

UNIT I

INTRODUCTION TO CAD Tool

#enthutech®

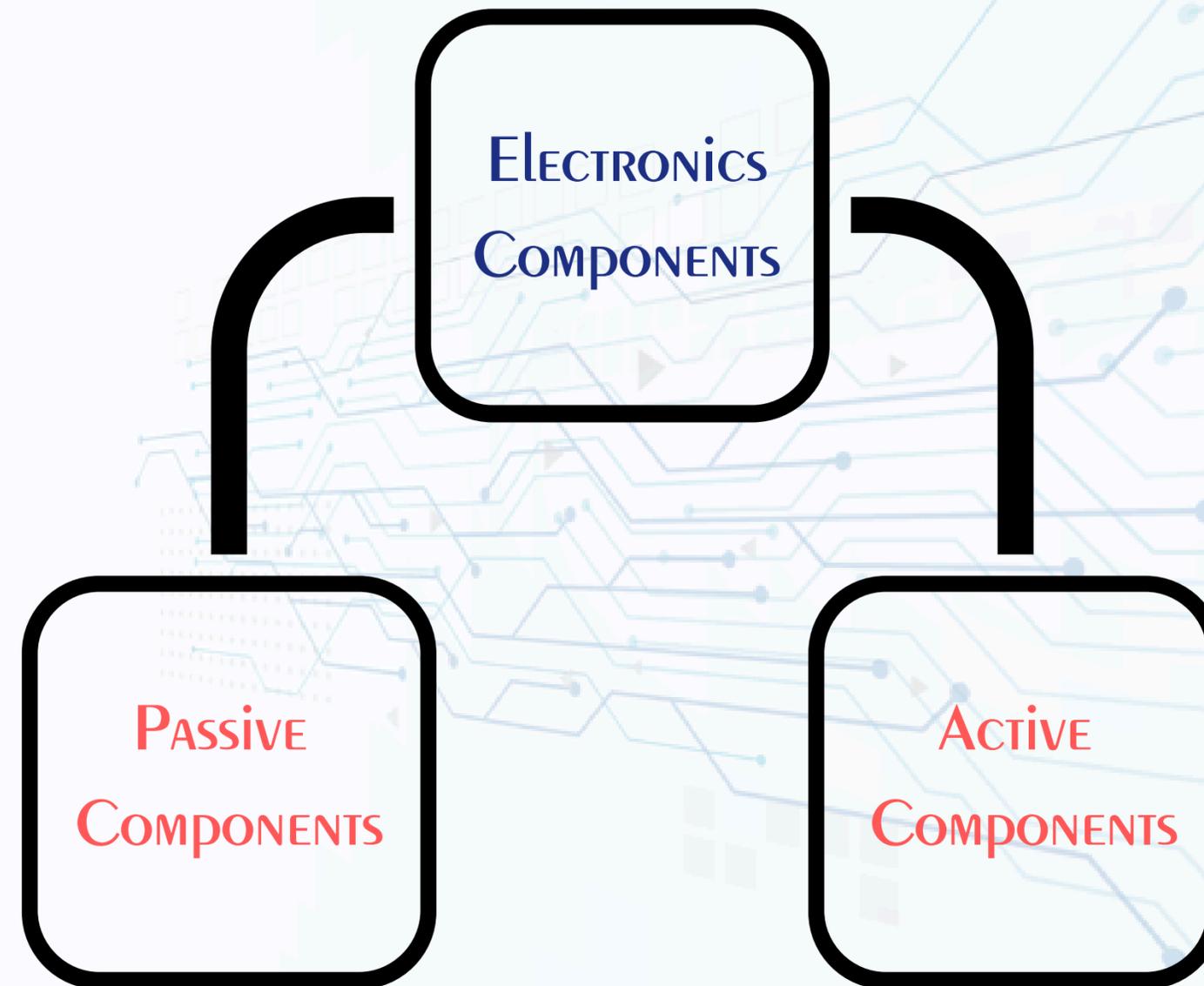
VOLTAGE

- Voltage is the electrical potential difference between two points in a circuit. It is the "push" that causes current to flow.
- Unit: Volts (V).

CURRENT

- Current is the flow of electric charge through a conductor.
- Unit: Amperes (A).

Key Components in PCB



#enthutech®

PASSIVE COMPONENTS

Components that do not require external power and cannot amplify signals; they only consume energy or store it temporarily.

Types

Resistor, Capacitor, Inductor, LDR, Variable Resistor, Switch, Etc.,

Active Components

Components that require an external power source to operate and can amplify or control electrical signals.

Types

Diode, Transistor, IC, LED, Battery, Etc.,

RESISTOR

- A resistor opposes the flow of electrical current.

Application

- Current limiting in circuits.
- Voltage dividers.

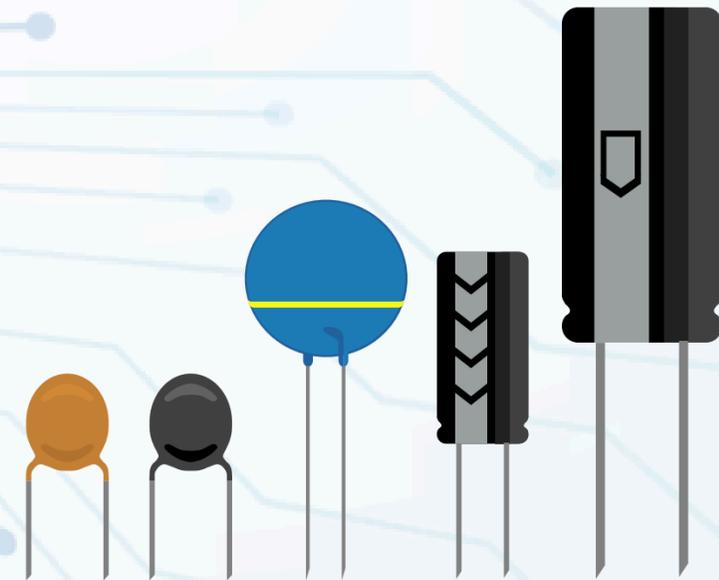


CAPACITORS

- A capacitor stores and releases electrical energy

Application

- Smoothing power supply noise.
- Timing and oscillation circuits.

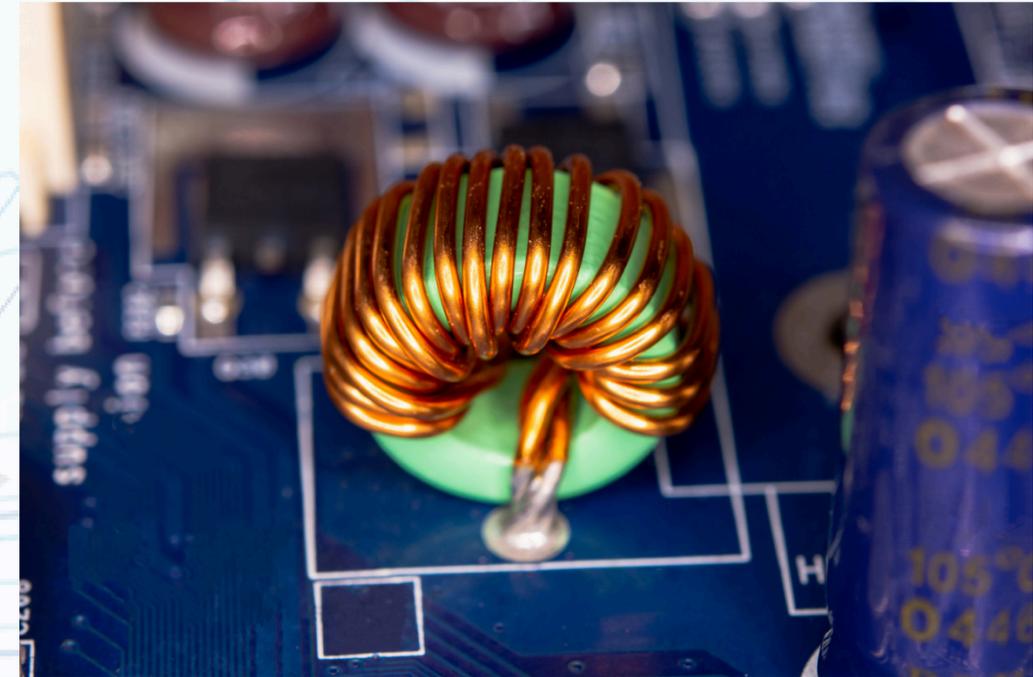


INDUCTORS

- An inductor stores energy in a magnetic field when current flows through it.

APPLICATION

- Filters in power supplies.
- Transformers.



Diode

- A diode allows current to flow in only one direction.

Application

- Rectifiers.
- Voltage regulation (Zener diodes).

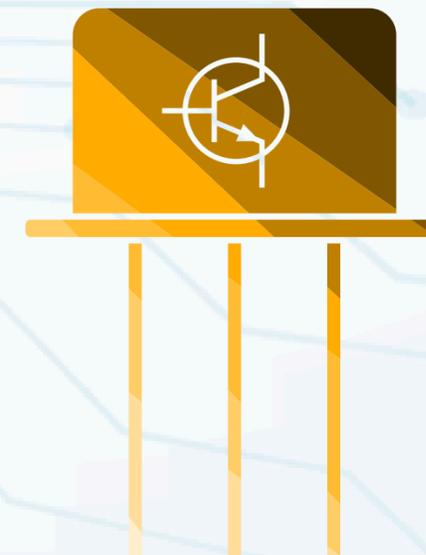


TRANSISTOR

- A transistor is a semiconductor device used for amplification and switching.

Application

- Amplifiers in audio devices.
- Logic gates in digital circuits.

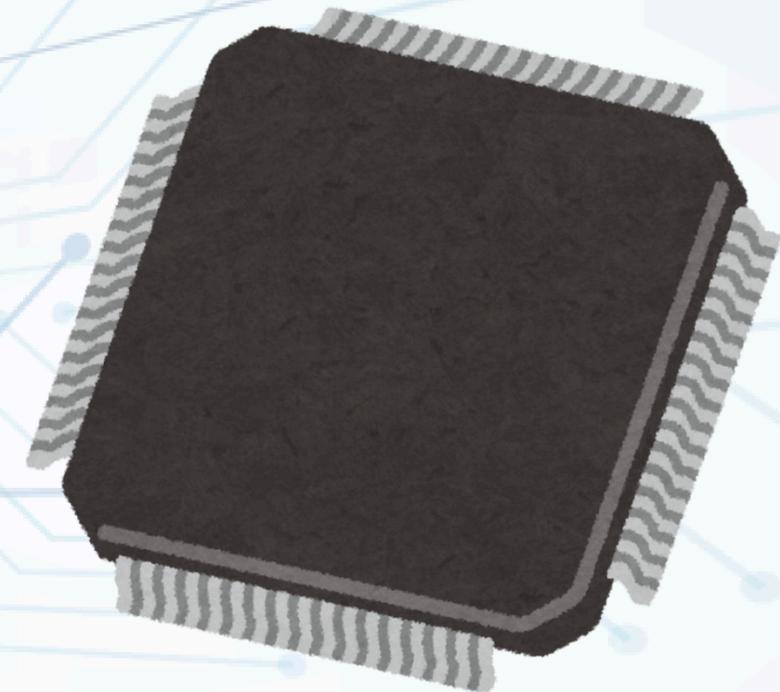


INTEGRATED CIRCUITS (ICs)

- ICs are miniature circuits with multiple electronic components integrated into a single chip.

Application

- Microprocessors, Memory Chips.
- Power management and control systems.



#enthutech®

SOFTWARE INSTALLATION PROCESS

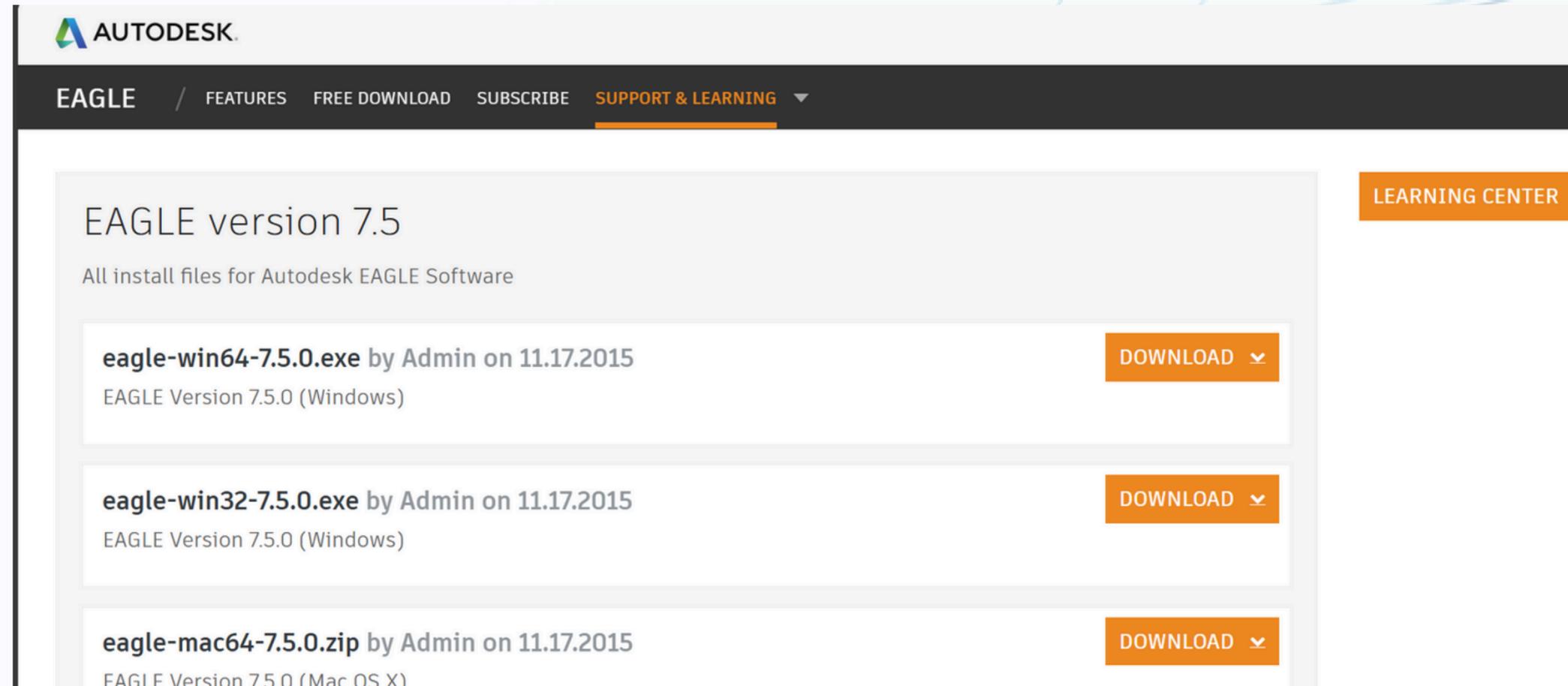
#enthutech[®]

EAGLE SOFTWARE INSTALLATION

The screenshot shows a Microsoft Bing search results page for the query "eagle 7.5.0 download". The search bar at the top contains the text "eagle 7.5.0 download" and the Microsoft Bing logo. Below the search bar, there are navigation tabs for "SEARCH", "IMAGES", "COPILOT", "VIDEOS", "MAPS", "NEWS", "SHOPPING", "MORE", and "TOOLS". The search results indicate "About 15,500,000 results". The top result is from Autodesk Eagle, with the URL "http://eagle.autodesk.com > eagle > software-versions". The main heading for this result is "EAGLE version 7.5 - Autodesk", dated "Nov 17, 2015", with a sub-heading "All install files for Autodesk EAGLE Software." Below this, there are two columns of links: "Manufacturing" (Increase your manufacturing throughput through automation, collaboration, and ...) and "Support" (Will saved libraries, PCB projects, and Library.io be usable after EAGLE retires. ...). There is also a "Suppliers" link (All prospective suppliers are required to register on our prospective supplier ...). At the bottom of the result, there is a link "See results only from eagle.autodesk.com" and a link to "element14 Community". On the right side of the search results, there is a "Related searches" box with the following items: "eagle 7.0 download", "eagle 7.7 download", "eagle 7.7 install link", "eagle 7.7.0 install links", "cadsoft eagle v7 download", "cadsoft eagle v7.7", and "eagle pcb".

#enthutech®

EAGLE SOFTWARE INSTALLATION

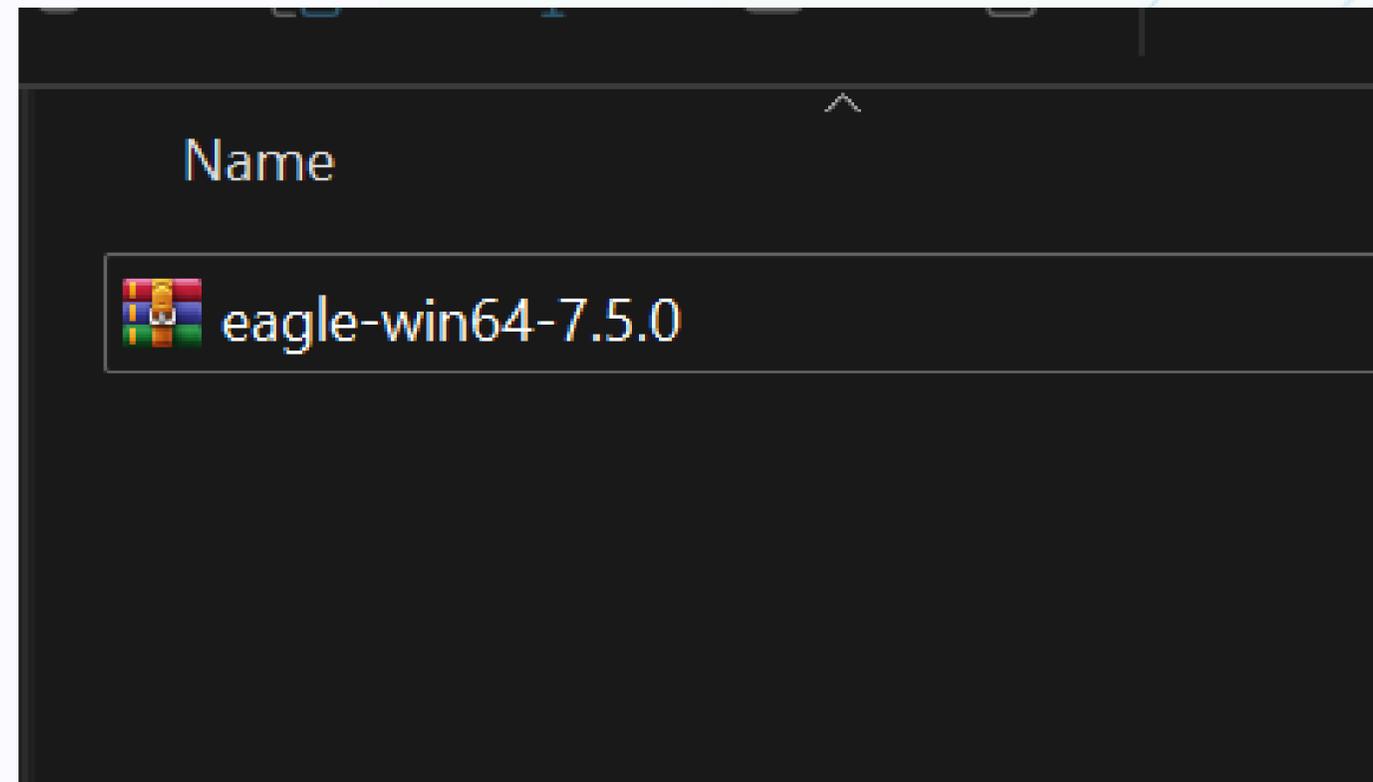


The screenshot shows the Autodesk EAGLE software download page. At the top left is the Autodesk logo. The navigation bar includes 'EAGLE', 'FEATURES', 'FREE DOWNLOAD', 'SUBSCRIBE', and 'SUPPORT & LEARNING'. The main content area is titled 'EAGLE version 7.5' and includes the subtitle 'All install files for Autodesk EAGLE Software'. A 'LEARNING CENTER' button is located on the right. Below the title, there are three download options, each with a 'DOWNLOAD' button:

- eagle-win64-7.5.0.exe** by Admin on 11.17.2015
EAGLE Version 7.5.0 (Windows)
- eagle-win32-7.5.0.exe** by Admin on 11.17.2015
EAGLE Version 7.5.0 (Windows)
- eagle-mac64-7.5.0.zip** by Admin on 11.17.2015
EAGLE Version 7.5.0 (Mac OS X)

#enthutech®

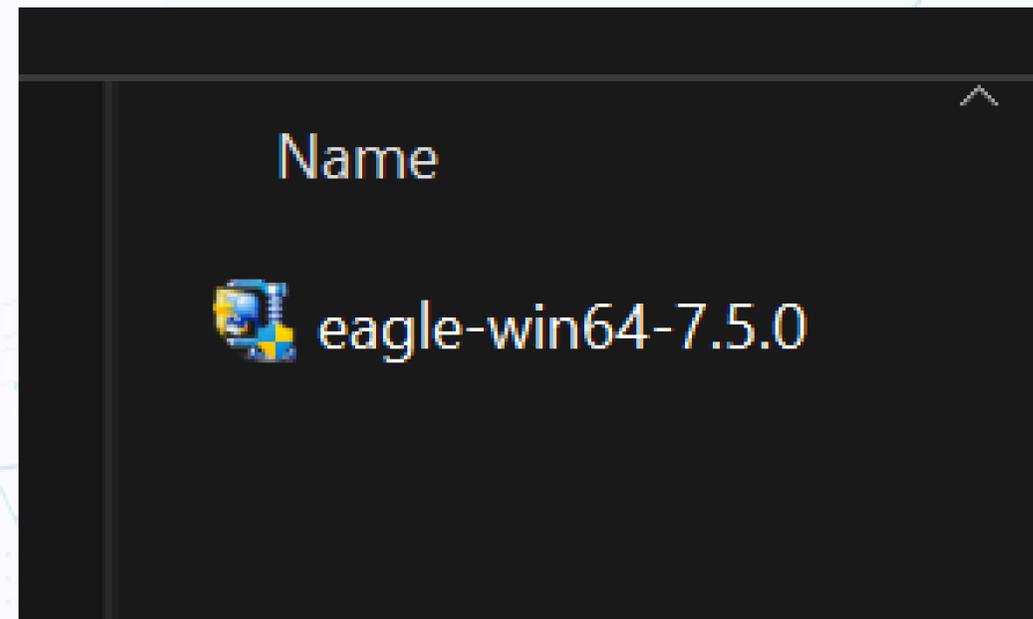
EAGLE SOFTWARE INSTALLATION



Extract the Rar file

#enthutech®

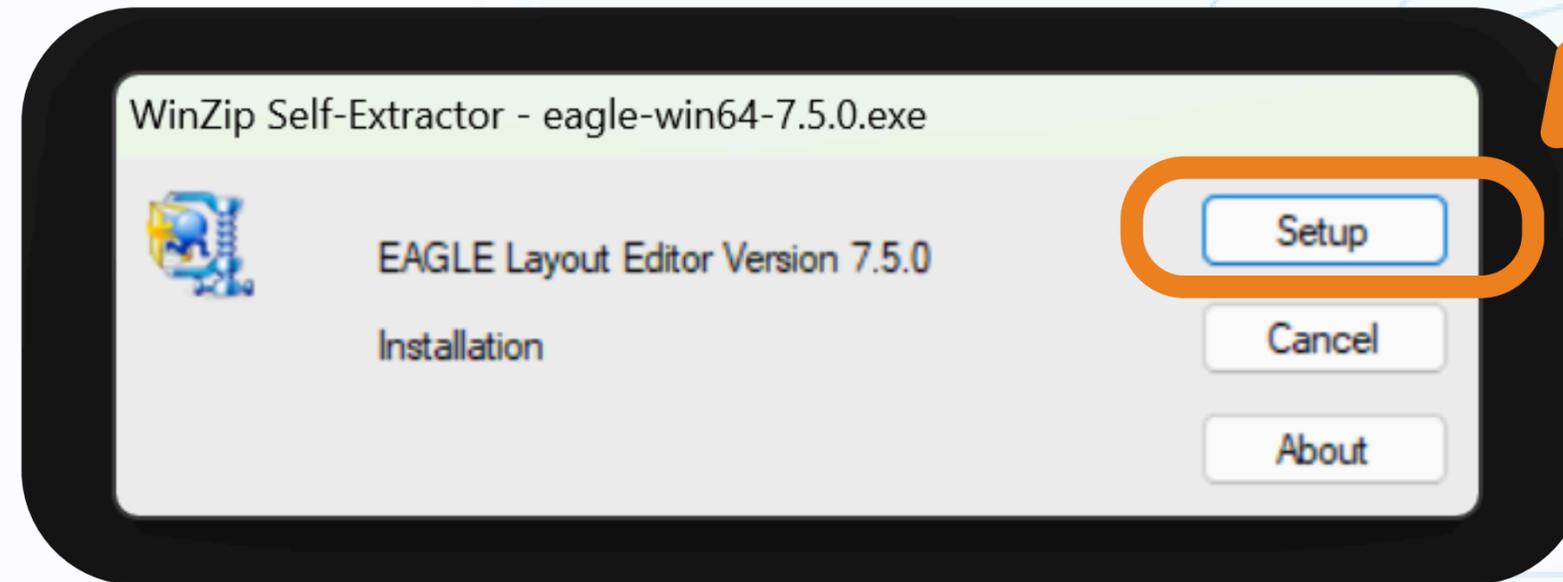
EAGLE SOFTWARE INSTALLATION....



Click the exe file

#enthutech®

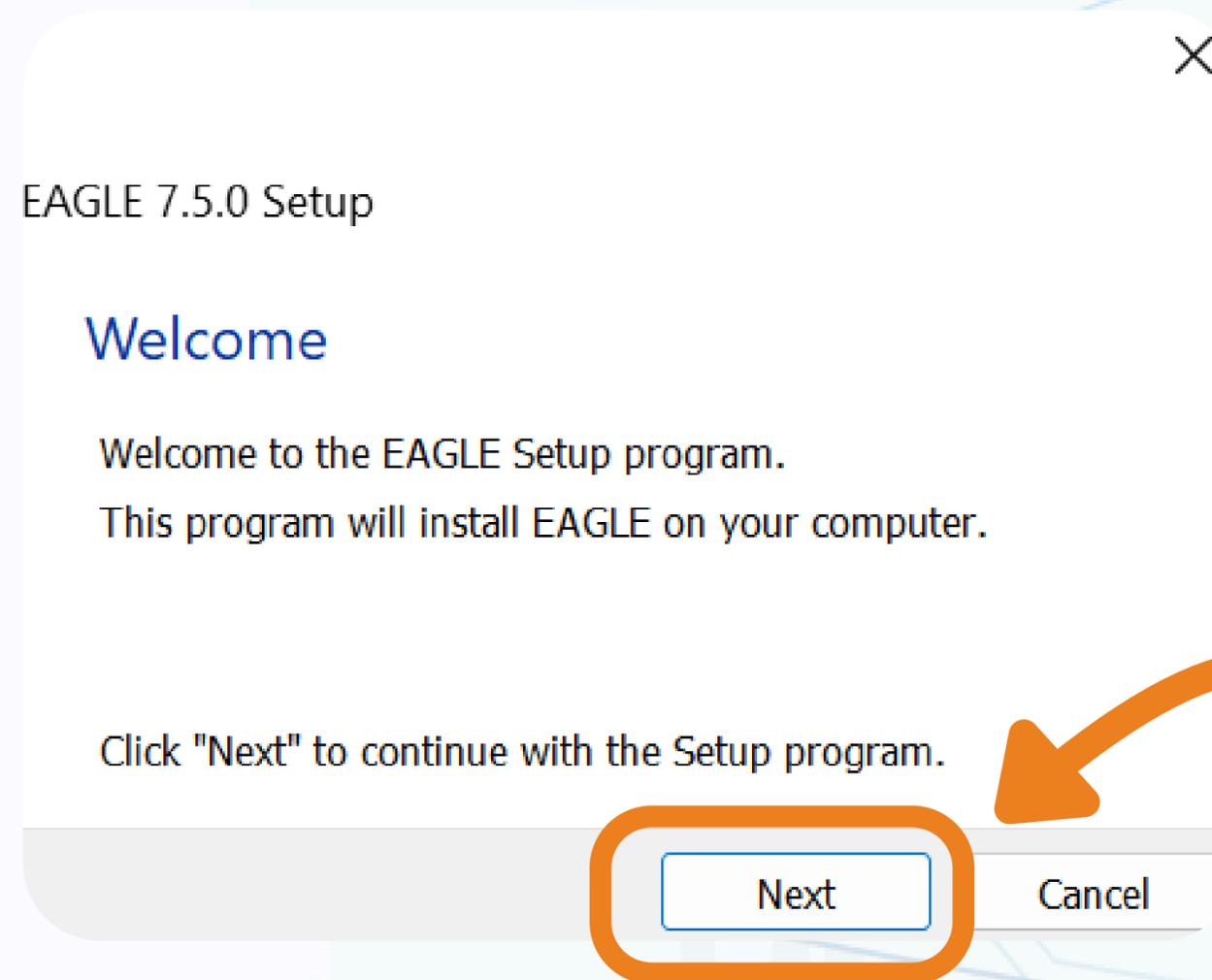
EAGLE SOFTWARE INSTALLATION....



Click the Setup option

#enthutech®

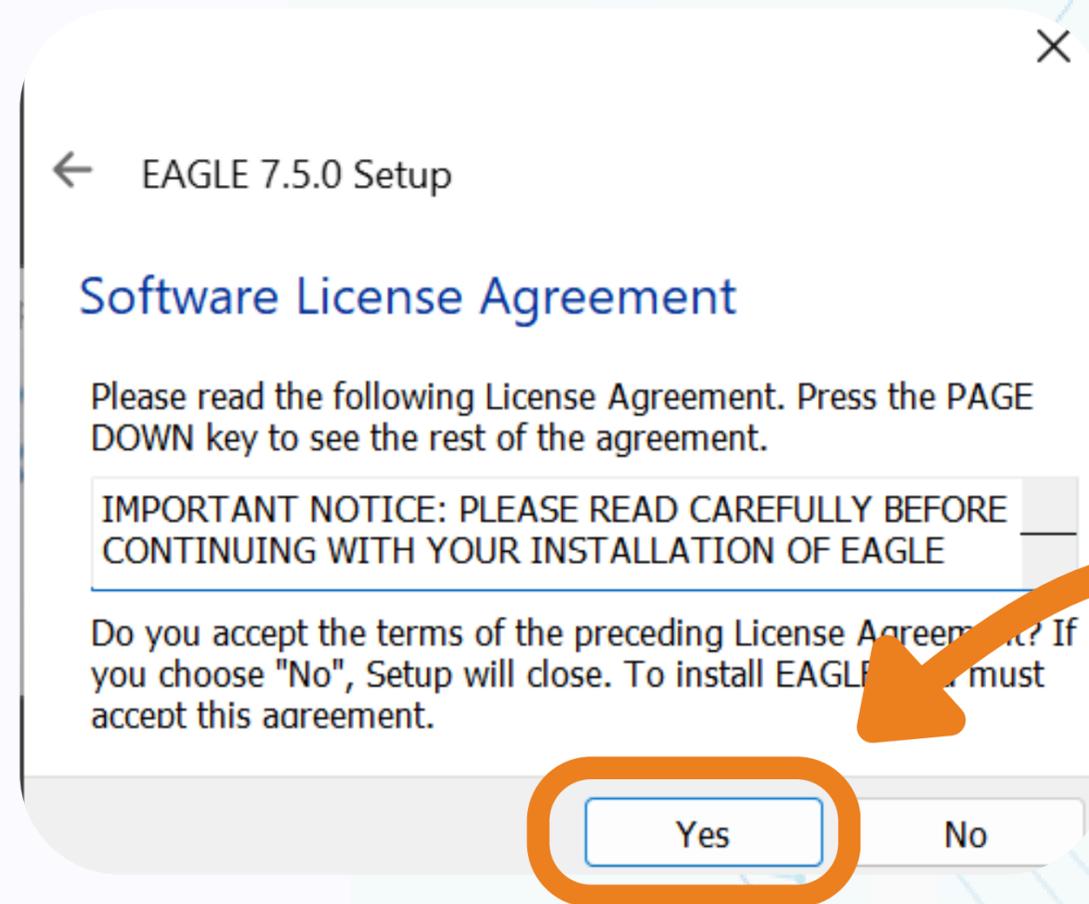
EAGLE SOFTWARE INSTALLATION....



Click the Next option

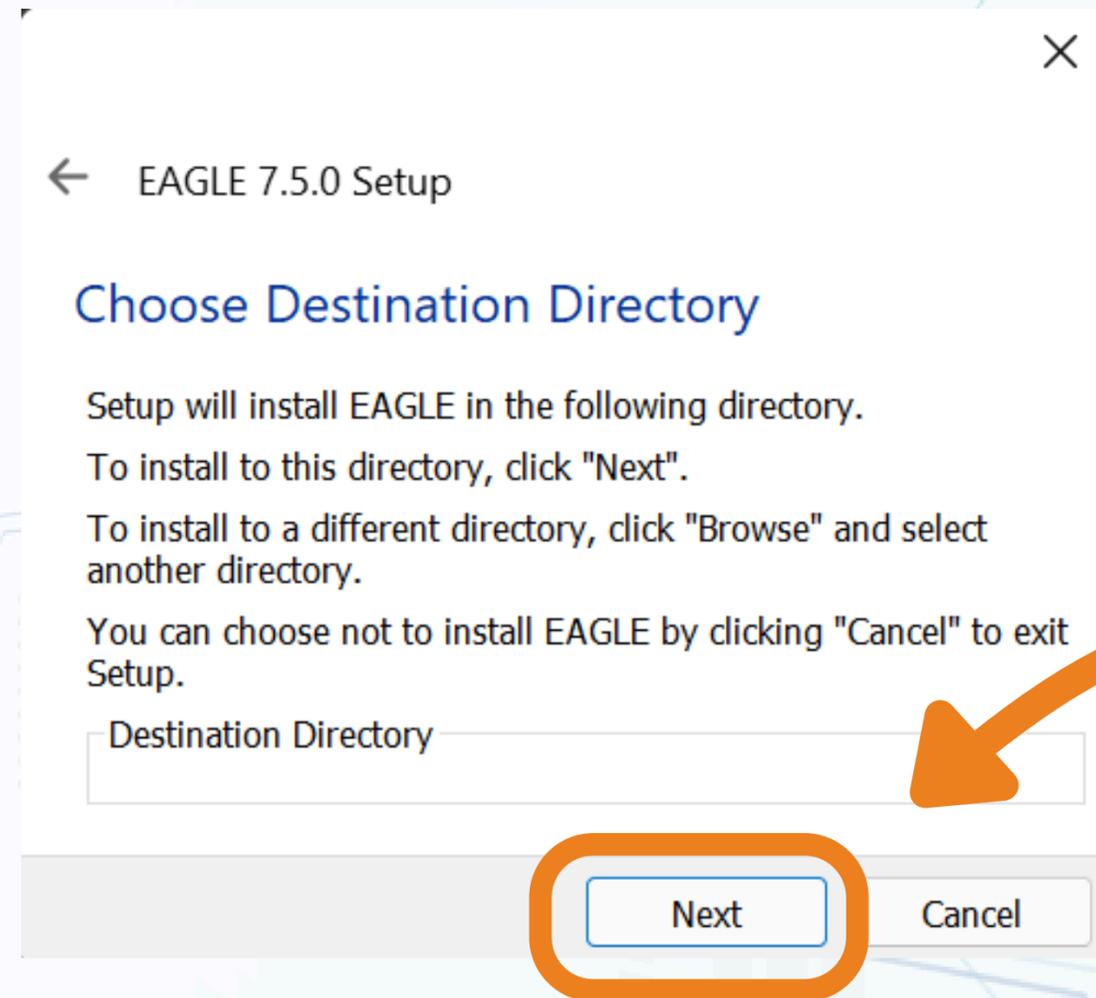
#enthutech®

EAGLE SOFTWARE INSTALLATION....



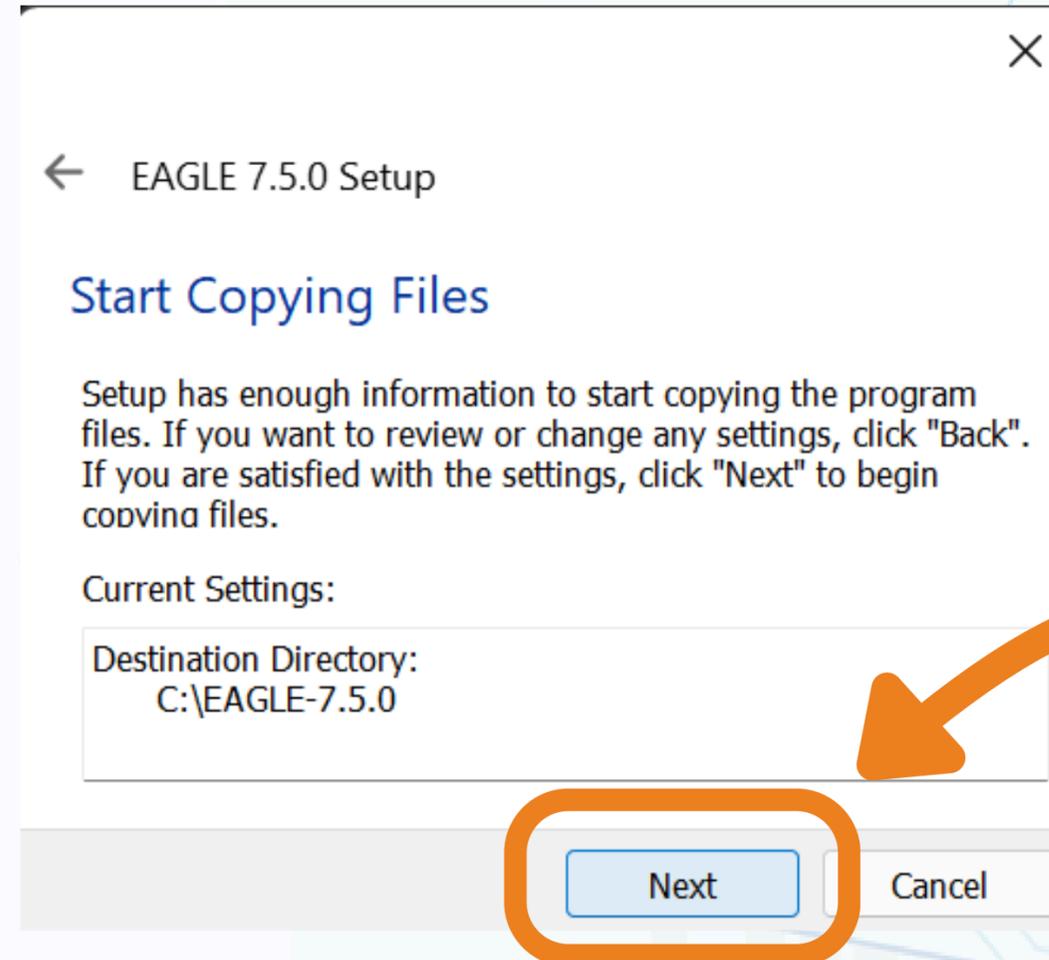
Click the Yes option

EAGLE SOFTWARE INSTALLATION....



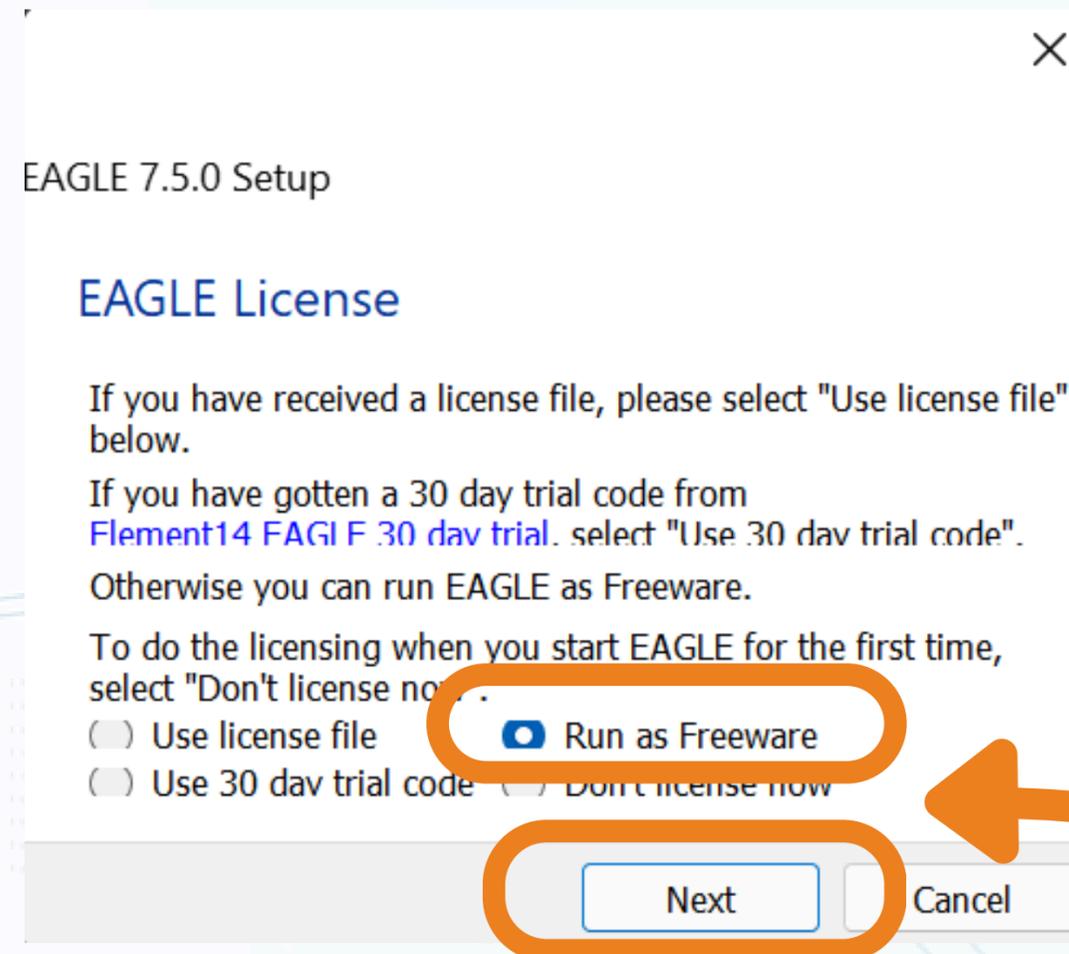
Click the Next option

EAGLE SOFTWARE INSTALLATION....



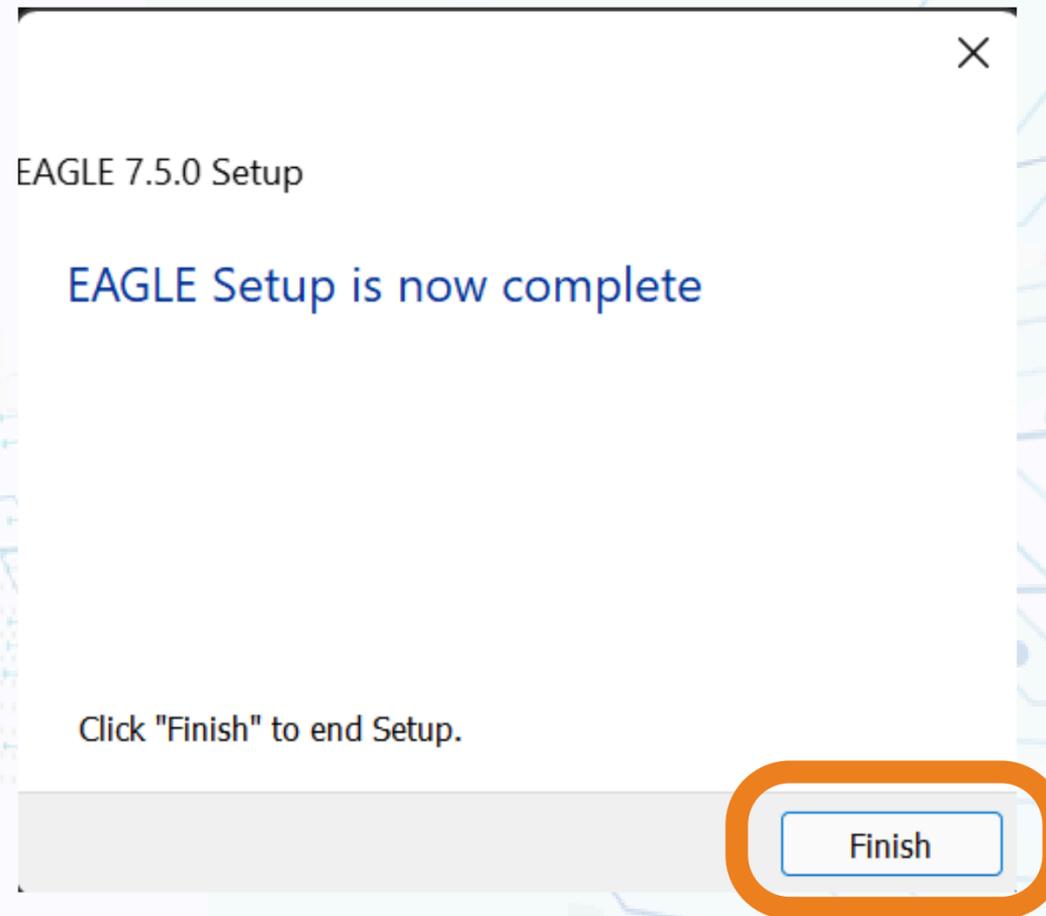
Click the Next option

EAGLE SOFTWARE INSTALLATION....



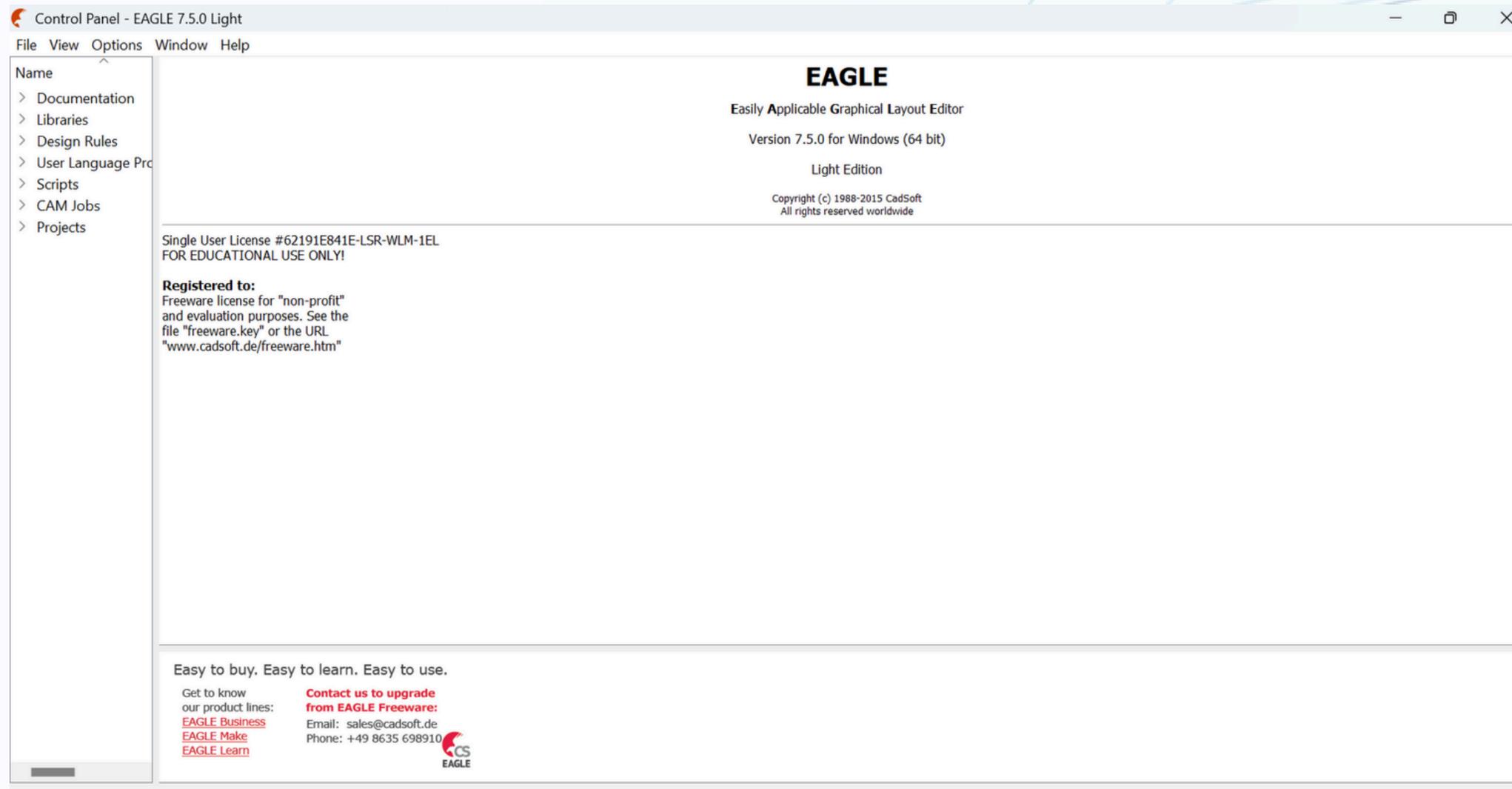
Select Run as Freeware and Click the Next option

EAGLE SOFTWARE INSTALLATION....



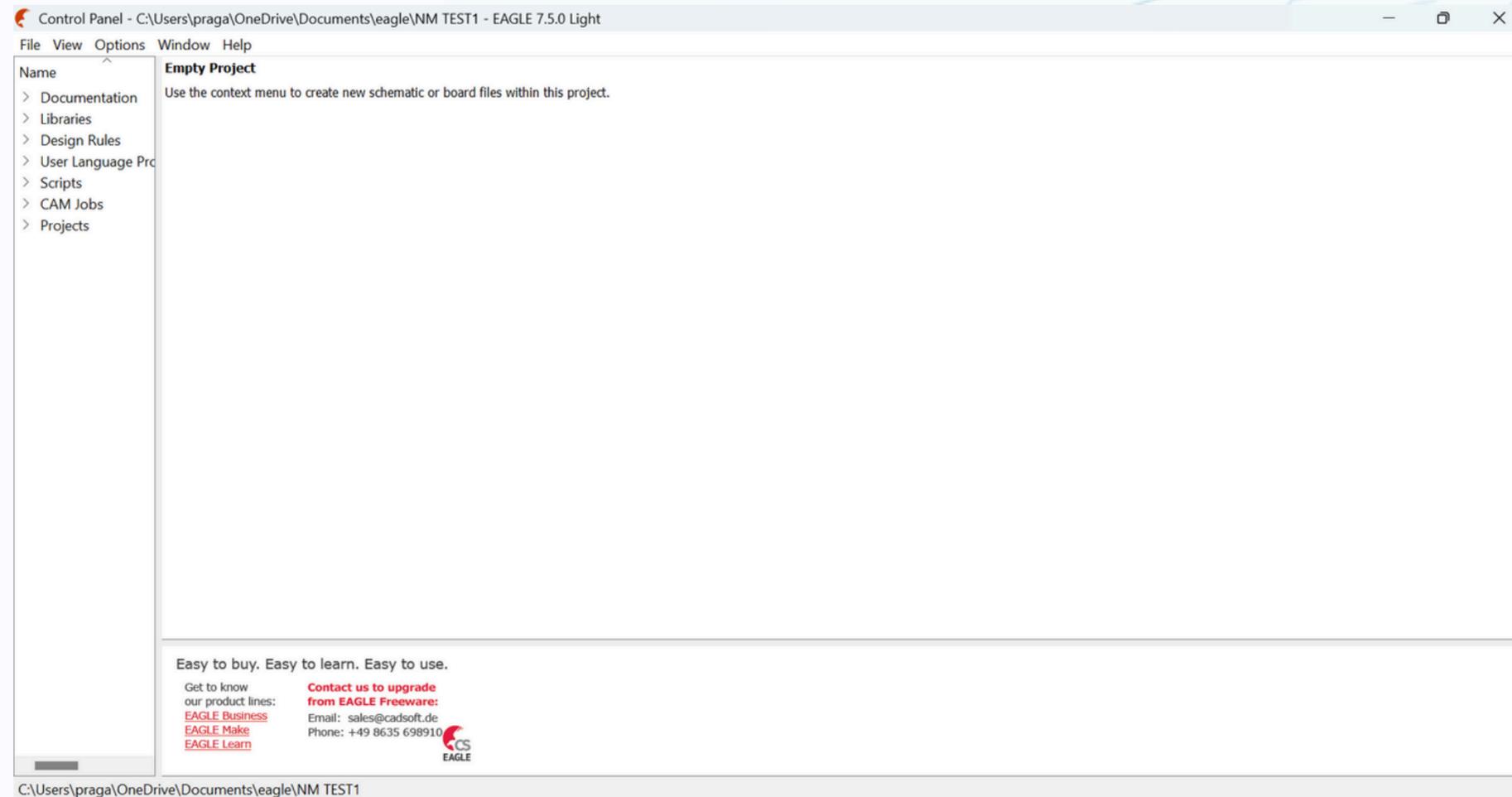
Click the Finish option

EAGLE SOFTWARE



#enthutech®

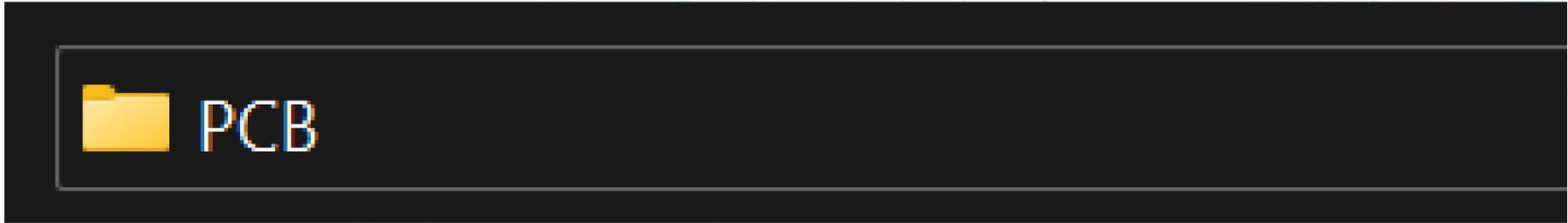
EAGLE SOFTWARE



Open the Eagle 7.5.0

#enthutech®

File Management

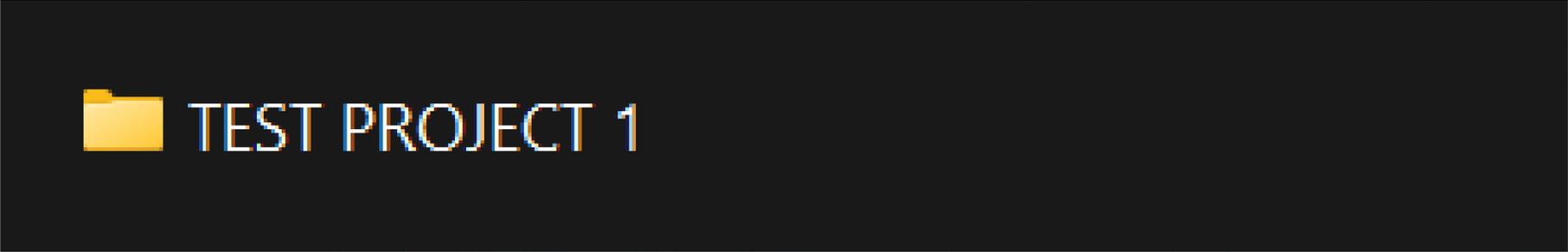


 PCB

Create a folder to store all the project files

#enthutech[®]

File MANAGEMENT

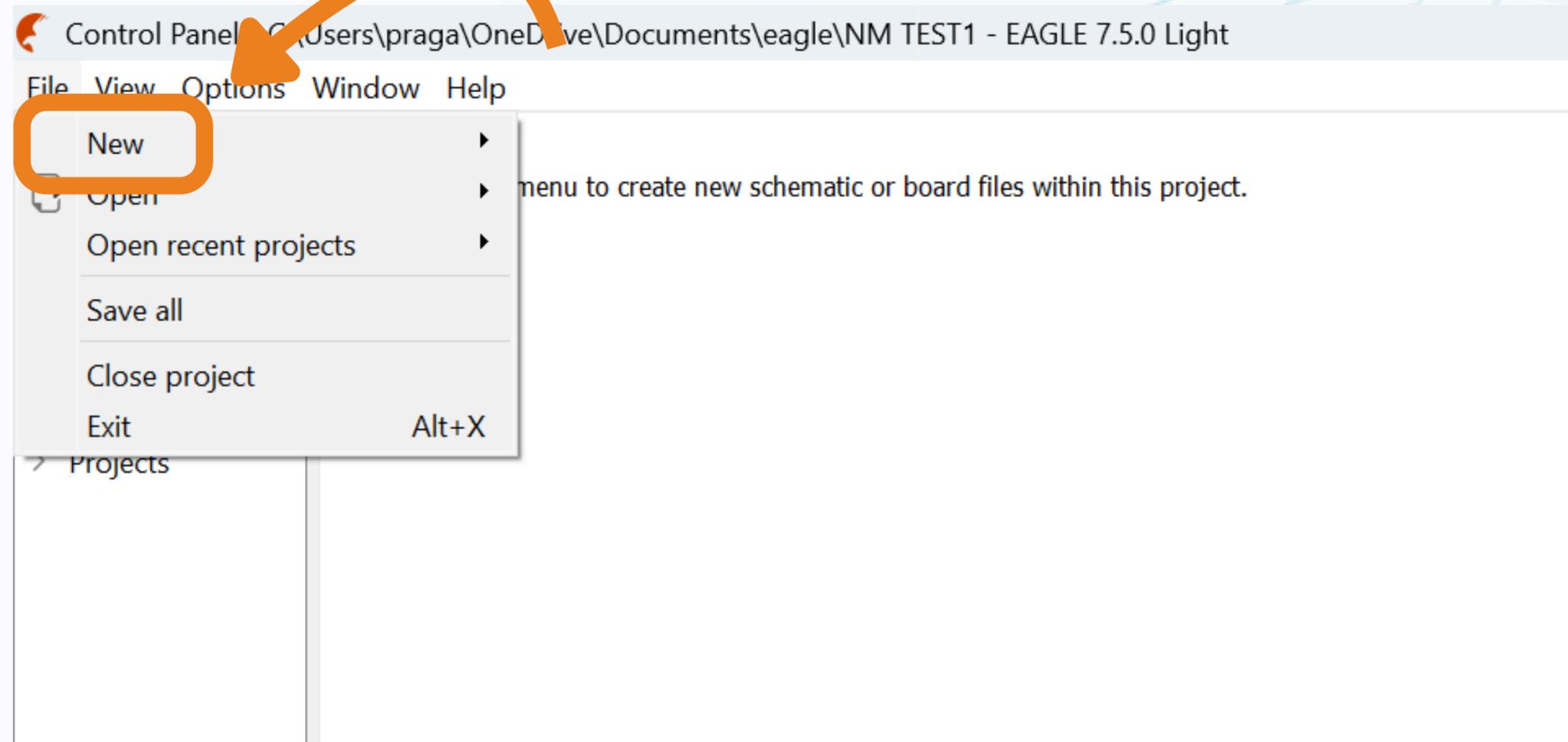


 TEST PROJECT 1

**Create a Project folder to store the
Design and fabrication files**

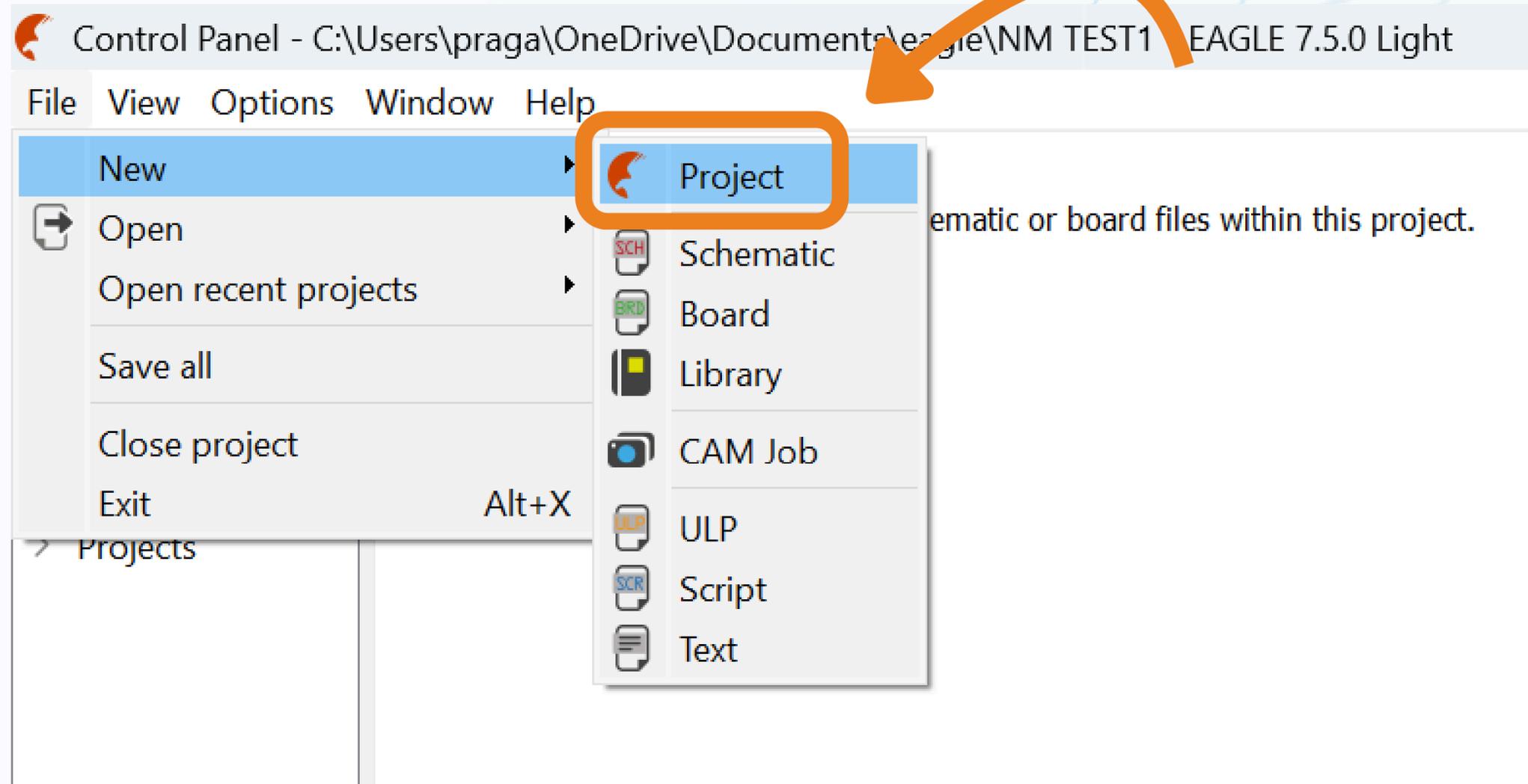
#enthutech®

NEW PROJECT CREATION



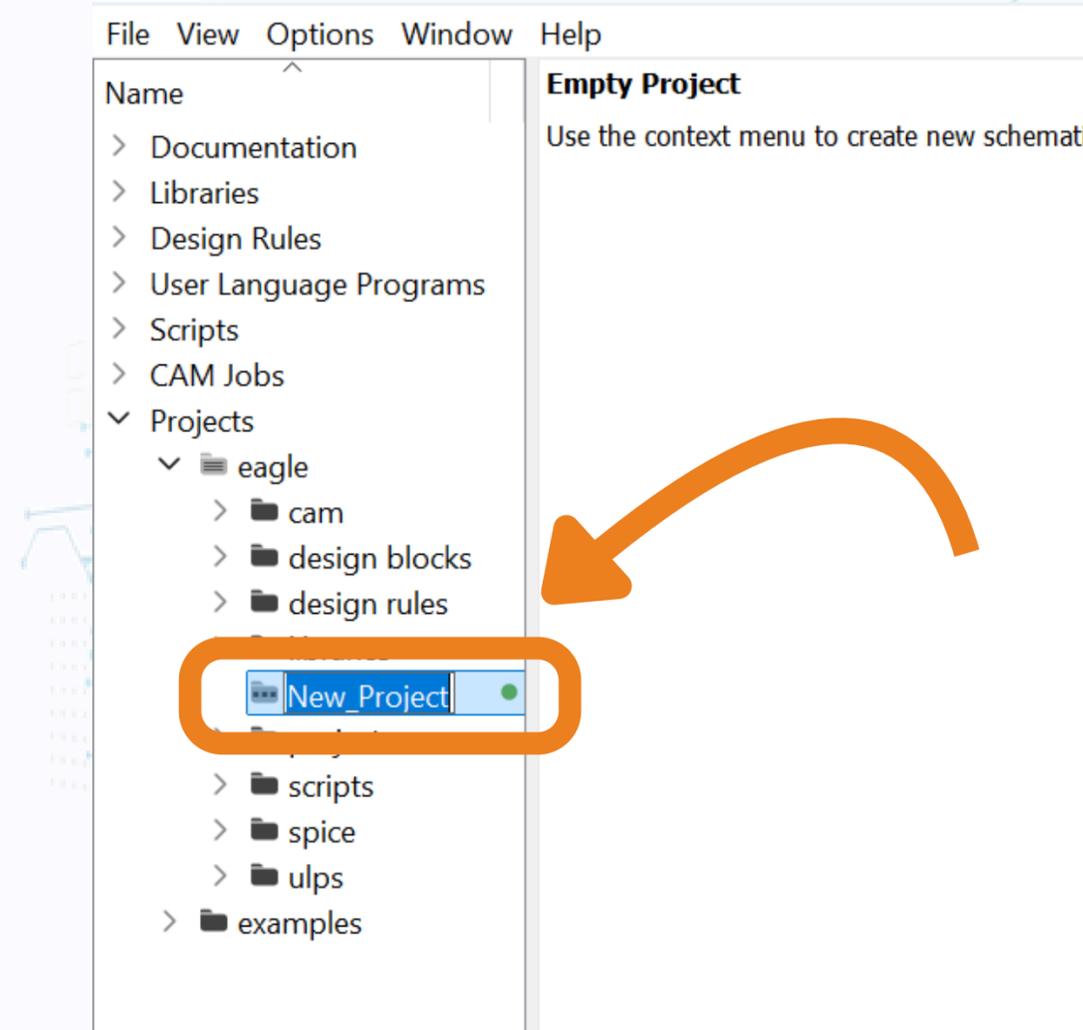
Click file > New

NEW PROJECT CREATION



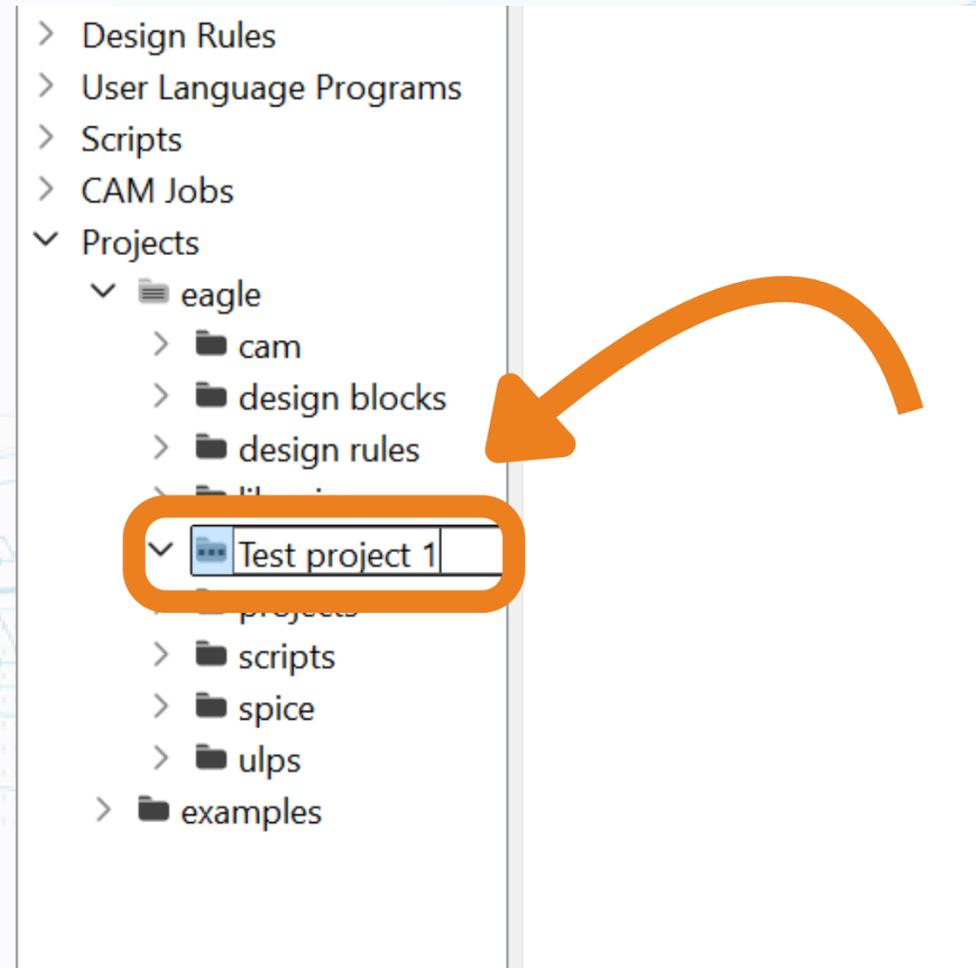
Click file > New > Project

NEW PROJECT CREATION



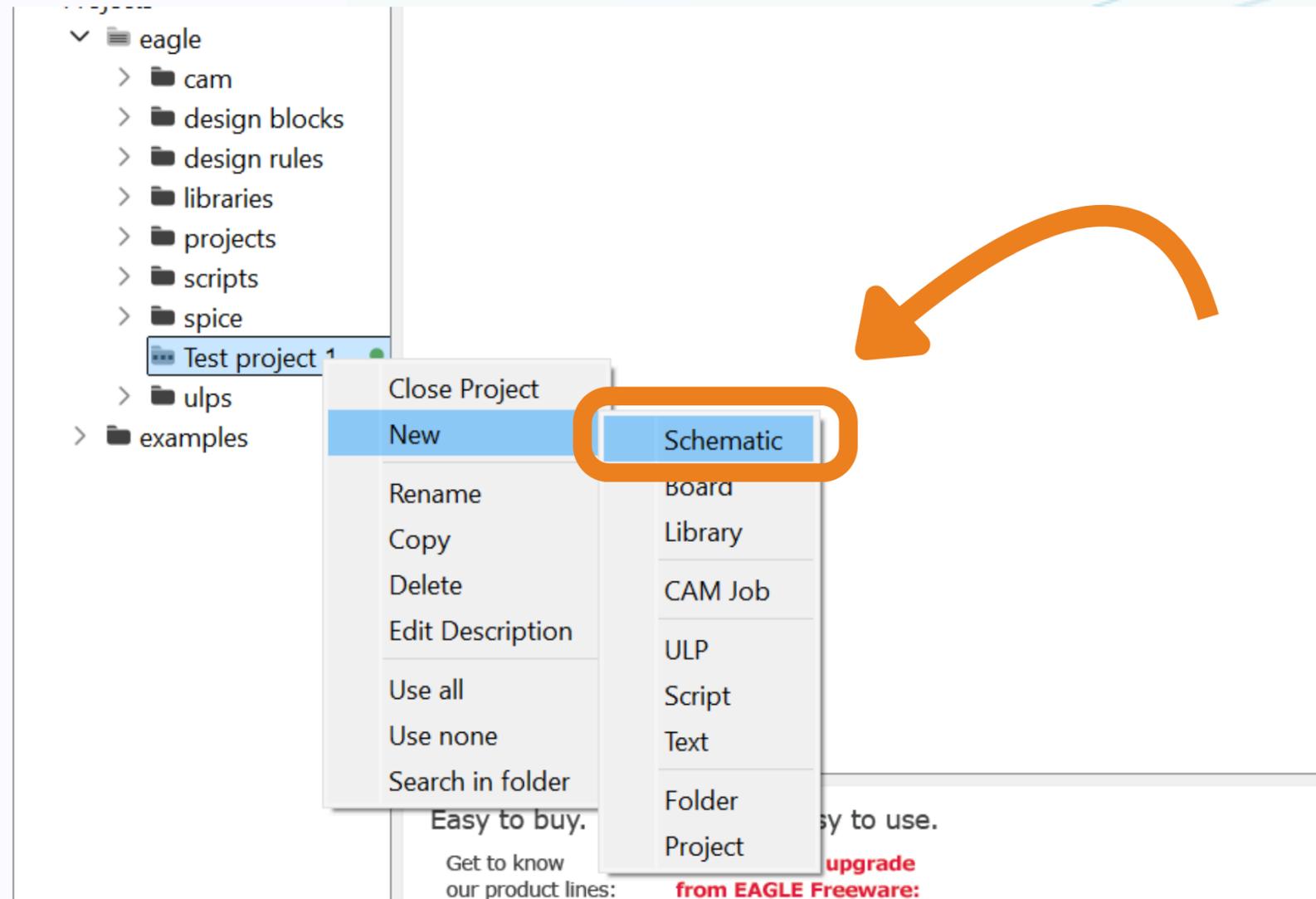
Rename the project file

RENAME THE PROJECT FILE



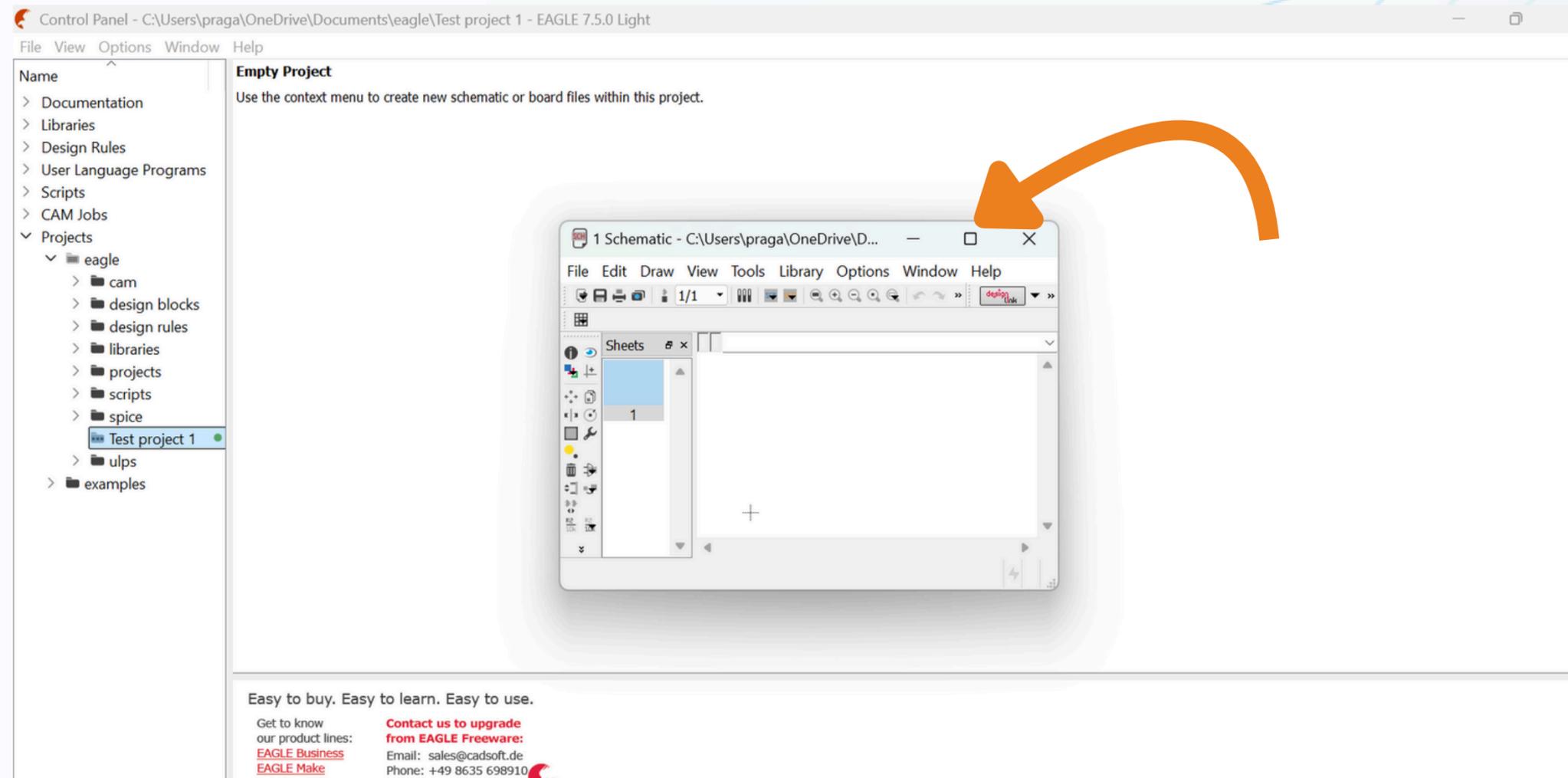
Renamed project file

CREATING SCHEMATIC SHEET



Project > New > Schematic

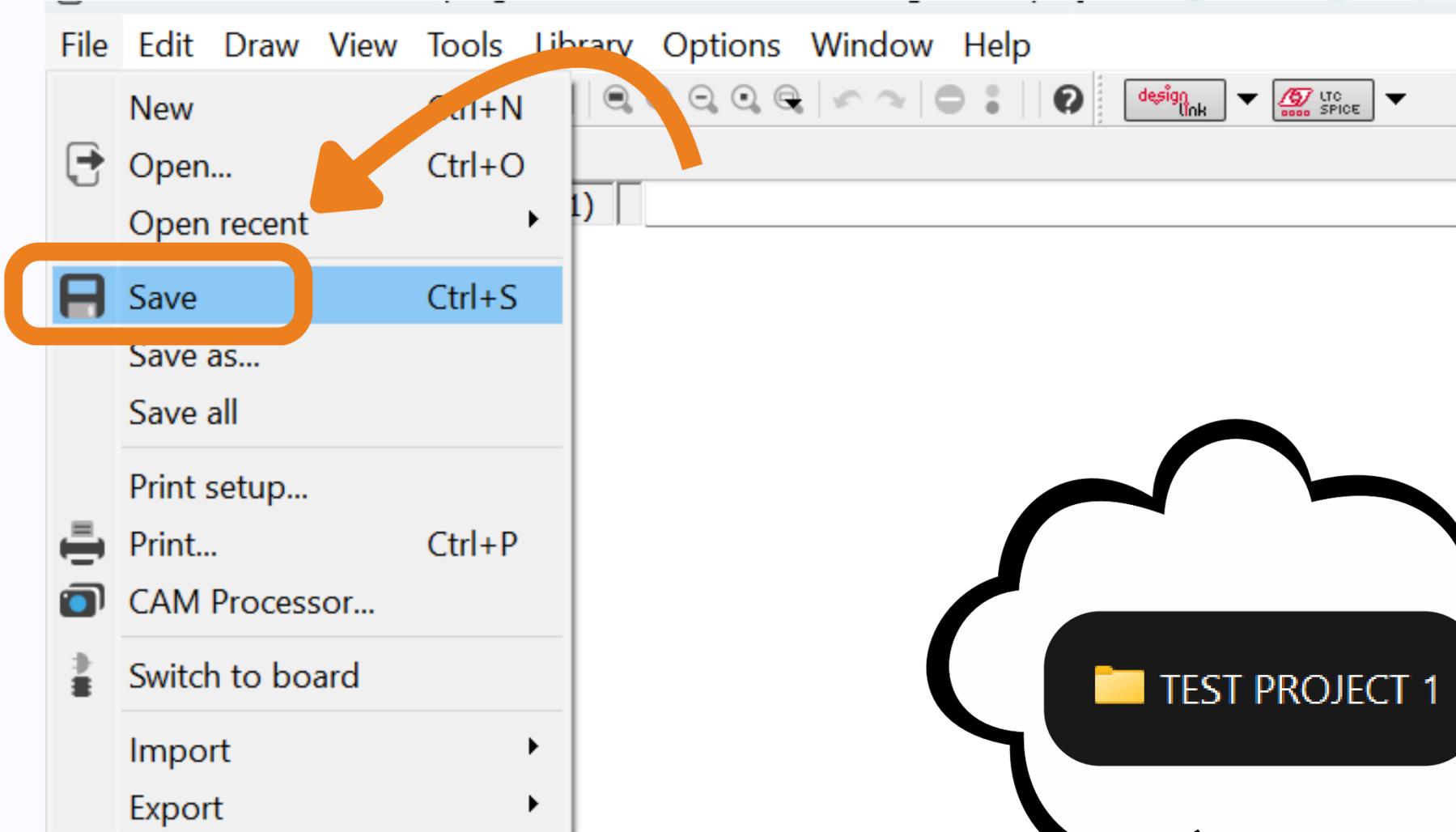
SCHEMATIC SHEET



Open the Schematic sheet

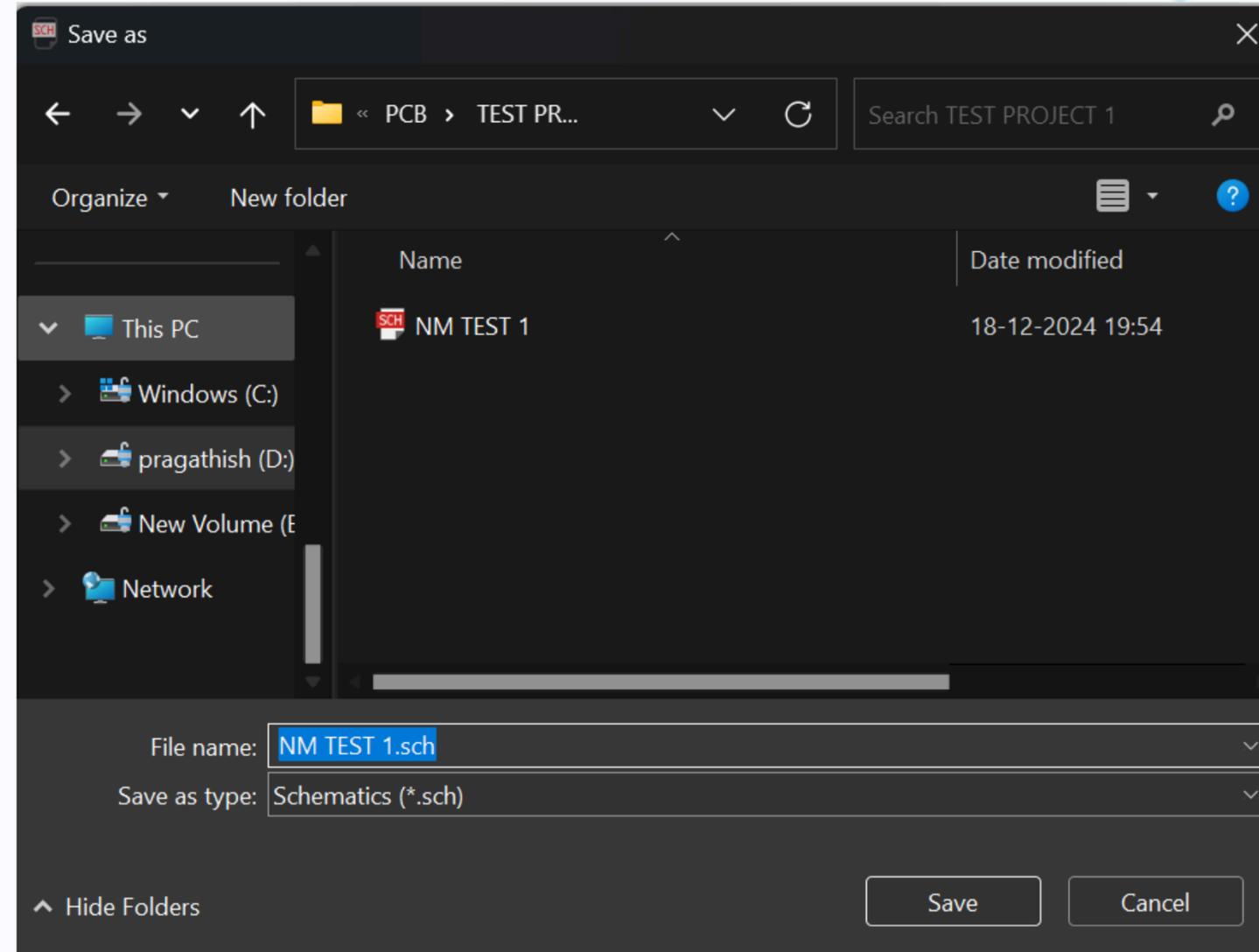
#enthutech®

SAVE THE SCHEMATIC SHEET



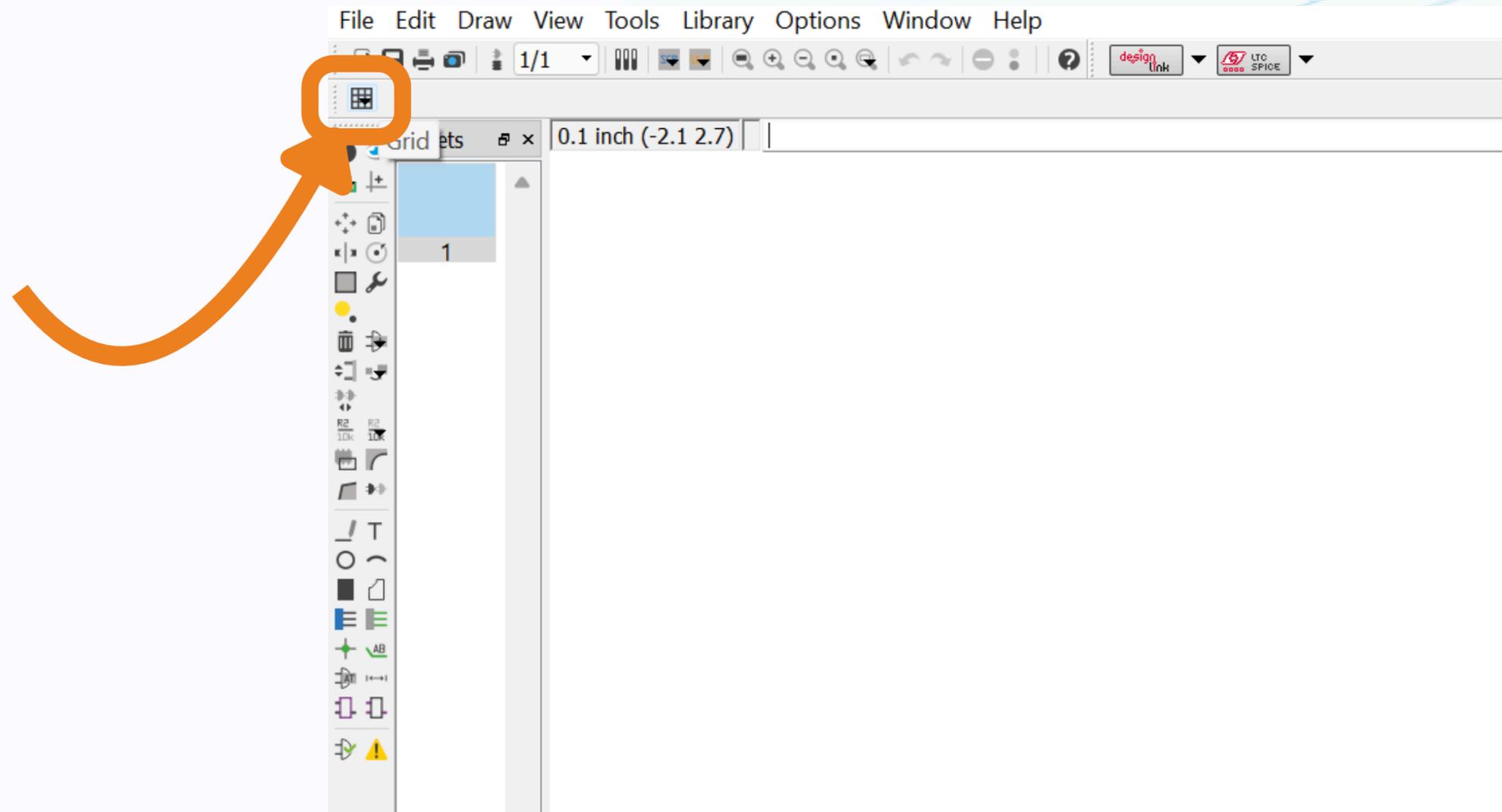
Save the Schematic Sheet in that folder(Test project 1)

SAVE THE SCHEMATIC SHEET



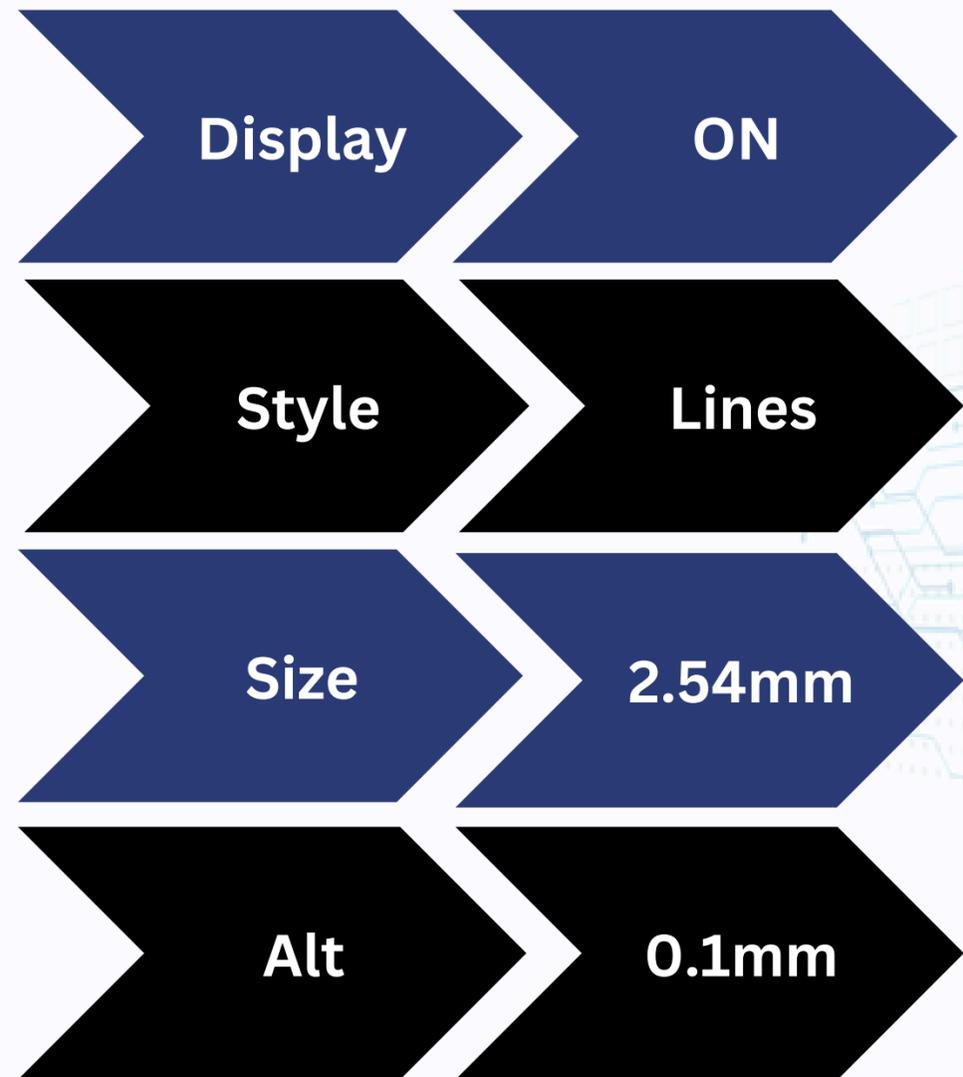
Save in that created file

Grid Configuration



Click the Grid option

Grid Configuration



SCH Grid

Display On Off

Style Dots Lines

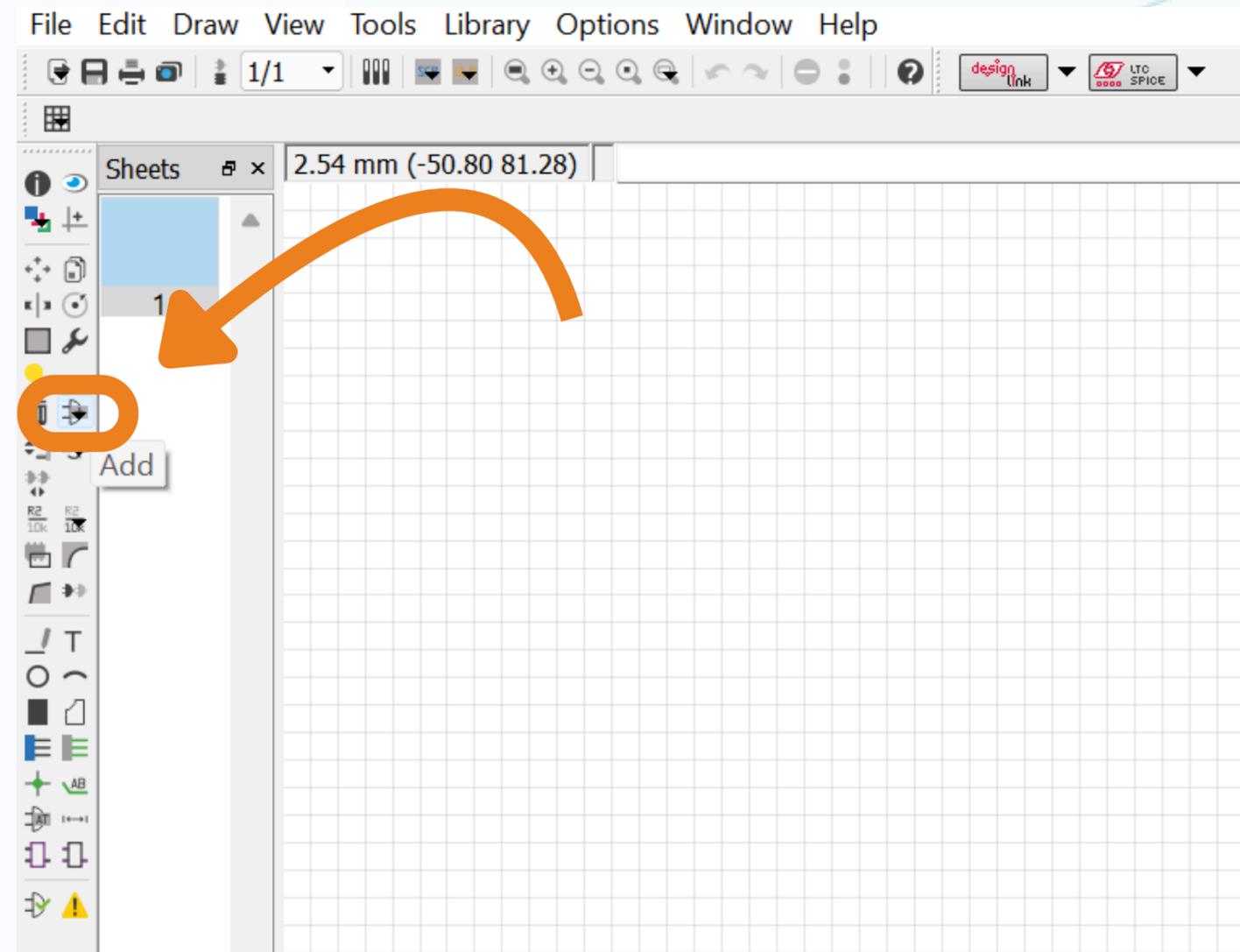
Size: 2.54 mm

Multiple: 1

Hint: It's strongly recommended to use the default grid in schematics.

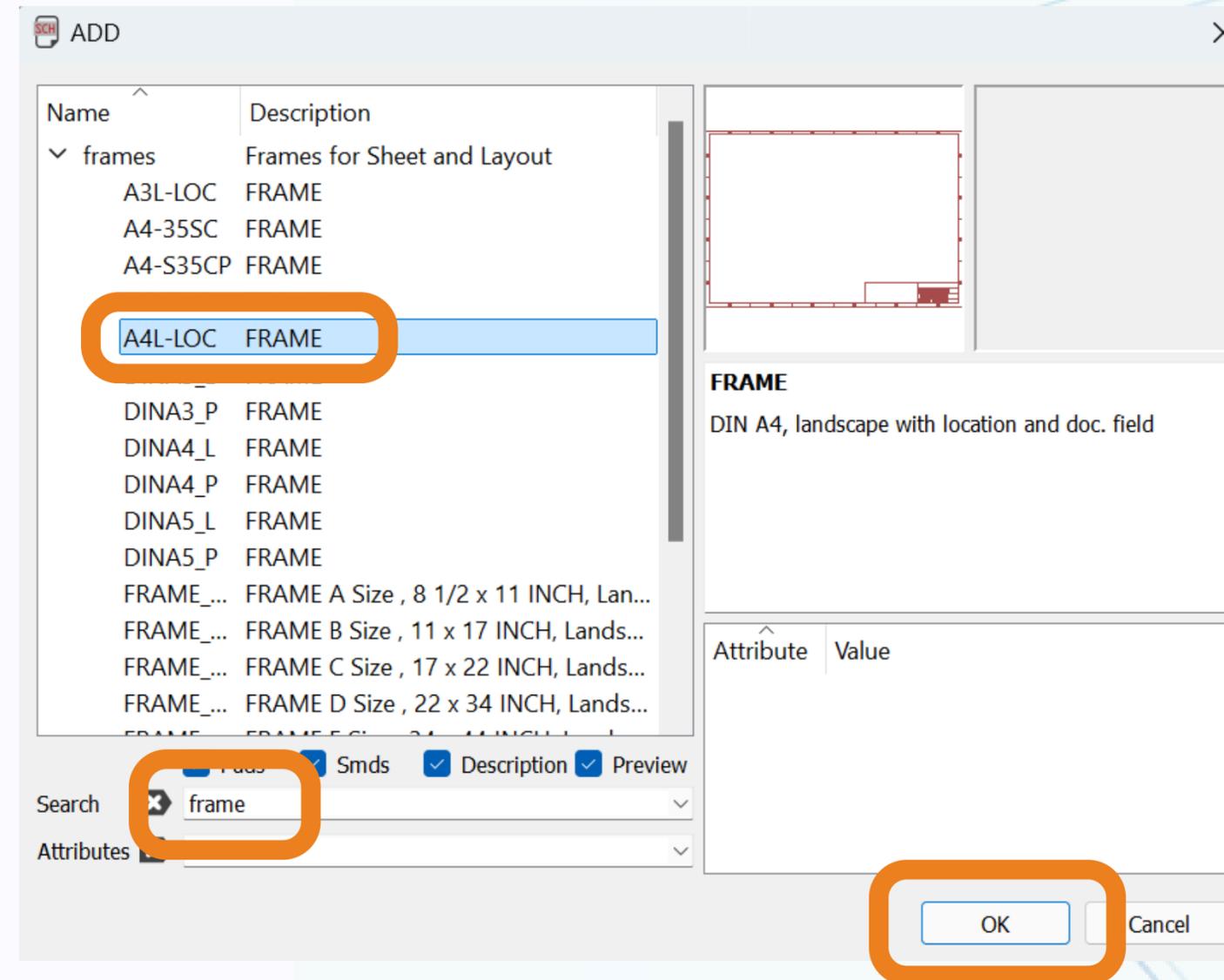
Alt: 0.1 mm

Add FRAME



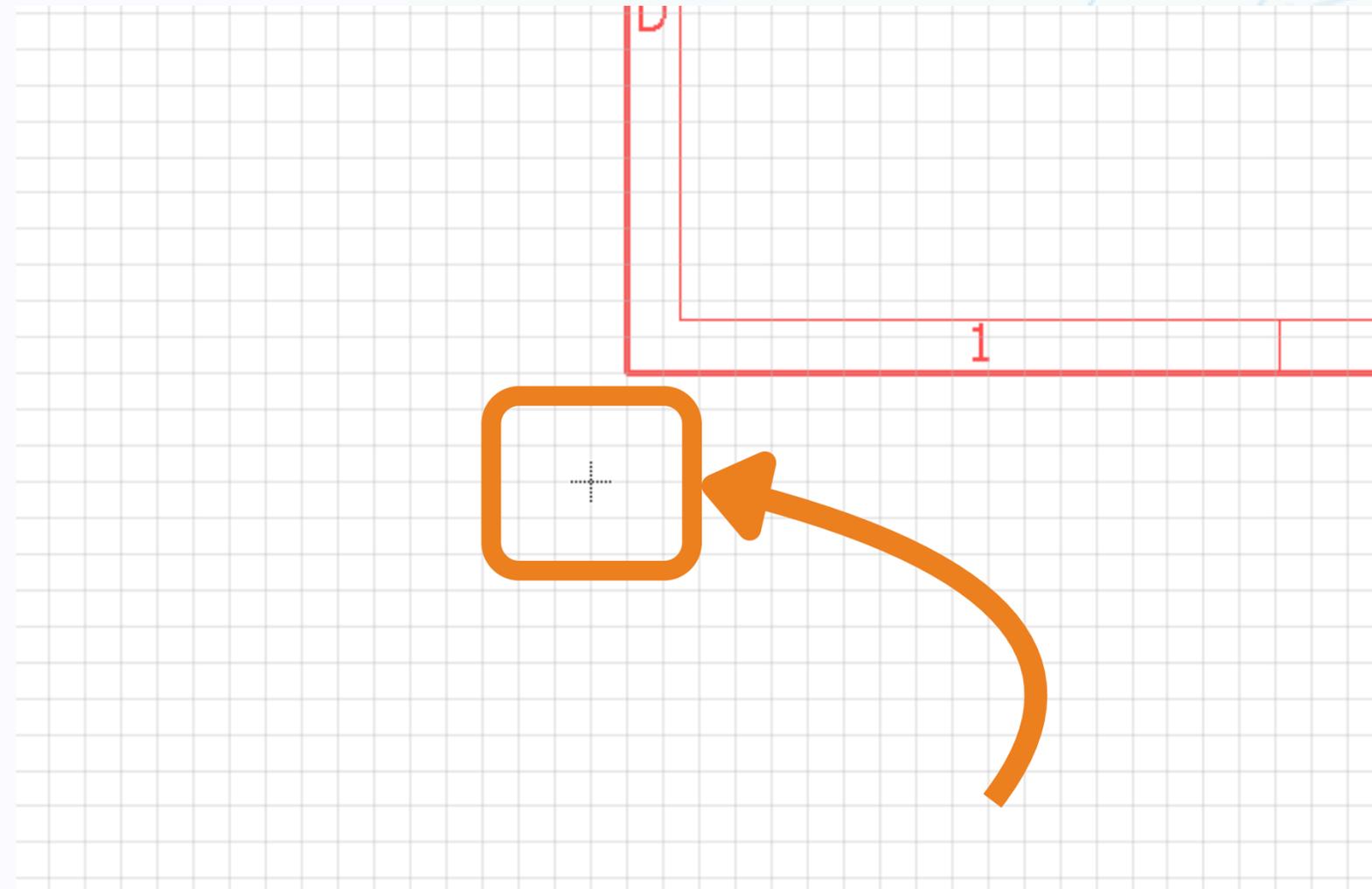
**Click the Add button to
select the components**

Add FRAME



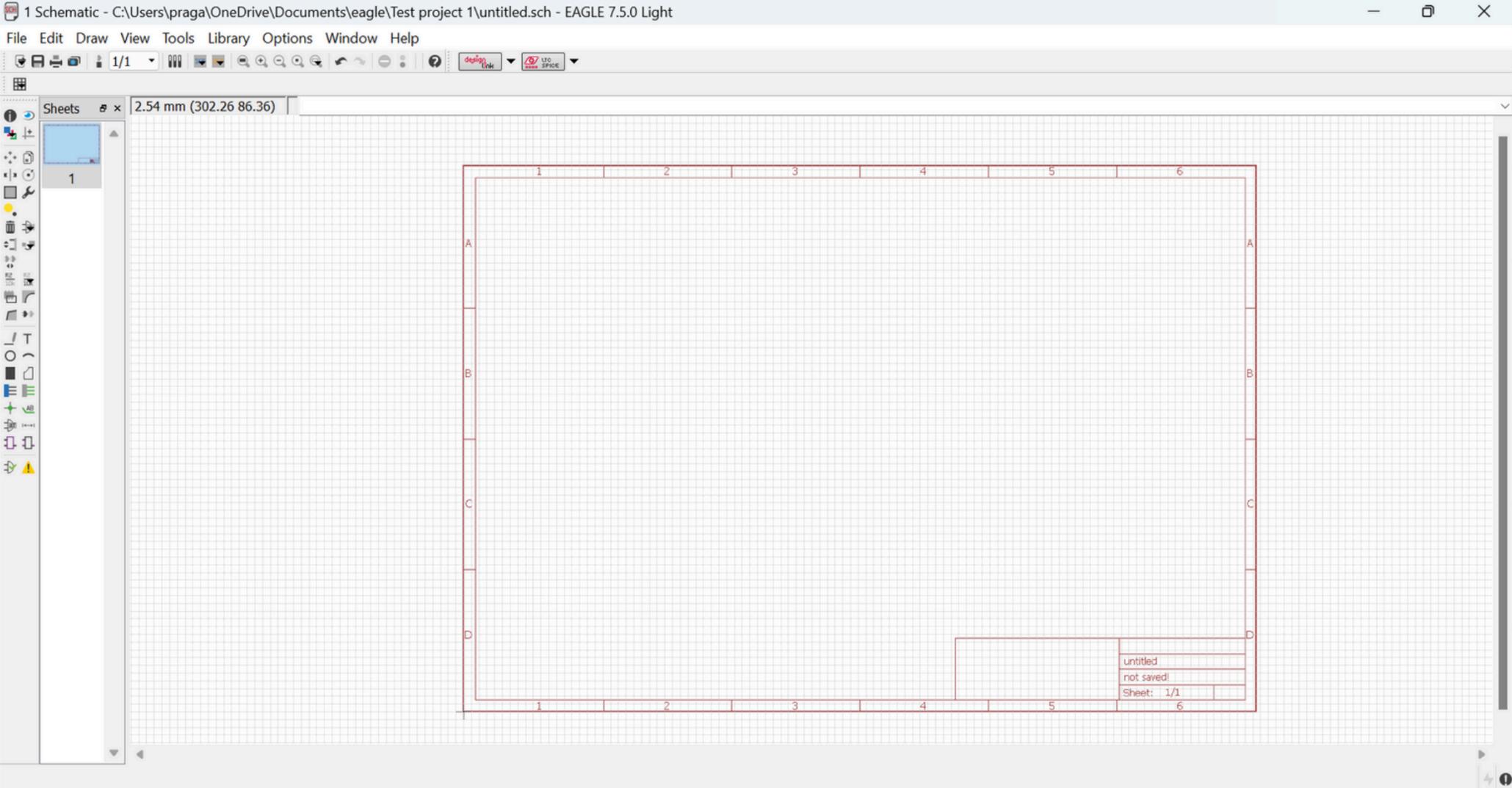
Search > Frame > A4l-loc > OK

Add FRAME



**Place the frame in that
Origin point**

Schematic Sheet



#enthutech®

 Snap EDA
<https://www.snapeda.com>

SnapMagic Search | Free PCB Footprints and Schematic Symbols

Design faster with SnapMagic Search. Download CAD models for millions of electronic components, including schematic symbols, PCB footprints, and 3D models.

Signup to the Snapeda library tool



Work email*

Username*

Company*

Password*

Confirm Password*

I'm not a robot 

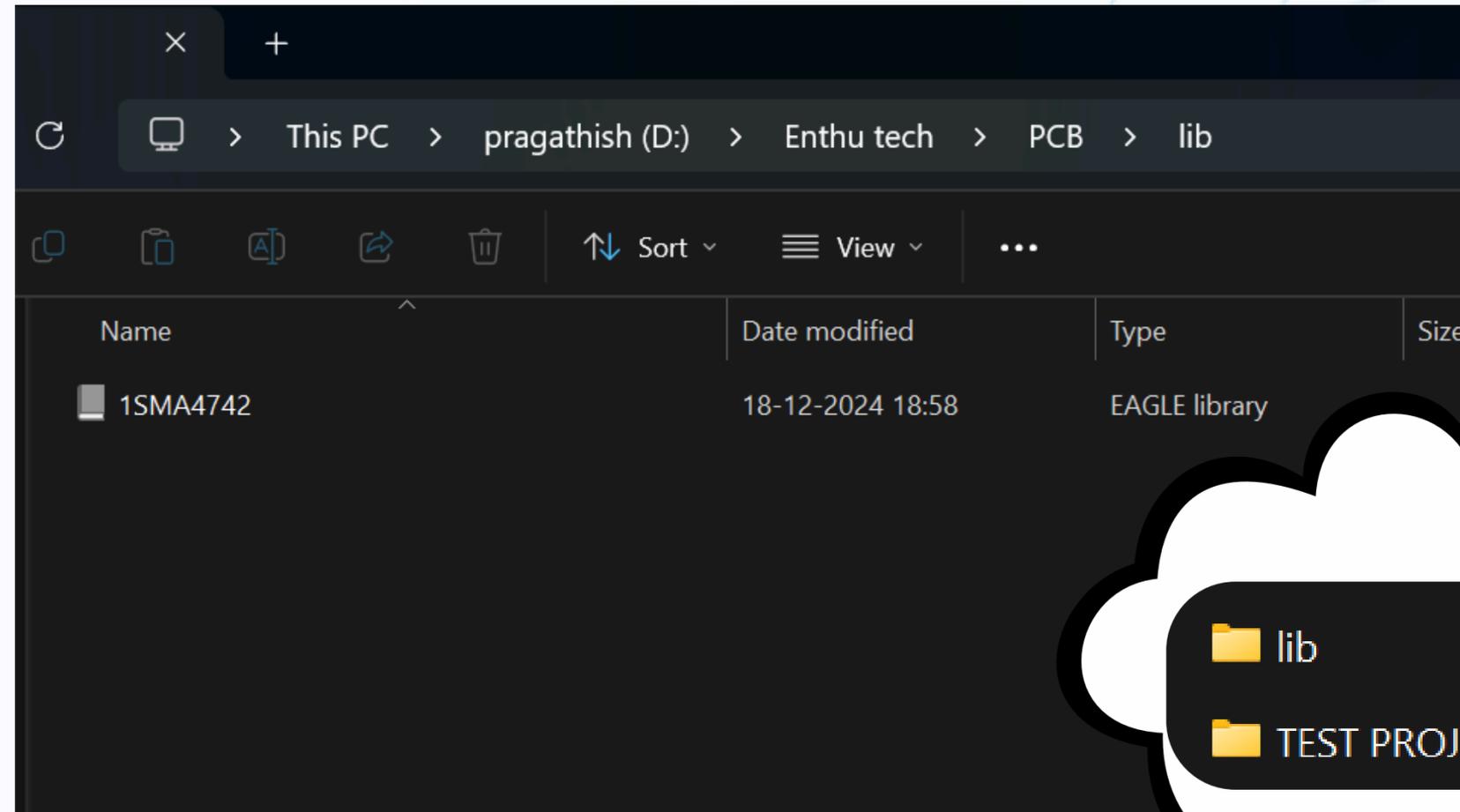
reCAPTCHA
Privacy - Terms

Library downloading

The screenshot displays a web interface for a component library. On the left, a sidebar shows the component name '1SMA4742', its description 'DO-214AC (SMA), 1250mW, 5%, Zener Diode', and availability status 'In Stock'. Below this are buttons for 'Add to Library', 'See Datasheet PDF', 'Pricing', and 'Pinout'. The main content area is divided into two tabs: 'Symbol and Footprint' (selected) and '3D Model'. The 'Symbol and Footprint' tab is further split into 'Symbol' and 'Footprint' sub-sections. The 'Symbol' section shows a schematic symbol for a Zener diode with an orange box around the 'Download Symbol and Footprint' button. The 'Footprint' section shows a physical footprint of the component with an orange box around the 'Download Footprint' button. An orange arrow points from the 'Download Footprint' button to the 'Download Symbol and Footprint' button.

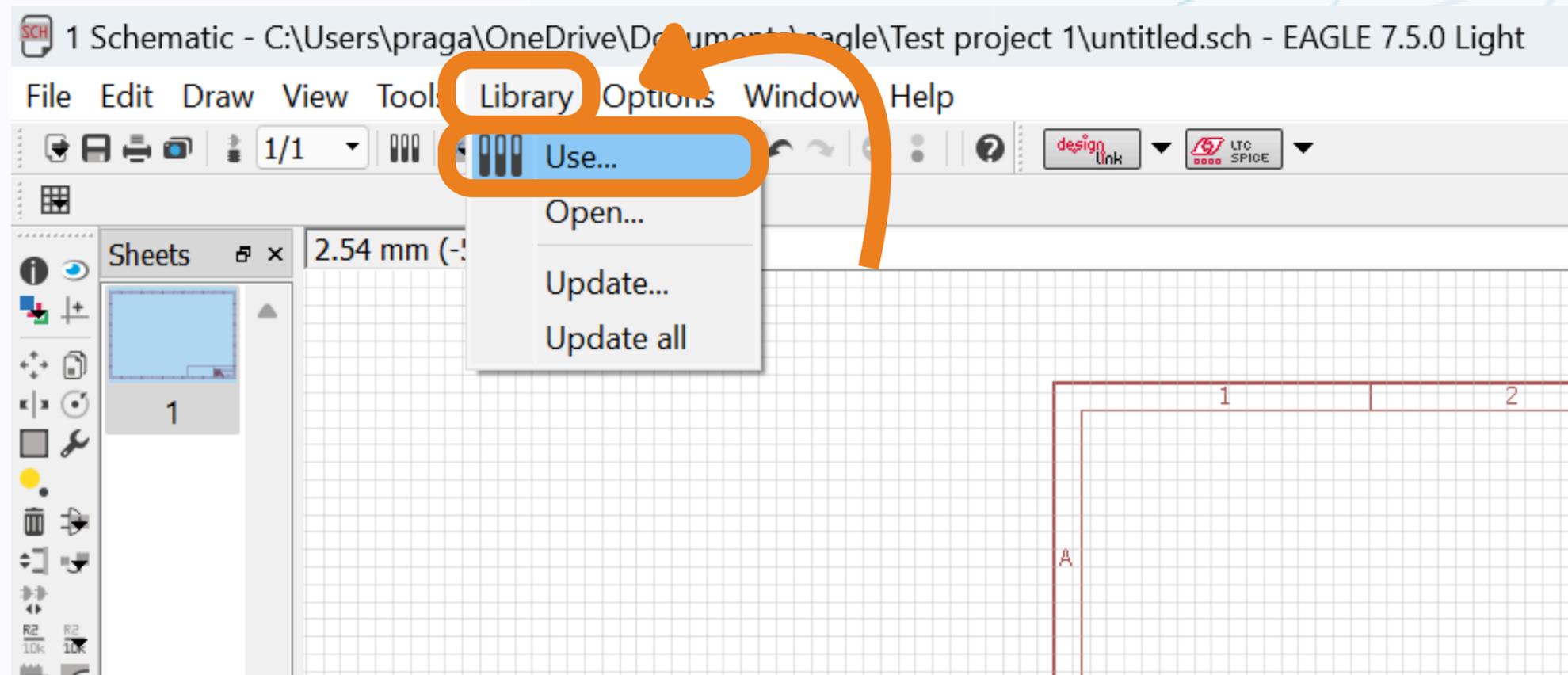
Download symbol and footprint

Library downloading



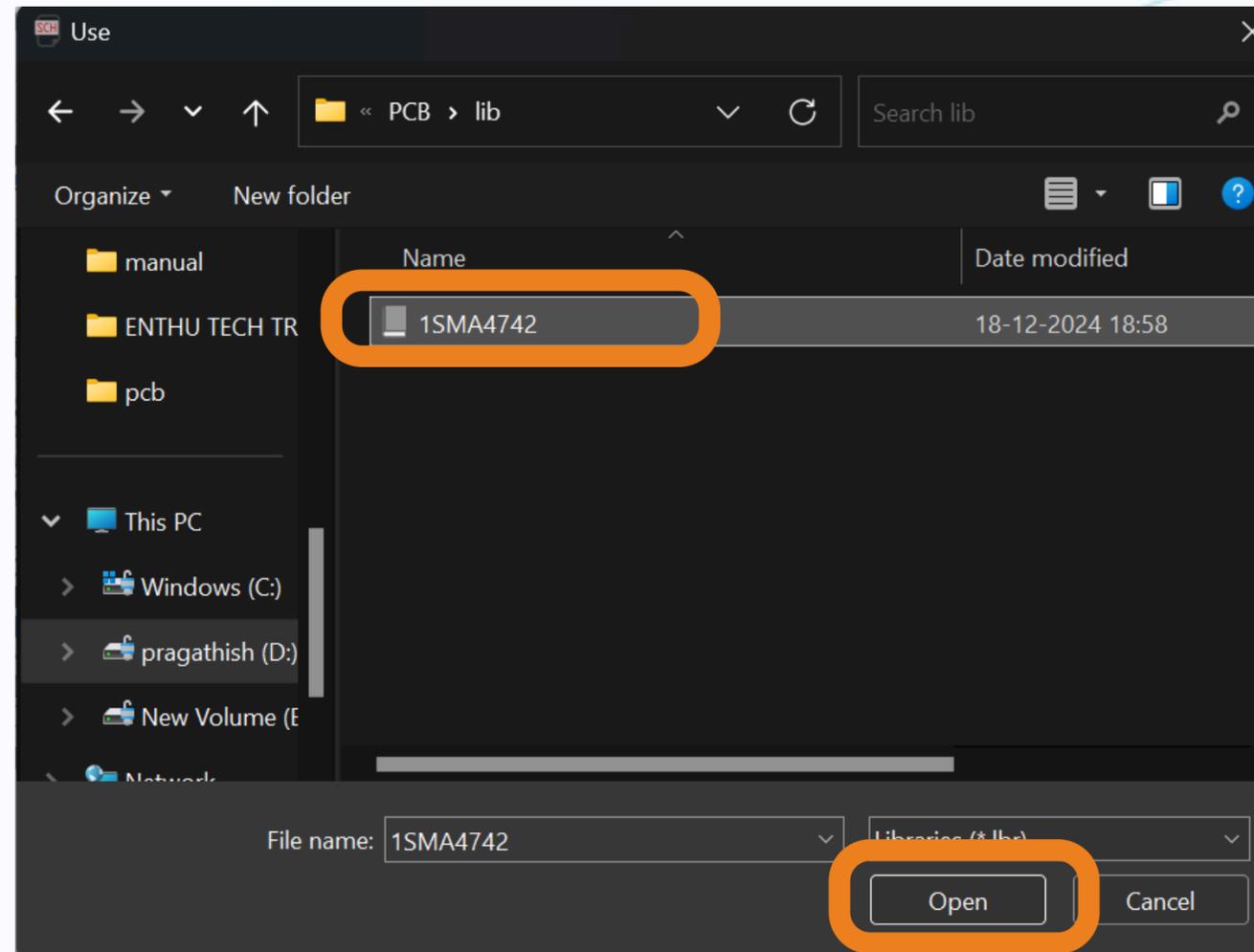
Save the library file in a separate folder

IMPORTING LIBRARY FILE



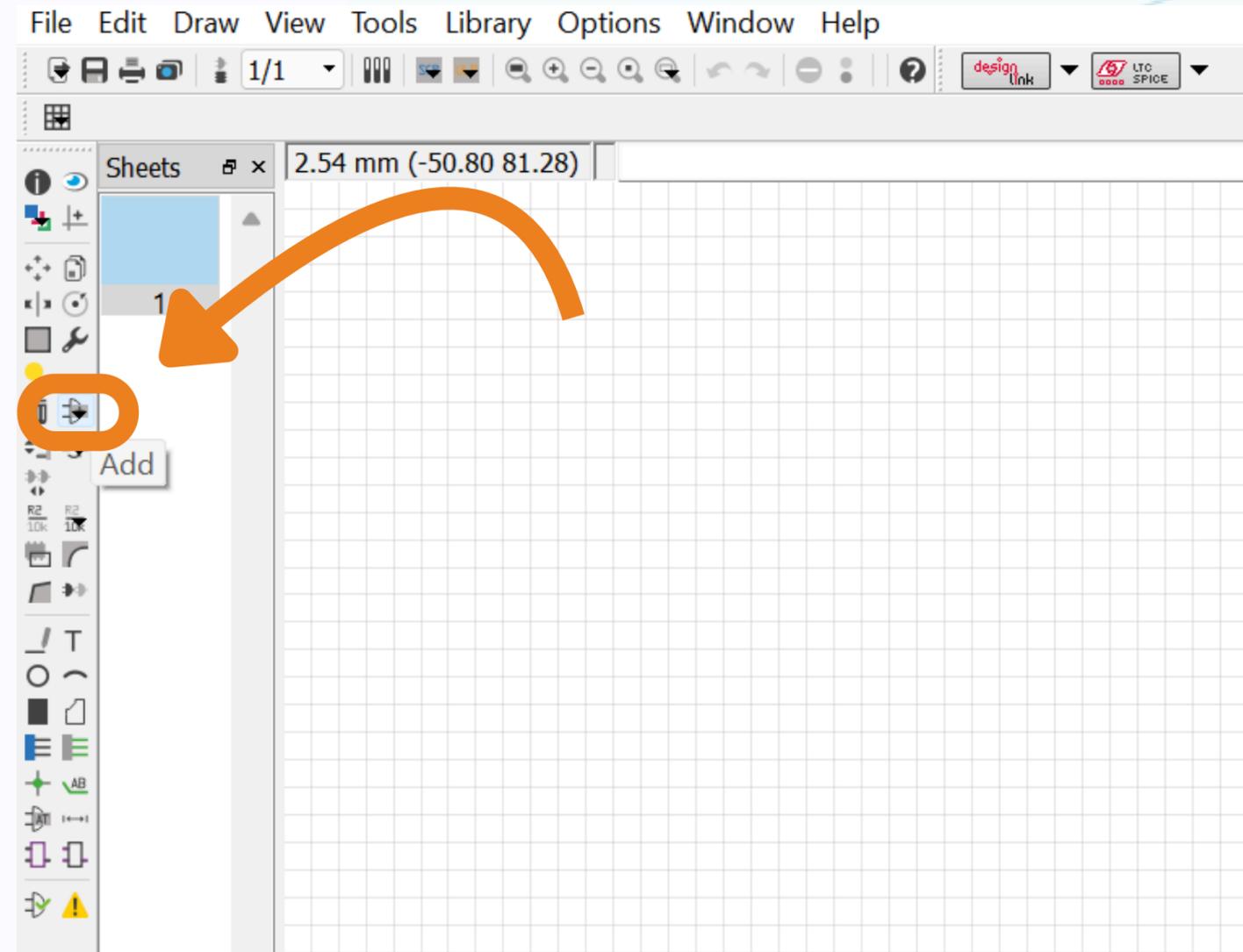
Click > Library > Use

IMPORTING LIBRARY FILE



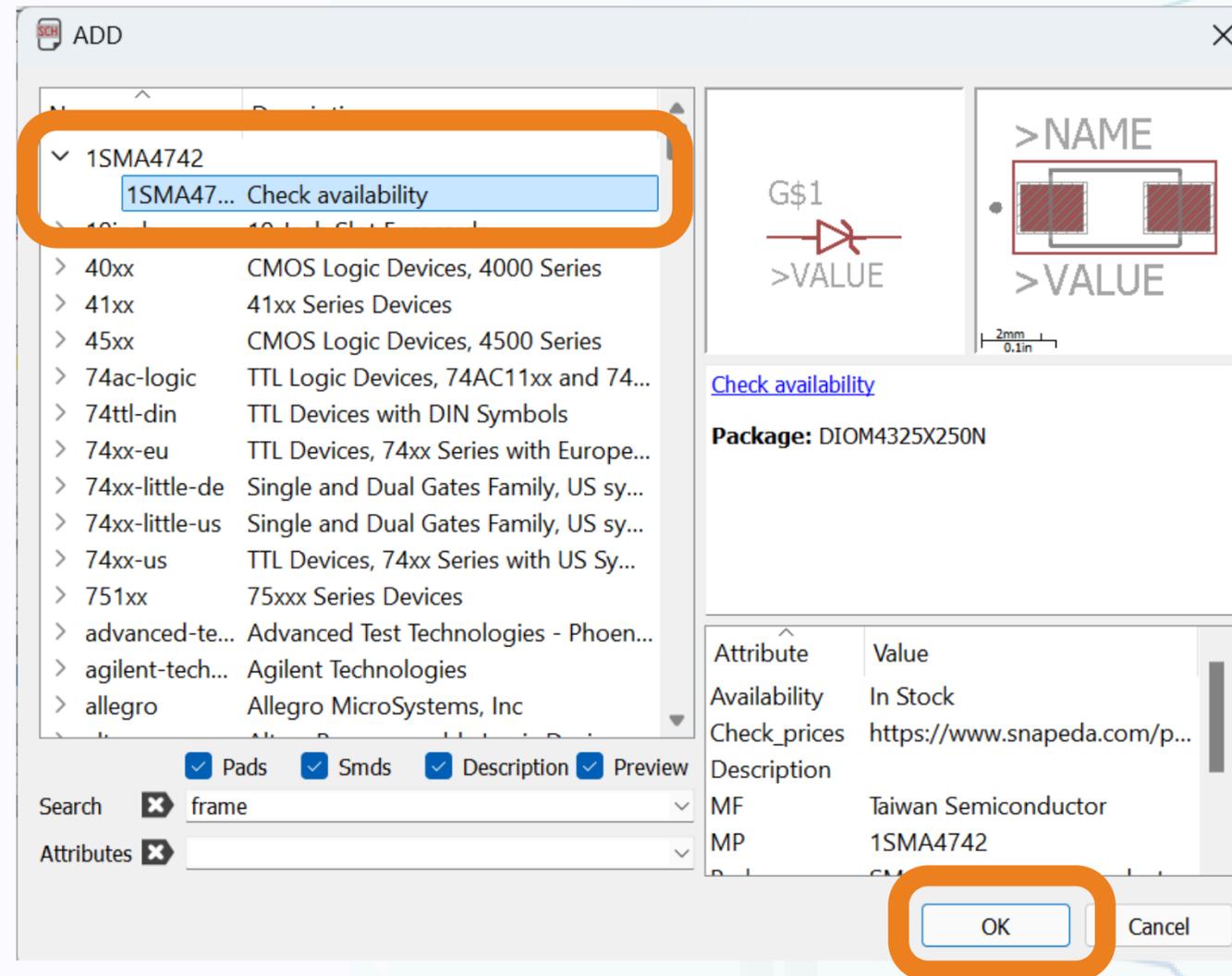
Select the Component > Open

Add COMPONENTS



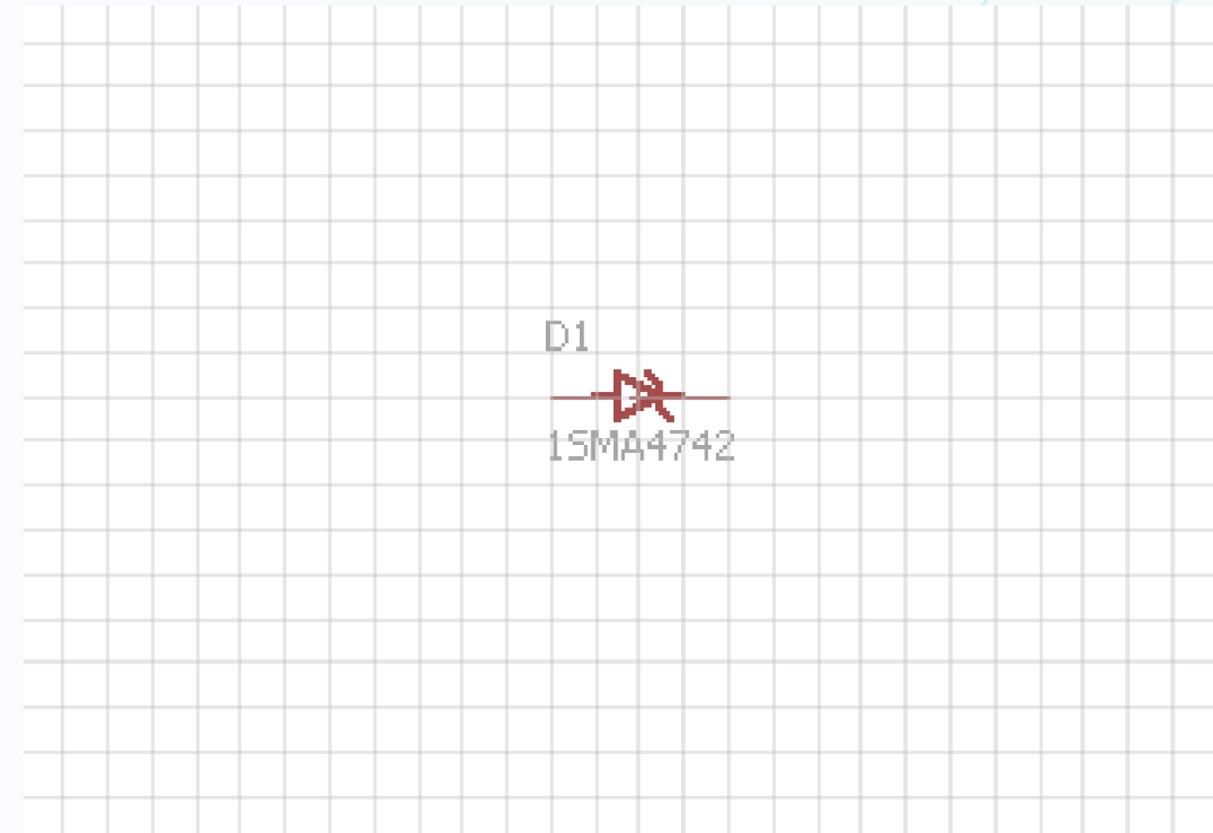
**Click the Add button to
select the components**

Add COMPONENTS



Select the component > Click OK

Component Import



**By this method you can import
multiple Components**

#enthutech®

Thank You

#enthutech®