Quick Guide to Finding S-Parameters with Ansoft Designer SV

1. Start Ansoft Designer SV
2. In the Project menu, select “insert circuit design” and choose “none” to get a blank schematic.
3. In the left pane, click the components tab for a list of components to place in your schematic. You can find transmission line models in the “Ideal Distributed” section of the “Circuit Elements”.
4. Double click on TRLE (transmission line, electrical length) or TRLK (physical length) to select. Click on the circuit anywhere to place the item.
5. Right click and “Finish” or press Escape to stop placing elements.
6. Double click on placed element to modify its parameters (such as impedance, wavelength, etc.)
7. To place a port, go to the Draw menu and select the “interface port” item.
8. You may have something that looks similar to this:

Now that you have your circuit set up, you want to run a simulation.

9. In the left pane, click the project tab. Select “circuit” right-click on “analysis” and select “add solution setup…”
10. Click next to accept defaults, then set your frequency sweep parameters (give start f, stop f, step f), then “ADD>>”, then OK, then finish.
11. Right click on analysis again, select “analyze” to run simulation at said frequencies.
12. Right click on “results” and “create report…” then pick either rectangular or smith chart, or data table to see the results of your simulation.
13. Add a “trace” for each parameter that you want to see (e.g., S11, S12, S22, S21…V1, I0…)
14. For a 3-port T, use 3 sections of transmission line, and use 3 ports total: