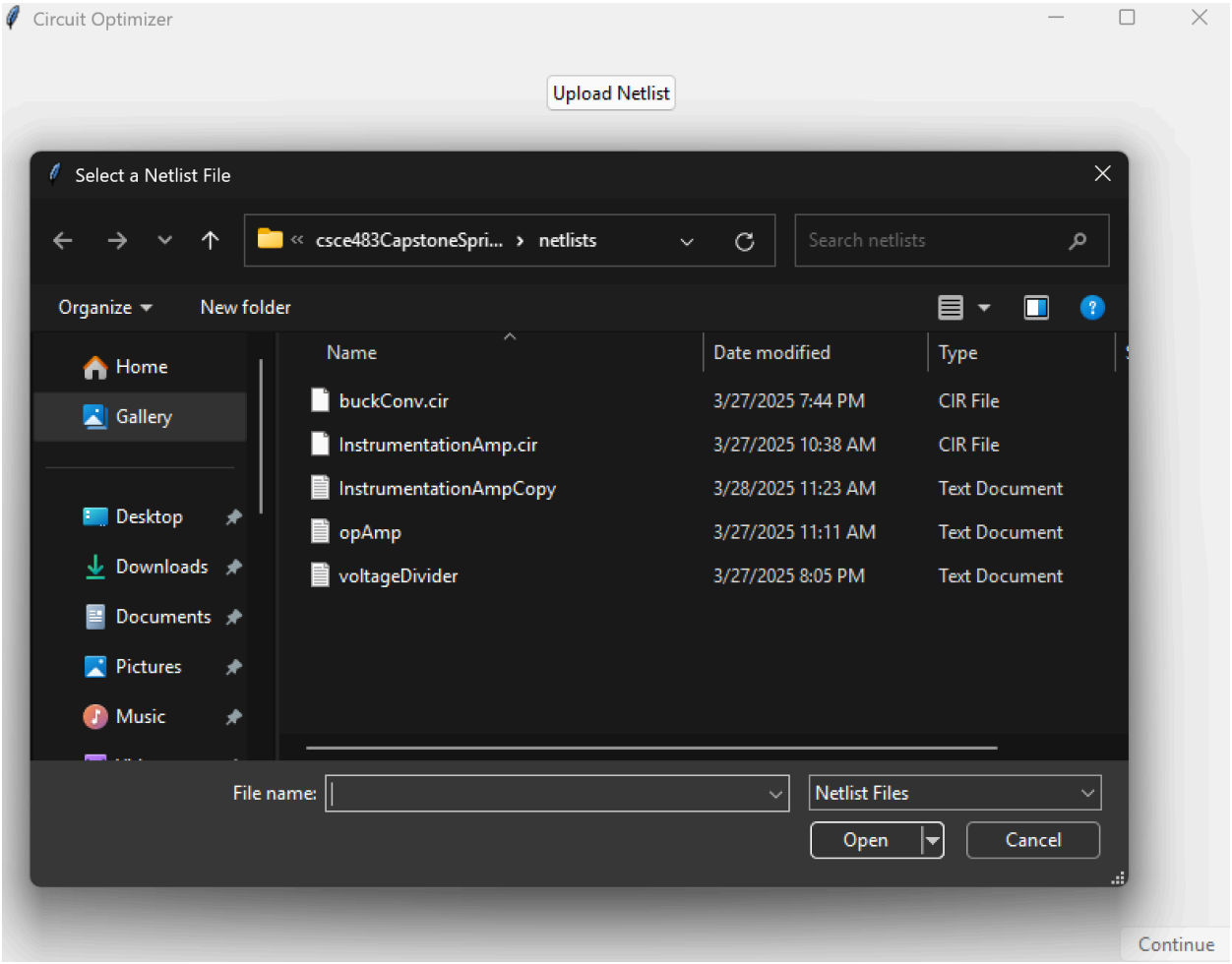
User Manual

Software Installation

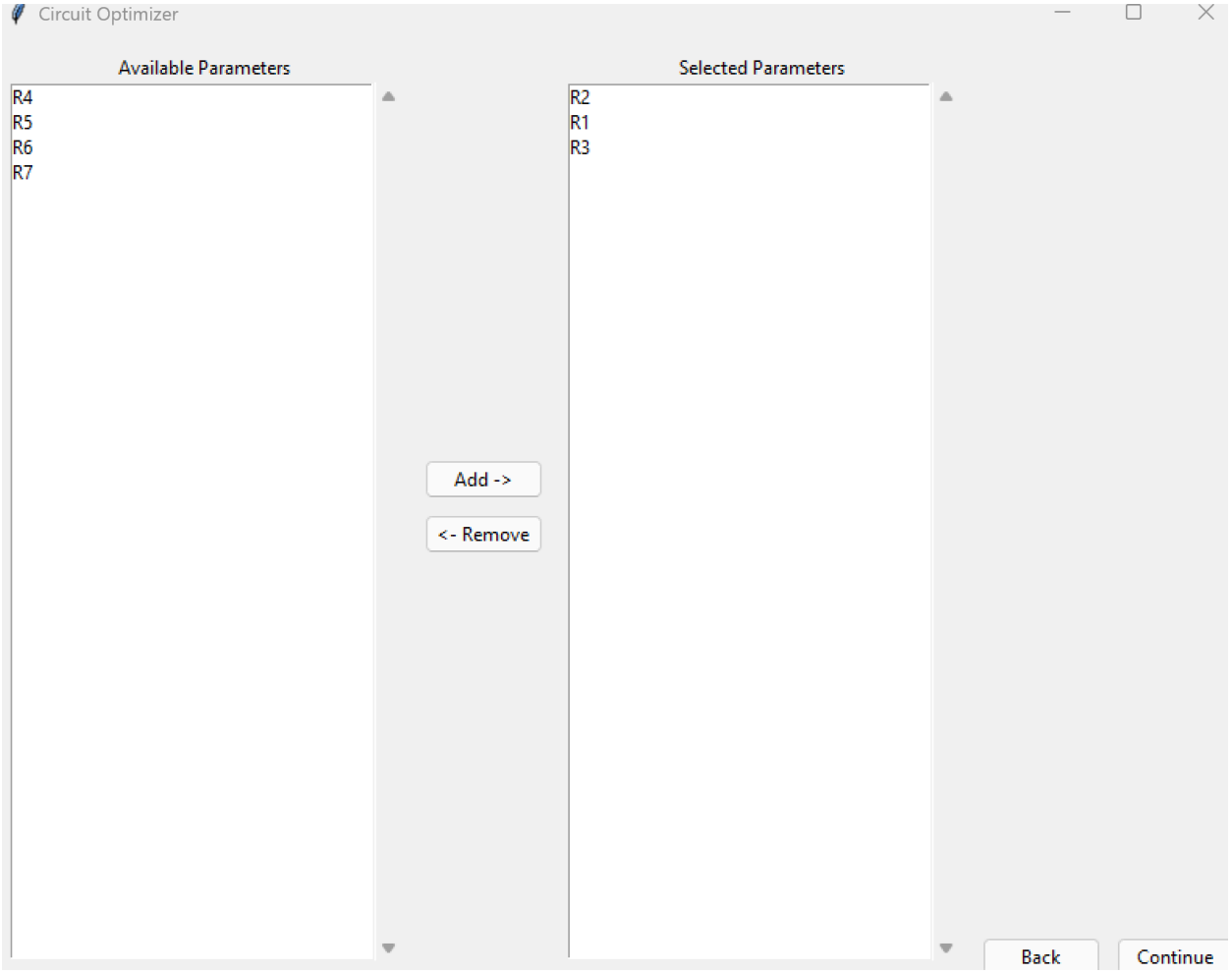
Our project consists of a Windows executable. Simply download the file, unzip the project, and run the executable. For our project, you will need a netlist written in Pspice syntax. We recommend using Qucs for a Windows schematic editor that plays nicely with Xyce. You can install Qucs using their website. In addition, your computer will also need to install Xyce. Xyce can be installed on Sandia's website. Xyce also needs to be in your path variable. Simply add the executable Xyce bin file in the path file. In order to check that Xyce is installed correctly, run the command *xyce -v* in the command line and make sure that command runs.

Operation Manual

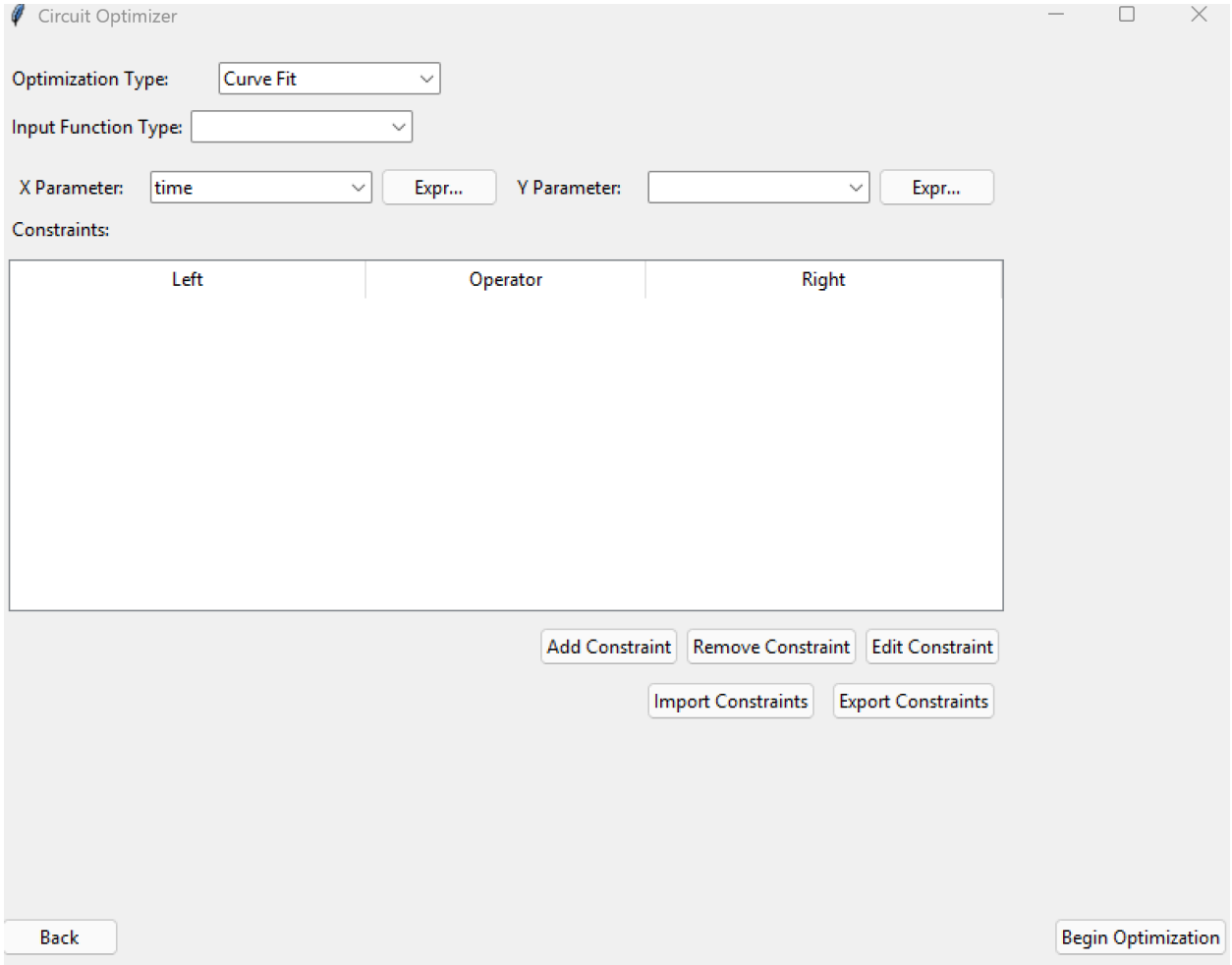
To use our software first upload the netlist. Our software supports Netlists with syntax that is supported by Xyce. For a more detailed description of what features are supported check out the reference guide. Please do not include .tran or .print commands in your netlist as our software will generate them automatically based on the target function input.



Select a valid netlist file and then press open. Afterward, press the continue button.

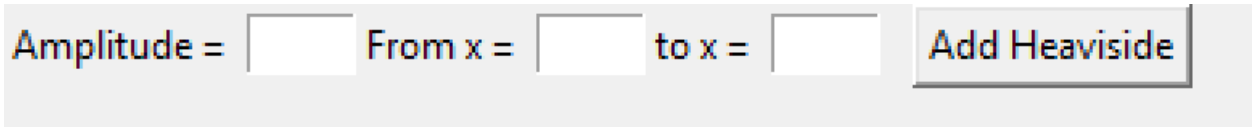


Parameters are R, L, or C values that can be optimized and changed. Select the parameters you want to optimize and move them to the selected parameters section. Parameters that are not selected will be kept at their original value and will not be optimized.

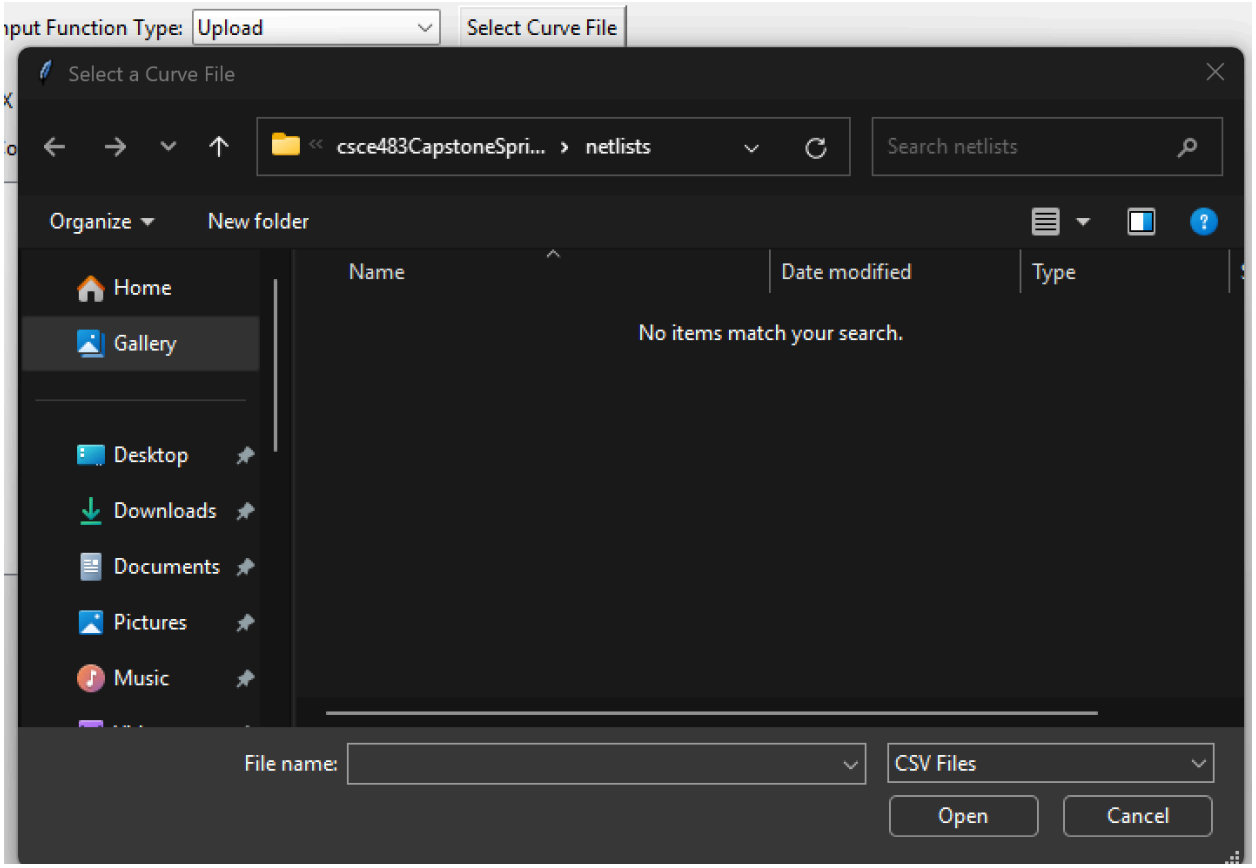


Here you specify the optimization settings. The target function is the type of the function that you want to optimize. The optimizer will try to make your Xyce output equal to the target function. When inputting your target function the specified x values will be the start and stop time of the transient simulation. The units are in seconds. We support three function types: line, heaviside, and custom function.

For the line type input the slope as well as the start and stop value for the line and then press Add Line.

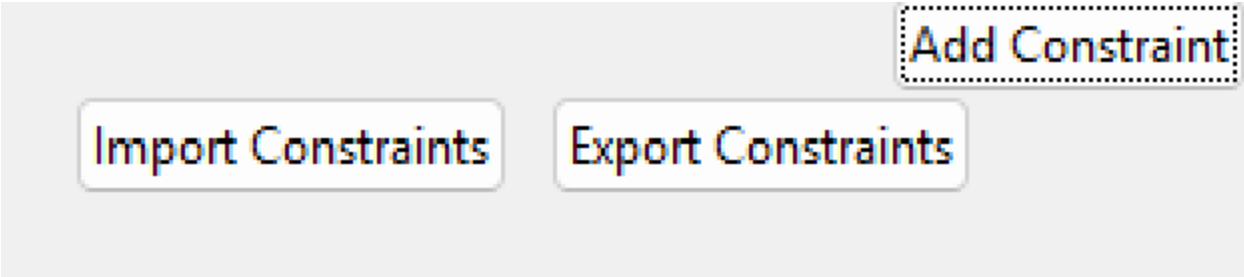


For the Heaviside set the Amplitude, start and stop values, and then press the Add Heaviside button.

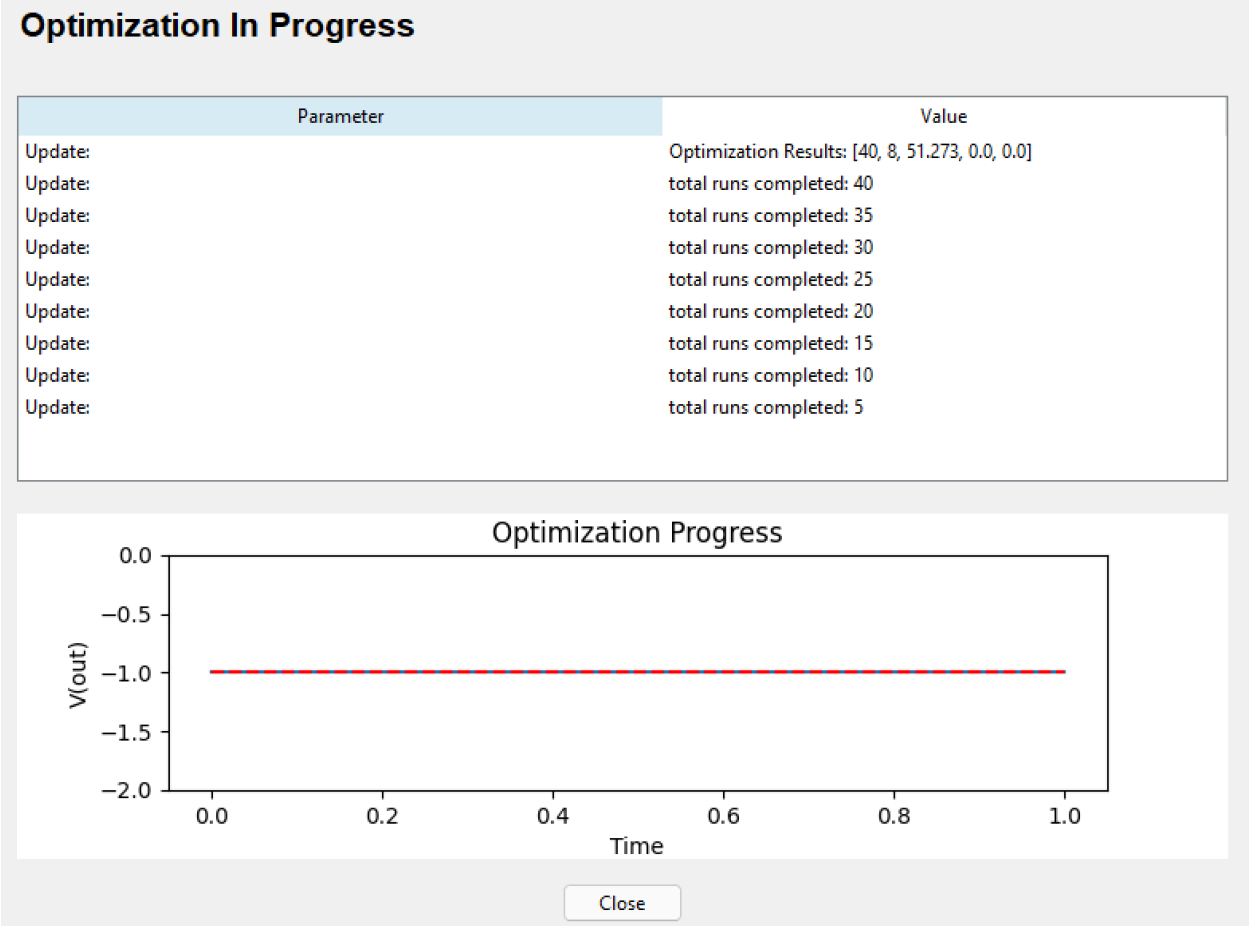


For Custom, press the upload button and then select a CSV file that has a function you want to optimize with. The Y-parameter should be the voltage that you want to optimize the function to. The units for the x axis will be seconds.

Press Add constraint to add a constraint. Constraints allow you to bound certain values for your optimization. On the left side you are allowed to put either an optimization parameter (one of the R, L, or C values you selected in the earlier page) or a voltage node. If you put an optimization parameter <= or >= support only constant numbers on the right side. If you have an optimization parameter (R,L,or C value) and put = you can put mathematical expressions featuring constants as well as additional optimization parameters (R, L, or C values). If you put a voltage node you can only use <= or >=. = Does not work for voltage nodes. In addition the right side can only be a constant value. If you input an invalid constraint the program will error and let you know.

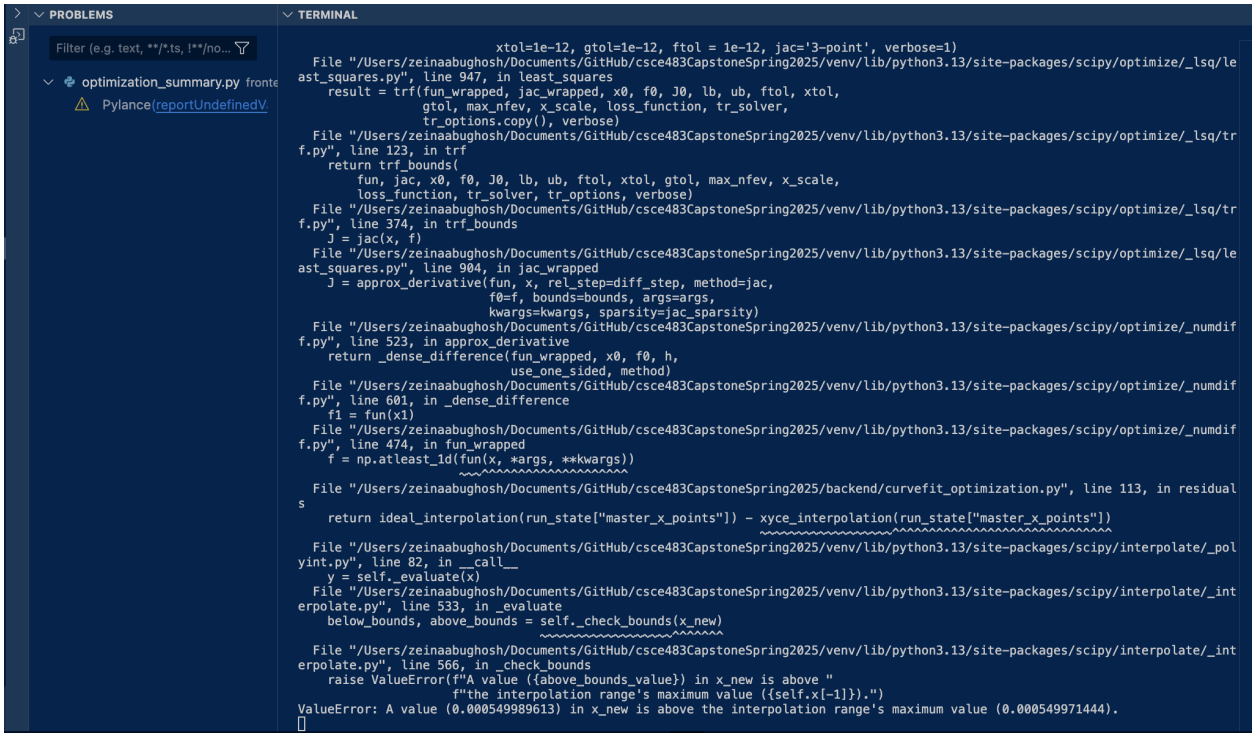


You can also import and export constraints using the buttons below. Imported and exported constraints only work for the same netlist.



Afterward, the optimization will be complete. The optimized netlist will be present as a copy in the original directory where the netlist is located, with copy.txt added to the end of the file.

Common Errors



If you get an error in the command prompt and the optimization summary page that talks about how a value is outside the interpolation range, that means that the Xyce run terminated prematurely. Usually the case of this is that the circuit parameters are too unrealistic to be simulated. The fix for this error is to add constraints to the circuit that make sure that each possible iteration of the circuit can be solved. You can see the netlist values that Xyce errored on by viewing the copy of the netlist file.