Using the MicroSim 8 Simulation Package

1. Every time you start your work, first run the **Design Manager** (Start menu > Wszystkie programy > MicroSim 8).
2. If you are beginning work on a **new design**:
   a) in Design Manager, create a new design folder:
      - choose File > New Workspace from the menu;
      - in the Location field enter—or choose using the "..." button—the path where the design is to be created (conforming to laboratory regulations);
      - click Create.

   b) run the Schematics application using an icon from the vertical toolbar on the left-hand side of the Design Manager window;

   c) each new schematic belonging to a project should be immediately saved to the previously created design folder; if a schematic has been successfully saved, its name shows up on the file list in the Design Manager window.
3. If you are continuing work on an **existing design**:
   a) in Design Manager, open the design folder selecting File > Open Workspace from the menu;

   b) open the schematic you wish to edit:
      - expand the file list in the Design Manager window and double click the name with .sch extension; or
      - run the Schematics application kusing an icon from the vertical toolbar on the left-hand side of the Design Manager window, then open the schematic selecting File > Open from the menu or using the appropriate icon.
4. Useful **keyboard shortcuts** in Schematics are: *Ctrl+G* – insert new element after list lookup (right mouse button exits), *Ctrl+P* – insert new element from the recently used list, *Ctrl+R* – rotate, *Ctrl+F* – flip, *Ctrl+W* – draw wires (double click or Esc exits).
5. **Simulation type and parameters** are defined in the Analysis Setup dialog (Analysis > Setup from the menu or the appropriate icon). The most frequently used analyses are: *DC Sweep* – DC component simulation for different values of a variable source or of a parameter of one of the standard elements; *Transient* – transient state simulation for elapsing time; *AC Sweep* – AC component simulation for variable frequency; *Parametric* – add-on analysis that enables to vary an additional parameter (apart from the main analysis variable).
6. The **PSpice A/D** simulator is started automatically from the Schematics application after choosing Analysis > Simulate from the menu or clicking the appropriate icon, or hitting F11. In the case of a textual circuit description, the PSpice A/D application should be started using an appropriate icon in the vertical toolbar of Design Manager, then opening the circuit description file choosing File > Open from the menu; if an already simulated circuit is to be reopened, it may be chosen in the bottom of the File menu.
7. At simulation stage, **errors** may occur that are indicated in PSpice A/D and Message Viewer windows. In the majority of cases the cause of an error may be found in the following manner:
   a) read and understand the error message displayed in Message Viewer window;

   b) locate the cause in the schematic by double-clicking the appropriate error message line in Message Viewer window;

   c) locate the cause in the textual circuit description included in the output file by looking for an "`ERROR`" message together with the "`---$`" marker that shows the exact location of the cause;

d) the textual circuit description may be opened by selecting File > Examine Output from the menu in PSpice A/D window or by selecting Analysis > Examine Output from the menu in Schematics.

8. After a simulation is successfully finished in the PSpice A/D window, **Probe** application is automatically started for circuits entered by means of a schematic. For circuits with only a textual description this application must be started with File > Run Probe from the PSpice A/D menu.

9. **Waveforms to be displayed** in Probe may be defined as follows:

   a) before running the simulation, set appropriate markers in Probe:

      - potential marker – V icon or Markers > Mark Voltage/Level from the menu,
      - voltage marker – Markers > Mark Voltage Differential from the menu (a first click sets the positive, a second one, the negative marker),
      - current marker – I icon or Markers > Mark Current into Pin from the menu;

   b) set markers after simulation is run, then waveforms are added to a current plot and a current axis and only appear after you exit the marker mode in Schematics (after placing markers hit Esc);

   c) in Probe using the Add Trace icon or Trace > Add from the menu, or Ins key, and choosing a quantity from the list (it is favourable to first hide all the model sub-circuit elements by unchecking Subcircuit Nodes) or entering a formula with the keyboard.

   The current marker only works for built-in Spice simulator elements; it does not work for elements modelled as sub-circuits which is frequently the case for power devices. For such elements the marker should be placed on another element leading the same current (if there is no such, insert a small resistor or a zero-value voltage source).

10. After a successive simulation, you may restore **plot settings** in Probe to what they were at the end of the previous simulation, by hitting F12. Any plot settings for a given set of plots can be saved and then restored using Tools > Display Control from the menu.

11. **Plot scale** in Probe is automatically adjusted so that the waveform with the greatest amplitude fits in the plot. In this scale waveforms with a smaller amplitude might not be visible. In order to observe them, the Y axis scale must be changed manually or they must be plotted in a separate plot (Plot > Add Plot from the menu) or in the same plot but a second Y axis (Plot > Add Y Axis from the menu).

12. If it is necessary to **reverse a waveform [**change its sign, e.g. `-V(3)`], the waveform symbol should always be put between parentheses together with the minus sign, e.g. `(-V(3))`.

13. **Cut**, **Copy** and **Paste** functions still work in Probe. To apply them to a waveform, click on its formula under the plot to mark it in red. A waveform is pasted to a current plot and a current axis; this way, waveforms may be moved between axes or plots.

14. New waveforms should be added an existing waveforms should be pasted only when all plots are in autoscale mode (View > Fit from the menu or the View Fit icon, or Ctrl+N). Otherwise the application may **become unstable** and close.

15. **Closing once opened** PSpice A/D, Probe, Message Viewer windows is senseless and considerably slows down the work with the package.

16. In the case of an application **not starting up**, it is best to close all the package applications including the Design Manager an start them again in a proper sequence. If this does not solve the problem, check if the settings server (PUM1 computer, stand N1) is on, then log out and log on again.