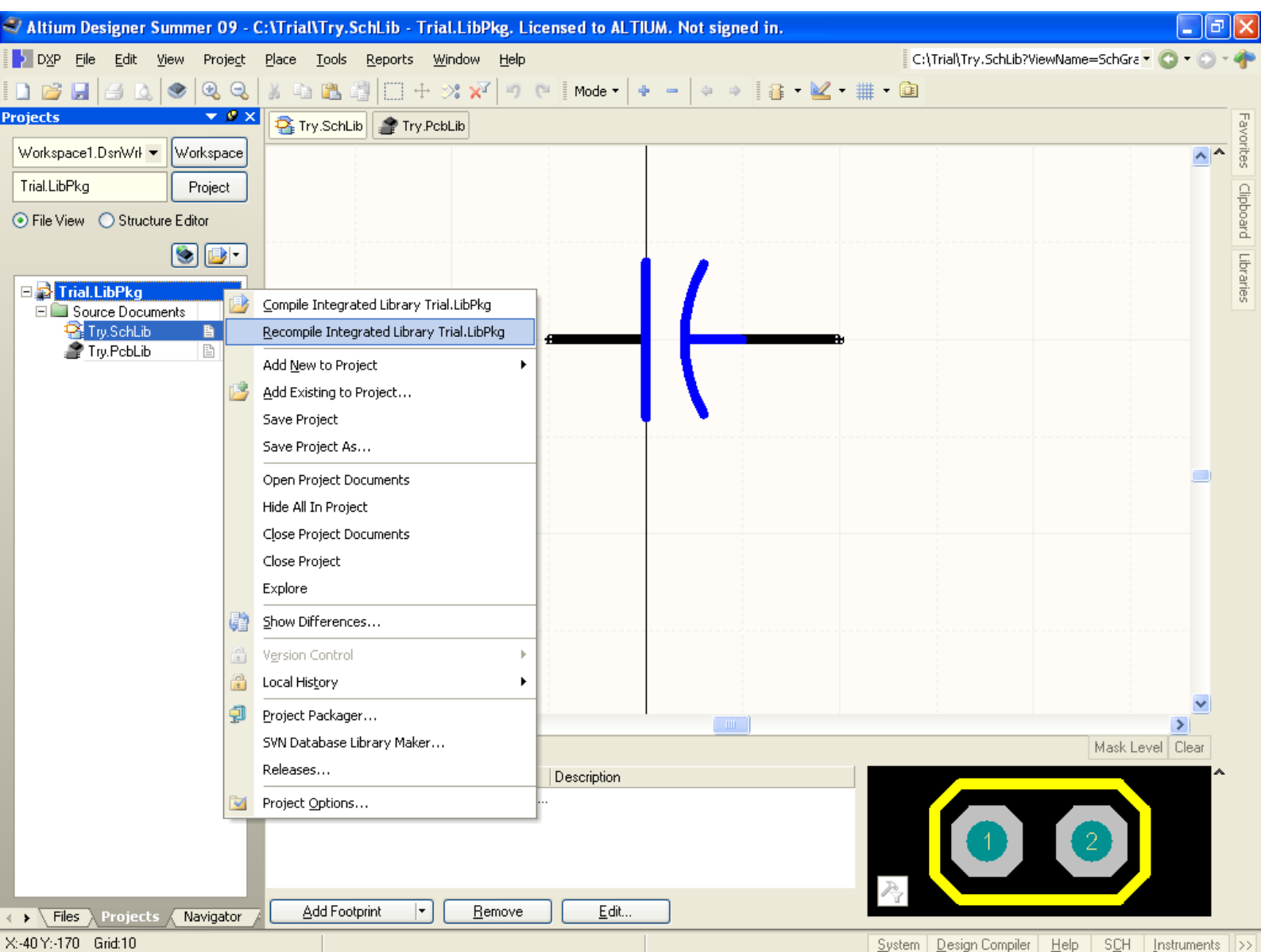
Then perform the same task of editing with all the components one by one .

Now go to the path as shown below :

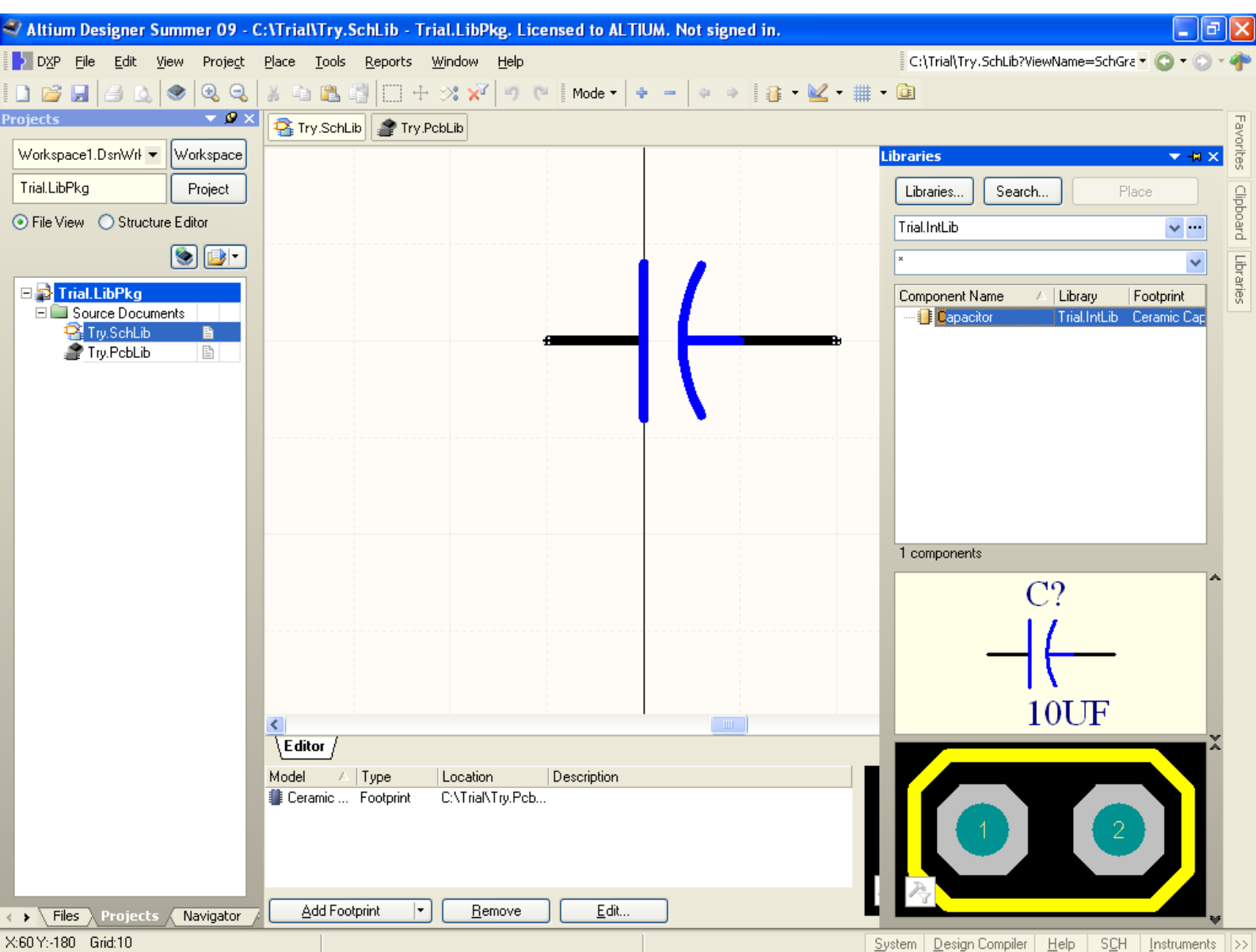


Then save the schematic library and the PCB library & save the project .

Now finally our library will be created by recompiling the project as shown below :



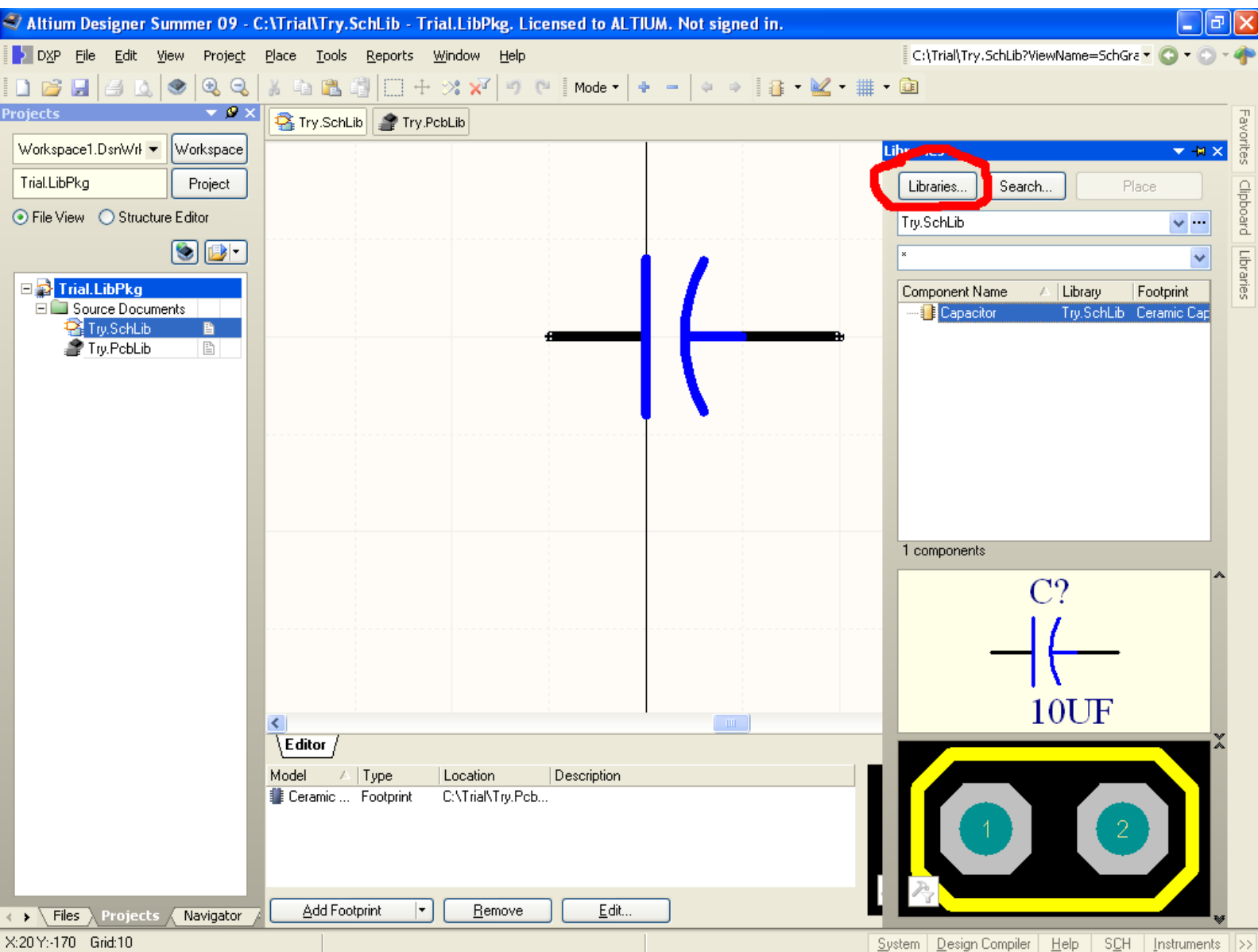
If the compilation is successful then a new library will be added in to the library on your right side and it will pop up :



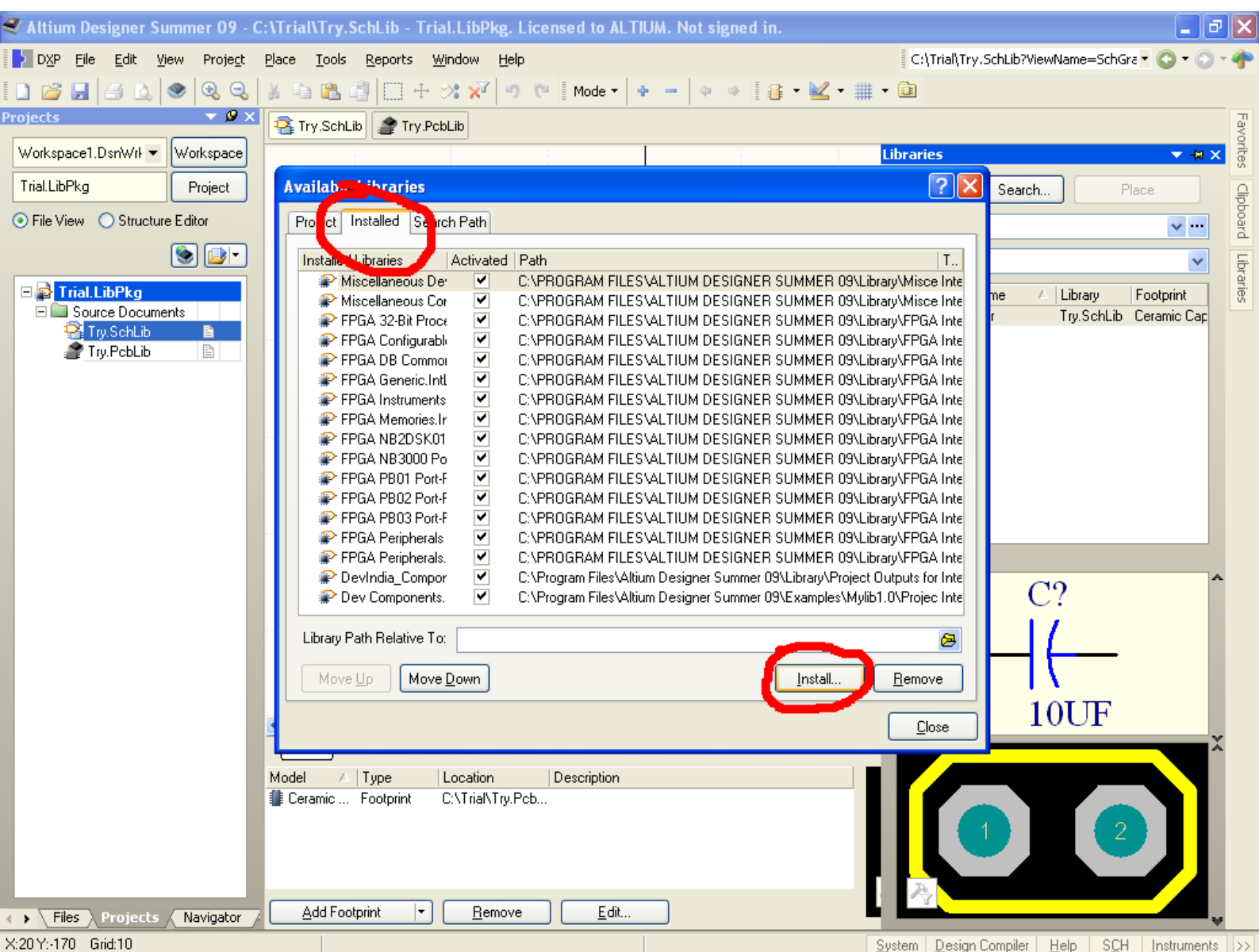
In the end library is made and the Integrated library will be created in the same folder where we had saved the schematic library & PCB library in this folder the Integrated library file will be in another folder named **Project Outputs for**
In this folder there is an Integrated library file.

Now to install the Library in other computer please copy the Integrated library file whose extension is .intlib and paste the library in the ALTIUM → Library folder .

In ALTIUM Software we need to install the same library go to Library → Libraries



Click on installed and then click on install :



Thereafter browse to proper path and then click on close . Finally the library will appear in library option on the right hand side.