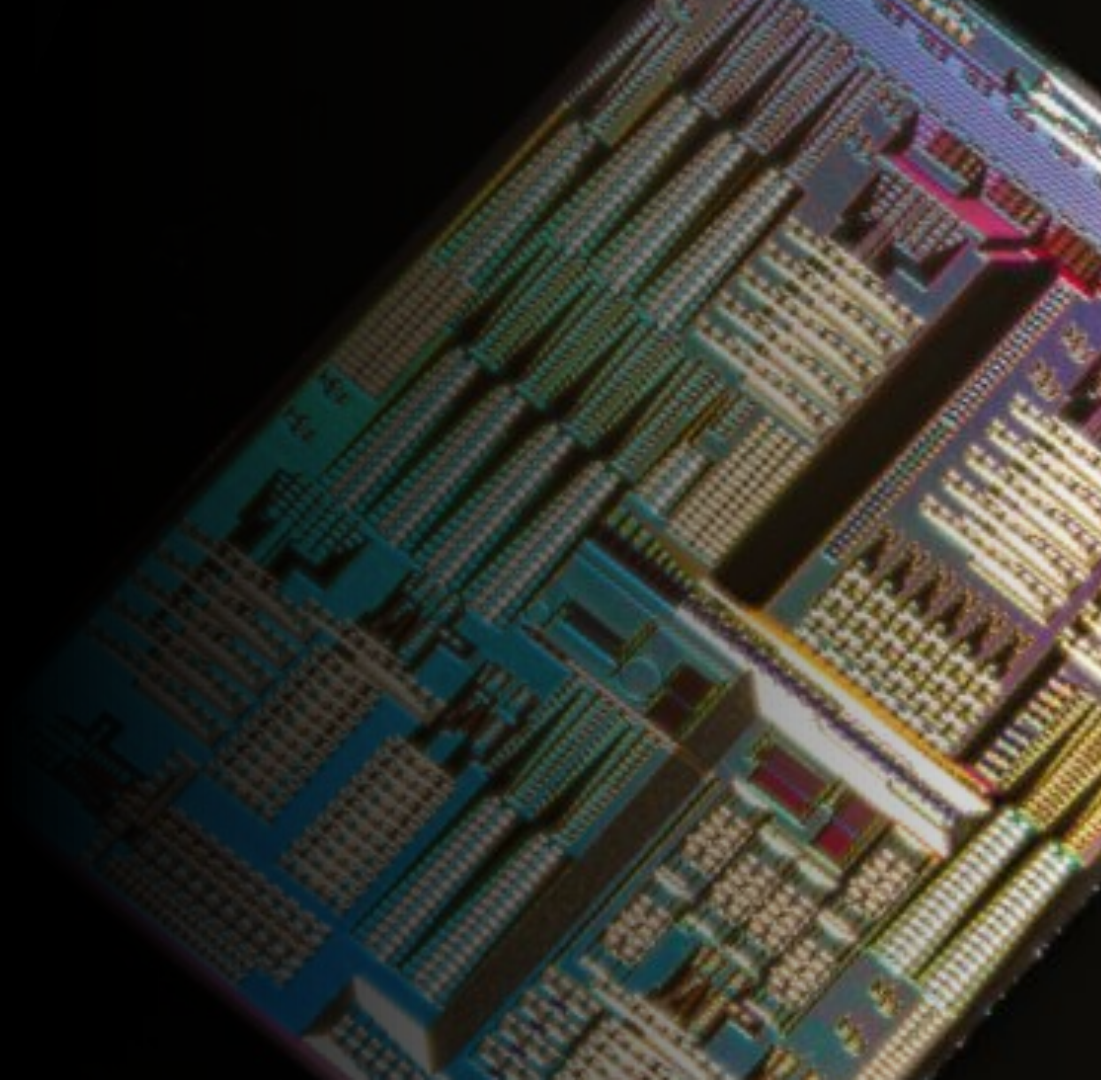
PyOptomip User's Guide

11/9/2022



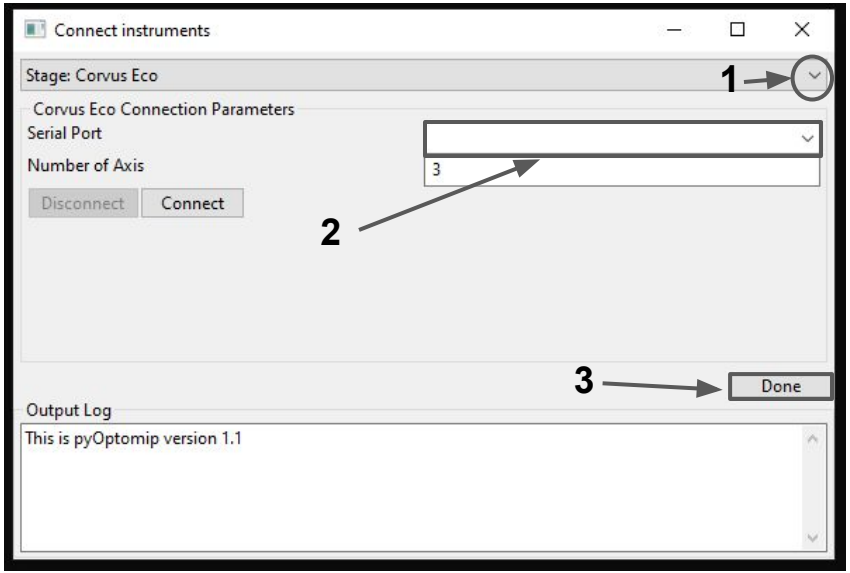
Purpose

PyOptomip is a python application which aims to facilitate testing of silicon electronic photonic integrated circuits by controlling testing equipment though a user interface.

Table of Contents

Getting Started

To begin using PyOptomip to perform measurements you'll first need to connect any instruments you will be using.



1. Select the device you wish to connect to from the drop-down menu on the connect instrument page.

2. Once the desired device is selected, enter its GPIB address in the designated space.

3. After all necessary instruments are connected, press done.

IDA Stage Instrument Connections

The GPIB addresses for the instruments at the IDA stage in KAISER 4060 are as follows:

Stage: Corvus Eco - ASRL7::INSTR

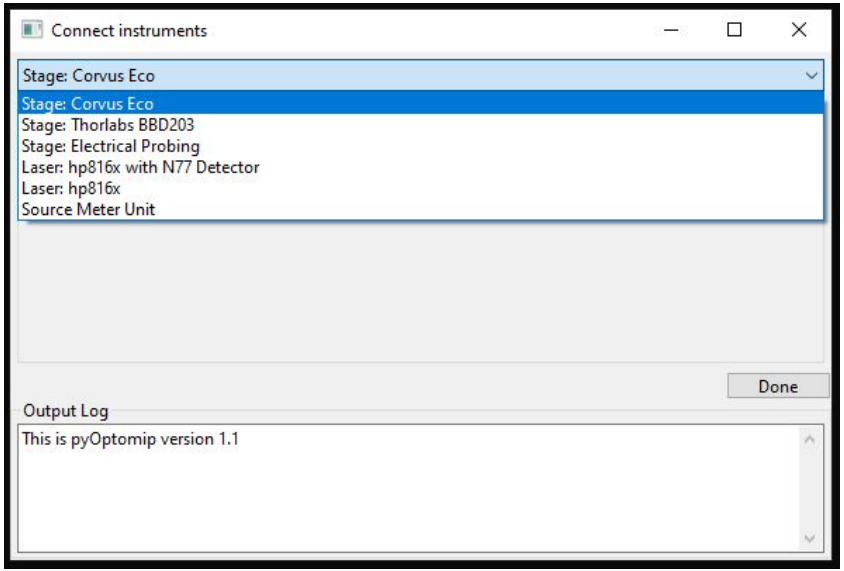
Stage: Electrical Probing - ASRL9::INSTR

Laser: hp816x with N77 Detector -

GPIB0::20::INSTR

USB0::0x0957::0x3718::MY48102149::INSTR

Source Meter Unit - GPIB0::26::INSTR



Instrument Control

PyOptomip has five tabs which are used to control instruments and automate measurements.

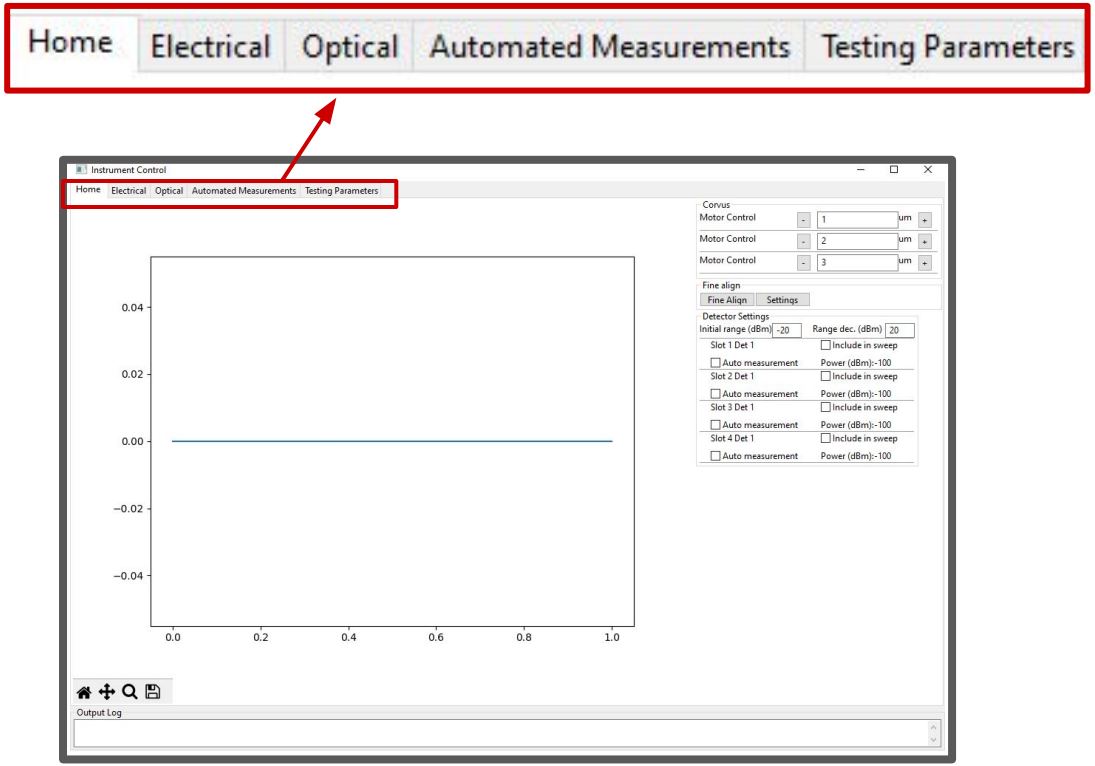
Home tab:
Stage control, detector measurements and settings

Electrical tab:
SMU control, IV sweeps

Optical tab:
Laser control, wavelength sweeps

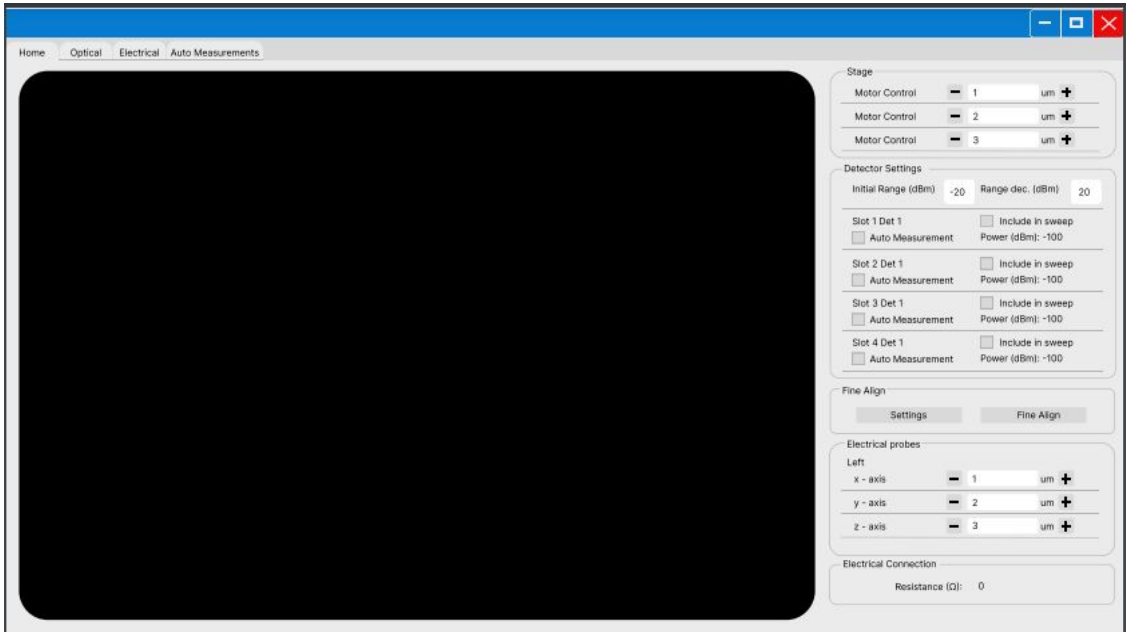
Automated Measurements:
Automated measurement settings

Testing Parameters:
Testing routine creation



Home Tab

The home tab can be used to move the stages on which the chip and wedge probe rest, respectively. It can also be used to gauge alignment with a device using the detector.

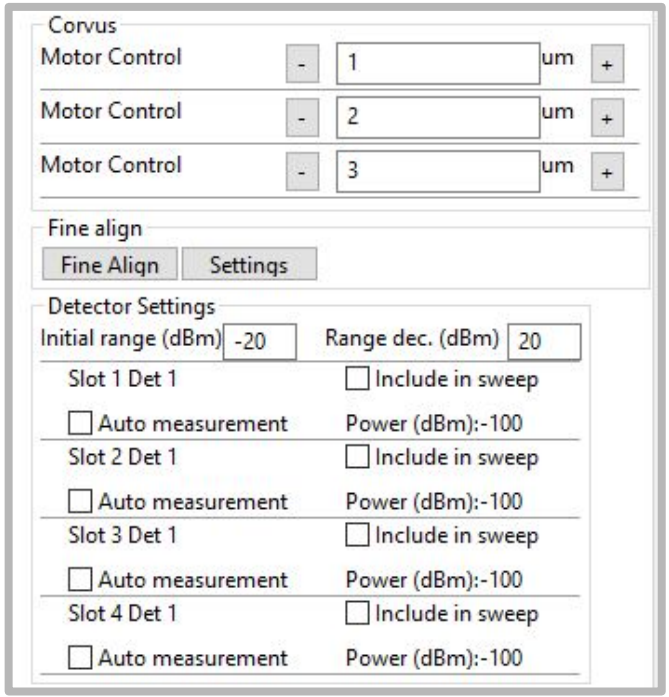


Corvus Eco

The three axes of the stage on which the chip rests can be controlled by entering a number of micrometres and pressing either the **+** or **-** button for that axis.

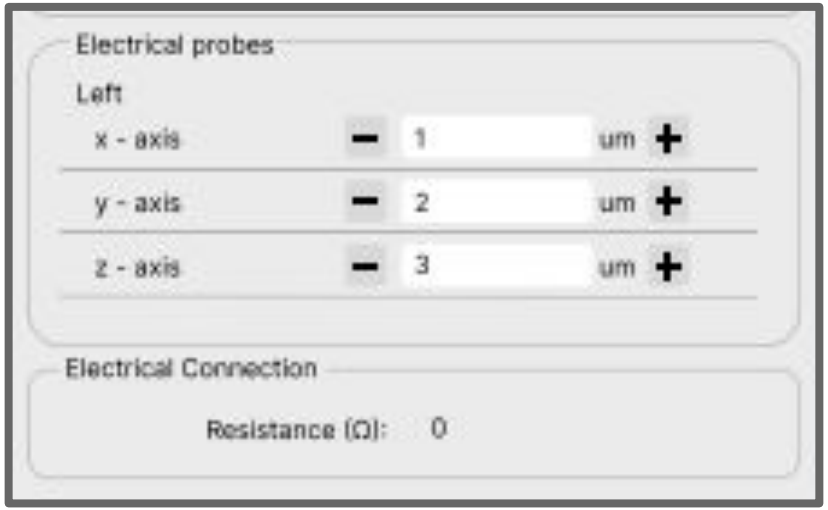
Select **Auto measurement** for the detector channels whose readings you want to see. You can use these readings to determine when the laser is more or less aligned.

At this point, **Fine Align** can be used to optimize the light coupled to the detector.



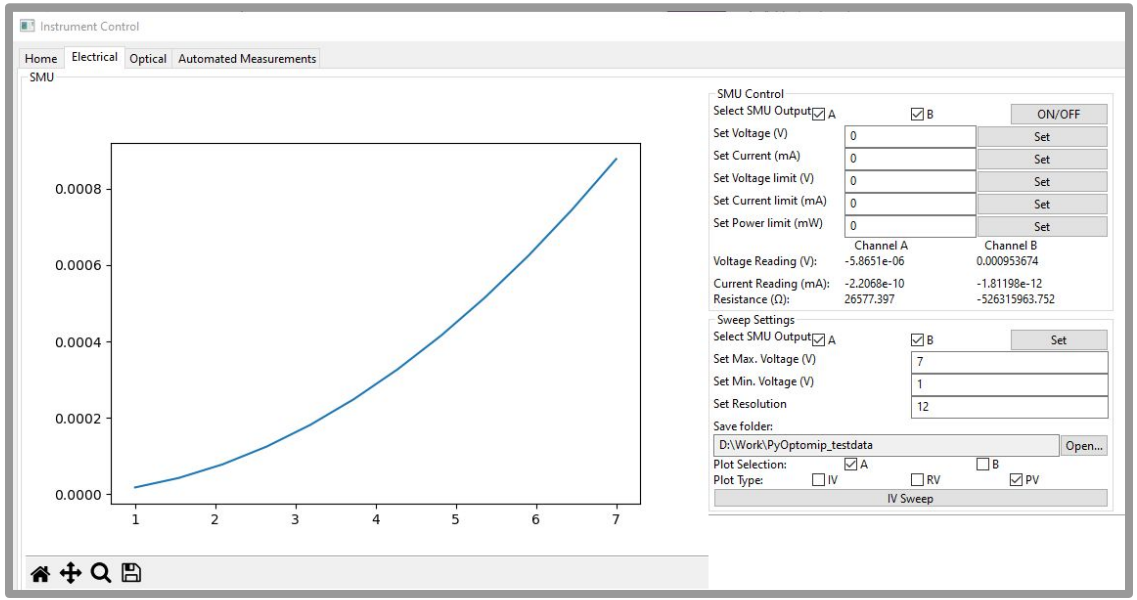
Electrical Stage

The stage on which the wedge probe sits can be controlled by entering a number of micrometres and pressing either the **+** or **-** button for that axis.



Electrical Tab

The electrical tab can be used to control an SMU to set the voltage and current on its channels. It can also be used to perform IV sweeps and plot the results.

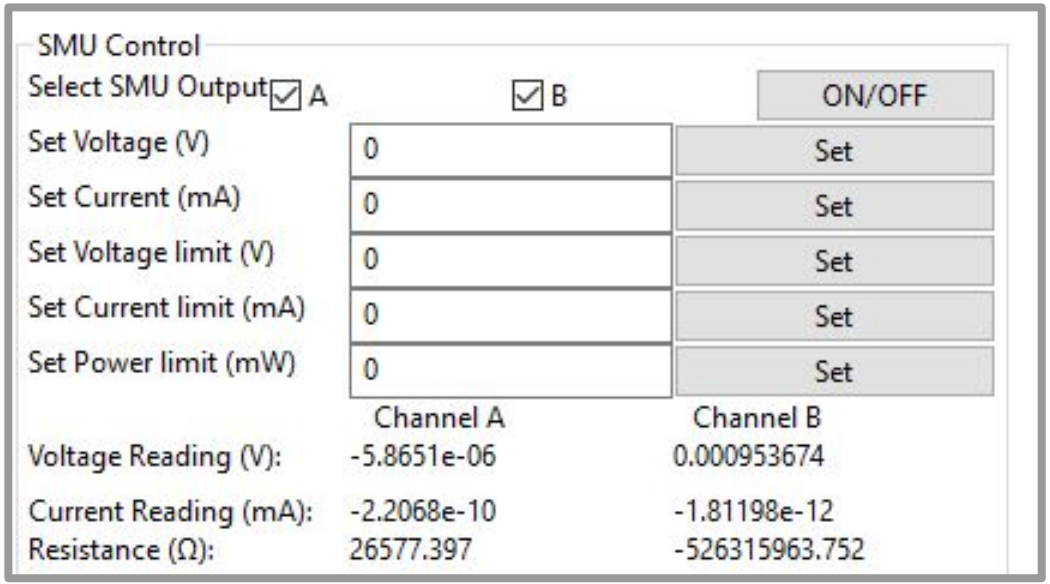


SMU Control

Channel A, or B, or both, of the SMU can be set by checking off the desired channel, inputting parameters and pressing **Set**.

Once parameters are set, the selected channels can be turned on.

Readings from each channel are displayed at the bottom of the panel.



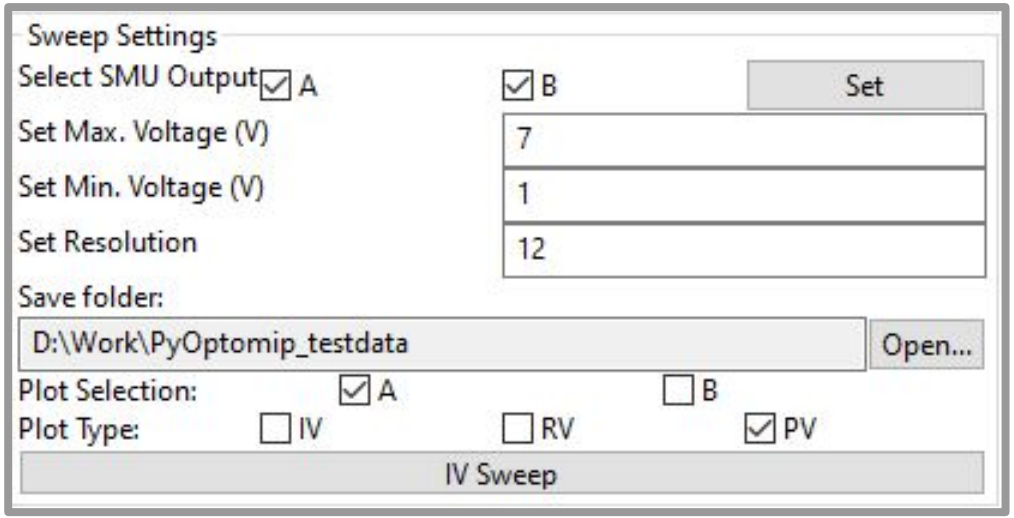
Sweep Settings

IV Sweeps can be performed with channel A or B by checking their respective boxes and clicking **Set**.

Choose the minimum and maximum Voltage in *Volts* as well as the resolution in *Volts*.

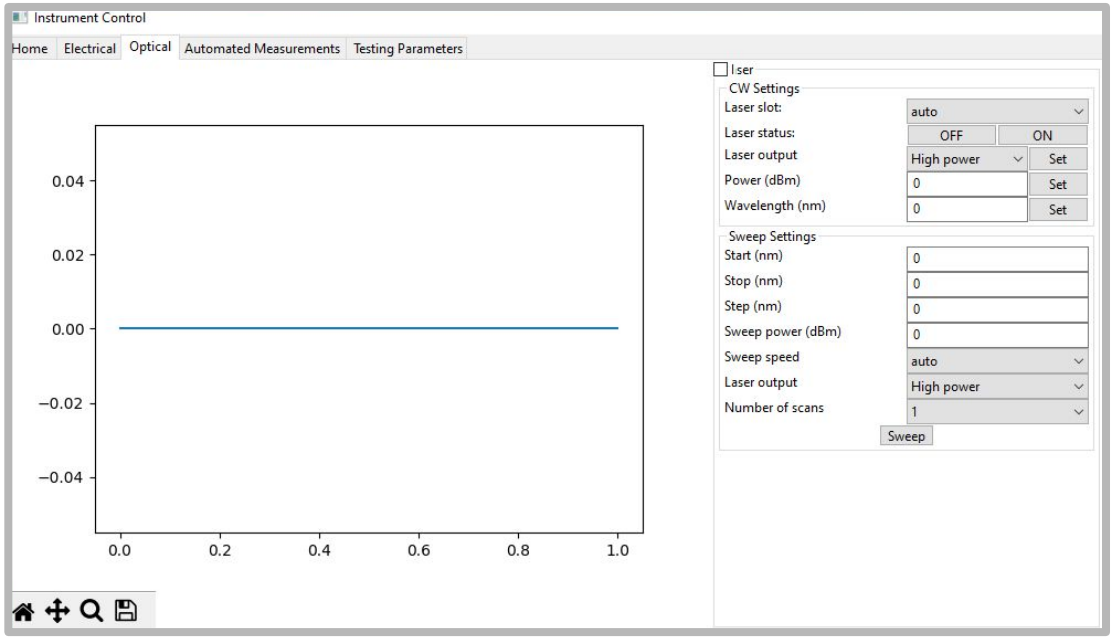
If you wish to save the sweep results You can choose a folder to do so.

You can also choose which channel to plot as well as the type of plot. When ready, press the **IV Sweep** button.



Optical Tab

The optical tab can be used to control the laser as well as perform and plot optical spectrum sweeps.



CW Settings

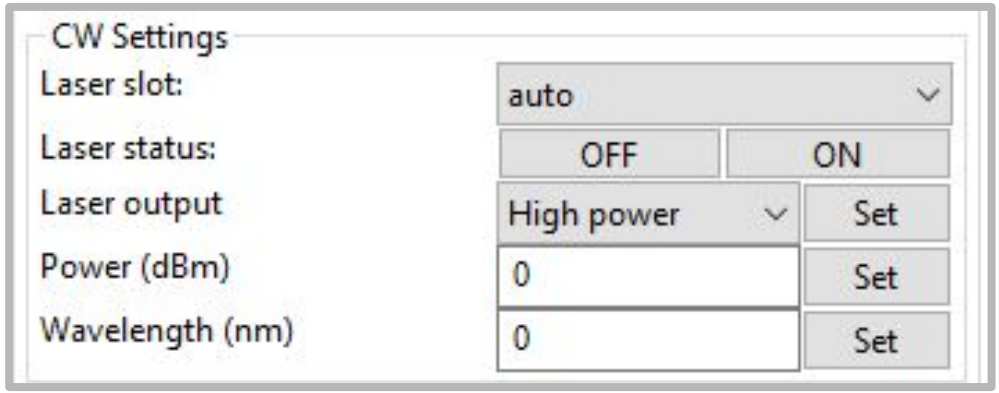
The laser output parameters can be set using the CW settings

The laser slot should be set to auto by default and should in most cases stay this way.

Use the laser status **ON OFF** buttons to toggle the laser On and Off

Select the desired laser output and that click the **Set** button to push output to laser.

The power and wavelength of the laser can be set in dBm and nm respectively.

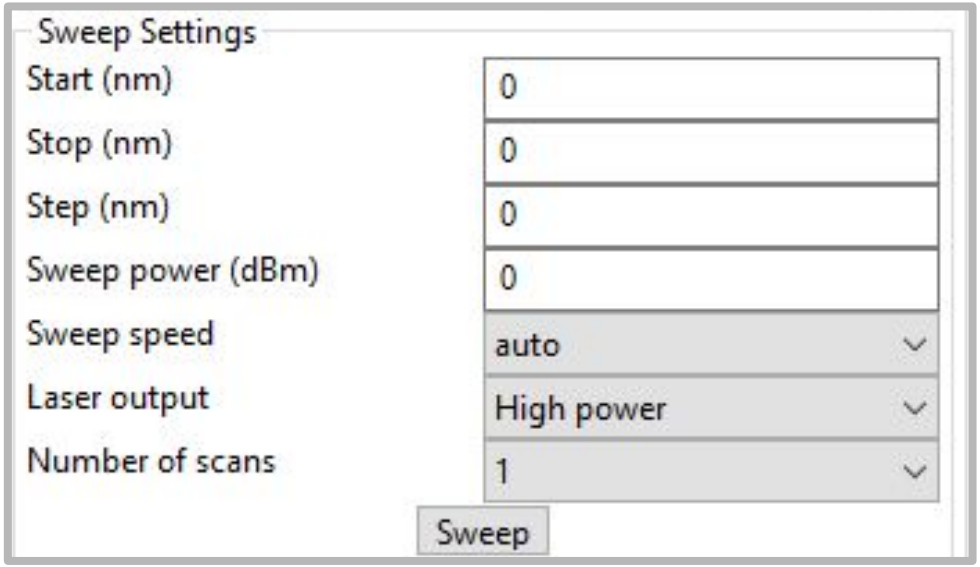


Sweep Settings

In the sweep settings frame all parameters required for an optical sweep can be set.

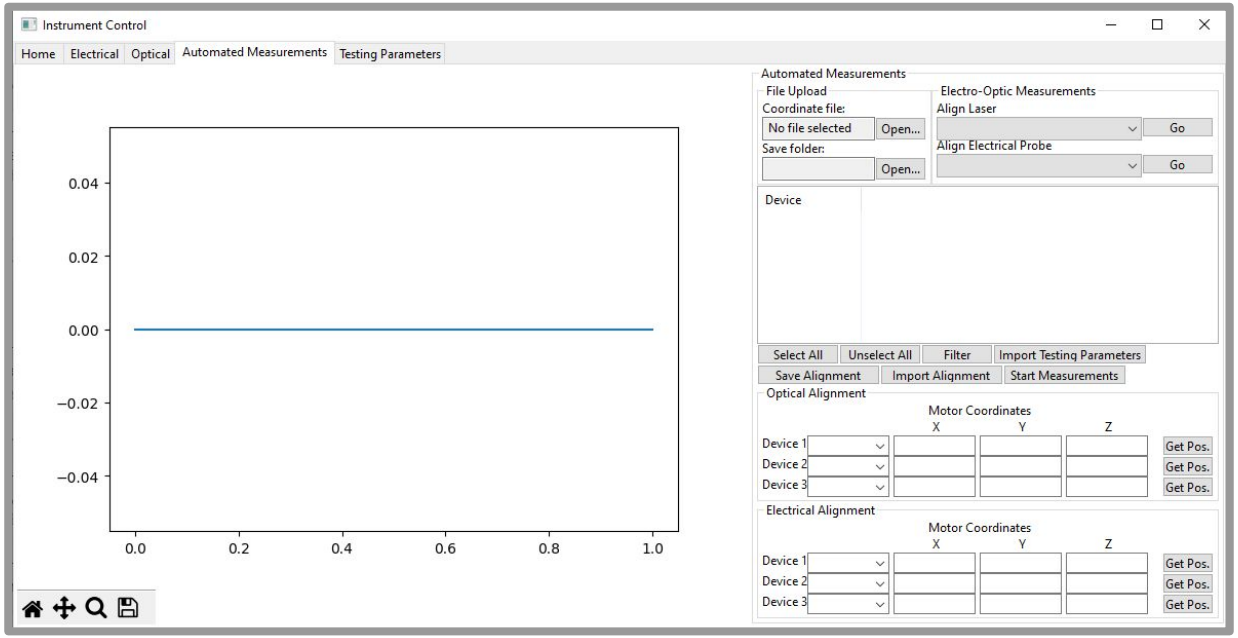
Starting wavelength, stop wavelength and step size between wavelengths can all be set here in nm.

Choose which detectors you wish to include in the sweep in the Home tab.



Performing Automated Measurements

The automated measurements tab can be used to align your equipment automatically to selected devices by creating transform matrices.



Labelling Devices

In order to perform automated measurements, you'll need to generate a coordinate text file from the GDS of the chip you wish to measure. You'll need to install the SiEPIC-Tools package for KLayout and the SiEPIC-EBeam-PDK to do this.

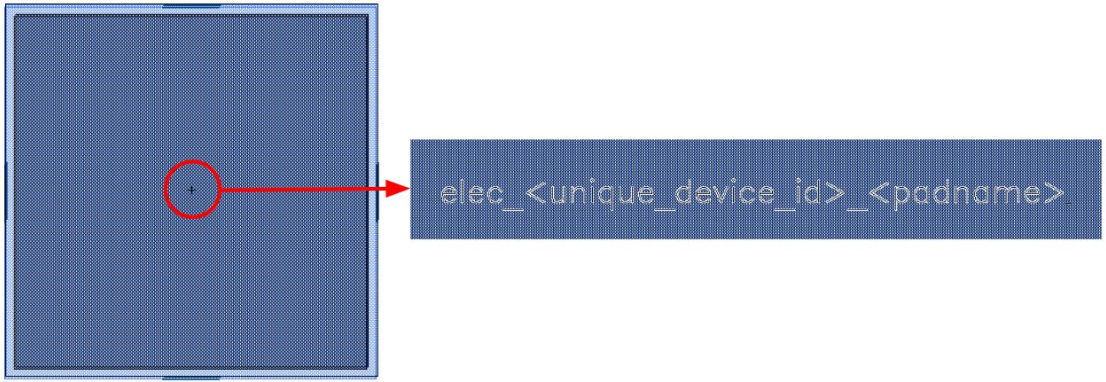
For every device in your design ensure you place labels for the optical and electrical connections as necessary. Make sure to follow the labelling standards and design standards outlined in the next pages.

Optical Labels

Optical labels should be placed at the tip of the grating coupler to be connected to the laser. Labels should follow the format ***opt_<polarization>_<wavelength>_<device>_<unique-device-id>***. Each electro-optic device should have an id which is unique from any other on the same chip.

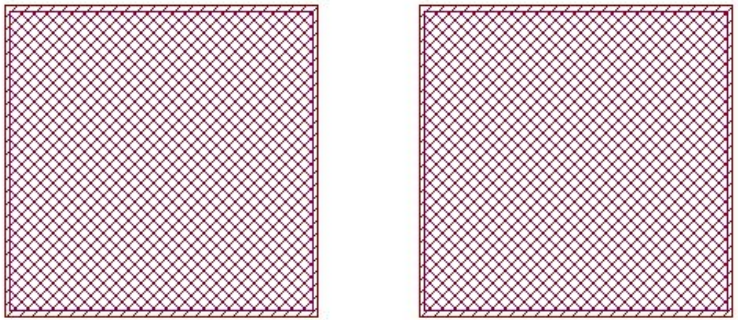
Electrical Labels

Electrical labels should be placed at the center of each bond pad and should follow the format ***elec_<unique-device-id>_<pad-name>***.



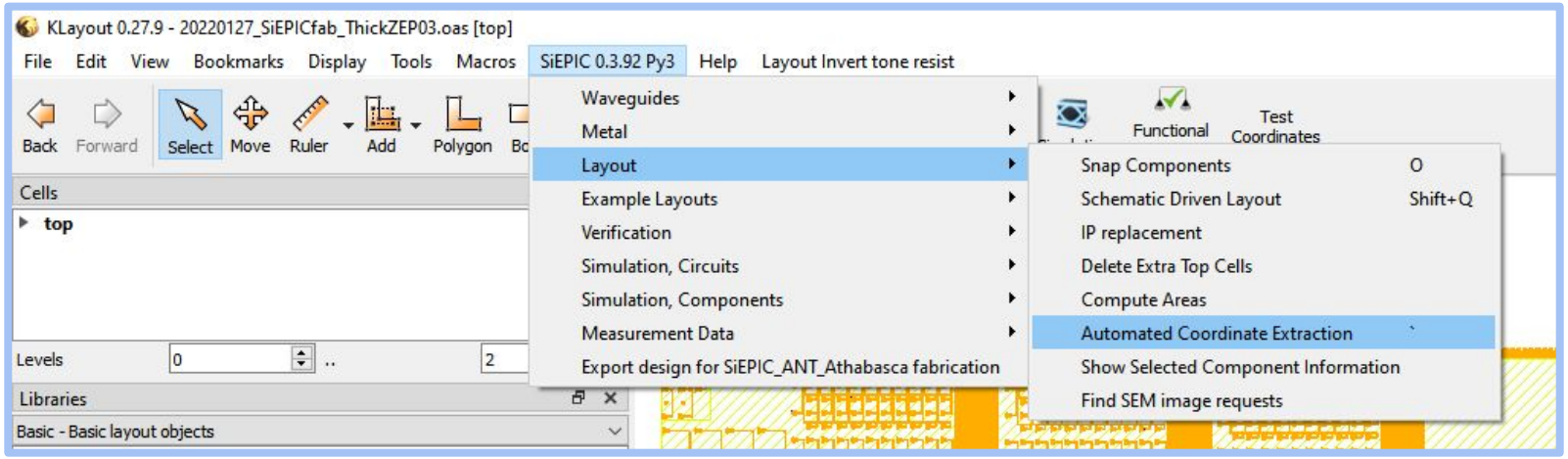
Electrical Pad Arrangement

Bond pads should be 75µm by 75µm and be spaced 100µm apart. There should be at least two bond pads and the leftmost pad should be signal ground.

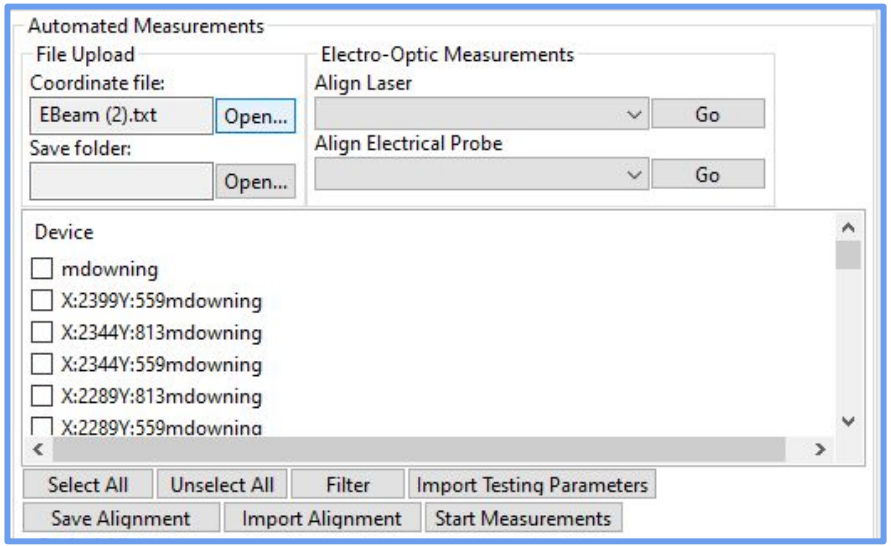


Generating Coordinate Files

Once you've finished your design and labelled all devices you can export a text file of device coordinates by clicking on the **SiEPIC** toolbar and then selecting **Layout** followed by **Automated Coordinate Extraction**. A pop up window will open which will allow you to review and then save your text file.



Importing Coordinate Files



Import the coordinate text file in the file upload panel. The checklist should populate with devices on your chip.

If there are multiple devices with the same ID, PyOptomip will distinguish them by appending the x and y coordinates from the GDS file to the front of each ID.

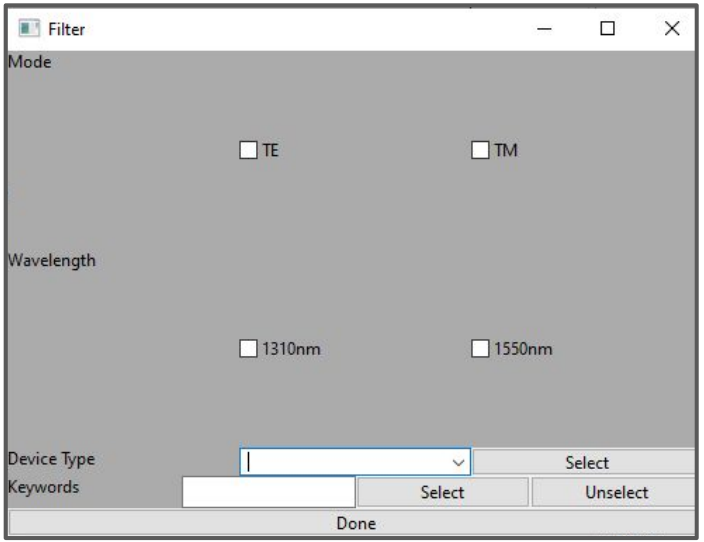
From here you can select which devices to perform measurements on using the ***Select All***, ***Unselect All*** and ***Filter*** buttons.

Filtering Devices

Within the filter frame you can select which modes and wavelengths you wish to measure. For example, selecting **TE** will select all TE devices.

You can also select the device types that you wish to include by choosing them from the drop down menu then pressing **Select**.

Keywords can be used to select or unselect any device which contains the desired keyword in its ID.

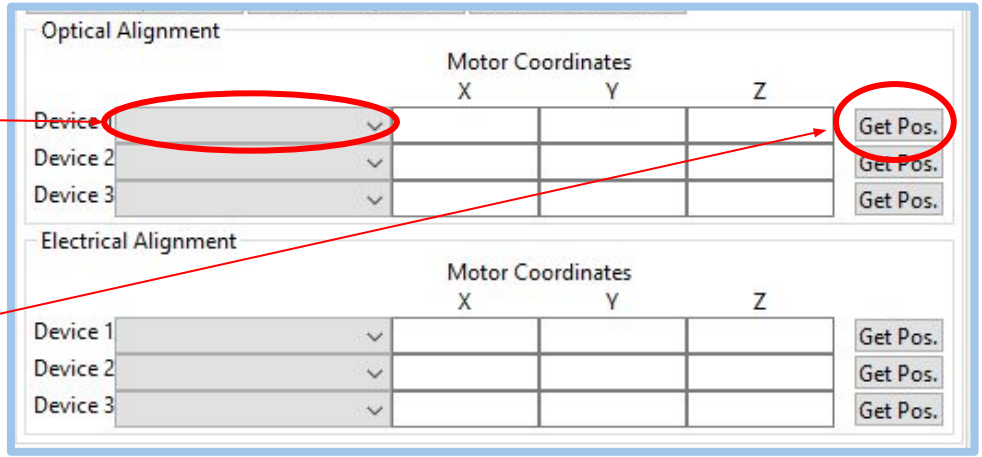


Creating Transform Matrices

Before automated measurements can commence the transform matrices must be created. Within the automated measurements tab are the electrical and optical alignment boxes. Align the fibre array manually (through the home tab) onto a device. Once the array is aligned, choose the device you have aligned to from the drop down menu and hit the **Get Pos.** button to save the current coordinates. Repeat this until all boxes have been filled.

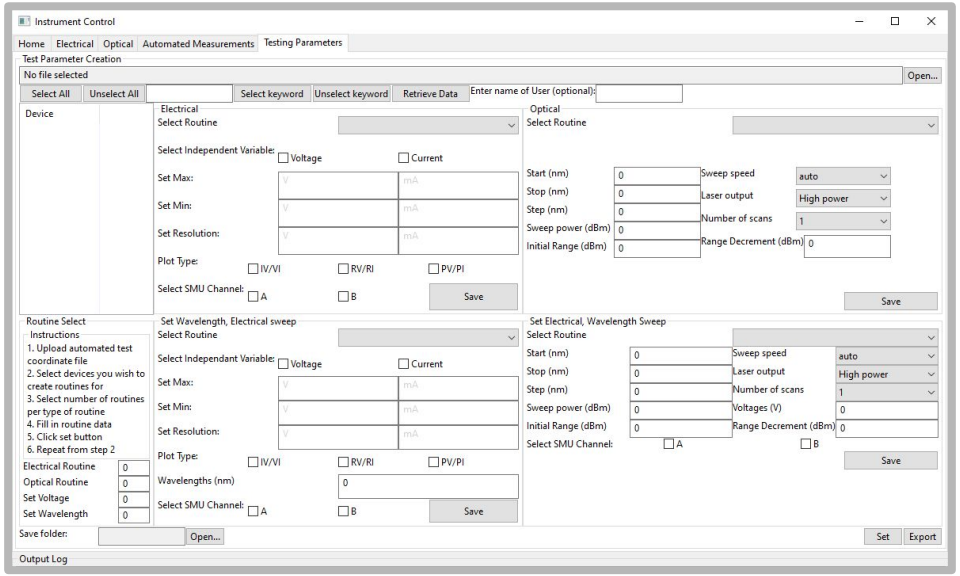
Choose alignment device from dropdown

Once the array is aligned click the **Get Pos.** Button



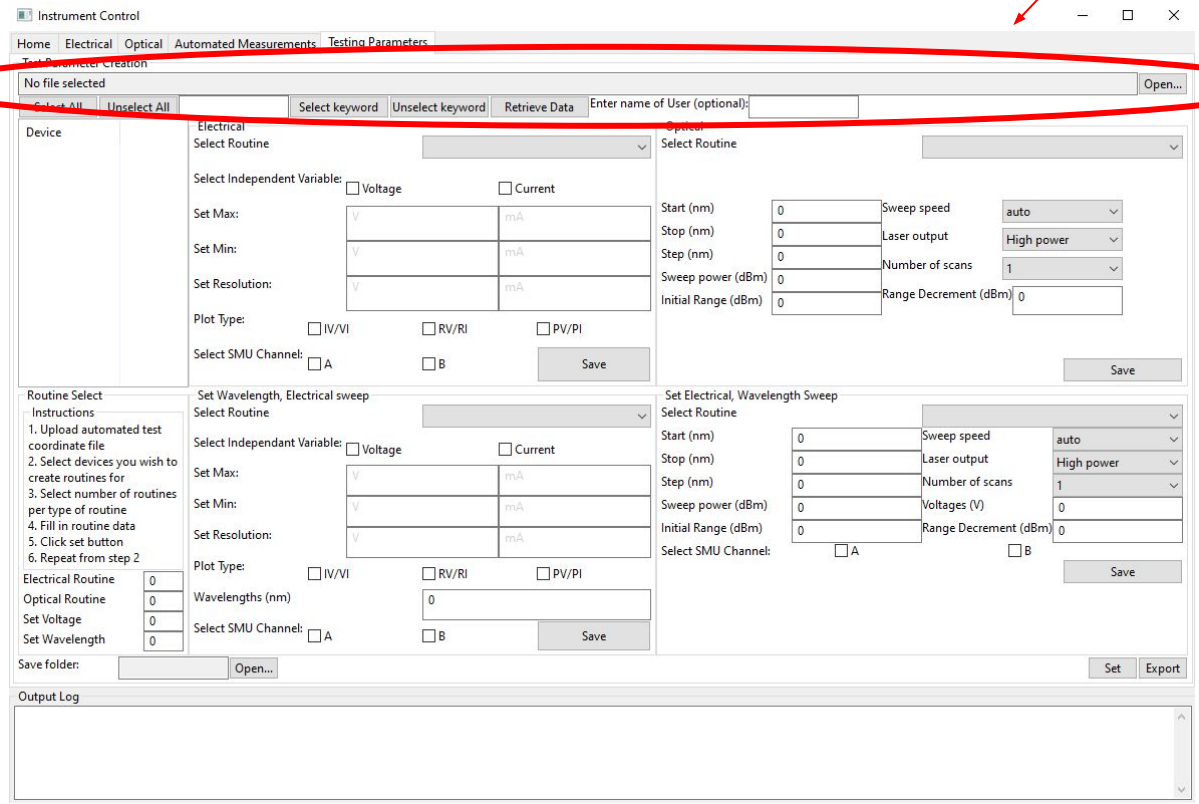
Creating Test Routines

Before the user can begin running automated tests they must create a set of routines for each device that they want to test. This is done using the testing parameters tab in pyOptomip or by running the standalone testing parameters python script.



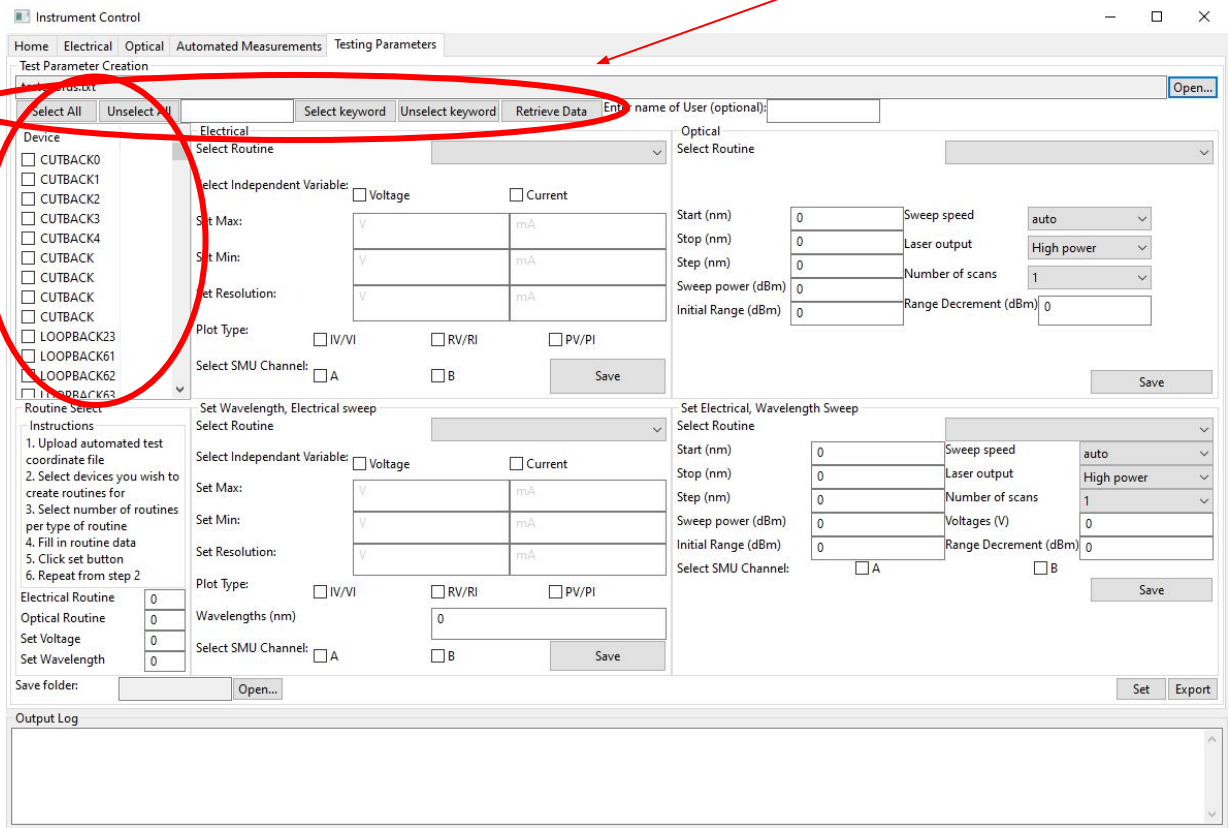
Uploading Device Coordinate Files

Use this to open folder and choose chip coordinate file



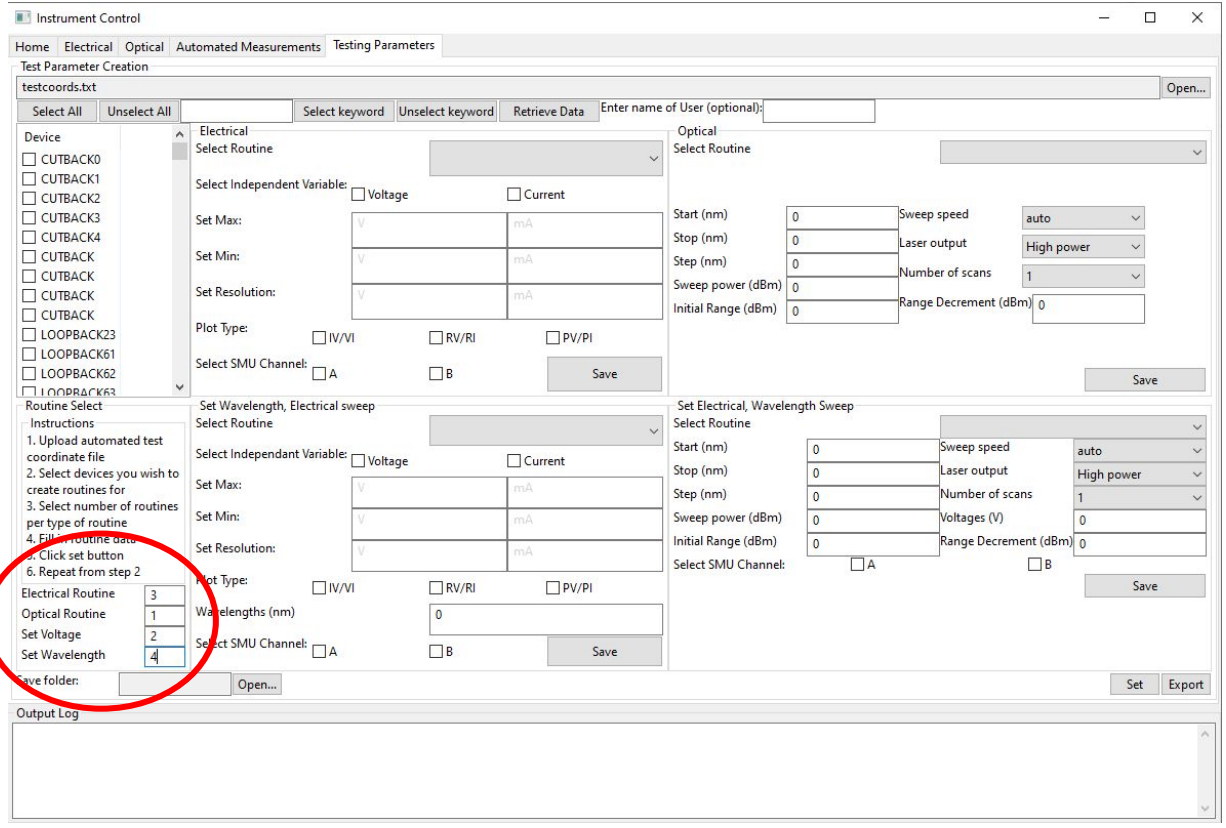
Selecting Devices for Routines

Use keywords in device names to select devices or select all



Manually select devices

Selecting Quantity of Routines

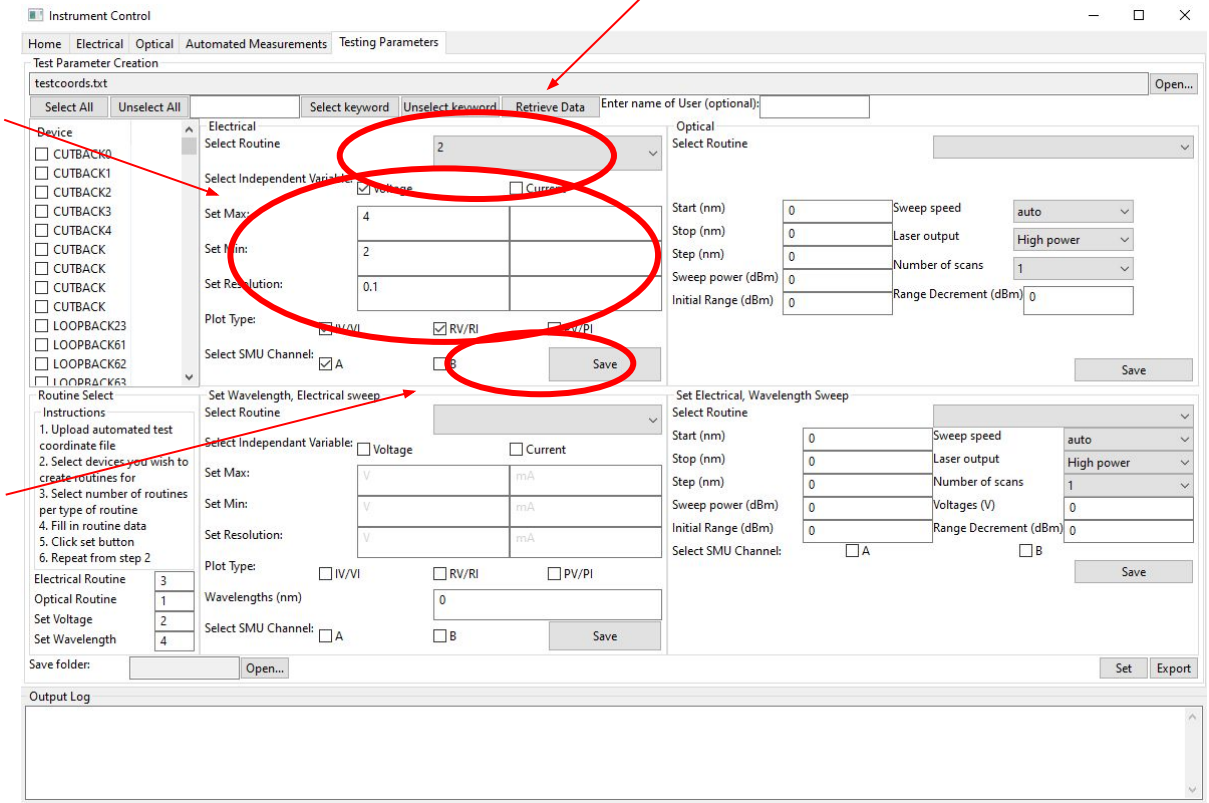


Setting Routines

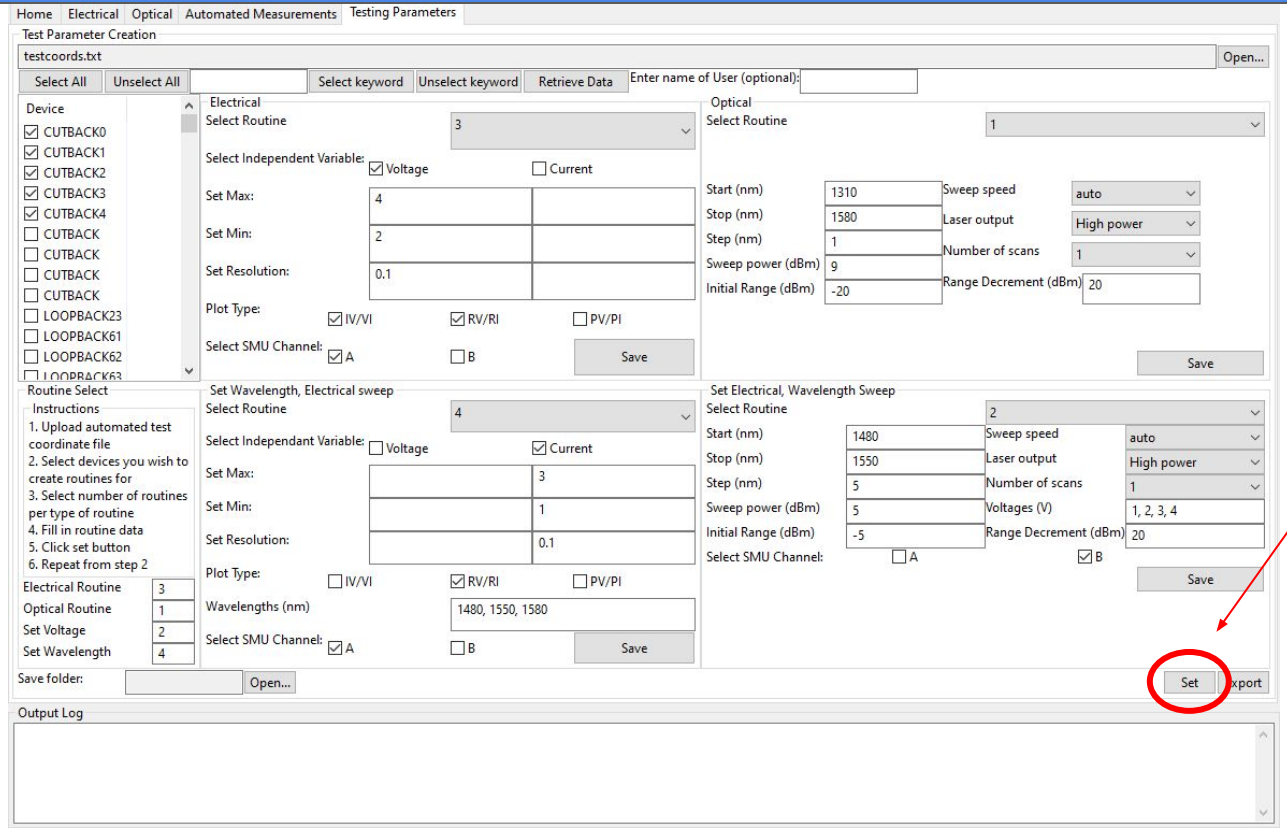
If more than one type of the same routine choose specific routine number

Input routine parameters

When all routines complete hit the save buttons to temporarily save the routine parameters



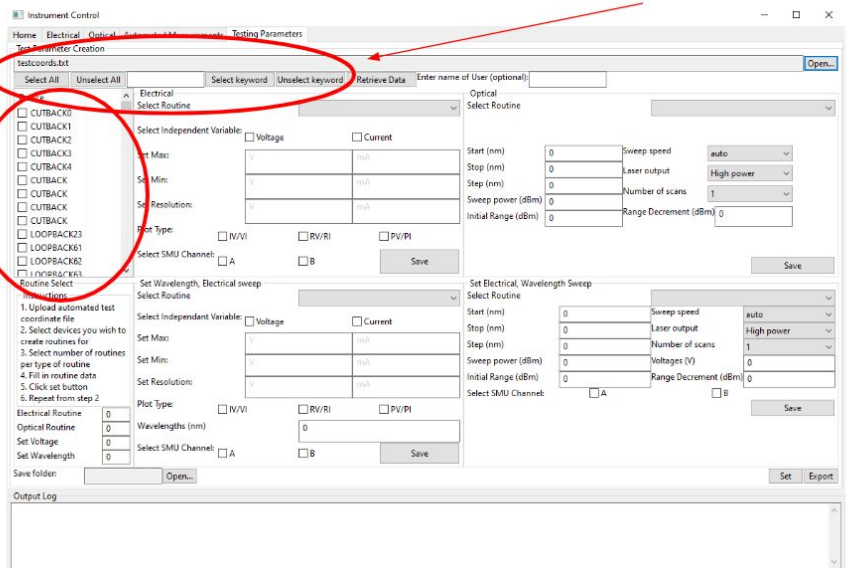
Setting the Data Array



Once all parameters have been set and saved hit the set button to upload data to file and clear software for next set of routines

Repeat until all desired device routines are set

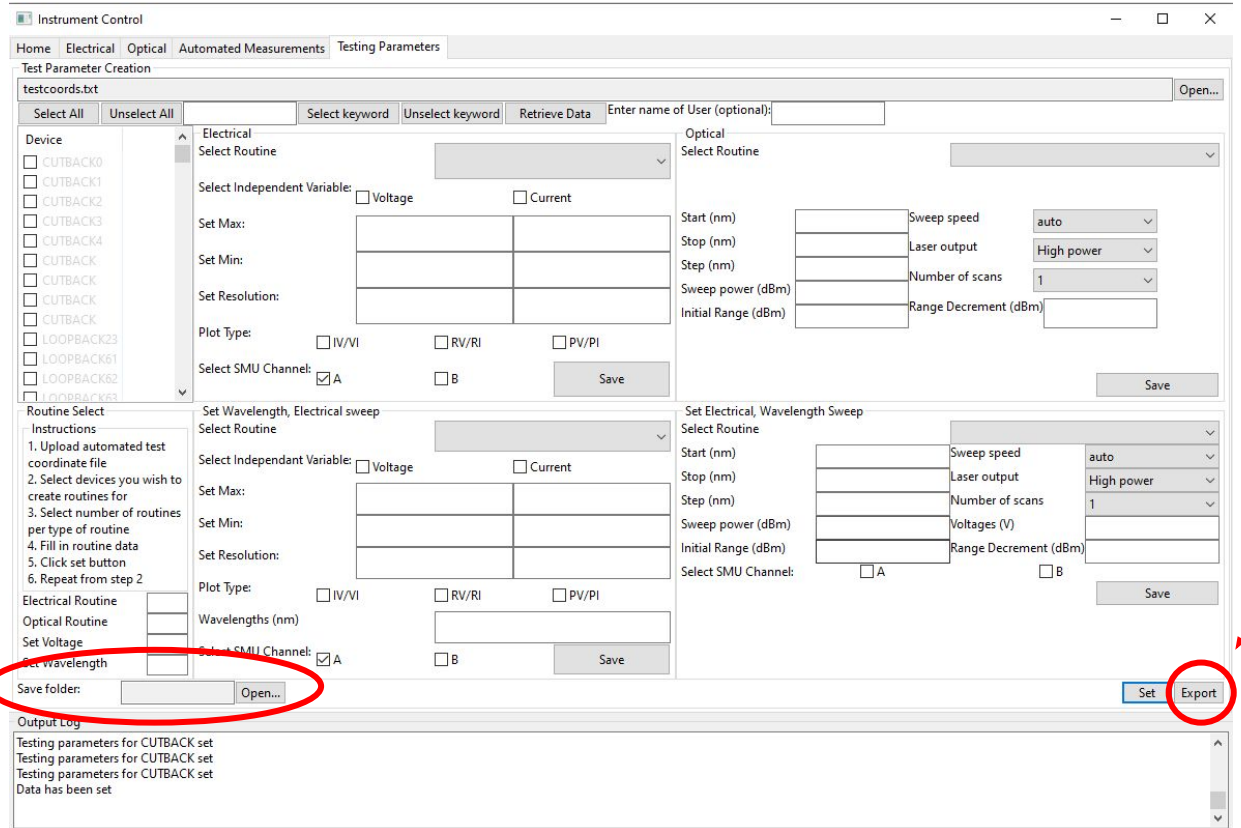
Use keywords in device names to select devices or select all



Manually select devices

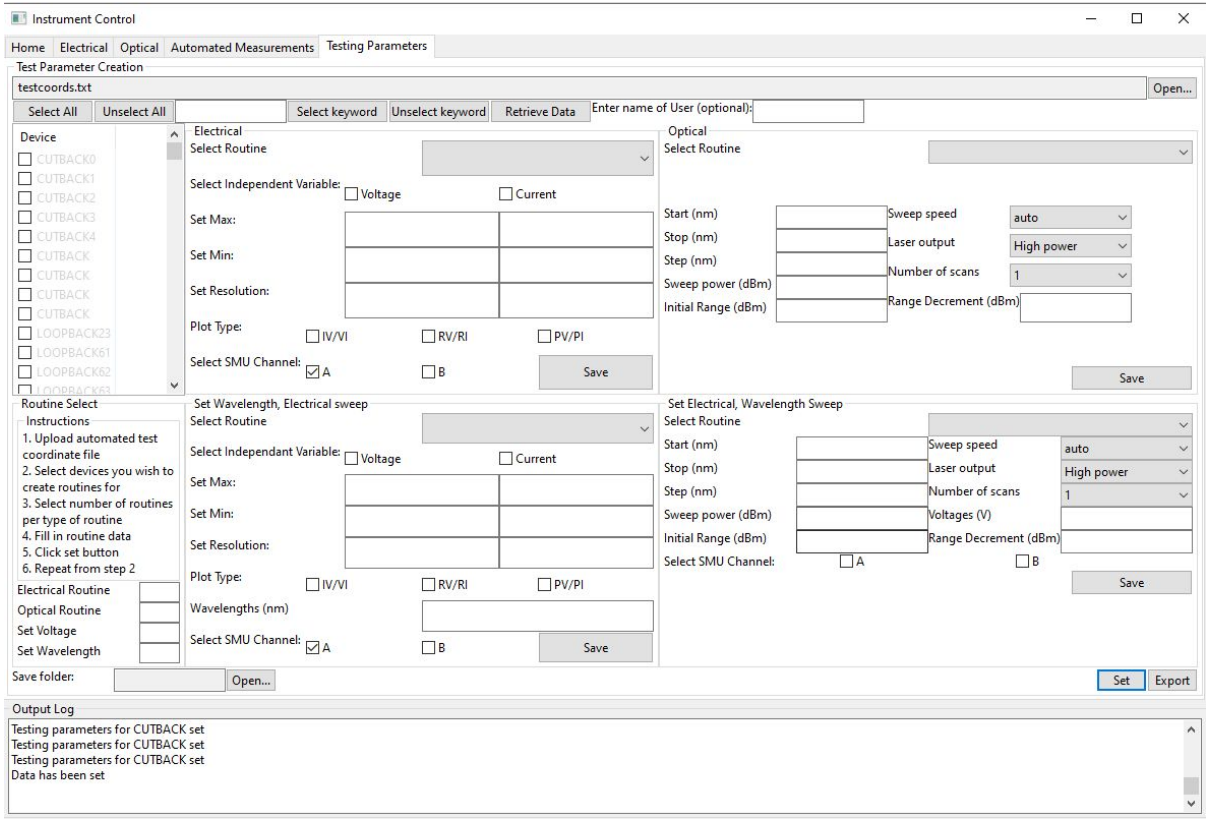
Finalize

If using the standalone testing parameters file, or just want an additional copy of the parameters saved in csv format choose save location



When all devices have been set, finalize by hitting the export button

Retrieving Data



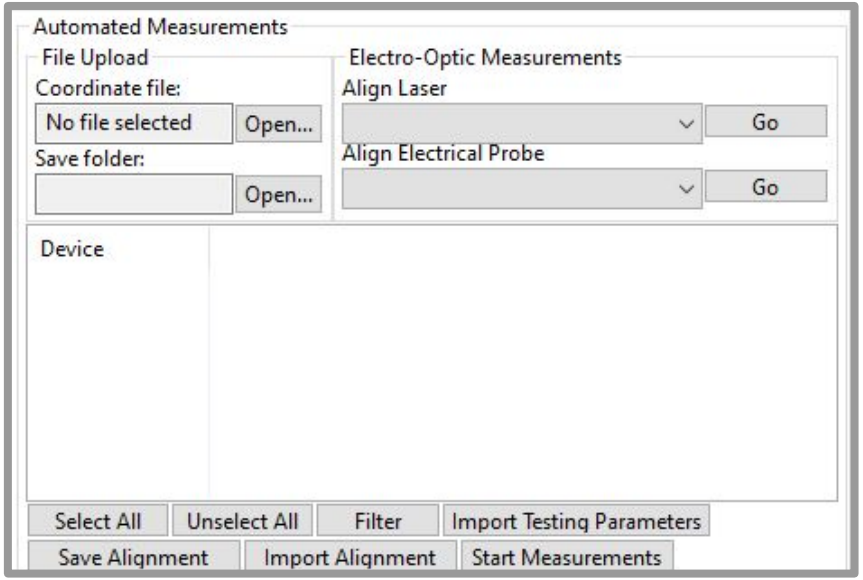
Starting Measurements

After uploading the coordinate file, creating the transform matrices and setting the testing parameters, you are ready to begin taking automated measurements.

Set the folder in which you wish to save your measurement data by clicking **Open...** beside the designated text box.

Next, import your testing parameters file using the **Import Testing Parameters** button.

Finally, select **Start Measurements.**



PyOptomip Github Repo

To make changes to the PyOptomip code or to view how the software works, all related PyOptomip content can be found in the following github repo:

https://github.com/SiEPIC/pyOptomip

The ida-main-py3 branch is the version currently running on the IDA stage in KAISER 4060.

Requirements

Before one can begin using PyOptomip on their PC they must ensure their computer has the required modules and API's. To find all required python add-ons and modules use the **requirements.txt** file located in the PyOptomip folder in the Github repository.