

EcuTester Software Manual

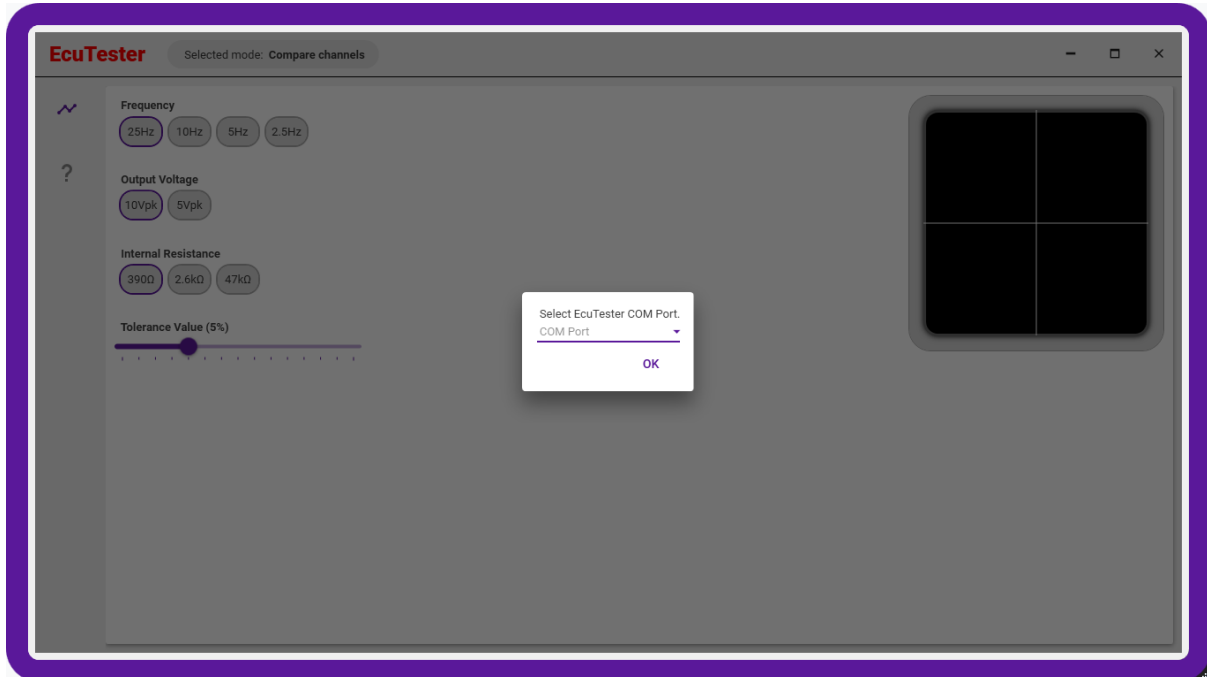
Revision 1.2.4

Table of contents

Table of contents	1
Software interface	2
Creating reference file	5
Testing PCBs	5

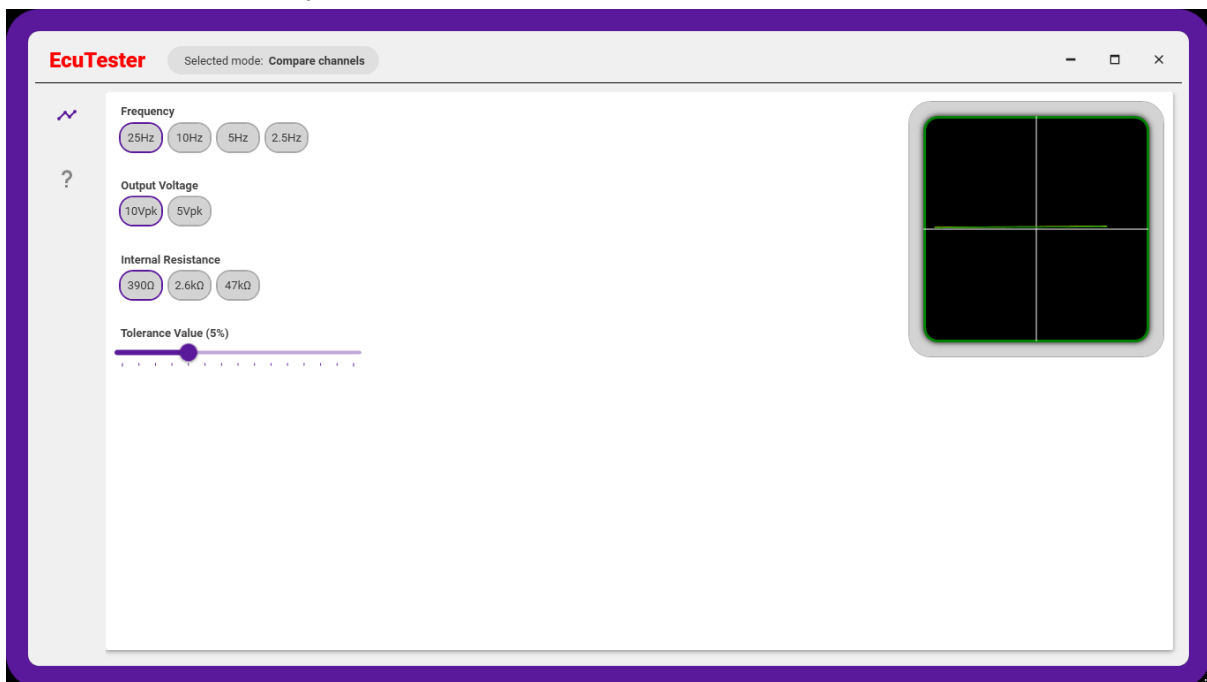
Software interface

After you open the software this interface will pop-up.



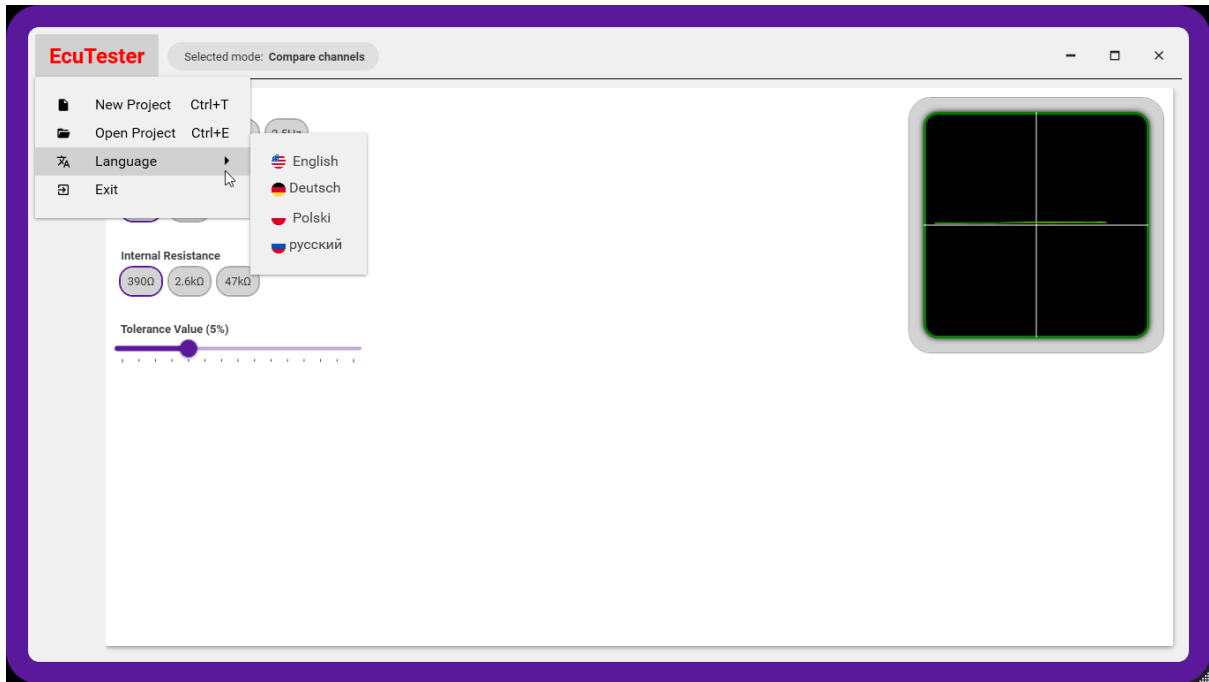
This means that you need to choose COM Port which is connected to the **EcuTester**. In most cases it will be only one port, but if you've more than one and you choose wrong it should inform you that you've chosen the wrong device.

After you select the **EcuTester** it should disable the blackout and show the curve on the right chart. If it does not show a curve it's probable that your **EcuTester** is not connected to power. Please connect your power adapter.



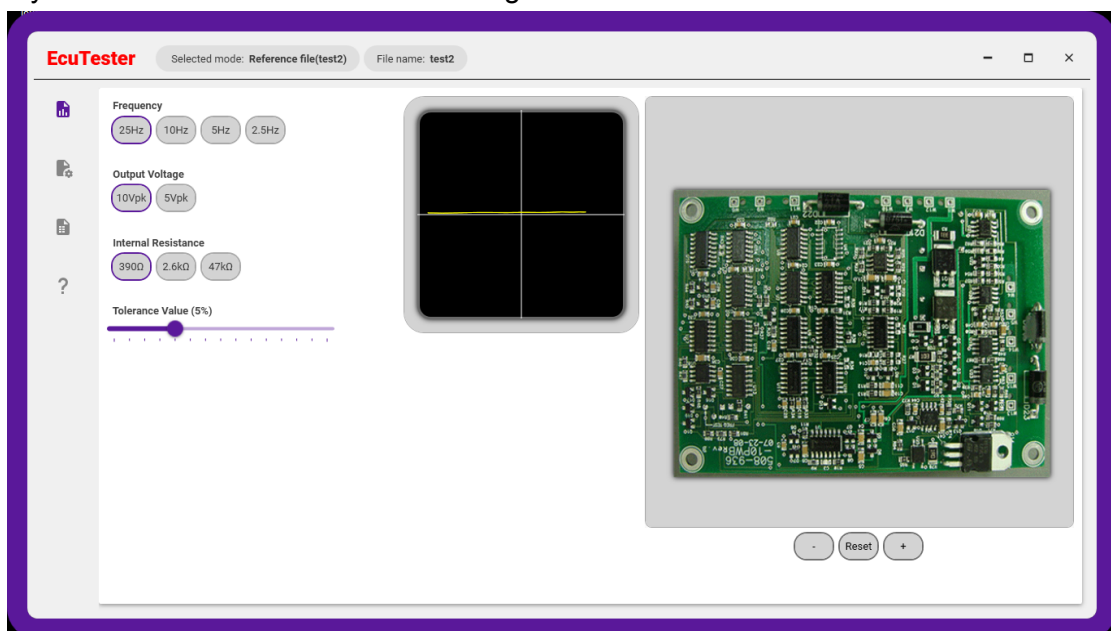
On the left you've icon panel to choose the current editor. By default the device is in "comparison mode" which allows you to compare channel A to channel B. Connection of channels is described in the help section indicated by ?.

To change language you can choose the red "EcuTester" logo, which is also used for project creation. Such a menu should pop up.



To create a new project simply click "New Project". Pop-up with some data will appear and you will be asked to select save destination and PCB image. You can change the PCB image later.

After you create a file the visuals will change into "file mode".

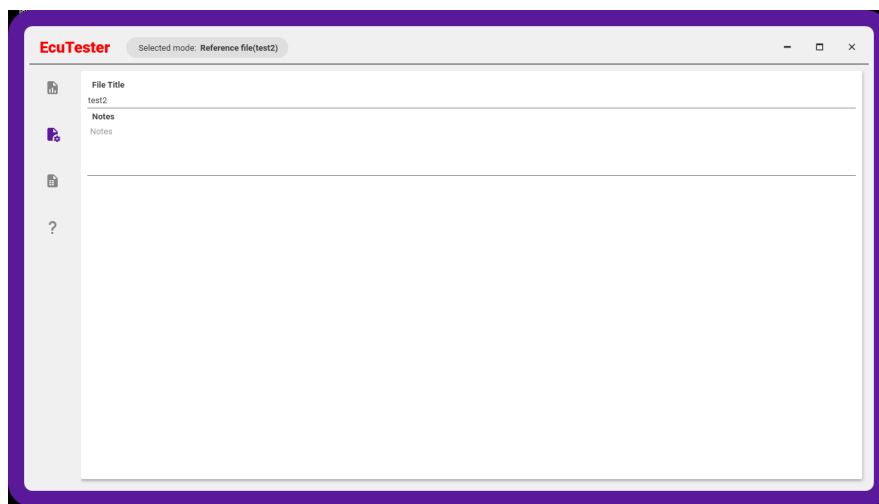


In file mode you can add test points to the PCB and measure them to save a “reference file” for future usage.

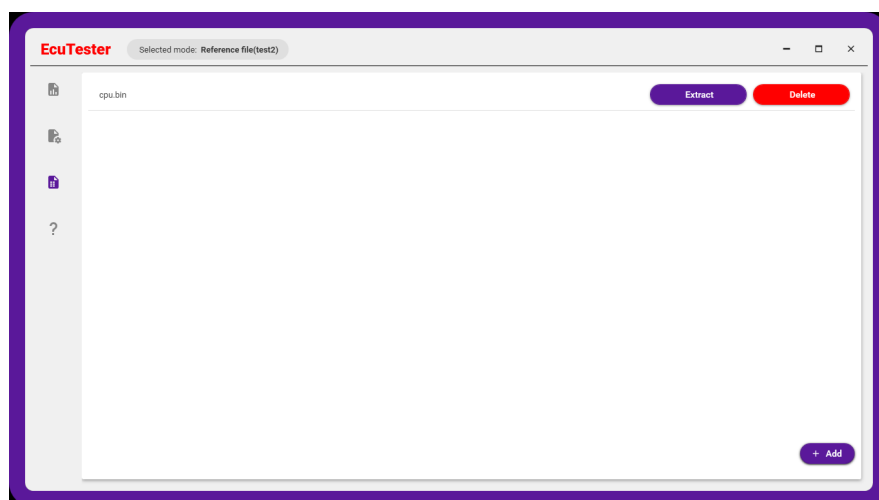
1. Adding a point: Hold SHIFT and click on the PCB image in the place you wish to add the point.
2. Removing a point: Hold DELETE and click on point you wish to delete
3. Zooming in/out: Use mouse wheel
4. Rotating PCB image: Hold SHIFT and use mouse wheel

For more PCB options you can also right-click on the PCB image. There will be more useful things like reset position, change image etc.

You can also move between points using arrows, but it's quite non-intuitive.



The next menu on the left bar allows you to add some notes and title to the file. By default title is the same as file name.



Next menu is reserved for adding binary files. This allows you to store several .bin, .hex etc. files for microcontrollers (but it's not limited by format, so you can also add data sheets etc.)

Creating reference file

To test a PCB, simply connect **EcuTester** to power and enter file mode (create a project with a PCB image).

Now you need to add points (by holding shift and clicking on the PCB image).

Now simply touch the pad where the point is located. The curve (yellow line) in the chart should change. While the probe is touching the pad and the curve is drawn, click SPACE to save point data. If “automatic” mode is enabled then software should automatically choose the next point.

If the chart is barely visible (flat line / vertical line) you can try to change settings like frequency, V_{pp} or internal resistance. It often helps to make the chart more round, thus decreasing the chance of measurement error.

Testing PCBs

You do this nearly the same as creating a file, however you don't click space. Device will automatically check if values are matching the provided tolerance. If the point is OK then it will turn green. Otherwise it will turn red.