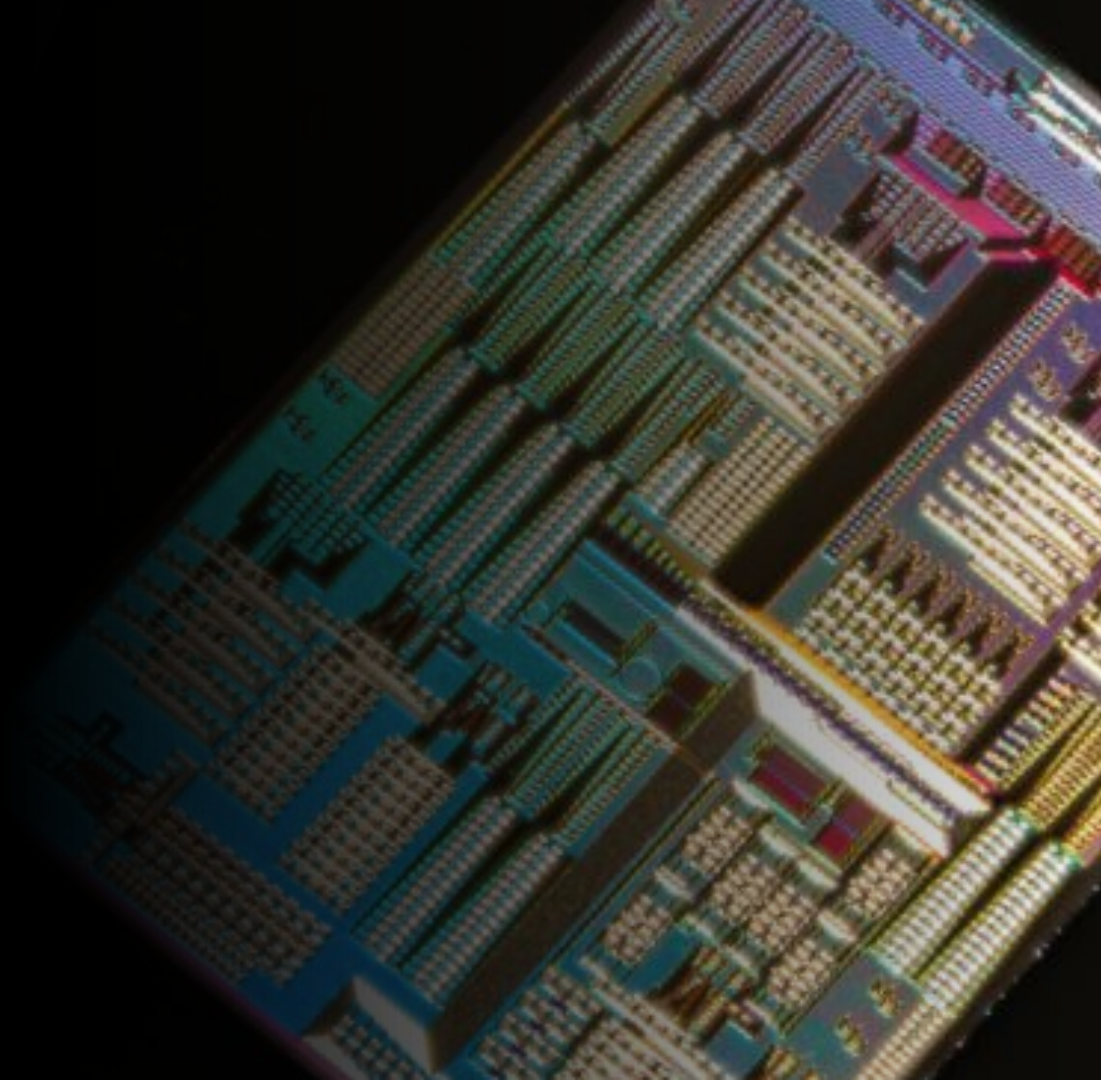
PyOptomip User's Guide

12/12/2022



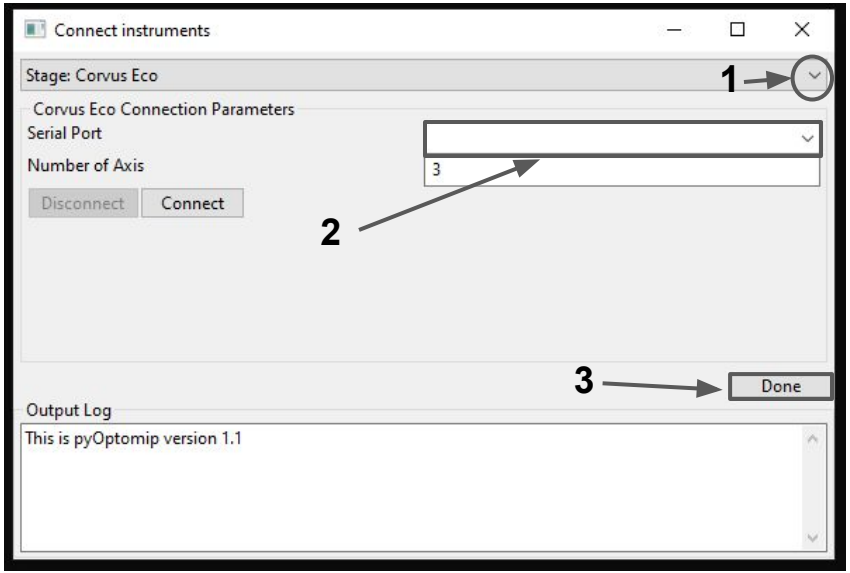
Purpose

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

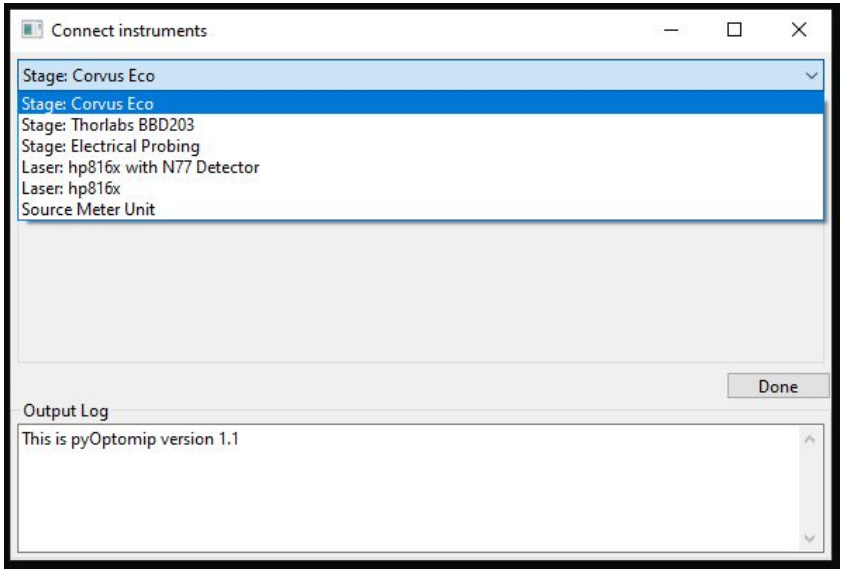
Stage: Electrical Probing - ASRL10::INSTR

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.

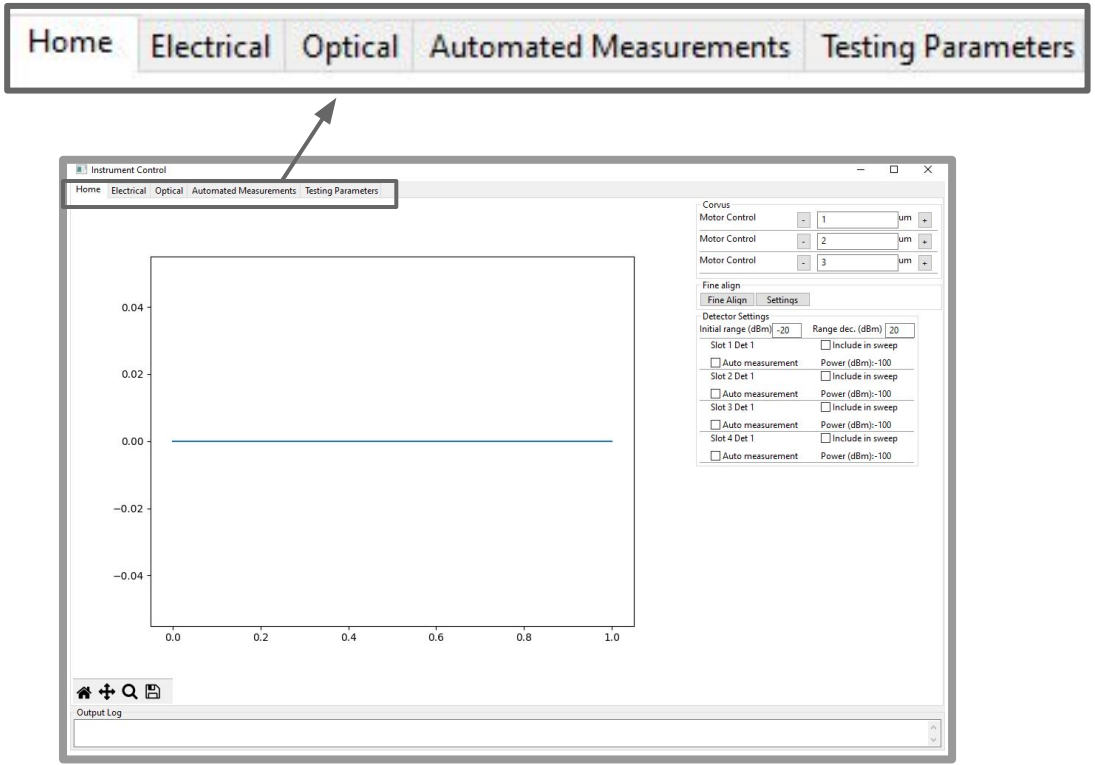
Home tab:
Stage control, detector measurements and settings

Electrical tab:
SMU control, IV sweeps

Optical tab:
Laser control, wavelength sweeps

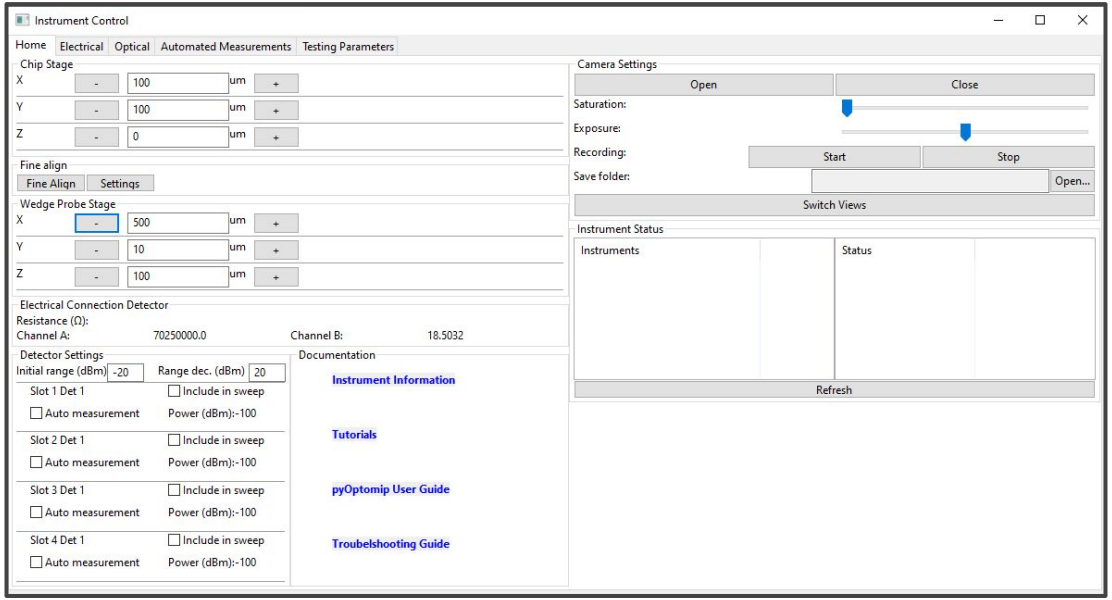
Automated Measurements:
Automated measurement settings

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 and SMU as well as the camera.



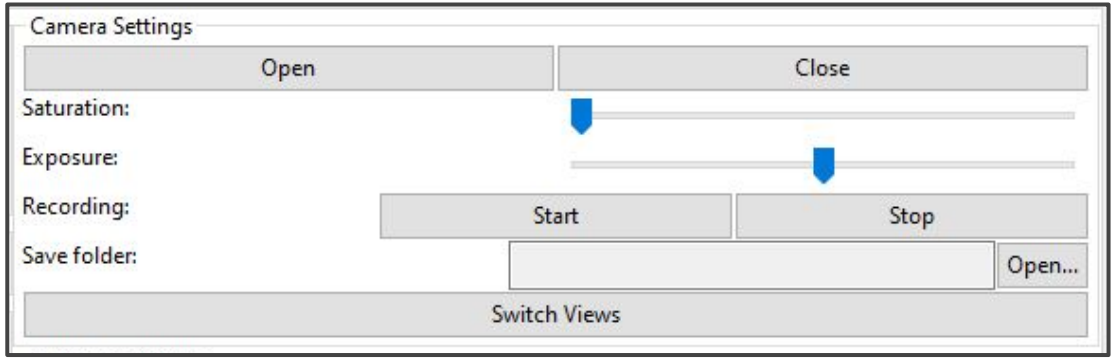
Camera Settings

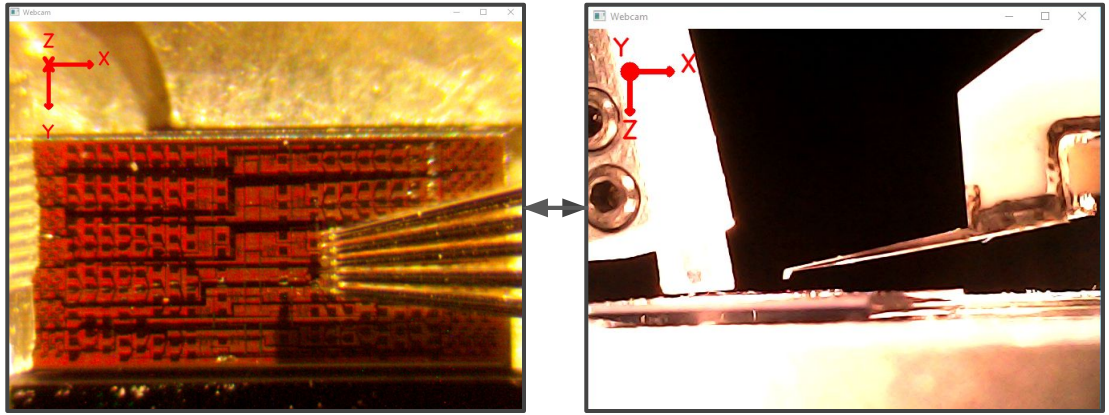
The Camera Settings can be used to adjust the exposure and saturation of the camera and to switch between views.

Begin by pressing **Open** and then use the **Switch Views** button to toggle between a top and side view of the chip.

Make sure to close the camera before exiting PyOptomip.

If you wish to perform a video of your measurements simply choose a folder in which to save your results and then use the **Start** and **Stop** buttons.



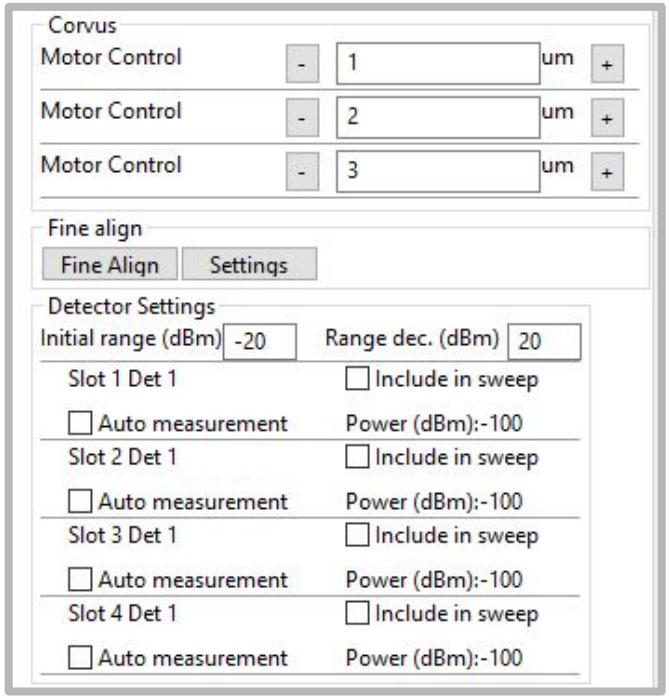


Corvus Eco

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.

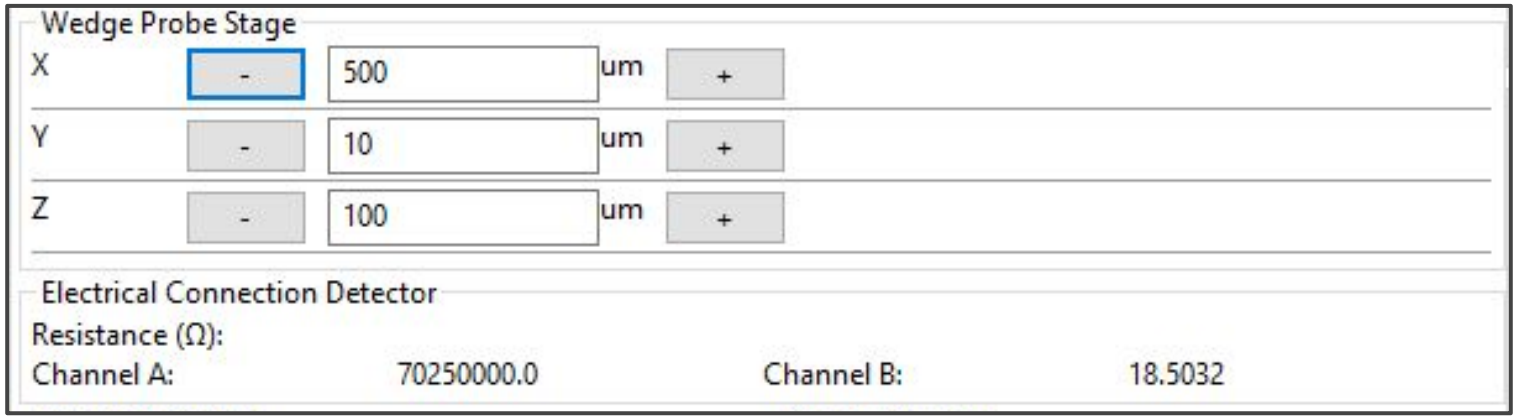
Select **Auto measurement** for the detector channels whose readings you want to see. You can use these readings to determine when the laser is more or less aligned.

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



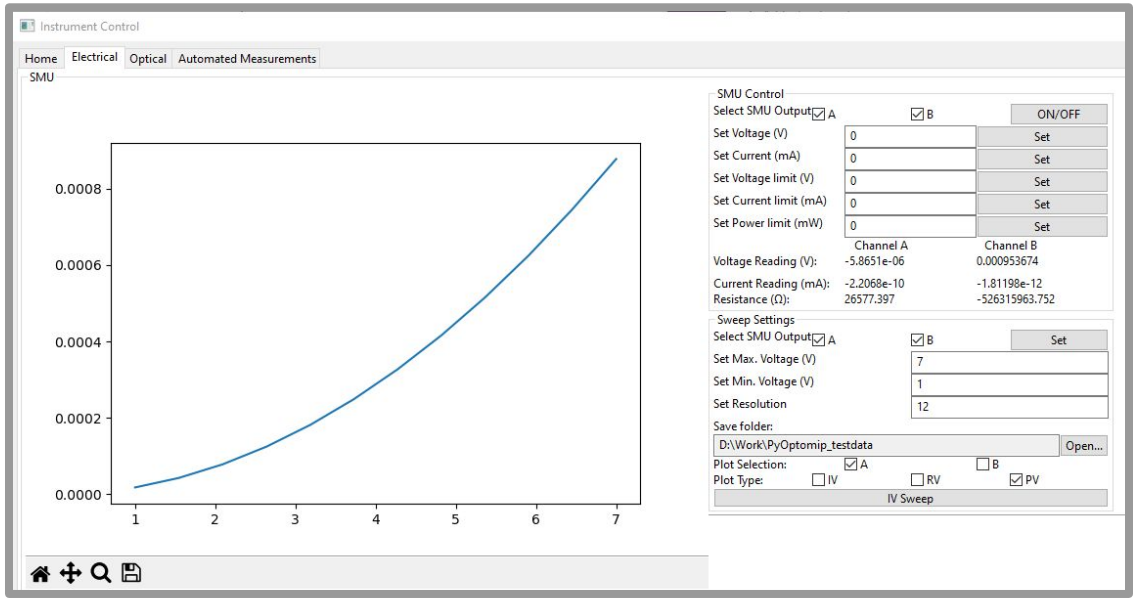
Electrical Stage

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. The resistance measured by the SMU is displayed to aid in aligning the probe.



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.

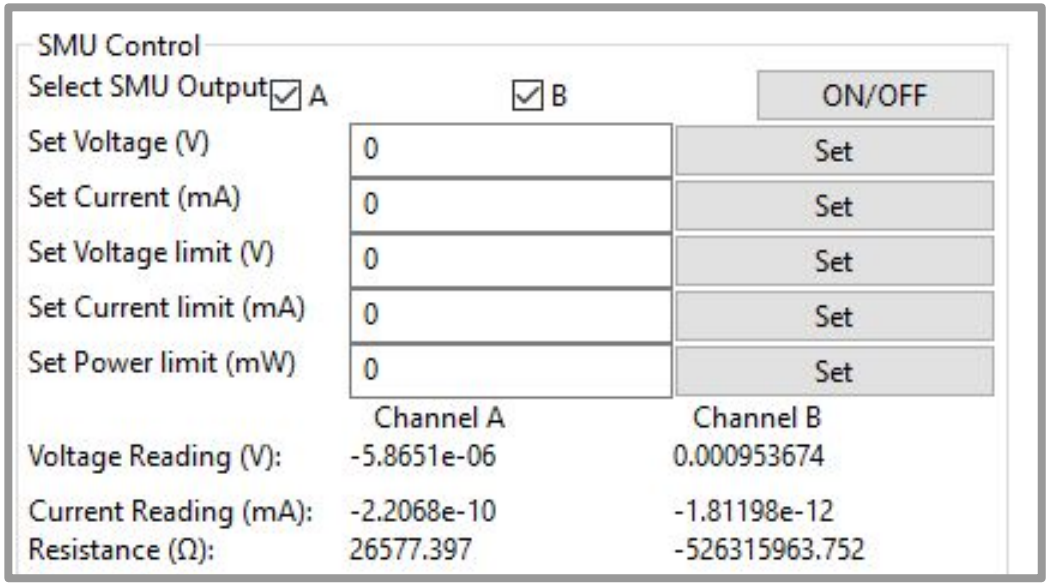


SMU Control

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.



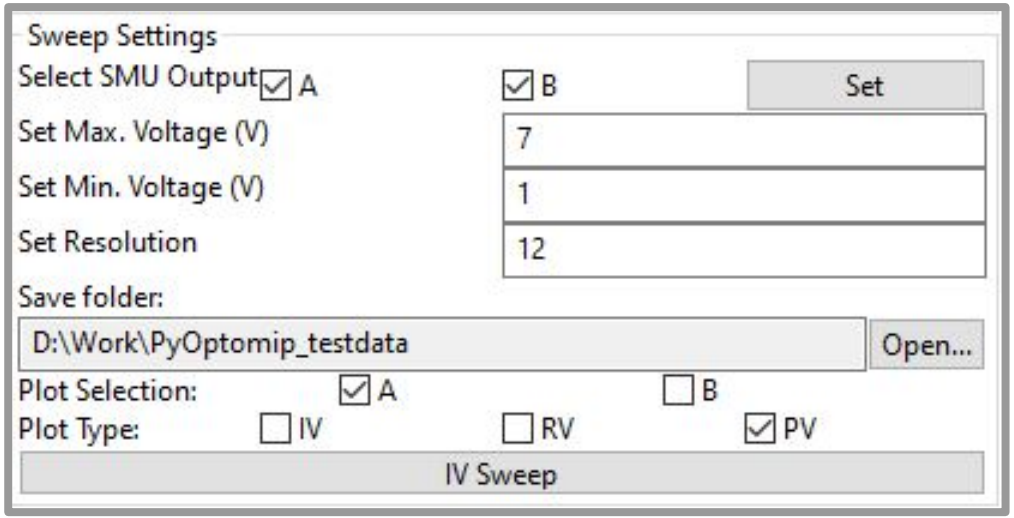
Sweep Settings

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*.

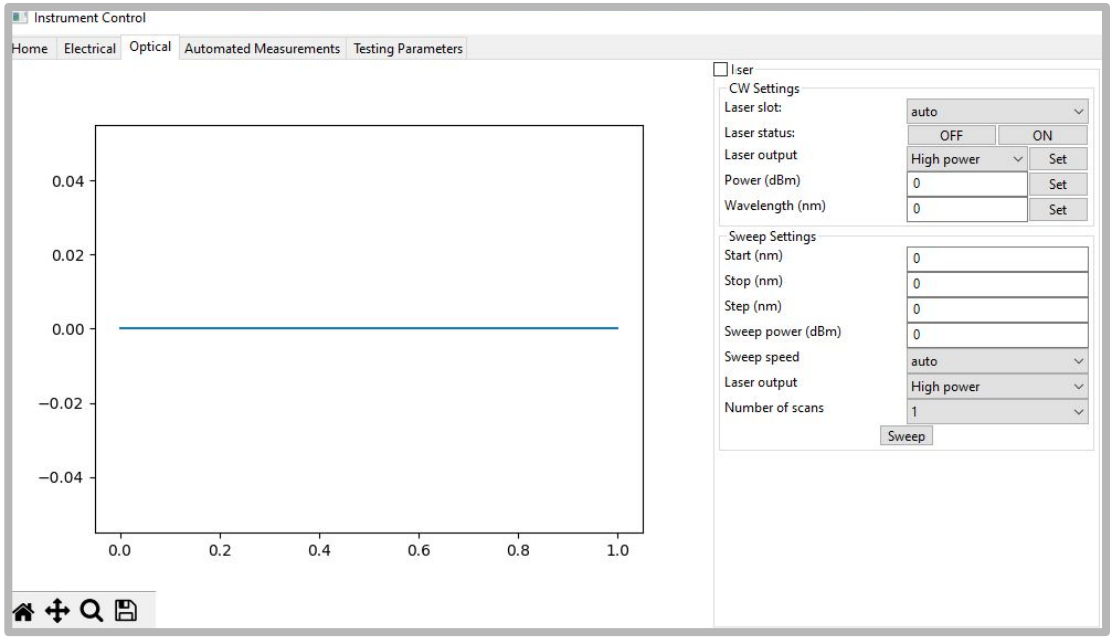
If you wish to save the sweep results You can choose a folder to do so.

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

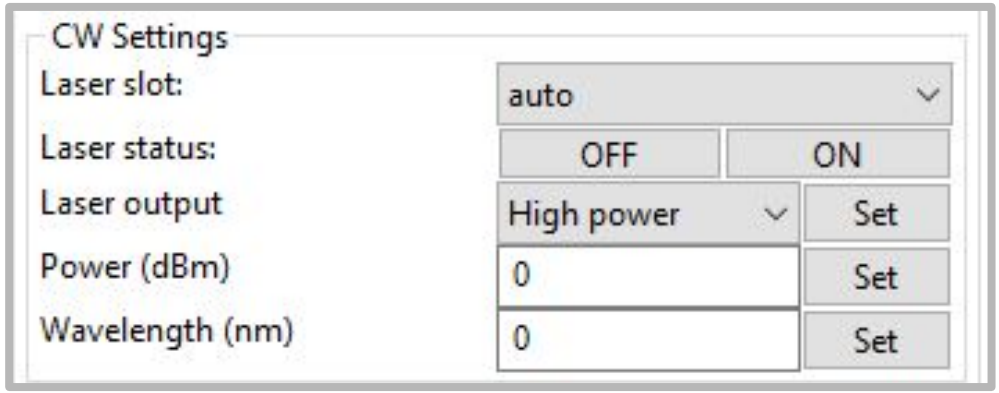
The laser output parameters can be set using the CW settings

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.

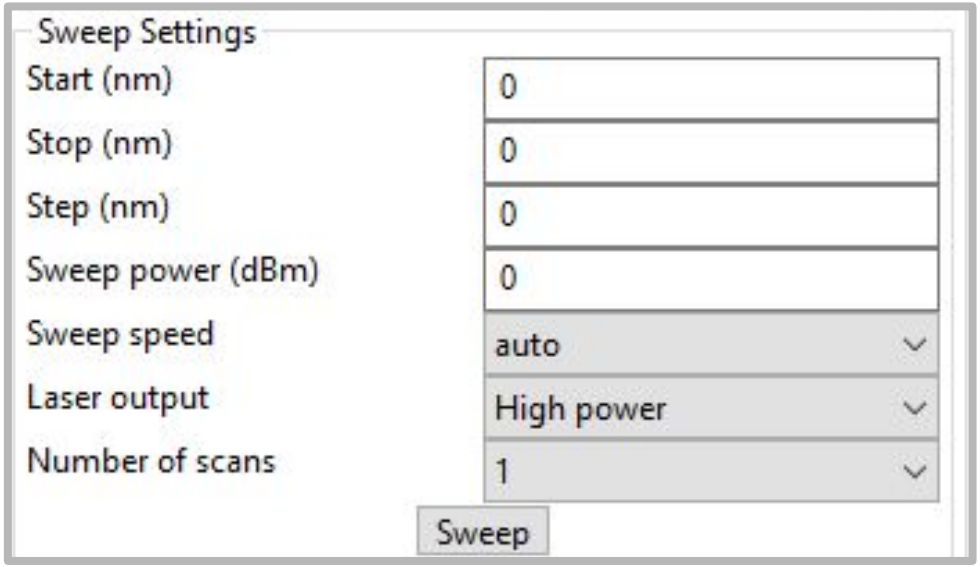


Sweep Settings

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.



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.

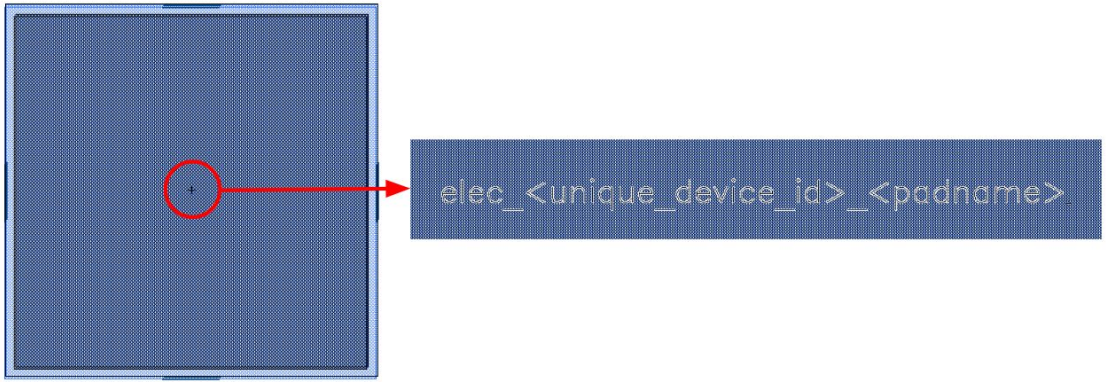
For every device in your design ensure you place labels for the optical and electrical connections as necessary. Make sure to follow the labelling standards and design standards outlined in the next pages.

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.

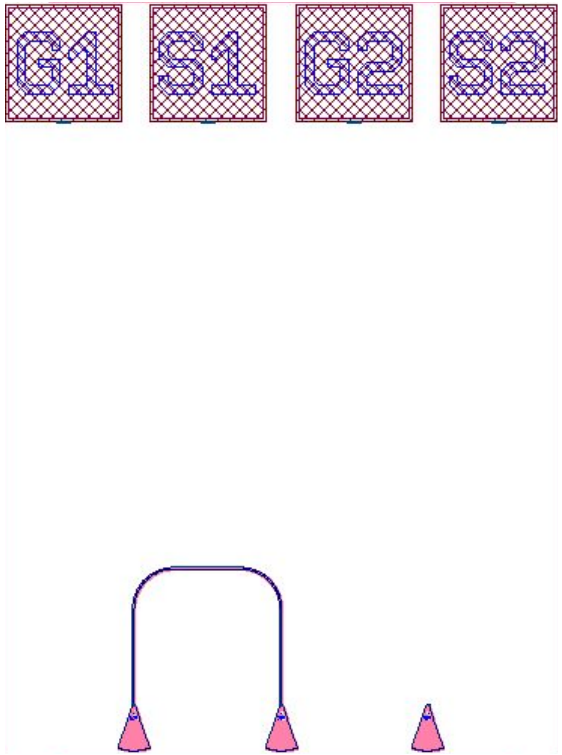
Electrical Labels

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



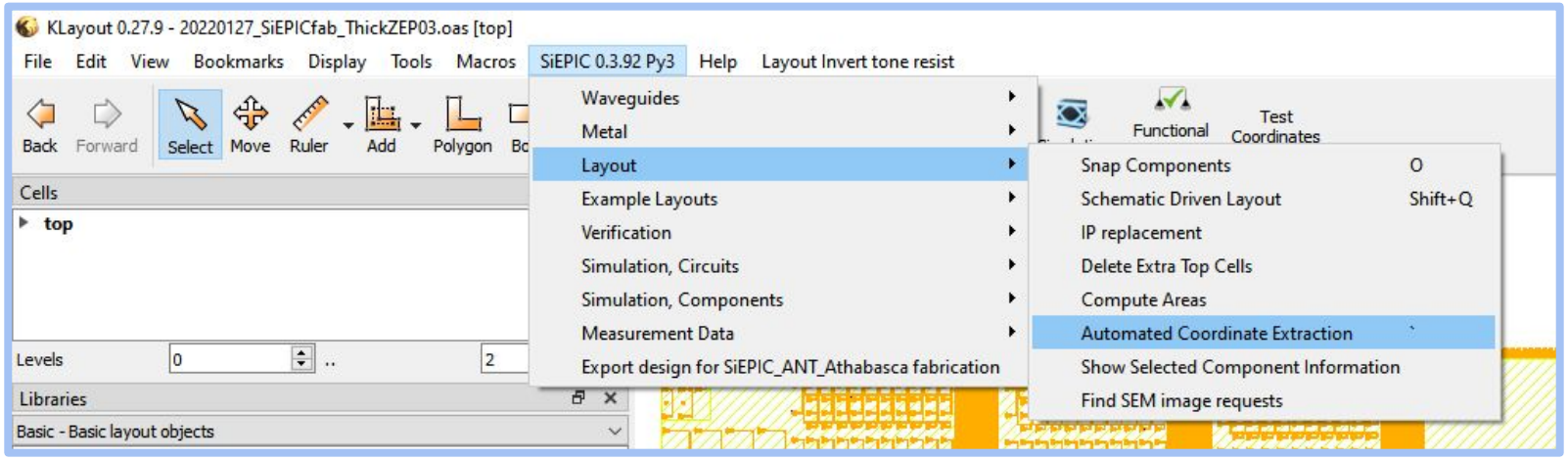
Electrical Pad Arrangement

Bond pads should be 75µm by 75µm and be spaced 125µm apart. There should be at least two bond pads and the leftmost pad should be signal ground. Ensure that all bond pads for a device are above the associated grating couplers in the orientation shown here and that there is adequate space between grating couplers and bond pads.



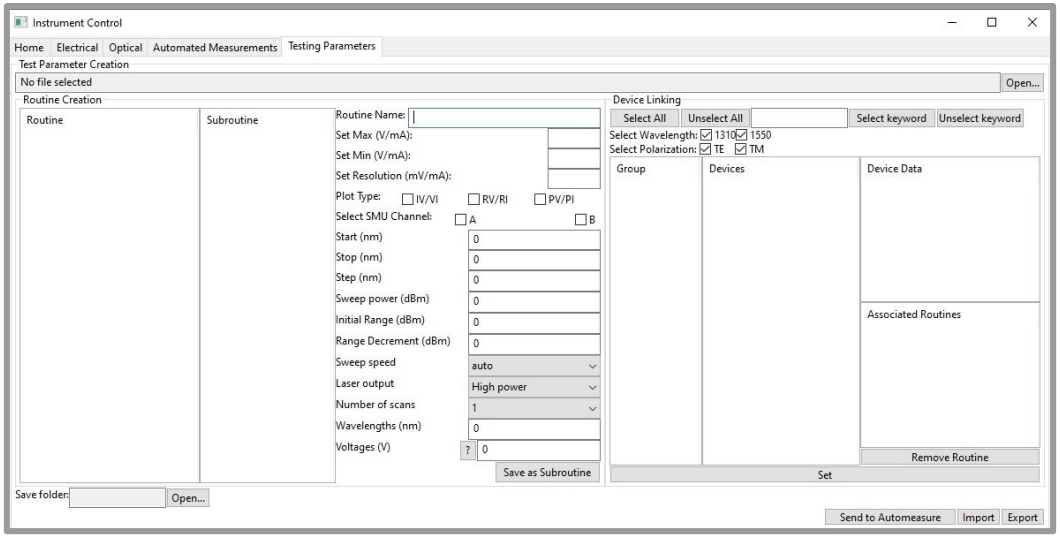
Generating Coordinate Files

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.

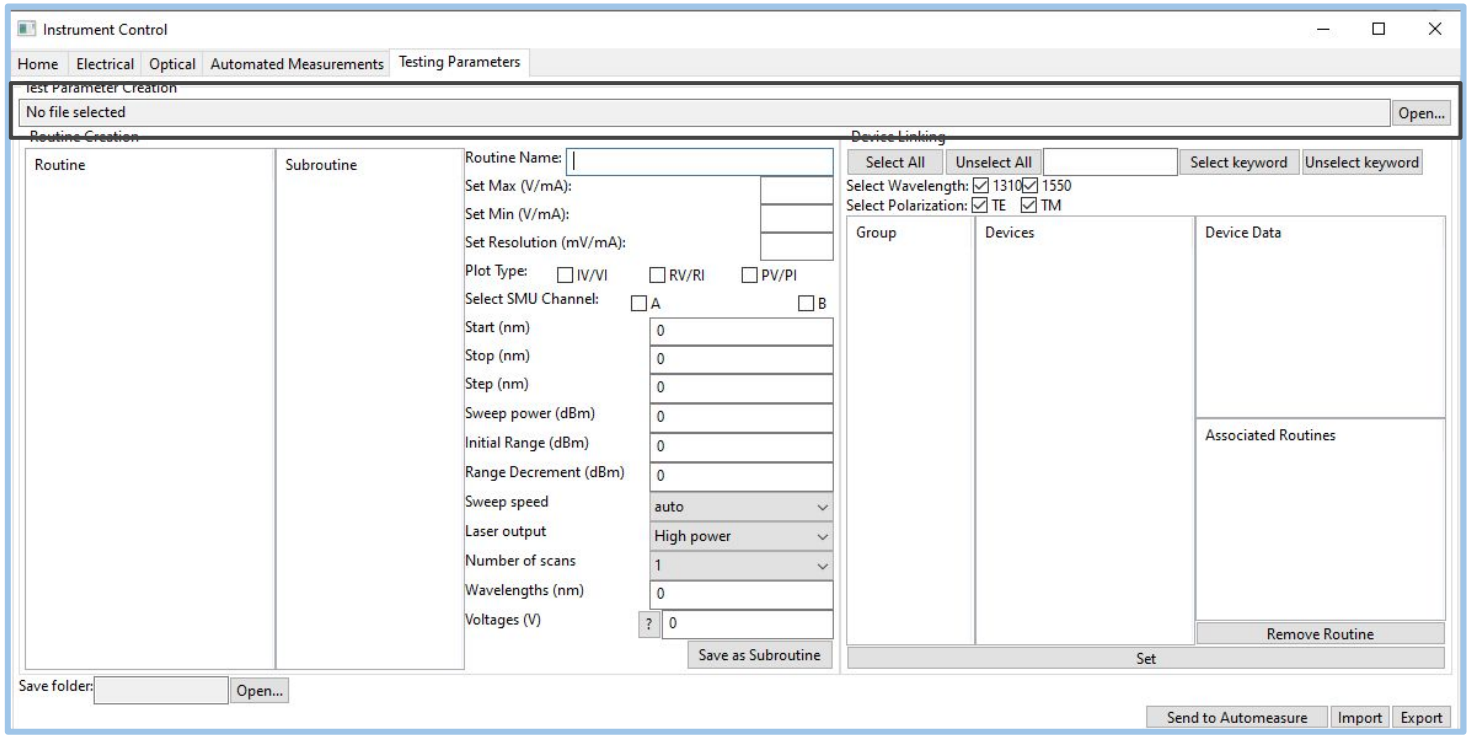


Creating Test Routines

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. The left half of the tab can be used to create routines while the right half can be used to associate routines with devices.



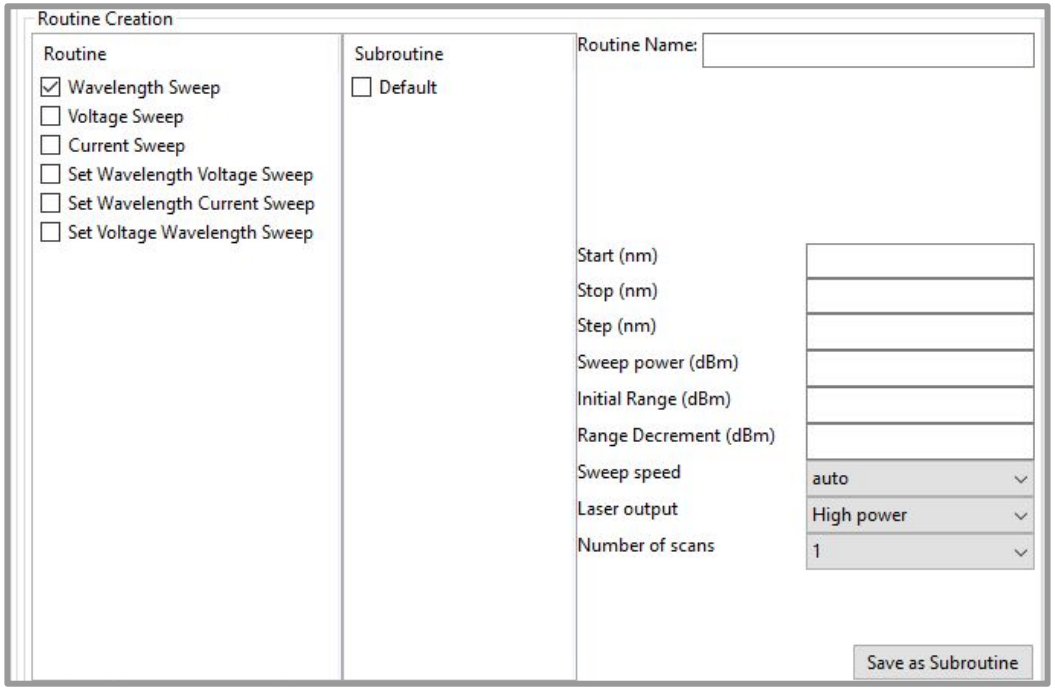
Uploading Device Coordinate Files



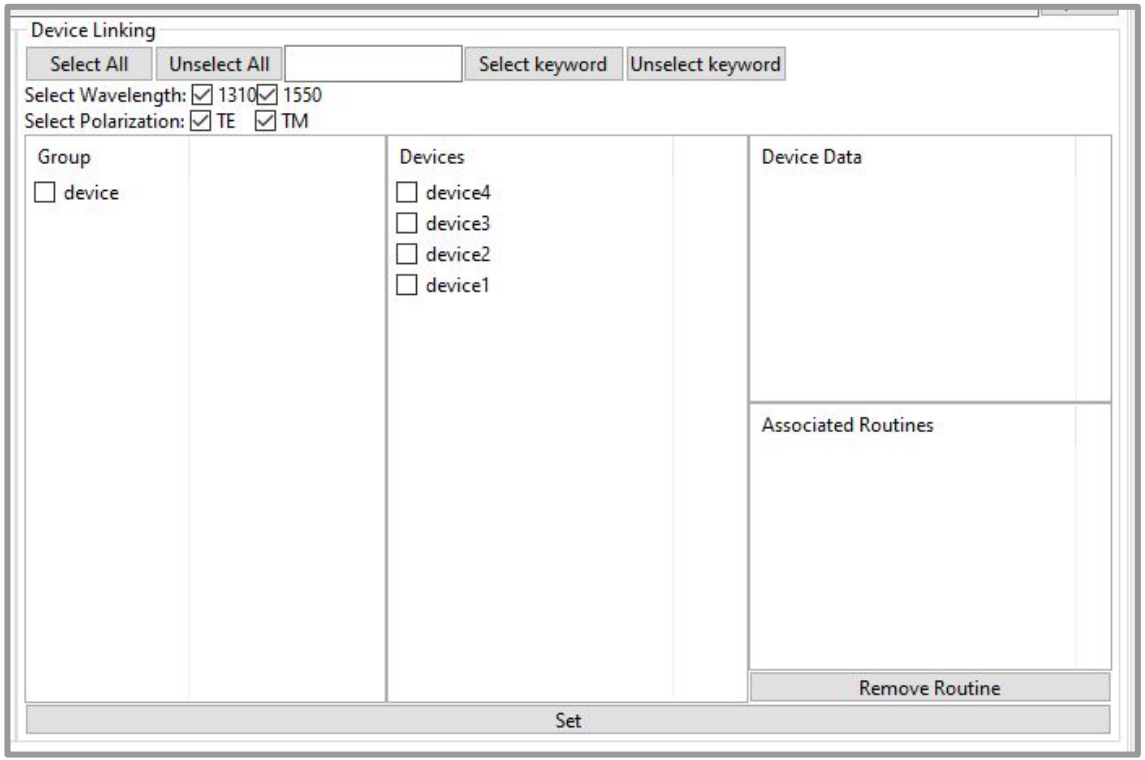
Use this to open folder and choose chip coordinate file

Creating Routines

Begin by selecting the type of routine you wish to perform. Next, give a name to your routine and and populate the parameters of the routine in the given text boxes. Finally, press **Save as Subroutine** and your routine will appear in the Subroutine column of the corresponding routine type.



Setting Routines

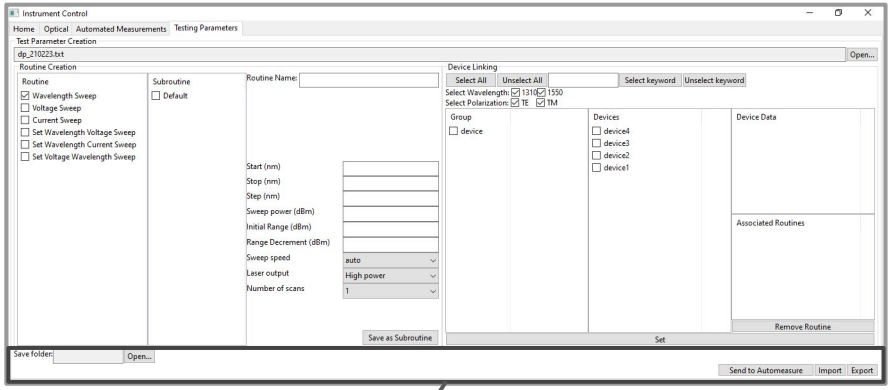


Next, select routines by checking them off and select the devices you'd like to associate with the routines. You can filter the devices by wavelength, polarization or by selecting the device type in the Group column. Click **Set** to link routines to devices. Once a device has associated routines you can view them by selecting the individual device. If you wish to remove a routine from a device you can use the **Remove Routine** button.

Exporting Routines

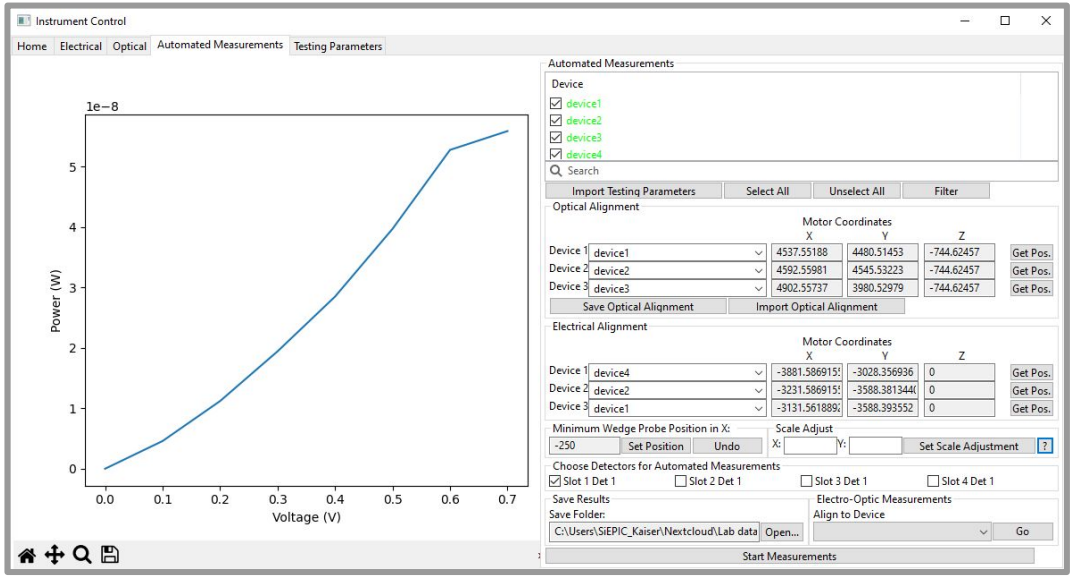
If you wish to save your devices and associated routines to perform measurements later, or just to have a back-up file, you can choose the folder in which to save your file and then press **Export**.

If you wish to perform measurements right away, you can send the routines and devices straight to the Automated Measurements tab with **Send to AutoMeasure**.



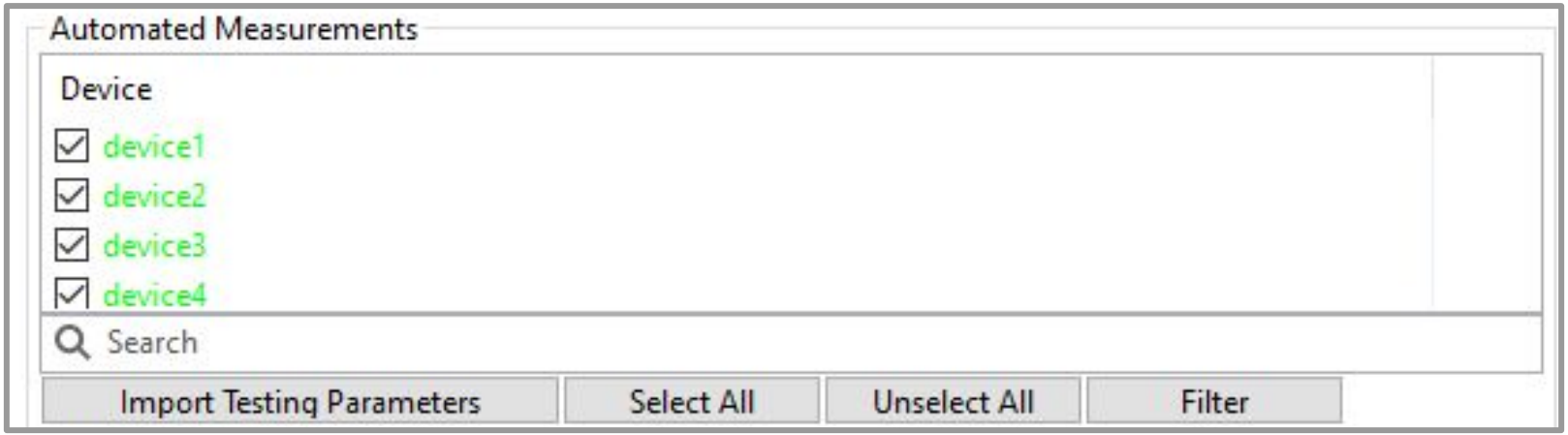
Performing Automated Measurements

To automate measurements of devices with the routines you've set in the Testing Parameters tab you'll need to go to the Automeasure tab. Here you can choose which devices to test and begin your automated measurements.



Importing Testing Parameters

If you created your testing parameters file in a separate instance of PyOptomip than the one currently running, you can import your file using the **Import Testing Parameters** button. All the devices should then populate in the device list at the top of the panel. Devices with associated routines will appear in black, while those with no routines will appear in grey. You can then select the devices you wish to test using the **Select All**, **Unselect All**, or **Filter** buttons.

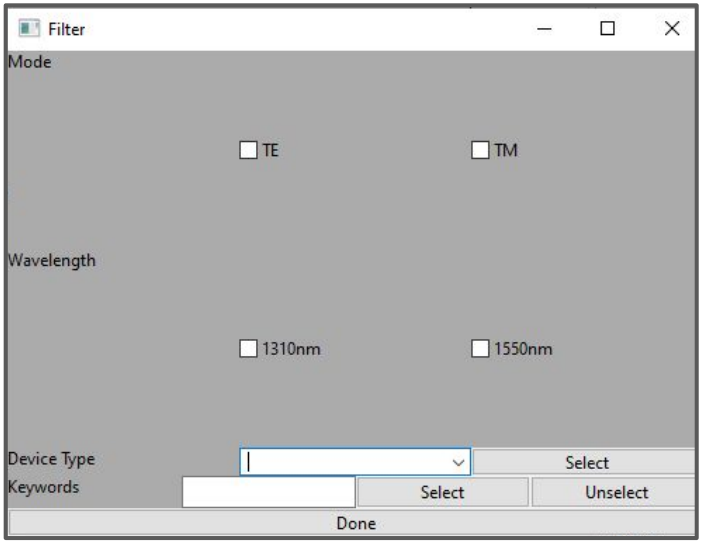


Filtering Devices

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**.

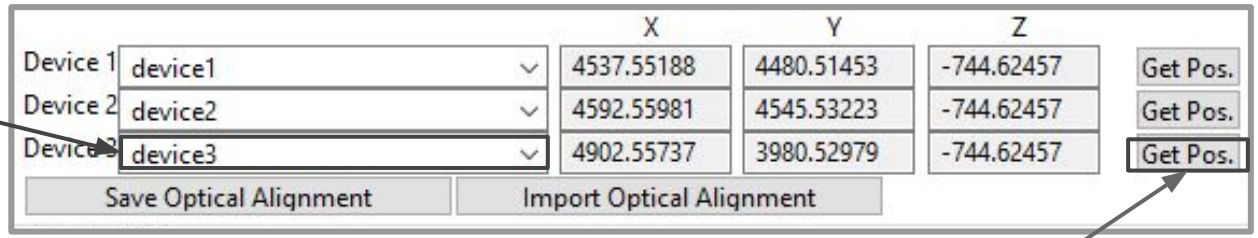
Keywords can be used to select or unselect any device which contains the desired keyword in its ID.



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. For the optical matrix there is the option to save the alignment and import it later.

Choose alignment device from dropdown

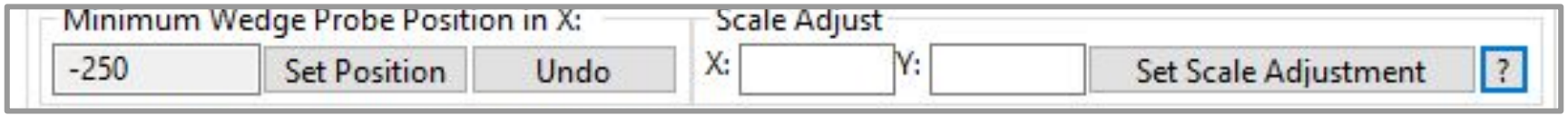


Once the array is aligned click the **Get Pos.** Button

Setting Wedge Probe Limit and Scaling

Move the wedge probe as close as you're comfortable getting it to the fibre array and the press the **Set Position** button to ensure the wedge probe never moves past this position.

If the axes of the wedge probe and stage are slightly misaligned you may need to adjust the x and y scales.

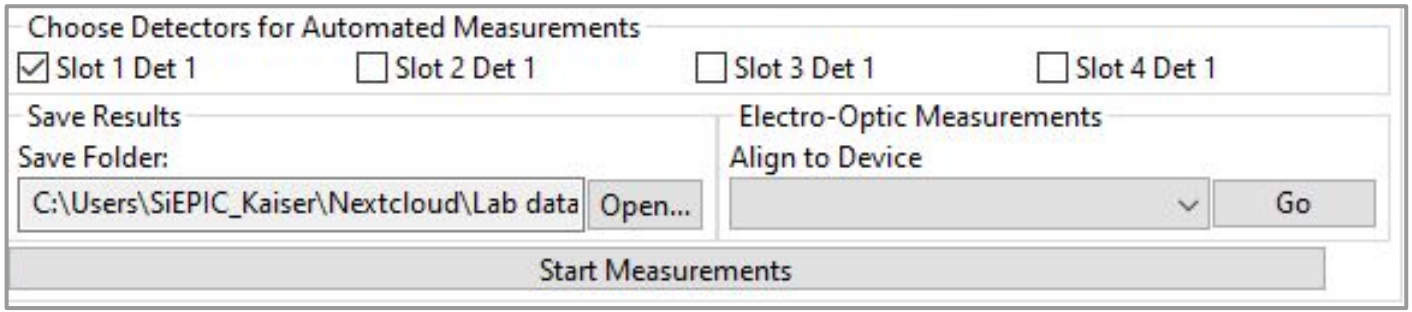


Starting Measurements

After uploading the coordinate file, creating the transform matrices and limiting the wedge probe movement you are ready to begin taking automated measurements.

Select which detectors you wish to use for measurements and then set the folder in which you wish to save your measurement data by clicking **Open...** beside the designated text box.

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 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.