

Catalogue of important PUFF21 Keystrokes and Commands (using PUFF under Windows7)

Remark: only use the content in the brackets as keystrokes

**Information: the catalogue consists of the following categories:
Layout / Part / PUFF / Simulation / Smith chart**

Subject	Layout	Command / Keystroke
Layout: change distance of port Connectors		Press F4 , move the cursor to the line „c“ and enter the new distance
Layout: change height of the PCB Connectors		Press F4 , move the cursor to the line „h“ and enter the new height
Layout: change dielectric constant of the PCB		Press F4 , move the cursor to the line „er“ and enter the new dielectric constant value
Layout: change metal thickness of the PCB metalization		<p>Use a text editor to open the „setup.puf“ file or the *.puf file of your actual project. Search the line for “mt” and enter the new value</p> <p>Example for mt = 35µm:</p> <p>mt 35Um {metal thickness, use Um for micrometers}</p>
Layout: change side length of the PCB		Press F4 , move the cursor to the line „s“ and enter the new value (remark: the PCB is square)
Layout: change surface roughness of the PCB metalization		<p>Use a text editor to open the „setup.puf“ file or the *.puf file of your actual project. Search the line for “Sr” and type in the new value.</p> <p>Example for a surface roughness of 2µm:</p> <p>Sr 2.000Um {metal roughness, use Um for Micrometers}</p>
Layout: create a connection between a point in the Layout and a port pin		Press F1 , then move the cursor with key_up / down / right or left to the point in the circuit and type in the port number. Confirm the procedure with < ENTER >
Layout: delete the connection between a port pin and a point in the layout		Press F1 , then move the cursor to the circuit point which is connected to a port. Press < Shift > , hold the Shift key and type the port number to delete the connection
Layout: delete the complete circuit in the layout		Press F1 , then type <CTRL + e>

Layout: delete part

Press **F1** and move the cursor to the connection of the part to be deleted in the layout (using the up / down / left or right keys).
Then Press < Shift > followed by the key which moves the cursor into the direction of the part

Layout: grounding of a circuit point

Press **F1** and move the cursor to the point in the layout (using key_up / down / left or right). **Then type < = >**

Layout: jump to the next node in the circuit

Type <CTRL + n> (when in F1)

Layout: Manhattan presentation for transmission lines

Press **F4** and **use the < TAB > key to toggle between Microstrip, Stripline and Manhattan**

„**Manhattan**“ means that the length of a transmission line used has the same value in the layout as all lumped parts (= 10% of board side “s”) and not the correct length as entered in F3. This gives a very compact layout and avoids the message “the part lies outside the board”

Layout: place part

Press **F1** and **move the cursor** to the actual point in the layout (using key_up / down / left or right).
Now select the desired part by typing the letter of the part in the F3 part list.
Then use key_up / down / left or right to place the part in the layout

Subject **Part**

Command / Keystroke

Part: coupled lines

Press **F3**. Enter “**clines**” or “**cl**” for a perfect line model, followed by the value of the “**Even**” impedance. **If the line impedance is equal to the system impedance of 50Ω, PUFF will automatically calculate the “Odd” impedance by using the formula**

$$Z^2 = Z_{\text{even}} \times Z_{\text{odd}}$$

and add the calculation result to the line properties.

If line and system impedance are not equal, the ODD impedance must be entered after the EVEN impedance, followed by the line length.

If you want an “advanced modelling” which includes all undesired effects, add an exclamation mark after “cl” and enter “cl!”

Part: Entering length in degrees for a transmission line

Press **F3**, then move the cursor to the line in the part list for a transmission line

After “tl” and the characteristic impedance type in the electrical length value followed by < **ALT d** >.

Example: tl 50Ω 90°

Part: enter new part incl. properties

Press **F3** and move the cursor with key_up / down to an empty line in the part list. Then type „lumped“ (the first letter “l” does also the job...) followed by the part value.

You may enter one or more parts in a line, using a series or a parallel connection.

For a resistor use the unit „Ω“ or „kΩ“ etc. OR use a value which is normalized to the characteristic impedance (e. g.: “1z” or “0.3z”)

For capacitors and inductors there are 3 different options:

Use either the correct unit (e. g.: pF, μH etc.) OR

Use the imaginary form (e. g.: +j10) OR

Use the imaginary normalized form (e. g.: +j5z)

You are also allowed to use complex series connections like “2+j4-j3z”

or parallel connections like 5pF||1μH||10k

(Type < ALT p > for “parallel”)

Part: equivalent series circuit for a given S11 or S22 value

Press **F2** and use **page_up** or **page_down** to change to the **desired frequency**. Toggle the Tab key to use the „**impedance form of the Smith-chart**“ (= point for Infinite on the right hand side).

Now use the cursor to **select an S-Parameter** (= S11 or S22). Type < = > and you get the part values (R, L and C) of the equivalent series circuit of S11 or S22 for the selected frequency in the Plot Window.

Part: equivalent parallel circuit for a given S11 or S22 value

Press **F2** and use **page_up** or **page_down** to change to the **desired frequency**. Toggle the Tab key to use the „**admittance form of the Smith-chart**“ (= point for Infinite on the left hand side).

Now use the cursor to **select an S-Parameter** (= S11 or S22). **Type < = >** and you get the part values (R, L and C) of the equivalent parallel circuit of S11 or S22 for the selected frequency in the Plot Window.

Part: extend part list

Press F3 and hit the Tab key for more parts to be entered

Part: „μ“ (e. g. for Micro Henry)

Type **< ALT m >**, followed by the unit.

Example for an inductor:
lumped 35μH

Part: microstrip option

Press F4 and use the **Tab key** to toggle Microstrip, Stripline- and Manhattan.

„Microstrip“ starts a calculation for a transmission line with a small line on the upper side of the substrate and an infinite ground plane on the lower substrate side.

The necessary entry in F3 has the form

tl characteristic impedance line length

The length can either be given in degrees (type < ALT d >) or in Millimeters

Example
tl 50Ω 90°

If you want to know the line's mechanical data (= width, length) move the cursor to this line in F3 and type **< = >**.

Caution:

„**tl**“ **simulates an ideal line without any losses.**

If you want to use a realistic model (including all losses, the dispersion etc.) then add **an exclamation mark (!)** to „**tl**“

Example:

tl! 50Ω 90°

Part: microstrip line with dispersion and attenuation

Type **tl**! (or only **tl!**) instead of „tl“ or “tline”.

Caution:

Before starting PUFF, open the setup.puf file or the PUFF-file of your actual project with a text editor.

Now enter the values for the PCB.

Example for RO4003 / 32mil at 2GHz:

er	3.550	{dielectric constant. er>0}
h	0.813 mm	{dielectric thickness. h>0}
s	100.000 mm	{circuit-board side length. s>0}
c	30.000 mm	{connector separation. c>=0}
r	0.010 mm	{circuit resolution, r>0, use Um for micrometers}
a	0.000 mm	{artwork width correction.}
mt	0.035 mm	{metal thickness, use Um for micrometers.}
sr	2.000 Um	{metal surface roughness, use Um for micrometers.}
lt	2.0E-0003	{dielectric loss tangent.}
cd	5.8E+0007	{conductivity of metal in mhos/meter.}

Part: Ohm („Ω“)

Type **<Alt + o>** after the part value in F3

Part: PARALLEL sign for lumped part parallel combinations

Type **<Alt + p>** (when in F3 and in the shown line of the part combination)

Example:

lumped 50Ω||1nH||10pF

Part: show the dimensions of a transmission line

Press **F3** and move the cursor to the line in the part list. Then type **< = >** and the mechanical dimensions will appear in the Dialogue Box.

Caution: for this operation there must not be an exclamation mark after “tl”

Part: stripline option

Press **F4** and type the Tab key to toggle **Microstrip, Stripline and Manhattan. Select “Stripline”**.

Now a transmission line with an “embedded central conductor in the centre of a substrate is used. On the upper and on the lower side of the PCB an “infinite copper ground plane” will be found.

Subject	Command / Keystroke
PUFF	
PUFF: end of your daily work	Press <ESC> twice
PUFF: help	Press <F10>
PUFF: loading a PUFF file	<p>Press <CTRL + r> (when in F3). Enter the file name, then type <ENTER>.</p> <p>(Caution: the file must always have the extension *.puf and must be saved in the PUFF folder).</p> <p>But it is not necessary to enter the „*.puf“- extension, the file name will do the job.</p>
PUFF: saving a PUFF file	Type <CTRL + s> (when in F2), enter the file name and type <ENTER>
PUFF: starting	<p>Start Windows 7, followed by the DOSShell. Click on the PUFF-Icon and the program will start. Confirm the start presentation with < ENTER > and use < ALT + ENTER > to get the full screen.</p> <p>Remark: The setup file is automatically loaded when starting the program</p>

Subject	Simulation	Command / Keystroke
Simulation: change size of screen		Type <Alt + ENTER> to switch between full screen and Window presentation of PUFF
Simulation: change of actual frequency		Press the page-up or page-down key and hold it down. Now the cursor runs into the selected direction and the actual frequency is indicated
Simulation: change start frequency		Press F2 and use key_up or key_down to reach the entry field for the start frequency on the horizontal axis of the plot diagram. Enter the new start frequency value and type < q > for a new simulation
Simulation: change stop frequency		Press F2 and use key_up or key_down to reach the entry field for the stop frequency on the horizontal axis of the plot diagram. Enter the new stop frequency value and type < q > for a new simulation
Simulation: change numbers of plot points		Press F2 and move the cursor to line " points ". Enter the number of simulated and plotted points (max. 1000)
Simulation: component - sweep		<p>At a selected frequency the value of a part in the circuit can be swept to find the optimum value. Frequency and values of other parts are held constant during this simulation.</p> <p>Start with Pressing F4 to enter the selected frequency fd for the component sweep</p> <p>Then go to F3 and move the cursor to the line with the part to be optimized. There add a question mark to the part value.</p> <p>Example: lumped ?10pF or tline 50 ?90⁰</p> <p>Press F2 and enter start and stop value for the part on the horizontal diagram axis.</p> <p>Example: 5pF.....15pF for a capacitor or 70⁰.....110⁰ for a transmission line</p> <p>Then type < p> or <q> to start the simulation</p>
Simulation: impulse response		Type < i > (when in F2)
Simulation: step response		Type < s > (when in F2)
Simulation: Printing of simulation		Type <print screen> and use "Irfan View" to work with the screenshot. Now you can cut out, copy, past, print, save ...
Simulation: start simulation and plotting		Type < p > (when in F2) and you can see how your PC is working and calculating. Type < q > (when in F2) to get the readout after the complete calculation. This reduces the calculation time to 10....20%

Simulation: start plotting after a modification and show the old and the new results

Type <CTRL p> or < CTRL q >

Simulation: quickplot

Type < q > or < CTRL q >.
<CTRL q> shows the old and the new simulation result

Simulation: equivalent series circuit for a given S11 or S22 value

Press **F2** and use **page_up** or **page_down** to change to the **desired frequency**. Toggle the Tab key to use the „**impedance form of the Smith-chart**“ (= point for Infinite on the right hand side).
Now move the cursor to **select an S-Parameter** (= S11 or S22). Type < = > and you get the part values (R, L and C) of the equivalent series circuit of S11 or S22 for the selected frequency in the Plot Window.

Simulation: equivalent parallel circuit for a given S11 or S22 value

Press **F2** and use **page_up** or **page_down** to change to the **desired frequency**. Toggle the Tab key to use the „**admittance form of the Smith-chart**“ (= point for Infinite on the left hand side).
Now move the cursor to **select an S-Parameter** (= S11 or S22). **Type < = >** and you get the part values (R, L and C) of the equivalent parallel circuit of S11 or S22 for the selected frequency in the Plot Window.

Simulation: add an S-Parameter to be plotted

Press F2 and move the cursor to an **empty s-parameter line in left upper corner of the screen**. There enter „**S**“ followed by the index of the new parameter (e.g.: S13)

Subject Smith chart	Command / Keystroke
Smith chart: change radius	Move the cursor to the line „ Smith radius “, when in F2. Enter the desired radius (= a value between Zero and 100)
Smith chart: change frequency of markers	Use “ page up ” or “ page down ” (when in F2) for “Frequency up” or “Frequency down”. The actual marker frequency is indicated in the dialogue window
Smith chart: full screen presentation	Type < Alt + s > (when in F2). The same command is used to switch back. Remark: use < Alt + Enter > to switch the complete screen between „full screen“ and „Windows“ presentation.
Smith chart: printing	Press tab < PRINT SCREEN > (when in F2) and use the IRFAN VIEW software to work with the screenshot (The software can be found on the CD). Cut out, save, print...do everything what you want. Remark: it recommended that you first change to the full screen presentation of the Smith chart with < ALT + S > to get a better resolution.
Smith chart: toggle between reactance and admittance presentation	Press Tab key (when in F2)