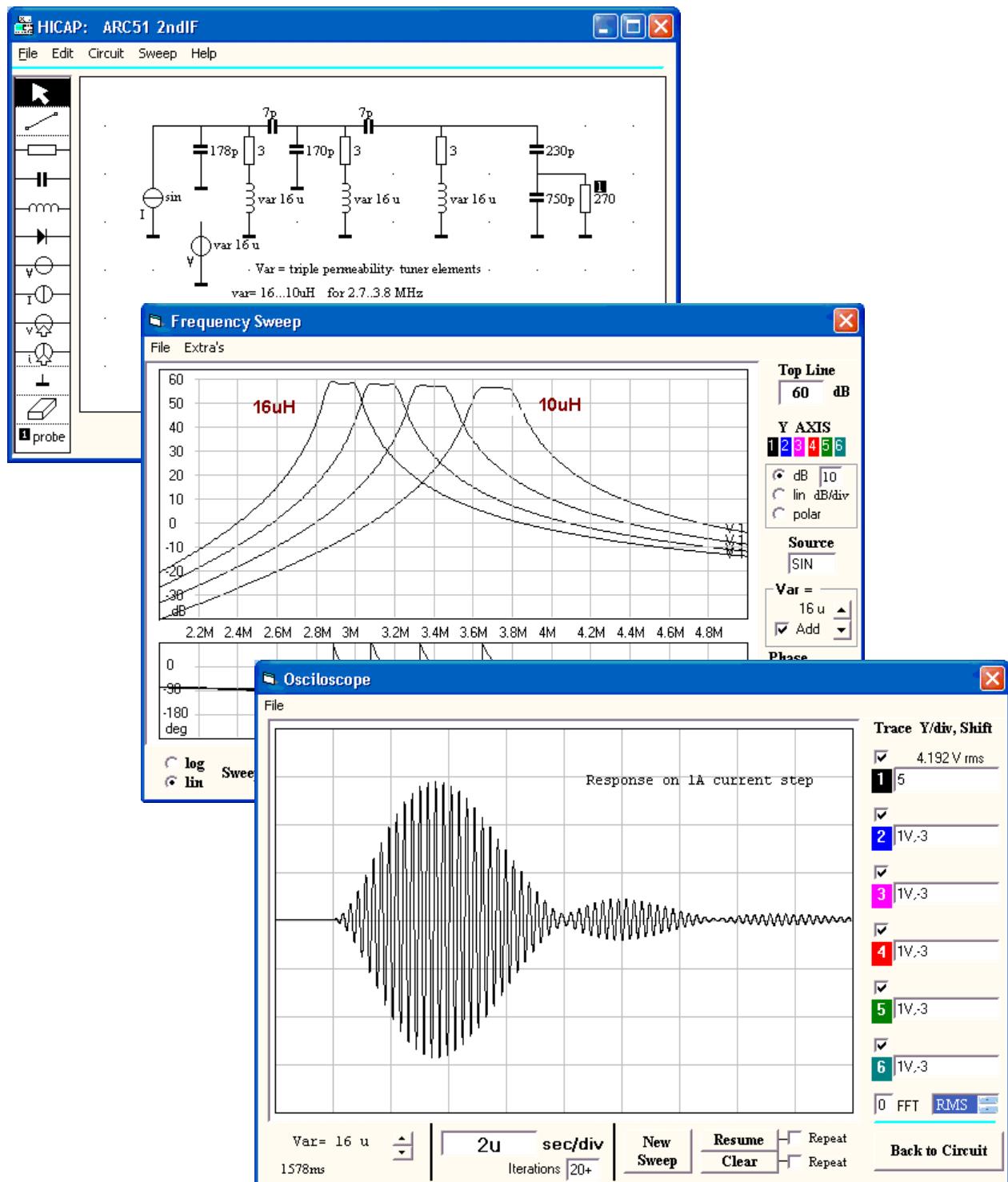


HICAP

Highly interactive Circuit Analysis Program
Version 1.9 august 2009 By K.Bouwknegt



Contents

1. What is Hicap
2. Types of Circuits , Size
3. Menu's
4. Numbers , Values, Polarities
5. Draw a circuit
 - Lines , Ground
 - Resistors
 - Capacitors
 - Inductors
 - Diode, Switch
 - Thyristor
 - Voltage and Current Controlled V or I Sources
 - V or I Generators (independent sources)
 - Subcircuits
 - Subcircuit Library
- and finally, define the ins and outs :
 - Inputs
 - Outputs (Probes)
- 6 Time Sweep
 - Basic oscilloscope settings
 - DMM
 - FFT
- 7 Frequency Sweep
 - Basic frequency sweep settings
 - Menu Extra's
- 8 Preferences
- 9 Shortcuts and mouse buttons
- 10 Clipboard use
- 11 Variable components
- 12 Calculate L or C for resonance
- 13 Errors
- 14 Version record

What is Hicap ?

Hicap is an electronic circuit analysis program for analogue, mainly linear circuits.

Hicap is fast, allowing real-time interaction with the circuit as done with real analogue circuits.

The analysis is optimised for linear circuit elements like resistors, capacitors and inductors.

Diodes are handled as resistors with two different values, depending on the voltage polarity across them.

Hicap has 3 main windows:

- The **Circuit Entry** window,
featuring 8 basic components
subcircuits (custom made or from a library)
and several waveform generators to drive the circuit
- The **Time simulation** window (the oscilloscope)
featuring in-sweep circuit modification, auto retrace and
post sweep calculations like rms etc. values or Fourier spectrum .
- The **Frequency analysis** window to show Bode and Polar plots
with on-the-fly cursor tip display of frequency, gain, magnitude and phase
while pointing at or moving over the plot, and a sub window to
show the transfer function formula for your circuit as a list of its numerator / denominator
coefficients.

Hicap has some additions specifically intended for Power Electronics. One is the 12 pulse gate control, which can be configured to drive up to 12 thyristors (in phase control mode) or IGBT's in pulselwidth control mode as used in dephased choppers.

Another typical feature for power electronics is the on-line rms. measurement of all traces, and FFT to find harmonics of waveforms.

Version

This help file was made for version 1.75, released 31-08-2003

Updated for version 1.9, released 03-03-2005

Author

Hicap was created in 1973 by Koos Bouwknegt for use on a mainframe. It was converted later to run on early desktop computers like the hp9845, and still later, on the Macintosh.

The program was rewritten in Visual Basic 5.0 for Windows 95/98/NT in 2000-2003

Version 1.9 was compiled with Visual Basic 6.0 for Windows 95/98/NT/ME/XP in 2005

You can reach me at :

a.k.bouwknegt@home.nl

Types of Circuits

Hicap is a linear circuit simulator. Linear circuits are built from linear components. With twice the input signal, all signals in the circuit double.

However, several non-linear components can be used:

- The diode, thyristor or switch. All are simulated by a resistor with either a low (R_{on}) value or a high value (R_{off})
- Capacitors can be limited in voltage
- Inductors can change their value above a certain current in either direction
- Dependant generators can produce a voltage or current which is a non-linear function of another voltage or current in the circuit.

Nonlinearities are not supported in a frequency sweep. Diodes etc. are simply "on" and non-linear-controlled sources become uncontrolled sources.

Nonlinearities in the time sweep reduce the accuracy of the output. Often you must use a much smaller timestep to get stable results.

The free version is limited to :

99 Components (resistors, capacitors, inductors, dependant sources, generators,
of which :

- 15 Capacitors + Inductors
- 14 diodes, switches or thyristors
- 4 generators
- 6 probes

Installation

PC requirements

Hic平 requires a PC running Windows 95, 98 ,Me, NT4 or XP

Screen size: any, 800 x 600 is fine. The component symbols look best with "small fonts"

Memory requirements are modest: 2 Mb disk space and 1 Mb Ram is sufficient.

Only the left mouse-button is used (This was originally Mac software)

Installation

Hic平 comes on two disks, or as a 2 Mb zipped file for electronic distribution. Insert Disk 1, and run the setup.exe program. This will create a new directory in the program group, and installs all needed files there.

Uninstall is supported as usual.

After the first run of Hic平, all circuit files (with .ckt extension) will get special icons, and can be double-clicked to open that file.

Circuit size

The amount of memory used is independent of the size of the circuit. The free version is limited to 99 components, 14 diodes, and a maximum of 6 probes.

Menu's in the circuit window

File menu

New	Remove existing circuit, start with empty circuit and default instrument settings Probe settings and generator settings stay where they were.
Save as ...	Save circuit on disk with specified (new) name
Save	Save circuit on disk, overwriting the last save.
Load ...	Get circuit from disk.
Print	Print circuit and last plot on the current network printer.
File names..	List of your most recent used circuits. A click will get one.
Quit	Stops the program. If the circuit is not saved, you are invited to do so. The closing cross of the circuit window has the same effect.

Edit menu

Undo	Replaces the last erased component or group of components, or subcircuit Undoes changes in the subcircuit symbol when you were editing it.
Cut	Removes the selected part of the circuit and puts it on the clipboard
Copy Circuit	Copies the visible part of the circuit to the clipboard as a bitmap (as big as the actual drawing field)
Copy Plot	Copies the last plot to the clipboard as a bitmap
Paste	Puts the circuit from the clipboard back in your circuit. You can paste the same circuit part multiple times, however Hicap will remove the labels to prevent multiple labels with the same number.

Circuit menu

New Subcircuit	Make a new subcircuit - both its contents and its symbol. Select 2, 4 or 6 nodes for your subcircuit
Library	Open the library with subcircuits.
Duplicate	Duplicates the selected component, even duplicates complete subcircuits

Preferences	Sets the preference for : dots in the circuit area, the type of resistor symbol (European or US) and whether you like to disable autorange. When disabled, both time and frequency sweep are less accurate, but the transfer function coefficients (an extra in the frequency plot) will be easier interpreted. All preferences are saved automatically for the next time you use Hicap.
NetList	List of components present in main or subcircuit- with location, type and value.
State	Lists all generator-, capacitor- or inductor voltages/ currents at THIS moment, even when you are in the middle of a sweep.
	More items are available in debug mode, but not of interest in normal use

Sweep menu

Frequency	Show the frequency analysis window. Select frequency range etc. there and make a bode plot.
Time	Show the oscilloscope where you can start a time sweep with "New Sweep", or resume an interrupted one with the "Resume" button.
Resume	Make a new (freq or time) sweep with existing settings, adding its traces to the current plot. Useful to resume or compare changes after a circuit modification, or even between two different circuits

Help menu

Index	The contents list of this help file
Search	Search for keywords in this help file
About ...	Informs about your Hicap version and its author. Further, it allows you to enter a few lines of text to document your circuit. This text is saved with the circuit file, and therefore specific for each circuit.
Calculator	Stand alone little spreadsheet to find the unknown from: resonant frequency, L and C when any two of them are known

Draw a circuit

The circuit is drawn by placing components and interconnections on the drawing area. Many short cuts are present in Hicap to make circuit drawing easier.

The toolbar on the left is the basis to enter your circuit :

- Arrow** 1. Presents **editable info** about each clicked component in your drawing area.
2. Double-click the arrow tool changes the cursor to a “+”, which allows you to **drag a selection rectangle** in the drawing area or in the subcircuit symbol area. Selected components or pixels can then be moved as a group, or deleted with the delete key or backspace key.
3. A click at an empty place in the drawing area while the arrow tool was selected and the shift key was pressed allows to enter **free text** there to document your circuit. The free text is saved, moved, selected and edited as a component.
- Line** 1. Make **dot-to-dot interconnecting** lines with a click in the line tool mode.
2. When you keep the Shift key pressed while you place a line, (shift-click) you make links for **long interconnections**, joining nodes that are separated by an obstacle. A link is a small arrow with a single character name. All links with the same character (case insensitive) are interconnected.
- Components** R, L, C, Diode, four types of sources - the voltage and current generators, the voltage and current dependant sources (symbols with input arrow) are created when the appropriate tool is selected and you click in the circuit. Components are placed between two dots (nodes) on the screen, and can be moved around, they “stick” to the left mouse button. Editable information is on the bottom line.
- Ground** Forces a node number to be zero. Ground symbols can help but are not required.
- Eraser** Click a component in this mode removes that component.
The last erased component can be replaced with the “Undo” command from the Edit menu.
- Probe** This last tool is used to attach a numbered label to a component, representing a probe in the main circuit, and resulting in a trace on the oscilloscope with the same number.
Only the voltage across the R, L, C, Diode, or source can be plotted, or the current through these components. This means all probes are floating- they are always related to a component; you cannot measure the voltage between two node numbers unless you create a component between them.
The current in a line cannot be measured either, so attach your current probe to a component with that current.
The created probe is a **voltage** probe, but when made **while the shift key** was pressed, it is a **current** probe.
When a component with a probe is clicked again in probe mode, then its probe is removed.
You have no control over the label number - simply the next free number is used.
All labels (1-9) in main and subcircuits can drive dependant sources, controlled switches, or generators with a probe input. Only labels 1-2-3-4-5-6 in the main circuit produce traces in the oscilloscope or bode plot. These are the real probes.

Arrow tool

1. Arrow

Just select the arrow tool and click on a component.

This presents editable info on that component.

2. Double click the arrow = Group selection

Double click on the arrow tool converts the cursor into a “+”. Now position the cursor somewhere in the circuit, press the (left) button, and drag the mouse over the circuit with the button still pressed. This creates a selection rectangle over the circuit. All components in this rectangle are now selected. They can be moved as a whole, or deleted with the **Backspace** or **Delete** key.

3. Add text

Select the arrow tool, and click at a free place in the circuit area with the **shift key** being pressed. Now type text in the circuit for documentation. This text is editable, moved and deleted like a component.

To add text in the time or frequency plot, click in the plot and simply start to type there.

Calculator

Suppose you have a capacitor, and want to know which inductor will give a tune at 10kHz.
The Calculator window helps you in finding component values. Select it with the last item from the "Help" menu.

The calculator fully stands alone, it does not interact with your circuit.

LC resonance frequency

The calculator uses the formula $F=1/(2 \pi \sqrt{LC})$. Enter L and C, and the resonance frequency is shown. If you enter a frequency, then the component value that was not locked is updated.

RC roll-off frequency

Click on the "L" changes it to "R", and the formula $F=1/(2\pi RC)$ is used instead.

Shortcuts

1. Summary of Shift-Click and Double-Click actions.

Double click

1. Double click the arrow itself gives "+" cursor to draw a selection rectangle in the circuit or subcircuit symbol
2. Triple click the arrow itself redraws the complete circuit window

Shift click

1. With arrow selected : enter free text
2. With line selected : place a link instead of a line
3. With probe selected : place a current probe instead of a voltage probe

Double click or shift-click

1. On an already placed generator opens its settings window
2. On a subcircuit symbol in the main circuit opens its contents and symbol-edit facility

2. Duplication

Duplication of a component or group of components is done by **cut** and **multiple paste** of a selection, or with **Circuit menu: Duplicate**

3. Use of mouse buttons

In the circuit window, the right-hand mouse button displays a floating menu with some options like cut-copy-paste, duplicate, and properties of the selected component.

4. Remove component(s)

Remove a component

Select the eraser tool from the toolbar, and click on the component to be removed. If a subcircuit is removed, and it has no clones, then the components in it are removed as well. Undo is possible for this action (edit menu: undo)

Remove a Group of components

A selection is removed with the **Backspace** key , the **Delete** key , or with “**Cut**” from the Edit menu. Undo is possible for this action (edit menu: undo)

A ‘cut’ selection can also be restored with ‘paste’, although the component(s) might re-appear at a different place than where they were cut.

Error messages

Source load conflict

The load impedance on both voltage and current sources may not reach zero either at DC or at infinite high frequency. So a capacitor (or series connected capacitors) in parallel with a source, or an inductor in parallel with a source will give an error message. Add some series resistance or change your unrealistic circuit.

Floating components or circuit parts.

All components and circuit parts MUST be interconnected at least at one node.

Single components with no nodes connected give an error.

Single node connected components, and short circuited components are disregarded.

Do not let the primary circuit and the secondary circuit of a transformer float with respect to each other. This will give an error like "Source Load Conflict/ open node"

DC floating nodes

This error occurs when the DC voltage on a capacitor is undefined, or when the DC current in an inductor is not defined. The first happens when you put two capacitors in series without a resistor in-between to ground defining the DC voltage level. The solution is to add a (high value) resistor.

A possibly infinite DC current gives the same error. Give inductors some copper-resistance.

Too many

Version 1.6x /1.7x allows the following number of components:

1. Up to 150 things (components + lines + grounds + subcircuits + free texts ...) of which
2. Up to 99 components, including those in subcircuits
3. Up to 6 probes, up to 4 generators
4. Up to 14 diodes
5. Up to 15 capacitors and inductors, however the total of inductors, capacitors, probes and diodes must be less than 24

Numbers, Values, Polarities

The **component value** is entered in the bottom line, the first 8 characters appear near the selected component.

Scientific notation

The value is a number. If it is followed by one of the following characters : p, n, u, m, k, M, G or T then the value is multiplied with the corresponding power of 10, so 2.2k is read as 2200

Anything after this character is **not** read as value. So 2k2 is read as 2k !!

A value like "2" or 2 ohm is also accepted. A leading zero is not needed, you can use .05 for 50mH

Initial value

Capacitor values can be followed by a **comma** and an initial voltage.

Inductor values can be followed by a **comma** and an initial current.

Anything after a **quote** is not read and may be used for comment.

The value of a diode is its "On" resistance. The "Off" resistance is by default 100,000 times the "On" resistance, or any **other ratio** you specify after a comma.

Sources

The value to be entered for the **generators** (source symbols without arrow) is a waveform type, like sine, step or battery. Once the generator has a name, you can double-click on its symbol to open a window with settings for that generator.

Dependant sources

The value for the **dependant sources** (source symbol with arrow) is the gain followed by a comma and a label number. If you use a negative label number, the source is current-controlled, otherwise it is voltage controlled.

A negative gain can be specified to invert the transfer from input to output of a dependant source.

The dependant sources routine is completely separated from the probes routine.

You can current-control a dependant source even from a voltage probe !

For instance, the following value is entered for a dependant voltage source between nodes 3 and 8 :

v(3-8) 10k, 3 This means that the output voltage of this source is 10,000x the voltage at label 3.

v(3-8) -1,6 The output voltage is minus one times the voltage over the component with label 6

v(3-8) 5m,-2 means the output voltage is 0.005 times the current through the component with label 2.

The value for the **diode** is its on resistance, the off resistance is normally 100,000x higher.

Example:

D(3-8) 0.1 This is a diode with forward resistance of 0.1 Ohm, and off resistance (leakage) 10kOhm.

Most real life diodes have a higher ratio between on and off resistance, but sorry.... the numerical precision of Hicap does not allow much more. However, the on/off ratio can be changed.

Polarity

The polarity of the plotted traced is such that the top or left node of a component is positive. For diodes the top or left node is negative. Sorry for inconvenience...

Current is plotted positive when from top to bottom in any component, including current sources. Voltage is plotted positive when top (or left) node is positive with respect to bottom (or right) node.

Example: A current source is terminated with a 1 ohm resistor. The source makes a positive, constant current of 1A. Current in the source is from top to bottom, thus from bottom to top in the resistor parallel to it. This makes a negative voltage both on the current source and the resistor. The resistor has negative voltage and current, thus dissipates power. The source has positive current (top to bottom) but negative voltage, thus is an energy source.

If you get traces that are upside down, use a negative probe sensitivity to correct.

Lines, Links and Ground.

Lines interconnect dots.

Repeated lines interconnect nodes that are far apart.

If you must cross other circuits, then use **links**.

Links are little arrows with a name. The first non blank character of the name identifies the link.

Links are created when you place a line with the **shift key** pressed. All links with the same first character in the same (sub)circuit are interconnected.

Ground is a special link, valid in all subcircuits. They are convenient, but not required.

Resistors

value looks like 22k.

You can use all prefixes like p for pico , n, m, k, M, G and T.

An embedded prefix like 2k2 is **not** recognized. Use 2.2k instead.

Comment

Anything you enter after a quote is ignored. This way, you can enter short comments after each component like

R 1-2 22k 'feedback resistor

The R and node numbers 1-2 are made by Hicap.

Capacitors

value is entered like C 1-2 0.1uF

Initial voltage

Every time the scope is triggered, all capacitor voltages start at zero. At automatic retraces, or when you resume a sweep, the capacitors maintain their acquired voltages of course.

When you want to start with an initial voltage, then add this voltage after the value, separated by a comma. For example, the following capacitor of 22 nano Farad starts at -5.5V

C 6-0 22nF, -5.5

Initial value conflict: If you have a loop of only capacitors, and they all have an initial voltage, and these initial voltages do not sum-up to zero, than Hicap will randomly disregard one of the initial values.

Capacitor voltage Limiting.

The voltage of any capacitor is limited when you add **L** after the value, followed by the maximum voltage on the capacitor in either direction.

The following capacitor has a voltage that is limited between +10V and -10V :

C 7-1 22 uF L 10

The following capacitor starts at 2.3V, and is limited at + or - 5V:

C 1-2 0.1uF, 2.3 L -5

Caution: The current in a capacitor during voltage limiting has some "crosstalk" to other capacitor voltages, especially when this current is high.

This bug is caused by the simple limiting of a state variable, without taking its current into account in the rest of the circuit. The correct way would be to substitute the capacitor by a voltage source during limiting and recalculating all state equations for that situation. This would slow down the sweep considerably, and the crosstalk is acceptable in many cases when the capacitor current is limited to a low value.

Inductors

values are entered like

L 1-2 0.1mH

Initial current

Every time the scope is triggered, all inductor currents starts at zero. At automatic retraces, or when you resume a sweep, the inductors maintain their acquired currents of course. When you want to start with an initial current, then add this current after the value, separated by a comma. For example, the following inductor of 50 microHenry starts at -25mA:

L 1-2 50uH, -0.025

Initial value conflict: If you have a node of only inductors, and they all have an initial current, and these initial currents do not sum-up to zero, than Hicap will randomly disregard one of the initial values.

Inductor saturation

Hicap can simulate saturation above some current in both directions. Saturation is indicated by the key letter S, followed by the saturation current and the saturation ratio Lunsat/Lsat.

Example. This 5 mH choke (initial current 1A) saturates to 2 mH above 12A in either direction. The saturation ratio is 5mH/2mH=2.5:

L 3-4 5mH ,1 S 12, 2.5

If the comma and the ratio are omitted, then ratio 5 is assumed, i.e. saturated inductance is one fifth the normal inductance.

Example. This 40mH inductor saturates at 50 mA (and is 8mH above this current by default)

L 5-2 40mH, 0.05

Diodes

The diode stands for a family of components that are implemented in HICAP as resistors that can switch between two values. The lowest value ("on" resistance) is specified in the infoline. The Roff is 100000 times Ron. Switching between low and high resistance can be based on current polarity for a real diodes, on voltage for a breakdown device, on other probe signals etc.

The simplest member of the family is the plain diode

Example: D 4-7 0.1

The "On" resistance is 0.1 Ohm. The default "Off" resistance is 100000 x Ron, so 10k in this example. Diodes accept another Roff/Ron ratio after the value, separated by a comma:

D 1-0 Ron{,ratio}

Alternatives

With a letter after the on resistance, the diode becomes a "switch" like component :

D 1-0 Ron, {ratio} F Fast diode
D 1-0 Ron, {ratio} M The diode is a switch that opens/closes with a mouseclick
D 1-0 Ron,xP The diode is a switch that is closed when probe x is positive (not zero)
D 1-0 Ron,xN The diode is a switch that is closed when probe x is negative (not zero)
D 1-0 Ron,xB The diode is a switch that closes when its abs.voltage is higher than x
D 1-0 Ron,pT The diode is a thyristor The pulse number p corresponds to a gate control unit
D 1-0 Ron,xG The diode is a thyristor, fired when probe x is positive
D 1-0 Ron,pQ The diode is a switch that is closed when pulse nr p is high

All switches are initially off, except the ,N diodes

All diodes and switches are open in a frequency sweep. They are resistors of value Roff.

The P,N,B,G and T types have a fixed on/off ratio of 100000 that cannot be changed.

The ,P and ,N switches can be controlled from a step generator, which has segments +1 to drive the ,P switch and segments -1 to drive the ,N switches. Zero segments leave both switch types off.

,P or ,N Probe Controlled switch

A switch driven by the polarity of a probe signal somewhere else in your circuit. The letters P and N define these probe controlled switches

D 1-2 0.1, 2P
D 1-2 0.1, 3N

After the on resistance, there is a comma, followed by the driving probe number, and a P or N. The P means that the switch is ON when the probe signal is positive; with the N, the switch is ON when the probesignal is negative. If you omit the probenumber, then the highest probenumber is used. Any ,xN switch is initially on, any ,xP switch is initially off.

,F diode

A normal diode takes one integration time step to start or stop conduction. This gives a problem when you make a chopper with a transistor and a diode. The transistor is a probe driven switch, for instance like D (3-0) 0.1, 2P When the transistor starts, the diode is still conducting, giving a short circuit for one delta-T and a surge current in both of them. When the transistor stops conducting, it takes one integration delta-T before the diode turns on, presenting a high impedance to the chopper inductor, hence a surge voltage. To prevent this problem, use a F(ast) type diode.

Syntax: D(3-4) 0.1 ,F

These diodes are normal diodes, except in the first delta T after a .P ,N or Q switch changes state:
When any ,P or ,N diode (switch) turns on, then all ,F diodes are turned off that same delta T
When any ,P or ,N diode (switch) turns off, then all ,F diodes are turned on that same delta T

With a number<10 in front of the F, the diode reacts only on the switch with that probe or pulse number.
Example:

D(3-4) 0.1, 2F

.B diode Break-over device or sparkgap

The ,xB converts the diode into a breakover device, normally off, but changes into a low impedance when the voltage x is reached. Conduction stops when the current becomes less than the holding current. The Break Over Device is symmetrical, and can conduct in either direction.

The syntax is:

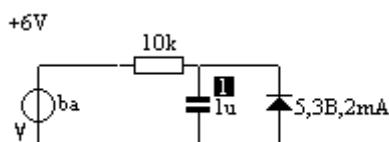
D 1-2 0.1, 8B , 10mA

The on-resistance is 0.1 ohm, the breakdown voltage is 8 V.

After a second comma, you may add the holding current. Conduction stops when the current drops below this level. If omitted, conduction stops at 0 amps.

Engineering notation (k.m, u etc) is not accepted for the breakdown voltage, but it is for the holding current.

A relaxation oscillator can be made with a battery, resistor, capacitor, and a BOD across the capacitor. The BOD breakdown voltage must be less than the battery voltage, and the holding current must be higher than the charging current.



-pG Thyristor

Turns on when probe p becomes high, and stays on as long as the current is in forward direction. No gate control is needed; the thyristor is driven by a probe.

.pT Thyristor

Turns on when gate control output p becomes high, and stays on as long as the current is in forward direction. A gatecontrol is needed.

.pQ IGBT

This switch is on as long as output p of the gatecontrol is high. A gate control is needed. When these "Q" type diodes are used, the gatecontrol is automatically converted from phase-, to pulse Width control.

Thyristors, and the gate control

Hicap has discrete thyristors, and a macro with a complete thyristor bridge in it.

A thyristor starts conducting when it gets a gate pulse while the anode voltage is positive (forward biased), and remains on as long as the current stays in forward direction.

Unfortunately, reverse current is detected after the calculation step, so a high short circuit current might develop in that deltaT. Although not visible in most cases on the oscilloscope, this can influence the rms and FFT readings over the sweep. It will be improved in Hicap 2.0

Discrete thyristors are triggered by some other signal or by a dedicated gate control.

They are entered as diodes when Ron is followed by ,xG or ,xT like:

D 2-3 0.1, xG This thyristor is triggered when probe x is positive (not zero)

D 2-3 0.1, xT This thyristor is triggered by gatecontrol output x (see below)

The off-resistance is 100000x the R-on. Be aware of the leakage current: with a forward (on) resistance of 1mOhm, the blocking resistance is only 100 Ohms !

A complete thyristor bridge can be built with individual "T" thyristors. Hicap has a special gatecontrol function, linked with a (mains) sinewave generator. This takes a lot of runtime, but allows to study unbalanced mains conditions, trigger failures, couplings, transformer arrangements etc.

#The Gate Control

The gate control is a normal sine generator with * or # added to its name. This gives that sinewave generator twelve extra internal outputs called pulse numbers. The first one (nr 0) is high during the first 30 degrees of the sine wave, nr 1 during the next 30 degrees, and so on. The gate control can be controlled by

- the value of the VAR generator if you used *, the asterisk, or
- the value of the first probe if you used #.

Depending on the type of switches used, either the complete pattern is shifted forward and backward when thyristors ("T") are found, or the width of the pulses is varied when IGBT's ("G") are found in the circuit. Only one gate control is allowed.

A. Phase control mode

When "T" thyristors are found, the gate control is in phase control mode. The gate control is part of a sine generator, usually one of the mains voltage generators. The name of that sine generator includes an * or #. All 12 gate pulses are normally 30 degrees wide, but any multiple of 30 degrees can be specified by the number directly after the * or #.

The firing angle is controlled manually by the VAR source (*) or by the first probe (#).

The phase control mode is intended for polyphase thyristor rectifier bridges.

NOTE - only the first 5 characters of a generator name are read, use no spaces in sin*2, otherwise the pulselwidth is not read.

Example: **V(3-0) sin*2** The phase shift (alpha) is controlled from the **VAR** generator. The 12 pulses have equal, fixed width of $2 \times 30 = 60$ degrees.

Pulse disable

A negative pulselwidth disables all gatepulses. Because only 5 characters are read, the only possible name for the synchronising sine generator is "si". Example: **V(3-0) si*-3**. This can be used in asymmetrical short circuit simulation, involving pulse blocking.

B. Width control mode

When "G" switches (IGBT's) are found, the gate control uses width control mode.

The 12 pulses (every 30 degrees) now have equal, variable width. This mode is intended for polyphase choppers.

Example: **V(3-0) sin*2** The width is controlled from the **VAR** generator. The 12 pulses have no common shift, and equal, variable width. In fact, the "2" is not read.

Controlling variable.

The pulse shift or width is always 30 degrees per unit of the controlling variable.

Example:

sin* The phase is shifted by the output of a variable DC generator with **30 deg / var**

This is a convenient way to play a little with phase control. To use it, create a Var generator with initial value =1, increment/decrement = 0.5, connected on one side to ground, other side floating. This will drive the gatecontrol, shifting all gatepulses by 15 degrees every time you press the up/down arrow buttons under the timeplot during a sweep.

sin# by the output of the first probe with **30 deg / division** whatever the setting of probe 1.

This way you can close a control loop. Put probe 1 on the output of your PI controller. Note that it is not the voltage of probe 1 that drives the gate control, but the divisions on screen. With probe 1 at 2 V/div, you need 2V from the loop to shift 30 degrees. With 1V/div you need only 1 V for the same 30 degrees, hence a higher loopgain !

The influence is the same for the # and * method, so you can find the correct thyristor nrs first with the manual (*) method before you close a control loop.

Defaults:

- If there is no sinewave generator, then the thyristor(s) are never fired.
- If there are sinewave generators, but none or more than one with an asterisk, then Hicap picks one as gate control
- If the sine generator name with the asterisk has no number after the asterisk, or a number after the fifth character, then pulse width = 1 (30 degrees) is assumed.
- If no pulse number is specified for a thyristor, it triggers on pulse 0

Or you can use the RECT generator. Place a voltage source in your circuit, and name it "Rect". No sinewave generators, gatecontrol, thyristors or transformers are needed. The RECT generator produces a voltage waveform like a 1, 2, 3, 6, or 12 thyristor bridge without overlap on a symmetric three-phase grid. This macro is 200x faster than using discrete thyristors and intended for use in control loop simulations. Learn more about this source [here](#)

[#] IDH_RectIntro

Controlled sources

A controlled source is a perfect voltage or current source, whose output is controlled by the signal from a probe (current or voltage) elsewhere in the circuit.

All 4 combinations (i.e. current controlled voltage source...) are supported.

Controlled sources are linear (with just a gain) **or** with some user-defined characteristic

Linear dependant sources

The symbols are similar to the generator symbols but with an arrow added. The component types are v and i. As value you must enter gain , driving probe.

Example: v 2-3 10k, 1

The voltage 2-3 is 10000 times the voltage across the component with label 1, even when label 1 was a current probe.

A negative labelnumber makes the source current driven :

Example: i 2-3 10k, -1

Current from 2-3 is 10000 times the current in the component with label 1.

Drive labels are always in the same (sub)circuit as the dependant source itself.

The linear dependant sources are always stable, and simulation speed remains high.

Non-linear dependant sources

These sources have capital V or I generator symbol without the arrow. Place the generator symbol in the circuit, and give it a name which has one of the following letter combinations in it:

re	rectifier	1-2-3-4-6 or 12 thyristor rectifier. Control is the firing angle
de	delayline	A delay line. Control is the input signal
cl	clipped	Output is an amplified, clipped probe signal
di	divider	Output is a probe signal divided by
mu	multiplier	Multiplies two probe signals
po	polynomial	$Y=ax^2+bx+c$
co	comparator	Like opamp without feedback
qa	quantizer	Quantized signal like ADC/DAC combination
sm	sliding mode	Huh
ag	Controlled Gain	Has gain as a tabled function of a probe signal
ib	Integrating Battery	Variable DC source with integrating control loop

The non-linear dependant sources might be unstable, unless the calcualtion step is made small.

This reduces simulation speed

Subcircuits

A subcircuit is a collection of basic components bundled as a group, with a user defined symbol like the opamp symbol.

Subcircuits are entered :

- with the "New Subcircuit" menu item
 - You have to fill it with components, and draw its symbol.
- or selected from the Library, a separate file on your hard disk.

A subcircuit interfaces with the main circuit by 2, 4 or 6 corner nodes and via ground.
Subcircuits can be stored in the Library.

The following applies within a subcircuit:

- Labels are local in a subcircuit.
- Subcircuits can not be nested.
- Corner nodes cannot be interconnected with other corner nodes within that subcircuit
- Do not ground the corner nodes inside a subcircuit
- Inside a subcircuit, only **labels** exist.(no probes !)
 - label numbers are **local** (each subcircuit has a different label 1)
 - labels only serve to drive dependant sources in the same subcircuit
 - they do **not** make traces at the oscilloscope or bode plotter .
- the **ground** symbol however is always the same, it is global.
- Linknames are **local**. (link a only interconnects with link a in the same subcircuit)
- subcircuits within subcircuits are not allowed (**no nesting**)

Create a subcircuit with the menu “new subcircuit” or select one from the library.

Open the subcircuit symbol by shift-click or double clicking on it. You get a window with a circuit drawing area for your subcircuit and a drawing area for its symbol.

The internal circuit is made the same way as in the main circuit.

- The symbol is a 41x41 pixel bitmap, created by clicking with the mouse (or moving with the mouse-button pressed)
- When the move started at a white pixel, then black pixels are made, when started at a black one, then white pixels are made as long as the button is down. Groups of pixels can be created after double clicking the arrow tool. Groups can be moved with the mouse, or cleared with the backspace key. You can Undo this clear.

Return to the main circuit with a click on the small symbol in the middle, or on the window's closing cross. You can now move the subcircuit or give it some name as a component.

Subcircuit orientation

The **2-node subcircuit** can be placed horizontal or vertical. Draw the symbol with the final orientation in mind. The internal schematic is drawn as horizontal one, but if you place the subcircuit vertical in the main circuit, the left hand node in the subcircuit is the top node in the main circuit.

The **4-node subcircuit** occupies the place of two vertical components.

You can not place other components along the left or right side of the 4-node subcircuit, but along the top or bottom sides it OK.

The **6-node subcircuit** occupies the place of 3 horizontal components. The middle one carries the symbol.

Placement of other components along its left or right side is possible. Although confusing, it is possible to place a 4-node subcircuit exactly over a 6-node subcircuit

Number of subcircuits.

Hicap can store 6 different subcircuits, each with a unique symbol and internal circuit. But.. you can use as many copies of the same subcircuit as you like.

For instance, you need 3 identical opamps. Create one of them manually (New Subcircuit) or load one from the Library. Create the other two by selecting the first one, then select "Duplicate" from the Circuit menu (or control-D).

This way, you get a family of subcircuits, all with the same symbol and circuit. If you double click one and change an internal component value, you shall see that this value is changed in all subcircuits.

Subcircuit with a parameter

There is one exception: if you give a component the value @, then this (part of the) component value is substituted by the subcircuits' name.

The subcircuit name up to anything non-numerical is used for the substitution.

Example Suppose you have a 2-node subcircuit with one resistor in it with value @k.

You use copies of this subcircuit at 3 places in your main circuit, with subcircuit names 3sub, 7sub and 12sub.

Then the first subcircuit contains a 3k resistor, the second one has a 7k resistor, and the third one has a 12k resistor.

Subcircuits Library

Subcircuits can be stored in a library. The library is a separate file in the same folder as Hicap.exe (probably in the programs directory). This file has a fixed name. If Hicap doesn't find this file, it creates a new (empty) one without notice.

The Lib is ONLY accessible from the main circuit. Its window pops up when you select the menu item "Library".

You can scroll through subcircuits, add or delete them, or use them (transfer one) into your circuit. When browsing through the library, the name of each subcircuit and its symbol are shown.

Close the Lib window with the closing cross in its upper-left corner.

To Use a subcircuit from the library in your circuit, click USE. The lib closes, and the last shown subcircuit now appears somewhere in your main circuit.

To Add a subcircuit to the library, handle as follows:

Select a subcircuit in your main circuit, its name must appear in the info line under the circuit. If you have no subcircuits, then create a New Subcircuit , fill it with components and symbol.

Then open the library

Click ADD

To Erase a subcircuit from the library, select one, and click ERASE. You will see the next subcircuit in the library window, or the first one if the last one was erased, or an empty window if you erased the only one.

If you load a subcircuit from the Library with a generator in it, then this generator has its default settings. Sorry, the library can not contain generator settings. This is only a library problem: also these generator settings are saved correctly along with your circuit.

Component value substitution

In some cases, it is convenient to change component values inside a subcircuit without having to open it.

For instance, you have a four diode rectifier as a subcircuit, and want to change the on-resistance of all diodes at once, or you have a transformer subcircuit and want to change the gains of the dependant sources inside it in order to change the transformer's turns ratio.

The procedure is as follows: Hicap substitutes any @ in a value of a component inside a subcircuit by the name of that subcircuit. So if you have a dependant source V 3-4 -@,1 and the name of the subcircuit was "10:1" then the dependant source has a value of -10,1. In fact, the @ is substituted by the subcircuits name from the first character until a character that is not 0123456789pnmkMTG or decimal point.

Hicap allows only 6 different subcircuits. With substitution you can make slightly different subcircuits in unlimited numbers.

Probes, the outputs of your circuit

Labels

The little numbered black squares are labels. They are the outputs of the circuit. All labels (1-9) in the main circuit and all (1-9) labels in any subcircuit can be used to drive dependant probes, to control switches, or to drive generators with a probe input.

Only the first 6 labels in the main circuit create traces on the time or frequency plots. These labels are called probes.

Where

Hicap is component oriented. Only voltages over a component, or currents in a component are recognised and can be calculated. Probes are always attached to a **component**.

Sorry, there is no current probe in a line, there is no voltage probe across any pair of nodes.

Attach a probe

Select "probe" from the toolbar, then click in the middle of a component to create a **voltage** probe across that component. If you click with the Shift key pressed, then a **current** probe through that component is created. You can't select its number, simply the next free number is created.

Removal

Click the component again in probemode to remove its label.

Time Sweep

TIME from the sweep menu selects an oscilloscope to show the time response of your circuit from generators as inputs, or as discharge of pre-charged inductors or capacitors.

This page informs how to:

1. Set the time per division
2. Set the vertical scale for each probe (i.e. each trace)
3. Start or resume the sweep, error indications
4. Repeat functions
5. Stop a sweep on-the-fly. Split timebase
6. After sweep calculations like FFT or RMS. value
7. Trace labelling

#1. Set Timebase

Time/div	The time per division in seconds, or use 10u if you want 10us/div
Resolution	The number of calculations per pixel. The higher, the more accurate and the wider the range of time constants in the circuit can be. A higher resolution takes more time on your PC. The maximum is 199. If you get a ' <i>time step too big</i> ' error (red square), then increase the resolution, or reduce the time/div.

During a (repetitive) sweep, the timebase speed can be increased/decreased in 0.5% steps with the up/down arrow buttons on the keyboard, provided your last click was in the oscilloscope screen.

2. Set for each trace the V/div and zero line

For each probe, the sensitivity (in V/division or A/division) can be entered, optionally followed by a comma and a number, indicating how many divisions the zero line will be above or below the centre line. (the shift)

Example: 10m,-3 is 10mV/div, and a zero line at 3 div. below the centre of the screen.

A zero sensitivity results in an error. Negative sensitivity inverts the trace.

The zero shift can be from -99 to +99 whole divisions.

A negative sensitivity inverts the trace like -10m, -3 It is good practice to add an A or V after the sensitivity to remind the type of probe like 10mV, -3

A short comment after the offset is allowed, and appears on your hardcopy.

Example: 10mV,-3 Vref

Further trace settings:

Click the checkbox above the channel nr, to turn the complete trace on or off.

Click the (coloured) channel nr itself, to toggle between dc and **ac coupling** of that trace.

3. Start or resume the sweep.

New Sweep	Start a new sweep on a fresh grid, with discharged (or pre-charged) capacitors and inductors, with generators starting from their initial values (like start phase for the sine generators)
Resume	Continue an interrupted sweep, or write a new sweep over the existing one. All currents and voltages stay where they were.
Clear	Removes all traces and text from the screen
Return to circuit	Close the oscilloscope window, and return to the circuit window. The close cross on top of the oscilloscope window does the same.

Errors are indicated by a red square:

-around a probesetting means it is missing or zero

-around the time/div indicates that time/div is too high, or the resolution is too small.

4. Repeated sweeps

The checkboxes near the buttons "Resume" and "Clear" cause a continuos repeat of these functions. This comes close to the real-time behaviour of an oscilloscope.

- **Resume with repeat:** When this box is checked, the sweep repeats itself forever, like a normal oscilloscope. (auto retrace) Uncheck the box by clicking again on it.

- **Clear with repeat:** Clear existing traces at the spot where the new traces are written. Otherwise, all repetitive sweeps add their traces to the screen.
Note: this adds 20% to your calculation time. Further, *everything* is erased on the plot including FFT results, text, labels, etc. If you use these, then do not use repeated clear. A running sweep is indicated by a walking blue line in Repeated Clear mode.

During (repeated) sweeps, you can change the time/div in 0.5% steps with the cursor up/down keys on your keyboard. This helps to make the plot "stand still" in non sweep-synchronised simulations. Up/Down keys are accepted only when the plot screen has the focus, so click there first.

5. Stop in the middle of a sweep

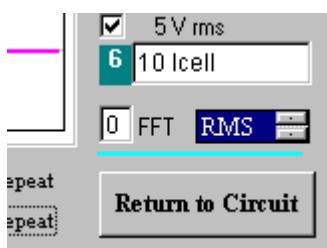
The sweep can be stopped temporarily at any time to **make slight modifications in your circuit**. For example, the load resistor of a power supply can be changed to simulate a load step. You can also change the time/div setting to make a **dual (split) timebase** sweep. A sweep stops at the tab-stop, visible as a red marker below the plot. A tab-stop is made with a click just below the grid before you start a sweep.

6 After sweep calculations

Hicap can calculate some figures over the last completed sweep. These are the DMM function (Digital MultiMeter on all traces) and the FFT function (Fast Fourier Transform of one trace)

DMM.

Select a function in the darkblue selection box below the trace definitions on the oscilloscope window by clicking the tiny up/down pads in it.



Choices are None, RMS, Average, MIN, MAX and Peakpeak. At the end of each sweep, these values are printed above the channel settings. When "Print" is selected from the circuit window, you get the circuit, the plot, and these values printed as well.

FFT

HICAP can print the first 50 harmonics of one trace in one sweep, assuming all sweeps are like this one. Enter a **probe number** in the box marked *FFT*. After the next sweep, a list with harmonics of that trace appears in the plot. When the FFT box is zero, the FFT function is off.

Clipboard

When you click the text "**Y/div, offset**" on the right of the oscilloscope screen, then the DMM results are put on the clipboard. This way you can document measurements with slightly different parameter "var" etc.

7. Trace labelling

Traces are recognised by colour. Additionally, it is possible to mark them with their probe identifier like V1, V2, I3 etc. There are two methods to do so:

1. If you are quick enough: Click in the plot while the sweep runs.

Traces are labelled at that point in time.

2. Set a tab (see "Stop in the middle of a sweep"). Once the sweep stops, click in the plot.

Traces are labelled at the tab position.

Frequency Sweep

Select FREQUENCY from the sweep menu.

Bode diagram as What-divided-by-What

Bode diagrams are relative values as a function of frequency. The relative value can be the ratio of a probe signal to the source level, or it can be the ratio of two probesignals. Hicap can show both, depending on the text you enter in the box on the lower right corner of the frequency plot screen.

- If this box is empty, you get all probesignals with respect to the only generator you have
- If the box has a generator name, you get all probesignals w.r.t. that generator as input.
- If the box is a probenumber followed by "/", then you get all probesignals with respect to that probesignal. The driving generator is not defined then, so better have only one in your circuit.

This active generator is the one and only input of the circuit for frequency analysis; all other *voltage* generators become (temporary) a short circuit, all other *current* generators become an open circuit. A current probe in such an open current source, or a voltage probe across such a null voltage source produces zero output, that is -640dB in the bode diagram.

1. Set -Start and Stop Frequency

lin or log sweep in X (sweep) direction. For lin sweep, enter grid line spacing in Hz.

2. Y-Calibration

Whatever the plottype is, the top entry box on the right side of the plot specifies the calibration of the top line in the grid or the outer circle of the polar plot.

In case of a **dB grid**, it's in dB, for the remainder it is the ratio from input to output. (whether in V/A, A/V,A/A or V/V) The dB selection allows setting of dB/div. (usually 6 , 10 or 20) Other lines are a fixed amount of dB lower, or in steps of 10 % for a linear Y-axis.

Polar plot

The polar plot has linear radial scale with outer radius equal to the topline of the **lin** grid

Errors. Hicap automatically tries to correct impossible entries without warning. If the grid doesn't look like you expected, check for wrong entries like Fstop < Fstart, negative or blank fields etc.

3. Trace labelling

Traces are recognised by colour like in the time sweep. You can also add probenrs to all traces. Click just below the lowest gridline of the phase plot and all traces are labelled by their probenumber at **that** frequency. Make a few sweeps to find the best position for the labels.

4. Multiple sweeps with slightly changed circuit

Make a first sweep. Change something in your circuit like a capacitorvalue or so.

Select **Resume Analysis** from the sweep menu. The second sweep will be added to the first one, so you can see what changes. Another way to make slight changes is to give one or two component values not afixed value, but a variable value.

5. Use of "var" in component values

Once your circuit has a (dummy) source with the name "var", then the variable "var" exists, and can be varied from the oscilloscope and from the freq. Sweep window. In the freq.sweep window, you get up/down buttons to change the value of "var" in steps you specified. Each click gives a new bodediagram. When you check the box "Add", then all the whole family of plots appears in one diagram.

#6. The extra's menu

IDH_Fsetting

IDH_Polar

IDH_Extra

Single Frequency	Opens a window in which you can enter a frequency. It immediately shows gain and phase of all traces at that frequency. When you move the mouse slowly over the plot under or around this window with the button pressed, you get the values at the cursor position.
Some Frequencies	Opens a window where you can enter some frequencies, separated by comma's. The gains and phases are immediately shown next to them. These lines are also printed on paper when you print your circuit and plot (max 3 probes, 8 frequencies). Further, the contents of the "some frequencies" window are pasted on the clipboard when you click the words "Enter frequencies, separated by commas".
# T(s) coefficients	Opens a window with the coefficients of the transfer function for the first two probes. The transfer function is not a function of s (the complex frequency $j.2\pi f$), but as a function of s times a constant. This autorange constant was used to make all coefficients closer to 1, giving a higher accuracy in the calculation process. You can switch-off the autorange facility in the preferences window, making the constant =1. The coefficients are presented in descending order of the power of s , followed by the common denominator coefficient and the numerator coefficient per probe.

Preferences

The following preferences can be checked:

Use node dots. Whether or not you like dots on your drawing board

Use US resistor symbol (the zig-zag type in stead of the rectangular one)

Use NO autoranging. This choice is only of interest for frequency analysis, and only when you want to see the coefficients of the frequency transfer function.

This transfer function is the quotient of two polynomials in s (the complex frequency $s=j.2\pi f$). Assume you have a seventh-order filter, then numerator and denominator polynomial are of the seventh order. The coefficient of s -to-the-power of 7 differs many orders of magnitude from that of s -to-the-power 0. This contrast causes calculation errors, which are kept small by using an internal frequency $f' = c. f$. The constant c is calculated before the coefficients are calculated to gain accuracy. Example: if the coefficient of $s^7 = 1e21$, and the (dc) coefficient of s^0 is only 1, then Hicap uses $f' = 1000 * f$ instead of f to bring the coefficients of the polynomials in f' to near-one.

A drawback of this autoranging scheme is that the coefficients look meaningless.

If you want to use the coefficients for something, switch off the autoranging.

Print Circuit as bitmap. When unchecked, Hicap prints the circuit to a laser- or inkjet printer as a vector file, giving crisp components and TrueType character values near the components. However, some printer drivers do not understand this format. In this case, you can check the preference to send a screen bitmap instead. Note that the plot is always sent as a bitmap to your printer, directly from screen.

Debug mode. This makes some extra items in the "circuit" menu visible:

sorted netlist. Hicap uses a tree model to set-up the Kirchoff equations. The tree connects all nodes, preferably via voltage sources and capacitors.

The elements of the tree are listed first, followed by the links.

Debug Data Can change

Switch The properties of all diodes, thyristors or switches

In the oscilloscope file menu, "DumpABCD" is now visible

Except for the dots, all preferences are normally off.

Preferences are saved in the registry.

Clipboard use

The clipboard can be accessed in several places

Circuit window: You can copy or paste
single components
or groups of selected components

Note: When you copy a part of the main circuit, and paste it in a subcircuit, then ...

Oscilloscope A click on the words "**Y/div, offset**" puts the actual values of some generators and all probe totals (rms, average etc) on the clipboard, separated by tabs. You can paste the results directly in a Word document. Doing so several times produces a table, useful to document the results of a series of small changes.
The generators that put data on the clipboard are (if present):
VAR generator: Value of VAR
iBatt generator: Value of Vdc
Steps generator: Frequency when in S mode

Frequency plot:

The contents of the "some frequencies" window are pasted on the clipboard when you click the sentence "Enter frequencies, separated by commas"

Description of the available Generators.

The Generators are the inputs of the circuit.

A generator has a round symbol with
a **V** near it for a voltage generator, or an **I** for a current generator.

The name you give the generator has 2 functions:

- It sets the waveform. If the name contains "si" it is a sinewave generator
- It identifies each source and must be unique (like SinR, sinS)

A generator marks the input when you make a frequency sweep.

For a time sweep, the generator (waveform) type must be known, like a sine-wave generator.

Like resistors have a value, generators shall have a name. The waveform is defined by the presence of one of the following, underlined two-letter combinations in the first 5 characters of this name. There is no difference between lower and uppercase.

Example: "Asteroid" contains "st" in the first 5 letters, so is interpreted as a steps generator.
You can select up to four generators from this list:

Independent generators

Step	Square, triangle, or stepped waves 1)
Sin	Sinewave generator
Var	Variable DC source
Bat	Battery
Noi	Noise source
Wa	Arbitrary waveform (graphic entry) 1)

Non-linear driven sources:

The driving probe is in the same (main or sub-) circuit as the source. Sources with two driving probes like the multiplier or the AGC have probe 1 always in the main circuit.

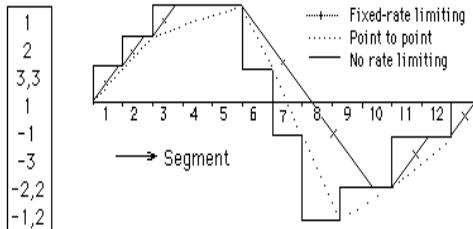
Mul	Multiplier
Del	Delayed source 1)
Pol	Polynome source ($y=ax^2+bx+c$)
Div	Divide source ($y=a/x + b$)
Rec	Rectifier 1)
Com	Comparator
Qua	Quantisizer
Clip	Clipper
Sm	Sliding Mode Converter 1)
AGC	Automatic Gain Control 1)
iBatt	Integrating Battery 1)

- 1) There can be only one generator of this type in your circuit, and only in the main circuit.
If you have more generators of the same type, give them unique names like bat1, bat2 ...
Generator abbreviations and defaults are stored in the gendata.txt file

The steps source

There is only one steps generator. The source generates up to 20 straight segments ("steps"), each with adjustable height, and a width that is a multiple of a standard width. When all segments are made, the cycle repeats itself. The steps source is reset to segment number one after triggering the oscilloscope.

Waveforms can be stepped, rate limited, or generated in a point-to-point mode:



Stepsheight entry field (left) and resulting waveform.

Stepheight Each line in the tall field represents a segment. The first number in each line is the stepsheight, valid for one step, unless followed by a comma and a second number, indicating the number of steps. Height can be positive or negative. Engineering notation like mA or kV is not allowed, but a decimal point is allowed.

After the height you may add:

- nothing* this segment has the standard width
- ,0 this segment has no width, to create vertical segments in PTP mode.
- ,*number* this segment has a width of *number* times the standard width.
- number* is an integer between 1 and 99.

Stepwidth can be either:

- number* like 10m for 10ms/segment. The stepwave is asynchronous when you retrace and the total cycle might not match the current sweep time.
- div* This keyword sets the basic stepwidth equal to the time/div of the sweep.
(synchronised)
- S 1 Sets internal stepwidth to one complete cycle per sweep.
- S 2 makes 2 cycles /sweep etc.

NOTE: A stepwidth less than two pixels or unreadable stepwidth is replaced by the default "div". A risetime less than 2 pixels is replaced by zero risetime also without warning.

NOTE 2: If you want to play with the timebase speed during sweeps in autoretrace mode, and you use the steps source, then its better to use a fixed stepwidth, rather than "div"

NOTE 3: If the stepwidth is S1, S2 etc then the steps source frequency will appear in the plot.

Risetime can be either:

- 0 Zero risetime saves execution time.
- number* Like 10u limits the rise and fall time to 10us/multiplier.
Rate limiting can extend to more than one segment.
- PTP Point to point mode connects the right hand top of each segment.

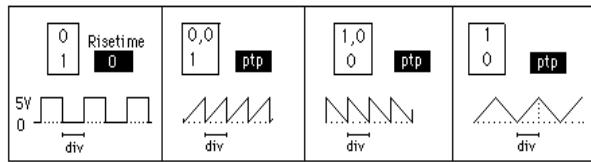
With risetime not zero, the steps source rises to the first stepheight as if it came from the last one. So if you want to start from zero with nonzero risetime, then make the last height zero, or add a last segment with zero height and zero width.

Multiplier The complete waveform is multiplied with "multiplier" for easy amplitude control. Engineering notation is allowed.

Step generator examples

The steps source can be used to generate square and stepped waves (in zero risetime mode), a sawtooth (in PTP mode) and a wide variety of other waveforms.

Try these examples with any timebase speed, stepwidth=div, multiplier=5, and risetime as given:



Sine generator

The **si...** generators have 4 parameters:

- | | |
|-----------|---|
| Freq. | in engineering notation like 3.3k for 3300Hz; |
| Start | phase in full degrees; |
| Amplitude | the peakvalue of the sinewave in engineering notation like 22mV.
It is in V for the voltage source, and in A for the current source. |
| DC | is the offset of the wave like 18 mV |

Variable DC

The **VArdc** source generates a dc voltage or current, that can be varied during the sweep from the oscilloscope.

The output is initialised on Start value when the time window is opened for the first time. Decrement or increment the value with the small buttons under the time or frequency sweep window. The actual value is displayed there as well.

The Var source is unique in that its output is available as a general purpose variable. This variable "var" can be used to change component values, or as manual pulse shift control (alpha) of the rectifier or the thyristors.

Noise Generator

Peak-peak amplitude and DC offset of **NOise** generators can be selected.

The third parameter is the distribution curve:

- 0 = Uniform distribution. The probability is equal for all values within the peak-peak range.
- 1 = Gauss distribution. The probability is inverse proportional with the amplitude.
This distribution was made taking the third power of a uniform distribution. It approximates white noise. When more than one noise generator is used, the noise is not white.

Battery

A fixed DC source, commonly known as a battery when it is a voltage source.

When the "battery" property is used for a current source, it can serve as a constant current sink.

Integrating Battery

This is a variable DC source with built-in controller designed to keep a probe at a reference value. This source is of interest only in repeated time sweeps. The source is updated after every sweep according to the formula :

$$iBatt = iBatt + K \cdot (\text{probe}(N) - \text{Ref})$$

This simulates a pure integrating control loop, with a bandwidth that depends on K, the circuit and your time/div setting. Parameters are:

- Probe number N that must be stabilised (not across the source itself !)
- The reference value and type for this probe signal
Example: 10 mV rms when you want to stabilize probe (N) at 10 mVrms. Properties like rms, av, min and max are recognised, rms is the default.
- The gain K . Start with a low value and increase until the loop becomes unstable.
- The initial and maximum value for the source, separated by a comma. Without a comma, there is no upper limit for the iBatt source.
Example: 10, 500 for startvalue 10, maximum value 500.

Multiplier

Returns the signal $y = a \cdot \text{probe}(1) \cdot \text{probe}(N)$ and limits the result to a specified maximum. Probe(1) is ALWAYS in the main circuit. Probe(N) can be specified in the same main or subcircuit as the multiplier.

If you add S to the probenr, then only its sign is used $y=a.\text{probe}(1) * \text{SIGN}(\text{probe}(N))$.

The sign function is +1 when the argument is positive, or -1 when negative.

The output of the multiplier can further be limited to a maximum value.

Polynomial source

Returns the signal $y = ax^2 + bx + c$ from a specified probe signal x. Output in Amps or Volts depends on the source type; input (Volts or Amps) depends on the driving probe type. Driving probe in same (sub) circuit as source.

Divider source

Returns the signal $y = a/x + b$ from a specified probe signal x. If this is a voltage source, and the probe senses the current through this source, then it becomes a constant power load, with the parameter 'a' as the constant power.

The output can be limited symmetrically to prevent chaos at or near zero input.

This limit further has a few options when you add P,N,A or B behind the limit value :

P : zero output when the input is negative

N : zero output when the input is positive

A : limiter value output when the input is negative

B: limiter value output when the input is positive

Delayed source

The DElay source duplicates the waveform of **another generator or probe** in the circuit, delayed by a specified time, entered in a notation like 1m, 8u etc. Delay can be zero, not negative, and is limited (with warning) to 10 divisions at res=1, or 5 divisions at res=2, etc. A warning is also shown if the specified driving generator is not found. The output of the delayed source is rubbish (zero) until the "delay" time has elapsed. Make a retrace if necessary. There is only one delay generator.

Comparator source

The COmparator source is driven by the polarity of the specified probe number.

The source produces the first output when this polarity is positive, or the second output level when driven with a negative signal. The comparator is evaluated every delta T.

Quantize source

This generator produces a QUantized copy of a probe signal like a ADC/DAC does.

Output Stepsize : The size of the steps on the output side

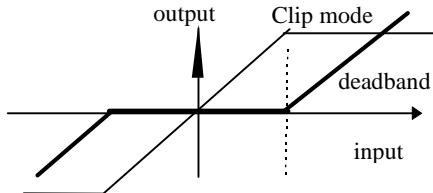
Gain : The gain from input to output

Offset : The offset, added to the output to position the zero step.

Probe : The input probe number.

Clip Source

The CLip source is driven by a specified probe times a certain gain until (output) set limits are reached. Probe is referenced by its label in the diagram. The clipsource output has 1 deltaT delay from its probe, and may be unstable when there is feedback from its output to the driving probe. Clipping the voltage of a capacitor is an alternative, but also can lead to instability.



When the (uppercase) letter D from Deadband is added to the driving probe specification, then this source only produces output when the input signal times gain is OUTSIDE the limits. This simulates a deadband instead of clipping.

Sliding Mode source

This source emulates a four-transistor DC/AC converter with hysteresis control. Together with a simple controller, the average value of the converter can follow a reference. Parameters are:

- Supply voltage Udc,
- Probe number N of the signal that controls the source,
- The hysteresis on this signal .

The polarity of probe 1 in the main circuit controls the polarity of the output. No other probe can be selected.

The signal probe is PN, the polarity probe is P1 and the procedure is

```
If PN > Vphys Then B =True Else If PN < -Vphys Then B = False
If P1 > 0 And B      Then U(Gen) = Udc Else U(Gen) = 0
If P1< 0 And Not B Then U(Gen) = -Udc
```

Controlled Gain Source

Returns the signal **y = probe(1) . function(probe(x))** . This source is similar to the multiplier, but optimised for Automatic gain Control.

Probe(1) is in the main circuit and used as signal input

Probe(x) is in the same circuit part as the multiplier, and used to control the gain.

The function is entered as a table of {probe(x),gain} pairs. If the probe(x) signal is below the lowest pair, than the gain is equal to the lowest pair. If the probe(x) signal is above the highest pair, gain remains equal to the highest pair. In-between pairs, gain is interpolated.

Up to 20 pairs can be entered in the field with the vertical scrollbar. Probe(x) value and gain are separated with a comma. Use one pair per line.

Example: (using probe 3 for control)

```
1,4
2,10
3,25
```

Up to 1 V from probe 3 gives a gain of 4 i.e the source produces an output of 4 times the probe 1 signal. With 2 V from probe 3, the gain is 10 times, etc.

There is no sorting algorithm, so an error is produced when the first arguments are not monotonic rising.

The AGC source has a test function:

- Type = 0 replaces the input probe signal by the constant "1".
- Type = 1 is the normal AGC mode.
- Type = 2 sets the gain to always 1

Rectifier

The REctifier generator simulates the waveform of a single or polyphase diode or thyristor rectifier. It doesn't rectify any signal in your circuit, it merely is a model that produces the DC side waveform of an ideal rectifier bridge. The only control input is the firing angle Alpha. Like all sources in HICAP, the rectifier generator is a pure voltage source, with zero output impedance in both directions. To simulate intermittent current operation, and to prevent negative dc current, a diode must be placed in series with this source. To prevent negative voltage, see below.

You can use only one generator with the "rectifier" property. Both the generator and the eventual driving probe 1 must be in the main circuit.

Rectifier types

Number of thyristors: select 1,2,3,6 or 12 thyristors or diodes per rectifier
A negative value places a freewheel diode over the output of the bridge.
Vac is the rms line-line voltage.
Freq is the mains frequency.

Alpha is the firing angle in degrees, limited to 0...172 degrees.

Alpha is: 0 for a diode bridge
number like 30 for a fixed firing angle of 30 deg
FP this keyword links alpha to the value of **probe 1** in the main circuit,
according to:
+ 3 divisions is 0 deg (diode bridge)
- 3 divisions is 180 deg alpha
VAR Output of variable dc generator. (1 volt = 1 degree)

Note that the overall gain is **30 deg/div** times the sensitivity of the first probe. The absolute voltage to get alpha=0 depends on the offset of the first probe.

The rectifier can be controlled by probe 1, and often becomes part of a control loop inside your circuit. The gatecontrol is of the 6 or 12 ramps type adding no time constants to the loop. The rectifier output voltage is $U=\sqrt{2} \cdot Vac \cdot \sin(\text{arc})$ where arc continuously increased with time, until (with a 6-pulse rectifier), arc> alpha+120°, then arc decrements by 60 deg..

The rectifier source is initialised at the beginning of the first sweep at the top of the (mains) sinewave at T=0, so the "sine" mode actually produces a cosine wave.

Arbitrary Wave Generator

Place a generator in the circuit, and give it a name with either "wa" or "pw" in it. Hicap recognises these two arbitrary wave generators

The waveform generator lets you create one cycle of a repetitive waveform. The waveform is edited by dragging the mouse. Move the mouse with the left button pressed creates one straight section.

The **vertical scale** (V/div or A/div) must be set, and

The **horizontal scale** (sec/div) can be set. If not, the horizontal scale is the same as the sweep timebase, so your arbitrary waveform has always one cycle per complete sweep, whatever the time/div setting. Up to Hicap 1.65, this was the only option.

There are buttons to scroll your waveform left, right, up or down in small increments. Keep the button pushed to autoscroll the waveform (dual speed)

There are two menu's

File menu

New-	erase the present wave, start with a zero line
Open	loads previous saved wave files
Trace 1	loads the waveform of trace 1 from your last time sweep
Save and save as..	store the wave
Print	prints all 480 points and the waveform on paper.

Manipulate menu:

Undo	Undo last change
Mirror 90 deg	after you created the first 90deg of the waveform, this function completes the waveform using 90 deg and 180 deg mirroring.
Mirror 180 deg	same, but only 180 deg mirroring.
Double	makes two periods of your wave.

90 degrees mirroring is done in two steps: First, $F(180-t) = F(t)$
the second step is 180 degrees mirroring, i.e $F(180+t) = -F(t)$

Control a component or generator during the sweep

"var"

The output of the Variable DC source in your circuit is available as a variable named "var" throughout the program. This variable can be used as a component value, or to control some of the sources.

Experimenting with a component while observing the time or frequency sweep is often a fast method to optimise some property of the circuit. This can be done by making one component variable, start the (auto-repeat) sweep, and play with the little up/down buttons under the plot.

Potmeter

Var can be used as a component value. You can have a resistor, capacitor or inductor with a value var or combinations of a constant and var.

Example:

```
R 2-3 var  
R 3-4 100k-var
```

If you initialise the variable dc source at the value 50k, and the stepsize as 1k, then you have made a 100k potmeter that can be changed in steps of 1%, and is initially in mid position.

There can be more potmeters, all on "one" shaft. The minimum value of "var" is clipped at 0.0001 (Ohm, Farad or Henry)

Gain control

A dependant source can have var as gain

Example: v 2-6 var, -1

The dependant voltage source between nodes 2 and 6 is var times the current in the component with label 1. "var" can be positive and negative.

Alpha control

Var can be used to drive the thyristor gate controls.

To drive the small thyristor bridge model, ("Rect" source), use **var** in the alpha field. (1 deg/V)

The large thyristor model is also controllable by **var** (30 deg/V). The synchronising sinewave generator must have a syntax like sin*2 with an asterisk in it.

Range

The up/down buttons in the sweep windows have no limit. Component values however can only be positive. Potmeters: in the example above, var= - 10k makes R 2-3 = 1micro-Ohm, R 3-4=110k

When **var** is used for alpha control, it has no limit; the same pattern repeats every 12 counts.

Exceptions

Only component values can be **var**, not the initial current or voltage of inductors or capacitors.

Diodes etc can **not** have var as on-resistance or any other parameter.

Saturable inductor can **not** have var as unsaturated inductance or any other parameter.

Version Record

1.74	5 aug 2003	Gate control has variable phase or width control
1.75	31 aug 2003	Most Recent Used circuits in Filemenu added
1.76	January 2004	Pulse Disable added
1.9	March 2005	Righthand mousebutton used for cut-copy-paste etc.

First released 2001