

ATP is installed on the software server remote.mtu.edu and is accessible via Remote Desktop. VPN may be required from off-campus. It may also be installed in computer labs in the department. The family of programs is within the Windows Start tab in a folder called ATP. Run ATPDraw - V7. If you'd like to install it on home or office computer, you must apply for a personal license at <http://eeug-test.hostingkunde.de/index.php/how-to/be-licenced> (Canadian/American Users Group), make sure you satisfy the licensing criteria, and then print and mail a signed copy to the users group. When you have received e-mail confirmation of your license, forward it to your instructor who will provide you with an installation CD. It is not legal to install it onto a non-MTU computer without an approved license agreement.

In the ATP Program group, there should be several options:

- ATPDraw V7: Graphical User Interface for building/editing/running ATP simulations.
- ATPDraw Manual in pdf format
- PlotXY: Basic very user-friendly plotting program (can paste Win Metafiles from here).

In general, everything can be done from within the ATPDraw program. Start **ATPDraw** by double-clicking on its icon in the ATP program group. Then...

- 1) Click on the blank sheet symbol to create a new simulation, or click on the file symbol to open an existing simulation. Edit/Draw the circuit and specify parameters. Use Save-As to save this *.acp file (or just click on diskette symbol to save changes to existing simulation). The file that the circuit diagram and parameters are stored in is referred to as a "project" file. These are kept in c:\atp\atpdraw\project*.acp (older version project files *.adp can also be opened)

- 2) Select ATP | Settings. There are several tabs. The first tab is the most important. There is a HELP button for each tab. Click on it for an explanation of the required data.

Simulation Tab - Choose reasonable values for Delta T (the integration timestep size) and Tmax (the length of the simulation). If Δt is smaller than needed, the simulation will take longer to run and you'll create huge bloated output files. If Δt is too big, this could result in large integration errors and incorrect results. Make sure that Δt is at least an order of magnitude smaller than the smallest time constant τ and/or the period of the highest frequency. Xopt is zero if you want to specify inductances in units of mH, or 60 if you want to use Ohms at 60 Hz. Copt is zero if you want to specify capacitances in units of μF or 60 if you want to use M-Ohms at 60 Hz.

- 3) To run the simulation and see the results:

- a) Select ATP | Run ATP (upper one in the list) - This creates ASCII/text input data file h:\atp\atpdraw\atp*.atp and then runs the simulation. Two output files will be created: h:\atp\atpdraw\atp*.lis is a text file containing a log of the simulation (i.e. a record of how the input file was parsed and interpreted and a record of how the simulation proceeded - this file can be referred to in case of simulation input errors), and c:\atp\atpdraw\atp*.pl4 which is a binary data file containing simulated waveforms. (Note: local installations use c: instead of h:)

- 4) When the simulation is done running, go to ATP | Plot to run the **PlotXY** program. PlotXY will automatically load the *.pl4 file that was just created. You can click on "LOAD" to manually select a *.pl4 file or add'l *.pl4 files that you desire to plot or overplot. Click on the Voltage or Current waveforms you wish to plot and the click on PLOT to display them. Experiment with turning the grid off and on, use the tracking cursor, zoom in by closing a window around the desired part of the waveform, etc. When you get the plot you want, clicking on "COPY" puts a Windows metafile into clipboard, allowing you to easily paste the waveform file into a Word document.