User's Guide

VacSim Multi

High & Ultra-high Vacuum System Design & Simulation

2 Signet Court, Swanns Road

CAMBRIDGE

United Kingdom CB5 8LA

Tel: +44-1223-516469

E-mail: info@softsim.com

Web site: http://www.softsim.com

Copyright

Getting Technical Support

For technical support, contact Technology Sources Ltd. at:

2 Signet Court, Swanns Road

CAMBRIDGE

United Kingdom CB5 8LA

Tel: +44-1223-516469

E-mail: info@softsim.com

Web site: http://www.softsim.com

TABLE OF CONTENTS

INTRODUCTION

Simulator Overview

The VACSIM MULTI vacuum simulator allows you to draw a schematic diagram of a vacuum system and to simulate the pumpdown performance. By changing the component parts and re-simulating, you can assess the influence of design changes on pumpdown times and ultimate pressures. Simulation allows you to develop a design best-suited to your particular requirements considerably faster and with less cost than the build-it-and-test-it approach.

VACSIM MULTI simulates pumpdown and pressure cycling in the high and ultra-high vacuum regimes using any combination of the 5 available working gases : Air, Hydrogen, Helium, Argon, Water and one user defined gas (Generic).

The stages in using the simulator are as follows:

1) Select some Components from the library of those available. These are represented as named boxes in the drawing window; they can be moved around by clicking and dragging with the mouse. Some components have parameters which can be set by the user (e.g. the length and diameter of a pipe); for such components you will be prompted to enter their values.

2) Each component has one or more connection ports (e.g. a

vacuum pump will have an inlet, outlet and on/off control). By clicking the mouse in one of these ports and dragging to another port on a separate component, a connection link is made.

3) You may add further components and link them up; you may also edit component parameters and delete components and links. When all components in the schematic drawing are fully linked, you can set the simulation run time, select the gases, and run the simulation.

4) As the simulation runs, new plotting windows appear and the pressures or throughputs at selected points in the system are displayed in the window. You can change the range of the plot to zoom in onto the regions of interest and measure the pressures on the plot by positioning the mouse cursor and clicking.

Chapter 3 GETTING STARTED - AN EXAMPLE PROBLEM illustrates the above process in some detail.

How the Simulator Works

The main VACSIM drawing window and associated interface allows the user to draw a connected set of items representing vacuum system components. Each component is associated with a mathematical model describing its general behaviour, and a set of parameters which specify the detailed behaviour.

The mathematical model is written in a format compatible with the SPICE electronic circuit simulator produced by the University of California at Berkeley. Traditionally, SPICE simulation is used to solve for Voltages and Currents in electronic circuits. However, there is an exact parallel between Pressures and Throughputs in vacuum systems; VACSIM exploits this isomorphism to simulate pumpdown performance.

When a schematic drawing of a vacuum system is completely linked up, VACSIM exports a set of SPICE Subcircuits, connected up to form a SPICE Circuit. The entire file is passed over to a SPICE simulator program (the “SPICE Engine”), based on release 3F4 of Berkeley SPICE). The SPICE Engine solves for the pressures and throughputs in a series of time steps and passes back the results of each time step to a set of Plotting windows which VACSIM has generated. The data values are plotted in these windows “on-the-fly” so you can monitor the simulation as it proceeds.

WHAT'S NEW IN VACSIM MULTI

The entry level vacuum simulator VacSim uses only one gas (effectively air).

VACSIM MULTI simulates pumpdown and pressure cycling in the high and ultra-high vacuum regimes using any combination of the 5 available working gases : Air, Hydrogen, Helium, Argon, Water and one user defined gas (Generic)

Throughout this User's Guide VacSim Multi will, for convenience, be referred to as VACSIM. This should always be understood as a brief form of "VacSim Multi" and not as the entry level product VACSIM.

Upgrades are available from VacSim to VacSim Multi.

NEW MULTIPLE GAS MODEL

VacSim Multi uses a multiple gas model. This is a major improvement over previous versions of VacSim, which only modeled a single gas, Nitrogen. This means you are now able to model systems with gases other than Nitrogen.

Five predefined gases have been provided with the standard component library. The gases are : Air, Hydrogen, Helium, Argon and Water.

One user defined gas (Generic) has been provided, but you will need to specify all the gas properties, and pumping characteristics, for this gas.

You may use any combination of the available gases in your simulation. For simple simulations you may only be interested in one or two gases, but more complex systems may require all gases to be simulated. To select which gases are to be used in the simulation, select the necessary gases from the Simulation Setup dialog box.

IMPROVED PLOT WINDOWS

In previous versions of VacSim, it was necessary to choose either a Pressure plotter to plot pressure values, or a General plotter to view other quantities. In VacSim Multi there is only one type of plotter which automatically uses the correct quantity, and selects appropriate units for displaying the results. It is now possible to selectively enable or disable traces in the plot window, so that the effect of certain gases, or certain inputs, can be more easily compared.

IMPROVED PARAMETER EDITOR

The component parameter editor has been revised to allow parameters to be entered for multiple gases. All the component parameters are now grouped by gas type, and are displayed on the screen in a single dialog box.

NEW USER DATABASE

The new database menu allows you to create new components from the parameters of a generic component. The components are saved into files with .USR extensions. Any file with a .USR extensdion will automatically be linked into the component database whenever the program is started.

FILE FORMATS

VacSim Multi uses a new file format. Files created with all previous versions of VacSim will automatically be converted to the new file format when they are loaded. The conversion process will automatically extend the old file, so that it may be used in a multiple gas simulation.

WARNING: The conversion process will copy the existing parameters into the gas "Air". All other gases will have their initial partial pressures set to the default values.

WARNING: If you use generic components, the file conversion process will copy your old parameters into the gas "Air". All other gases will have their parameters set to defaults. You will need to enter parameters for the other gases.

You will not be able to load files created with the NEW file format (Multi), into previous versions of VacSim.

SYSTEM REQUIREMENTS & INSTALLATION PROCEDURE

Software Requirements:

VACSIM runs under Microsoft Windows 95, Windows 98, Windows NT V3.51, Windows NT V4.0.

Hardware Requirements:

VACSIM requires an IBM PC or compatible machine with a Pentium Processor, equivalent or better.

You will need a minimum of 16Mb of RAM to run VACSIM.

The VACSIM executables and associated files occupy about 4.5MB.

Installation Procedure

To install VACSIM, start up Windows, insert the VacSim Multi CDROM, select the CDROM using Explorer and double click on "vacsimex.exe" . Installshield will open, and will prompt you for information as it is required. By default, all simulator files are stored in the same directory. The installation program will automatically load INSTALL.HLP which will give a more detailed description of what to do.

At the end of the installation process, the program will generate a message box informing you that installation is complete. You should then be able to run the simulator from "Start/ VacSim Multi/VacSim32".

Unless a dongle and valid codeword file is present, the simulator runs in demonstration mode. You can draw and simulate simple systems, but you are prevented from saving the model files or their corresponding output plots. To enable all features you must install the Dongle and a valid codeword. See: Installing the Dongle and a Codeword.

Installing the Dongle and a Codeword

The dongle fits in series on the standard parallel port. The dongle software automatically detects the location of the dongle on machines with several ports. In addition, you will require a codeword file which is stored in the same directory as the vacsim.exe executable. The name of the file is VACSIM.CWD. Technology Sources will ship the codeword with the dongle; refer to *Getting Technical Support* on page 2 of this manual for the address and telephone number.

A dongle device driver appropriate to the operating system is also required. Three different operating systems are currently supported: Windows 95, Windows 98 and Windows NT (V3.51, V4 and later The appropriate drivers (and instructions for installing them) are contained in the WIN31&95 or WINNT subdirectories on Vacsim Disk 1. Refer to the help file INSTALL.HLP for a detailed description of how to install the dongle device drivers.

For Windows 95 and Windows 98 run VXDINST.EXE in the WIN31&95 directory of Vacsim Disk 1. This installation program will ask you to restart the computer. After you have restarted the computer, you will need to enter your codeword into the codeword file VACSIM.CWD using notepad.

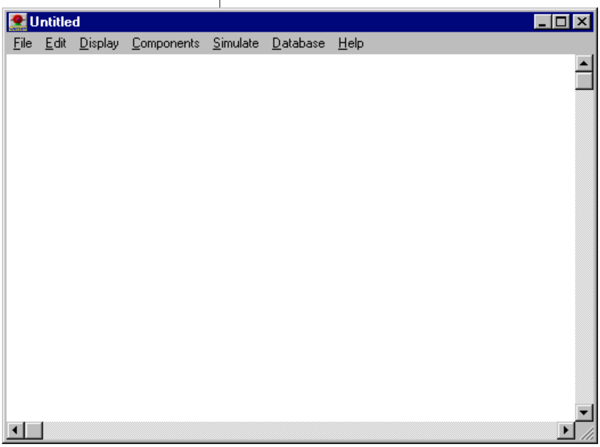
Windows NT users will have to log on as the system administrator, and install the device drivers from the control panel. In version

3.51 you must select Drivers, and in version 4.0 you must select Multimedia and then Devices/Multimedia Drivers. Enter the location of the DK12DRV and OEMSETUP.INF files (enter A:) and select OK. From the list of drivers choose the 'DK12 DESkey Driver' and select OK. Set the driver options to match your LPT configuration and Press OK. The driver should now be installed.

GETTING STARTED - AN EXAMPLE PROBLEM

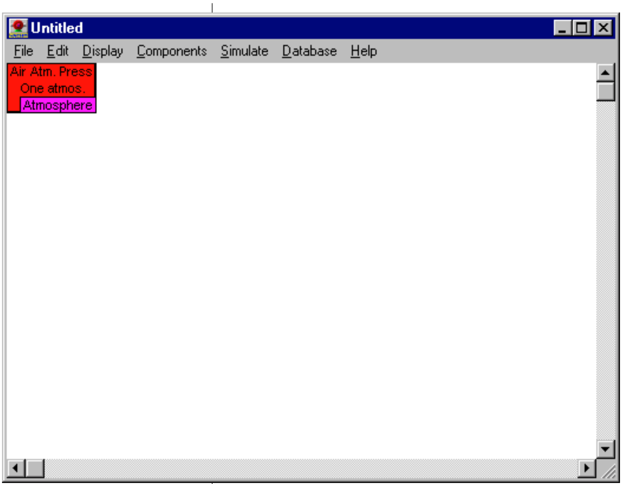
Overview

The following Step-by-Step Procedure describes the setting up of a simple problem and illustrates some of the features of the simulator. If you are using online help, you can display the procedure in a suitably sized window adjacent to the VACSIM drawing window and scroll through the steps as required. Note that if the program is running in demonstration mode due to the lack of a dongle or valid codeword, certain operations described (mainly associated with saving files to disk) will not operate and will result in a warning message. Nevertheless, the simulation program will still run.



Step-by-Step Procedure

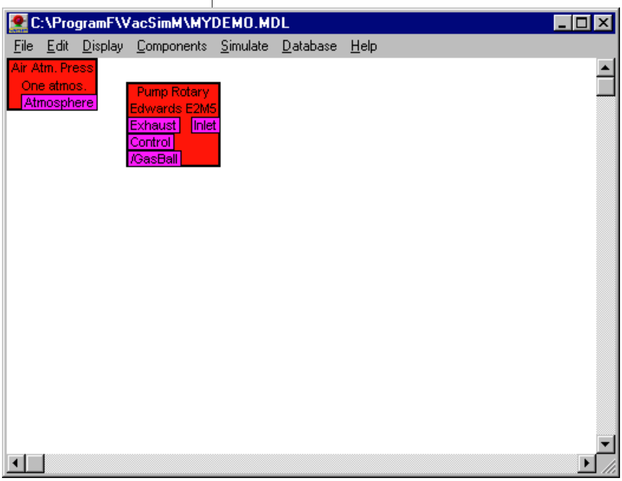
1. Having started VACSIM, the main drawing window should appear bearing the title "Untitled". If you have accidentally started the simulator with an existing drawing, select *File/New* from the menu to discard the drawing and

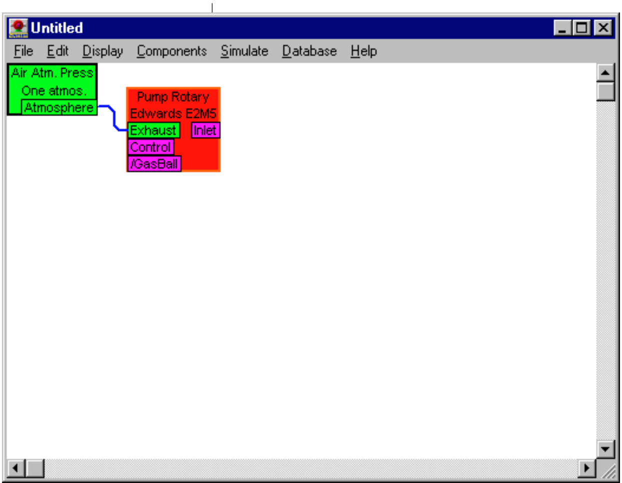


start with a clean sheet.

2. Click the left mouse button somewhere near the top left corner of the drawing window. This defines the insertion point about which a new component will be centred. Now select *Components/Add Component* from the menu; a list of available components will appear. From this list, select *Air*, either by single clicking with the left button on the word *Air* followed by a click on the *OK* button, or by double clicking on *Air* with the left mouse button. A second Component Selection dialog box appears from which you should select *Atm.Press*: this represents air at 1 standard atmosphere pressure, comprised of all available gases. A red box should appear in the drawing window; it will have one connection port which is magenta.

3. Repeat the previous step, selecting Pump for the first component selection and Rotary for the second. A scrollable list of different rotary pump models is presented, from which you should select *E2M5*. At this point, you will be prompted for the Initial Pressures - that is the pressures of the available gases, in the internal volumes of the pump when it is first switched on. Select *OK* to take the default of 1 atmosphere. A red box with 3 connection ports should appear.

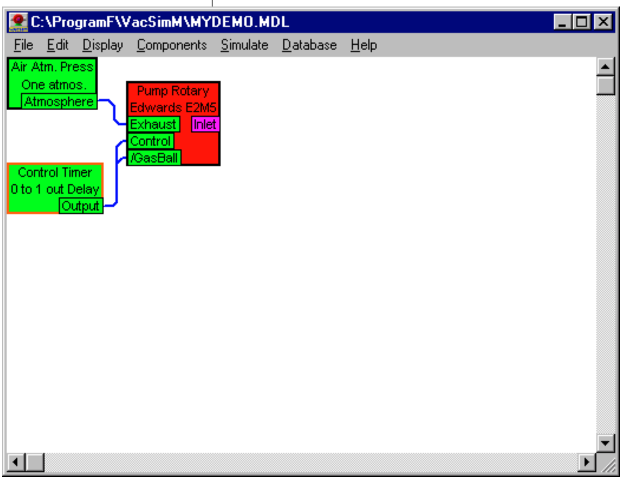


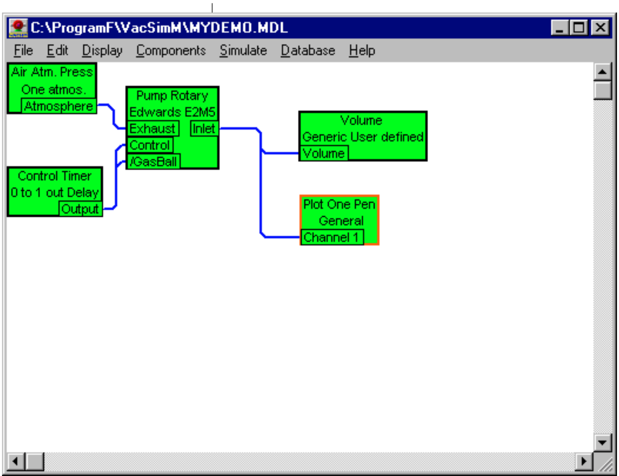


4. Re-position one or both components by clicking the left button down whilst the tip of the arrow cursor is over the **Red** area of a box and then dragging the mouse until the box outline is in the desired position. Releasing the mouse button completes the move.

5. Now link the Atmosphere port of the Air box to the Exhaust port of the rotary pump (the pump vents to air). Do this by clicking the left button down when the cursor tip is over the magenta Atmosphere port and drag the mouse until the cursor tip is over the Exhaust port of the rotary pump. Then release the mouse button: a connecting line is drawn. Note also that the linked connection ports turn from magenta to green and that the Air component box turns from red to green, signifying that all its ports (all one of it) are connected.

6. At this point, you can save the model by selecting <u>*File/Save As*</u> from the menu. A file dialog box appears, listing the existing models. You should type in a new name for the file with a **.MDL** extension. e.g. MYDEMO.MDL. Then select *OK* to save it. Note that the drawing window title now contains the full file name. If the program is running in <u>demonstration mode</u>, you will not be able



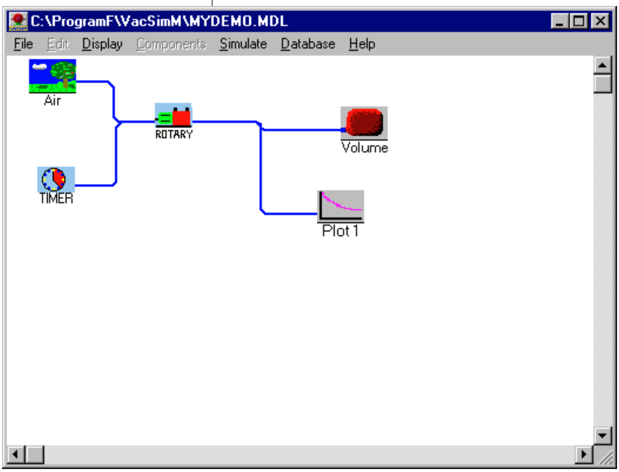


to save the file.

7. Add more components:

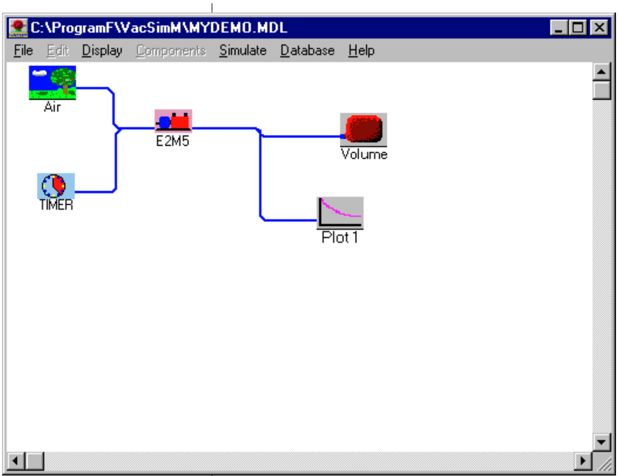
· *Control/Timer/Delay*. Take the default delay of 0 seconds. Link its output to the Control of the rotary **pump and to the /GasBall control input of the rotary** pump.

· *Volume.* In the parameter entry dialog, you will see that the parameter "Volume" is in a section called "General". This means this parameter applies to all the available gases. In the "Air" box outline click on the down arrow in the drop down list box containing the default units of atmosphere; click on the torr unit. Enter the value 760. Press *OK* to accept this default. Link the port of the volume to the rotary pump inlet. If the program is running in demonstration mode, a message box will inform you that editing Generic components is disabled and a default volume of 10 litres at 1 atmosphere initial pressure will be created.



· Plot/One Pen. Accept all defaults, then connect this to the Inlet of the Rotary pump (this is equivalent to connecting it to the Port of the Volume because the link lines represent logical connections and have infinite conductance).

8. If you are not



running in demonstration mode save the modified model using *File/Save*; it is now ready to simulate. However, before doing the simulation you may care to try out some of the other features of the drawing window. These are described in the next few steps.

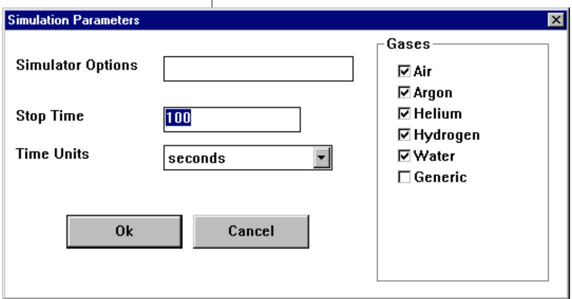
9. Select *Display/ System Bitmap* from the menu. The schematic boxes are replaced by bitmap representations of the items. Selecting *Display/ Component Bitmap* shows specific icons for individual components. The Rotary Pump bitmap is replaced by one bearing the E2M5 legend. If a specific component bitmap is not found, the System bitmap is displayed. Note that you cannot edit the drawing or add components when bitmap images are displayed (the main menu bar items are greyed). Select *Display/Colour Schematic* to get back to the standard view. Selecting *Display/Mono Schematic* shows the schematic diagram in black and white - this is useful when printing the diagram to a black and white printer c.f. *File/Print*. Select *Display/Colour Schematic* to get back to the standard view.

10. You can select a component within the drawing as the current component by clicking the mouse within its main body area (not within a port). The outline of the component changes from black to red. Select the Rotary pump, then select *Edit/Delete* from the menu. The pump and all its connection links disappear. Select *Edit/Undo* to restore them. You can also *Cut* and *Copy* the current object and *Paste* the copy back into the drawing using commands from the *Edit* menu option. Selecting multiple components is achieved by dragging a chooser rectangle around the target

group, or by CTRL-clicking unselected components. When multiple components are selected the outline of each selected component changes colour from black to yellow.

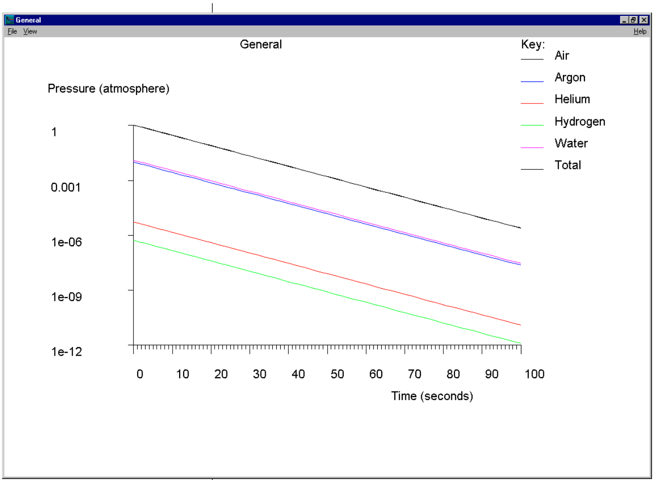
11. To select a link line, as opposed to a component, click the mouse button when the cursor is over the line. The selected line changes from blue to red. *Edit/Delete* deletes the link and *Edit/Undo* restores it. There are no Cut, Copy and Paste operations available for the link lines. By default the vertical section of the link line is drawn midway between its end points. If you click on the link line and drag the mouse, the vertical section is repositioned to the horizontal position of the mouse. On releasing the mouse button, the line will be re drawn at that position unless there is an overlap between two lines not sharing a common end point, in which case the vertical section will be moved automatically to eliminate the overlap.

12. Select the Timer as the current component and select *Components/Edit Parameters* from the menu. If a component does not have any editable parameters, the menu option is greyed. Double clicking with the left mouse button achieves the same result. The *Components/Substitute Component* lets you replace a component of one type with another of the same type (that is, the same number and types of inputs and outputs) so that you can swap one manufacturer's pump for another. The links to other components within the drawing are retained.



13. The *File/Printer Setup* menu selection allows you to configure your printer prior to issuing the *File/Print* command to print the schematic drawing. You can also save a bitmap representation of the drawing to the clipboard by using the *File/Clip* menu command.

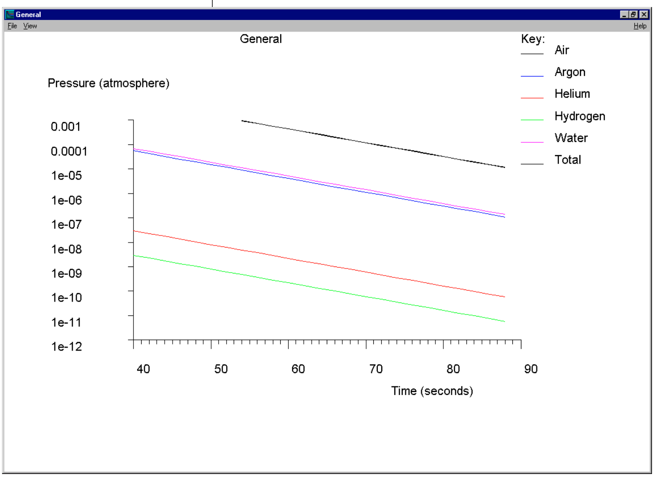
14. Save the modified model



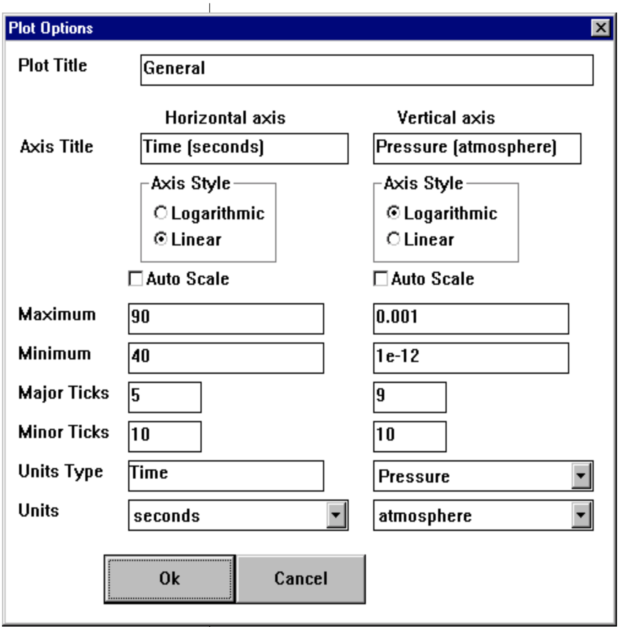
using *File/Save*. Set up the simulation run time parameters by selecting *Simulate/Setup* from the menu. Accept the default options by selecting *OK*. Select *Simulate/Go* to start the simulation. There will be a slight pause, then the SPICE Engine program will execute. Wherever there is a Plotter in the schematic drawing a small plotter window will appear and data from the SPICE Engine will be plotted in it.

15. You can Pause the simulation by selecting *Simulate/Pause*. Selecting *Simulate/Continue* allows the simulation to run to the end. Note that when the simulation is paused you can use *Simulate/Stop* to abort the run before the simulation has completed.

16. You can click and drag the corner of the plotter window to increase its size.



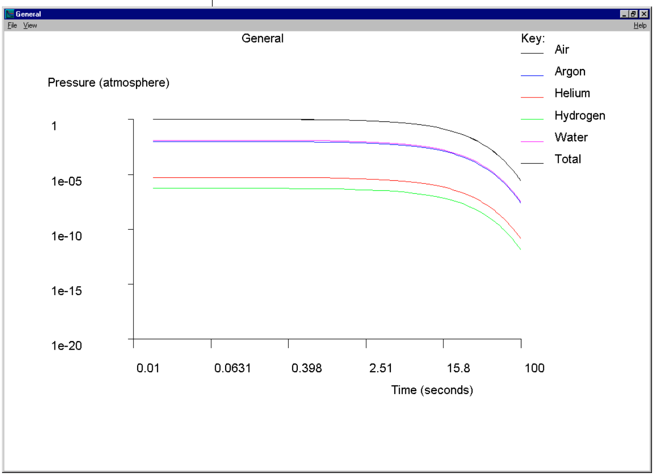
You can magnify a region of the plot by using the zoom feature in the plot window. Move the cursor to one corner of the region you wish to magnify, click down on the left button and drag the cursor until it reaches the opposite corner of the region of interest, then release the mouse button. The plot will be re-drawn to include the



region you selected. To undo the effect of zooming, select View/Unzoom from the menu. To examine the co-ordinates of the current cursor position, select View/Position from the menu.

17. You can call up a dialog to change the plot axes style, units of measure, units type and titles by selecting View/Set Axes. As an example, try changing the axis style on the horizontal axis from linear to logarithmic by clicking on the *Logarithmic* radio button in the *Horizontal Axis Axis Style* group. Uncheck *Autoscale,* set *Minimum* to 0.01 and *Maximum* to 100. Select *OK* to close the dialog and make the change take effect.

18. Clicking on the left-hand button in the top right corner of the Plot window will iconise the window and cause it to be placed over the schematic diagram box for the plotter. If you scroll the main drawing window using its scroll bars, the icon position will scroll to follow its schematic box.



19. To leave the program select <u>*File/Exit*</u>. If the drawing has changed since you last saved it, you will be prompted to save it, if not the program will exit immediately.

Refer to <u>Where to go next</u> for references to other useful documentation.

Where to go next

VACSIM is shipped with a few example schematics; these are called DEMO1.MDL, DEMO2.MDL etc. Others illustrate the use of particular vacuum components – ROTARY.MDL, ROOTS.MDL etc. You should look at these to gather some idea of how they work and read their associated notes DEMO1.WRI, DEMO2.WRI etc.

Refer to CHAPTER 4 - VACUUM SYSTEM DESIGN GUIDELINES for a general introduction to modelling vacuum systems.

CHAPTER 5 on VACUUM SYSTEM COMPONENT MODELS gives details of how the various components are modelled and highlights important aspects of the individual models.

CHAPTER 6 - DETAILED PROGRAM REFERENCE describes the VACSIM user interface and system operation in detail.

VACUUM SYSTEM DESIGN GUIDELINES

Purpose of Simulation

The purpose of your simulation is to model a system with sufficient accuracy to allow you to make design decisions based on the model.

The model is **NOT**, and should not be, a perfect representation of the real physical system. You should only include those aspects of the system which you consider to be significant.

The Multiple Gas Model

VacSim Multi now uses a multiple gas model, rather than the single gas model, which was used in previous versions of this program. This means VACSIM can now simulate the effect of pumps having different characteristics for different gases. Since the computational overhead of using many gases is very large, and due to a relative lack of information about various components, only a number of gases have been provided in the standard libraries.

A "Generic" gas has been provided, so that you may specify a custom gas, to use in the simulation. This may only be used if all components in the simulation are Generic. The Setup Simulation dialog will not allow you to select the "Generic" gas if any component is not generic. For example, you will need to use Air - Set Press, rather than Air - Atm Pres. This is necessary so that the partial pressure of the generic gas in air may be specified.

Modelling Vacuum Components

VACSIM provides a set of component parts which can be added to a system. These have been developed and tested in different system models and give satisfactory convergence behaviour.

The GENERIC component models allow you to specify all the parameters which are passed to the SPICE model of that particular component type. This flexibility carries with it the possibility of creating unrealistic system models (see Avoiding Convergence Problems) so you must choose the parameters for your GENERIC components with due care.

Modelling Vacuum History

Materials absorb gas when at atmospheric pressure and liberate it when placed in a low pressure environment.

The rate of evolution of gas depends upon the material and the concentration of absorbed gas within the layers close to the surface. This concentration is determined by how long the material has been exposed to normal atmospheric pressure. Hence the outgassing performance is a function of the complete “vacuum history” of the materials in a system.

VACSIM’s SPICE models can model this type of outgassing/absorption behaviour, but for simplicity assume that the materials have been fully equilibrated at a user-specified pressure at the start of the simulation. More elaborate vacuum histories can be generated from this initial state by using a series of timers to turn pumps on and off (See Control and Logic).

Control and Logic

VACSIM’s libraries include control and logic elements including:

- Timers

- Comparators
- Boolean logic devices

These devices can be used to start and stop pumps, or open and close valves, based upon time delays, pressure levels, or combinations of these entities.

Avoiding Convergence Problems

The SPICE circuit simulator (Berkeley SPICE 3F4) on which the SPICE Engine is based has been used to simulate electronic circuits for many years. General experience indicates that the numerical algorithms converge to a satisfactory solution, provided the system model does not contain any “unphysical” behaviour.

An example of an unphysical system model is two volumes at different pressures separated by a valve of infinite (or unrealistically high) open conductance. The simulation will proceed satisfactorily until the valve is opened, at which point a convergence error may occur as the simulator tries to model an infinite gas flux.

Error messages are logged in the SPICESTD.ERR simulator status file. The “Timestep too small” message is typical of a convergence error. On rare occasions convergence problems result in floating point errors which SPICE does not trap; in these cases Windows will produce an appropriate message.

The standard component models in the Component Library have been designed to minimise the risk of convergence problems. However, where components have user-editable parameters, those parameters (such as volume, conductance and outgassing rates) do give the scope to generate unphysical models. These types of components should be used with care, and if convergence problems occur should be the prime suspects for the cause.

The Role of the Vacuum System Designer

VACSIM has been designed to help vacuum system engineers build complex computational models through an easy to use graphical user interface and to evaluate the relative merits of different vacuum system configurations by comparing the results of simulations.

Only **you**, the system designer, can define the relative importance of the many different performance aspects of a system. VACSIM is a tool to help you investigate how changes in your design affect **your** measure of performance. You can then make the design decisions based on system cost, pumpdown time, ultimate vacuum etc.

VACSIM will attempt to model whatever you specify, whether or not it is what you intended to specify. As with all simulation exercises, you should bear this in mind when interpreting the results of a simulation.

Technical References

The principal technical references used to develop the SPICE models of vacuum components were:

John F. O'Hanlon "A User's Guide to Vacuum Technology"
ISBN 0-471-01624-1 John Wiley & Sons, Inc. 1980.

Alexander Roth "Vacuum Technology"
ISBN 0-444-10801-7 North Holland 1976.

VACUUM SYSTEM COMPONENT MODELS

General Description

The SPICE models of vacuum components represent the pressure and the gas flow in a vacuum component by the voltage and electric current in an equivalent electronic circuit. SPICE supports a range of components, including resistors, capacitors and non-linear voltage-controlled voltage sources. Complex mathematical models approximating the real behaviour of vacuum components have been constructed by combining these elementary components.

Events can be triggered by pressures crossing threshold values, on elapsed time or on logical combinations (AND, OR, NOT) of these conditions. All control signals use a positive logic convention so that applying a logic level of 1 to a control input labelled "Open" on a normally closed valve will cause the valve to open. Similarly, a comparator whose output is labelled P < Pset will give a logic 0 output when P >= Pset and a logic 1 when P < Pset.

The SPICE models of most vacuum components are "physical" models rather than "parametric" models. Refer to the Technical References for the basis of the models.

A physical model of a vacuum component has processes occurring within it which are isomorphic with the flow of gas in the component. In contrast, a parametric model is simply a "curve fit" to the performance data for the component over a specified region of its operation.

The main advantage of using physical models is that a large number of similar components can be represented by one underlying model. We refer to the underlying model as the Generic model. Specific components are modelled by substituting a small number of parameters into the Generic model. The parameters are derived directly from the physical characteristics of the component.

The Generic rotary pump model has parameters for the Compression Ratio, Pump Speed and the Initial Pressure of the internal volumes of the pump. This model can represent virtually all rotary pumps and their initial conditions.

Where a GENERIC component is made available in the Component Library, you have access to all the parameters so you can construct a specific model for a component which is not already contained in the Component Library. The underlying generic model is extracted from the Model Library.

Generic Components

The following components are included in the system component and model libraries.

AIR

Air - Atm. Press
Air - Set Press

VOLUME

Volume

BACKING AND BOOSTER PUMPS

Booster Pump
Drypump

Roots Pump

Rotary Pump
Scroll Pump

HIGH VACUUM PUMPS

Cryopump
Ion pump
Sublimation Pump
Turbo-molecular pump
Diffusion Pump
Baffle
Diffusion pump valve
Diffstack

OUTGASSING

Outgassing - bakeable
Outgassing - fixed temp

PIPEWORK

Pipework - aperture
Pipework - elbow
Pipework - straight

VALVES

Valve

CONTROL

Heater
Timer
Trip
Current Pulse
Current PWL

MISCELLANEOUS

Logic
Log_ramp
Gauges

Plotters
Leak - constant

TEST SCHEMATICS

Pump test schematic
Outgassing test schematic

Usage of these components

Air - Atm. Press

This model is used to represent connection to the atmosphere for rotary pump exhausts, air admit lines, etc. It takes no parameters and is set at one standard atmosphere. When multiple gases, are being simulated, then the partial pressures of the gases are set so that they repesent the partial pressures of air at sea level.

The partial pressure of water vapour is set to 1165 Pa (8.75 Torr) which represents a humidity level of 20% at 293K. If you are using Water Vapour in your simulation, and the conditions in your simulation, do not match these assumptions, then you will need to use the Generic, Air - Set Pressure Component.

If you are simulating a generic gas, then you will need to use the Air - Set Pressure component, rather than Air - Atm. Pres

Simulation Parameters

None

Air - Set Press

This component models an infinite volume of gas at a given pressure. It can be used to model pressures other than 1 atmosphere, or can be used to change the composition of the air being simulated.

You can change the gases that are used in a simulation by editing the "Selected Gases" in the Simulation Setup dialog.

If you are using a Generic Gas, then you must use Air - Set

Press, rather than Air - Atm. Press

Note that once set the pressure in this component will not change during the simulation. It can be thought of as an infinite volume containing gas at the set pressure.

Simulation Parameters

Pressure

The partial pressure of each gas present in the infinite volume.

Volume

This can be used to represent the gas capacity of a volume the conductance of which is large compared with the elements that are connected to it. Such a volume would be expected to have a uniform pressure throughout.

This component may be used with the "Generic" Gas.

Simulation Parameters

Pressure:
The pressure in the volume at the start of simulation.

Model parameters

Volume
The internal volume of the region.

Booster Pump

This model simulates the Edwards range of mechanical booster pumps and is not currently available in a generic form. The Booster pump uses a hydrokinetic coupling between the pump and the drive motor, to ensure that the motor does not overload at high inlet pressures. As a result, the booster pump and backing pump may be switched on together at atmospheric pressure. Generic Gases are not supported for this component.

Simulation Parameters

Initial pressure:
The partial pressure of the pump internal volume at the start of

simulation.

Mains Freq:
This specifies the supply frequency of the electrical supply. Typical values are 50Hz or 60Hz

Drypump

This model simulates the Edwards range of oil-free rotary pumps and is not currently available in generic form. The drypump uses a combination of roots stages and rotating claw. The DP range of drypumps are intended for chemical pumping and have lower compression ratios than the QDP range. Generic Gases are not supported for this component.

Simulation Parameters

Initial pressure:
The partial pressure of the pump internal volume at the start of simulation.

Mains Freq
This is the frequency of the electrical supply. Typical values are 50Hz or 60Hz.

Roots Pump

This model simulates the Leybolds range of roots pumps and is not currently available in generic form. The Leybold Roots Pump uses a pressure release valve connected from the inlet to the exhaust, to prevent the pump from overloading at high inlet pressures. The valve opens if the pressure difference exceeds 40 mBars.

The roots pump is designed to be used in conjunction with a rotary backing pump, and may be switched on simultaneously with the rotary pump. The pump may be controlled using the logical "Control" input.

This component may not be used in a simulation that uses the "Generic" gas

Simulation Parameters

Initial Pressure
The partial pressure of the pump internal volume at the start of simulation.

Mains Freq
The frequency of the electrical mains supply. Typical values for this parameter are 50Hz and 60Hz.

Rotary Pump

The model is effective for both single and two stage pumps. Gas ballast operation can be turned on/off through a /GasBall control input. This can be linked to the same control source as the Control input if gas ballast operation is not required since the input is logic low for gas ballast.

This component can be used in a simulation that uses a Generic Gas. Before you can select the "Generic Gas" for simulation, ensure that all components in the simulation are Generic Models. You will need to use the Generic User defined rotary pump, so that the pumping characteristics of the new gas can be entered.

Note: The Edwards SPEEDIVAC-2 pump is called SP2 in the component database.

Simulation Parameters (gas specific)

Initial pressure:
The pressure of the pump internal volume at the start of simulation.

Model parameters (general)

Diameter
Provides the speed limiting effect of the input aperture at low pressure.

Mains Freq
This is the frequency of the AC electrical supply. Typical values are 50Hz and 60Hz.

Model parameters (gas specific)

Compression ratio
Defines the ultimate vacuum for the pump.

Speed
Sets the pumping rate at input pressures near exhaust pressure.

CRgasball
Defines the ultimate vacuum with gas ballast on.

Pipe Cond
Specifies the conductance of the gas in a pipe in the viscous region. (See Pipework Model)

Deriving from Manufacturers data

Compression ratio.
Divide atmospheric pressure by the quoted ultimate vacuum using the same units to obtain a ratio.

Speed
This model assumes the motor speed, and hence pump speed, are a function of the supply frequency. This parameter specifies the 50Hz pump speed. If the pump is run at 60Hz, then the value used in the simulation will be this Speed value multiplied by 60/50. (ie 20% faster)

Diameter
The input port diameter may not represent the actual input restriction, the value may require adjustment to get best fit to the manufacturer's speed/pressure curve.

See Pump test schematic

Scroll Pump

This model simulates the Edwards range of Scroll Pumps, models ESDP12 & ESDP30A, and is currently not available in Generic form. The Edwards scroll pump uses intermeshing orbital and stationary scrolls to form non-contacting crescent shaped gas regions which are continually compressed. This pump requires no sealants and lubricants, and provides an oil free alternative to oil sealed rotary pumps.

Simulation Parameters

Initial pressure
The partial pressure of the pump's internal volume at the start of simulation.

Mains Freq
This specifies the supply frequency of the electrical supply. Typical values are 50Hz or 60Hz.

Cryopump

This model is for a two-stage closed circuit pump. Pump cooldown and crossover are modelled but gas release on warmup is not. The changes of gas vapour pressure with temperature are used as the basis for the model for gases pumped by condensation and freezing. For gases such as Hydrogen and Helium, which pump by adsorbtion onto activated charcoal, the model computes the equilibreum pressure of the charcoal from the temperature. The dynamic temperatures of the two stages are evaluated taking into account radiation and gas load heat input. The vapour pressure of the gases are calculated from the second stage temperature. If the gas pressure in the pump chamber exceeds the nitrogen vapour pressure gas is absorbed into the pump. Gas is not released in the opposite circumstance since this would require a knowledge of the accumulated gases in the pump.

The Temp1 and Temp2 connections are used to plot the dynamic temperature of the 1st and 2nd stages. This component may be used with the "Generic" Gas.

Simulation Parameters (Gas specific)

Initial pressure
The pressure of the pump internal volume at the start of simulation.

Simulation Parameters (General)

Hot
When set at 1 the pump is initialised at room temperature,

set to 0 the pump is initialised at its operating temperatures.

Model parameters (General)

Internal volume

The internal volume of the pump

TC1

The thermal capacity of the first stage of the pump

TC2

The thermal capacity of the second stage of the pump

TCON1A and TCON1B

The thermal conductance from the first stage to the refrigerator is split, TCON1A is from the first to second stage of the refrigerator while TCON1B is from this point to the first stage array.

Rad input

The total radiation heat input to the first stage.

TR1

The zero heat load temperature ratio across the first stage of the refrigerator.

TR2

The zero heat load temperature ratio across the second stage of the refrigerator.

TCON2

The thermal conductance from the second stage of the refrigerator to the second stage block.

TCON3

The thermal conductance from the second stage array to the block.

TC3

The thermal capacity of the second stage array.

Model parameters (Gas specific)

Speed

The pumping speed for each gas

Tmin and Tmax

These two constants describe the vapour pressure curve for gases pumped by condensation, (eg Ar, N2, H20) but for light gases such as H2 and He which pump by adsorbtion, these parameters describe the adsorbtion equilibreum presssure curves.

Specific Heat

The specific heat of the gas. This can be found in most databooks.

Heat VapFus

This value is the sum of the heats of vapourisation and heats of fusion for the gas.

Pump At Stage1

This parameter specifies where the gas will condense. If 1, then the gas will condense on the inlet array, otherwise if it is 0 the gas will be pumped at the second stage array.

Pump By Adsorbtion

This parameter specifies the mechanism used to pump each gas. If this value is one then the gas will be pumped by adsorbtion onto activated charcoal. If this value is zero then the gas will be pumped by condensation and freezing. The value of this parameter determines how the parameters Tmin and Tmax will be used.

Deriving from Manufacturers data

TC1 and TC2

These may be estimated from cool down times and the thermal conductivities. The cool down time constant is the product of thermal capacity and conductance and is in seconds.

TCON1A and TCON1B

The sum of these may be estimated from the change in first stage temperature with radiation heat input. The rise in second stage temperature with radiation input is likely to be less than that of the first stage, the ratio of TCON1A to TCON1B can be calculated from this.

Rad input

This should include radiation from the pump wall and from the work chamber.

Speed

Use the manufacturer's specified speed for each gas

TR1 and TR2

These are not usually given in manufacture's data but may be given in technical briefs or applications articles. The first stage zero heat load temperature is usually around 35K with the second stage at 10K. With ambient at 300K this gives TR1 = 8.57 and TR2 =3.5.

TC3 and TCON3

Crossover is usually quoted for the gas load that will increase the second stage temperature to 20K. The heat from this gas load will hit the second stage quite rapidly so that the heat does not have time to diffuse into the total thermal capacity of the second stage. To predict the crossover behaviour the model introduces a small thermal capacity, TC3, into which the heat is initially transferred before diffusing through the thermal conductance ,TCON3, to the second stage.

The total heat energy in that gas load (that in the gas plus latent heats of fusion and vaporization) with the allowable temperature rise will give the thermal capacity (joules per degree K), TC3. The thermal conductance, TCON3, will affect the recovery time but data on this is likely to be hard to find. Some small fraction of TCON2 is probably the best guess.

Tmax and Tmin

(for vapour pressure Ar, N2, H20, Pump By Adsorbtion = 0)

The temperature Tmax can be found in most databooks specified for each gas as "temperature nbp liq" (eg O'Hanlon), but alternatively can be approximately read from a graph by taking the point at which the vapour pressure curves cross 1 atmosphere. Tmin can be read from a vapour pressure curve by reading off the temperature at which the gas reaches 1e-14 atmospheres.

Tmax and Tmin
(for adsorbtion equilibrium pressure H2, He, Pump By Adsorbtion = 1)

Gases such as H2 and He are pumped by adsorbtion onto activated charcoal, which is usually situated on the rear side of the second stage array. At any time, molecules will enter and leave the surface, and the ultimate pressure is reached, when the two flows are in equilibreum. This pressure is dependant on the Temperature and the amount of gas already adsorbed. This model does not model vacuum history, so it will be necessary to choose a point near the operating point, say 200 sccm H2/gm charcoal. Plot the curve of equilibreum pressure vs Temp and read off the temperatures at which it crosses 1atm and 1e-14 atm. Use these temperatures for the two parameters Tmax & Tmin respectively.

Ion pump

The generic ion pump model gives reasonable results across the range of pump types from minimum input data. At high pressures where flashover and damage might occur in a real pump the model exhibits zero pumping rate.

This component may be used with the "Generic" Gas.

Simulation Parameters (gas specific)

Pressure
The partial pressure of the pump internal volume at the start of simulation.

Ultimate Pressure
The pressure (low) at which the pump speed for a particular gas falls to zero.

Model parameters (general)

Volume

The internal volume of the pump

Model parameters (gas specific)

Speed
The maximum speed of the pump

Deriving from Manufacturers data

Volume
This is not usually quoted but may be estimated from the pump body external dimensions.

Speed
This may be quoted at a number of pressures and for different gases, take the highest value given for Nitrogen (usually at around 10e-6 millibar) to give a reasonable approximation for dry air.

Ultimate Pressure
May be extrapolated from manufacturers pump speed curves.

Sublimation Pump

The sublimation pump model was designed to model a Titanium Sublimation Pump, but could equally well model sublimation pumps which use materials other than Titanium, if the appropriate parameters are specified. This model simulates the rate at which titanium atoms are generated at the filament, and the build up of atoms on the pumping surface. At low pressures the model is limited by the pumping surface area, whilst at higher pressures the model is limited by the titanium sublimation rate.

This model does not model molecule replacement at the pumping surfaces, but it does model the build up of atoms on the pumping surface. The coverage of titanium atoms is limited at different levels for the different gases.

Model Connection Ports

Current
Filament control input port. An electrical current must be provided to heat the filament. The model assumes that a current of 1 Amp is equivalent to a sublimation rate of 1g/hour.

Inlet

Pump inlet port. Connect this to the pump chamber. If the pump is housed inside the test chamber then set the Area Inlet parameter to a large value.

Model Parameters (General)

Area Pump

The area of the pumping surface. If the pump is enclosed, then use the surface area of the pump chamber. If the pump consist of an open filament in the test chamber, then use the surface area of the test chamber.

Area Inlet

The inlet to an enclosed pump is modeled as an aperture. This parameter specifies the area of the aperture, for enclosed pumps. If the pump is not enclosed, then use the cross sectional area of the chamber

Model Parameters (Gas Specific)

Stick

The sticking coefficient of the gas. This represents the probability that a molecule striking the surface will combine with a titanium atom, for a well covered surface.

n

The number of titanium atoms needed to pump one molecule of each gas.

eg. $N_2 + 2Ti \Longrightarrow 2TiN$ so n=2

Ap Mol Cond

The conductance of the gas in the molecular flow region of an aperture. This value can be computed as 1000*v/4 where v is the average velocity of the gas.

Capacity

The capacity of the pump is specified as the maximum coverage of atoms per square cm of pumping surface for each gas.

See Also

Current Pulse
Current PWL

Turbo-molecular pump

This model gives good results for most types of pumps, including those with molecular drag stages.

The component is also available in a Generic form, and may be used in a simulation that uses the "Generic" gas provided that the User Defined component is used.

Simulation Parameters (gas specific)

Initial pressure:
The pressure of the pump internal volume at the start of simulation.

Model parameters (general)

Volume
The internal volume of the pump

Ultimate
Sets the practical ultimate vacuum for the pump.

cp - Critical pressure
Sets the backing pressure for the knee in the compression ratio to backing pressure curve.

Stages
Gives the slope of the compression ratio to backing pressure curve for

pressures above the knee (cp).

Rupt
The time for the pump rotor to come up to 90% speed

Model parameters (gas specific)

Compression ratio
Sets the theoretical ultimate vacuum for the pump.

Rate (Pump Speed)

Sets the pumping rate at the middle of the working pressure range.

Deriving from Manufacturers data

Compression ratio
This is usually stated for a variety of gases and/or given as a graph against backing pressure. (For "Air" use the nitrogen value)

Note: The ultimate pressure quoted for these pumps cannot be used as an indication of compression ratio as they are limited by materials outgassing.

Ultimate
The practical limit for ultimate pressure is usually quoted at some number of hours following bakeout. The "Ultimate" parameter represents this limit. For detailed analysis during pumpdown material outgassing should be linked to the pump to represent its' internal surfaces. The ultimate pressure is limited by the compresssion ratio of the lighter gases, and materials outgassing. The model will always limit at the ultimate (total) pressure, but in a more accurate analaysis including outgassing, and with the light gases being simulated, the model should naturally reach a total pressure which puts the model at the point of limiting.

Speed
This is usually quoted and given as a graph against inlet pressure. Again use the Nitrogen figure as a guide for dry air.

Volume
The internal volume can be estimated from the external dimensions of the pump.

Rupt
If the Manufacturer quotes other than to 90% speed then some adjustment must be made to his figure to approximate the 90% time.

cp
Where a log-log graph of compression ratio against backing pressure is given take the pressure at the intercept of the straight portion of the falling part of the curve with the horizontal line at the maximum compression ratio . In the absence of this information

use the built in default value.

Stages
Where a log-log graph of compression ratio against backing pressure is given take the slope (decades of compression ratio per decade of pressure) of the falling part of the curve. In the absence of this information use the built in default value.

Power
The maximum power supplied to the pump by the power supply.

See Pump test schematic

Diffusion Pump

The diffusion pump model includes an oil backstreaming output which can indicate whether or not the pump is being used under correct operating conditions. Oil flow into the backing line is not included. The heater is on for an input greater than 0.5.

This component may be used with the "Generic" Gas.

Simulation Parameters

Initial pressure:
The pressure of the pump internal volume at the start of simulation.

Model parameters

Compression ratio
Sets the theoretical ultimate vacuum for the pump.

Speed
Sets the pumping rate at the middle of the working pressure range.

cp - Critical pressure
The backing pressure above which pumping stops.

Stages
Relates to the effective number of stages in the pump and sets the rate at which pumping speed reduces with rising inlet pressure in the overload region.

Warmup
The time taken for the heater to bring the pump to its' operating temperature.

Backstream
The rate at which oil backstreams from the pump under normal operating conditions.

Deriving from Manufacturers data

Compression ratio
This is not usually quoted for diffusion pumps and must be calculated from the ratio of backing pressure to ultimate pressure. The ultimate pressure is a function of the oil used.

Speed
This is usually quoted and given as a graph against inlet pressure. The curves are usually given for air which is appropriate to the Vacsim simulator.

cp
Critical backing pressure

This may be quoted for no load or full load, if both are quoted use the no load figure.

Stages
This gives the rate of fall of pumping speed with rising inlet pressure and should ideally be taken from a log-log graph of speed versus inlet pressure.

The default value of 4 gives acceptable results for most pumps.

Backstream
When quoted usually given in mg/sq.cm/min. This figure should be multiplied by the pump input area and entered in gm/min. In the absence of any data 0.001 mg/sq.cm/min is a typical figure.

Warmup
Use the quoted figure.

See Pump test schematic

Baffle

The baffle model is intended for use with the diffusion pump model and gives the oil backstreaming attenuation as well as limiting the pumping speed due to finite conductance.

This component may be used with the "Generic" Gas.

Model parameters

Conductance
The baffle conductance.

Atten
The attenuation in backstreaming rate across the baffle.

Deriving from Manufacturer's data

Conductance
May be quoted directly or as a reduction in pumping speed. If given as the latter then the conductance can be calculated from the relationship that the reciprocal overall speed is the sum of the reciprocal pump speed and reciprocal baffle conductance.

Atten
If not quoted as a ratio may be given as a maximum backstreaming rate in mg/sq.cm/min. Divide this by the pump backstreaming rate to get a ratio.

Diffusion pump valve

This valve model is intended to be used in combination with the diffusion pump and baffle models. It behaves as a normally open valve but includes integration of the oil backstreaming from the pump to give the total mass of oil accumulated on the system side of the valve.

This component may be used with the "Generic" Gas.

Model parameters

Area
The effective area of the valve when fully open.

Mol Cond

Molecular conductance of a circular aperture, see Pipework Aperture

Visc Cond
Viscous conductance of a circular aperture, see Pipework Aperture

Diffstack

The diffstack model represents a combined diffusion pump, baffle and valve. The parameters it takes are as for the diffusion pump but the speed is that for the combination and the backstreaming rate is that above the baffle.

If multiple gases are used, then the backstreaming rate will be determined as the sum of the backstreaming rates for the individual gases. When plotting this quantity, only the "Combined" value (ie total value) will have any meaning. It is recommended that you disable all the partial traces, in the plotter to speed up the simulation, by editting the plot component. (see Plot)

This component may be used with the "Generic" Gas.

OUTGASSING
Outgassing - bakeable
This is a three part model giving an idealised model of the outgassing process for an inner surface with the outer surface at atmosphere.

The three components are surface desorption, diffusion and permeation.

Surface desorption gives a 1/T dependence with time and models both adsorption and desorption.

Diffusion gives a 1/sqrt(T) time dependence and is bi-directional while permeation gives the steady state diffusion through the material by gas from outside.

The model has a temperature input which modifies the rates of all three components. This can be used to model baking the material (or cooling it) to reduce materials outgassing.

Simulation parameters

Starting pressure

The simulation starts as though the inner surface had been at this pressure for an infinite time.

External pressure

The pressure of the outside wall of the vacuum chamber. This parameter is used as part of the diffusion and permeation models. Usually this parameter will be set to represent 1 atmosphere with the standard gas compositions.

Model parameters

Area

The surface area presented to the vacuum.

Thickness

The thickness of the vacuum wall. Care must be exercised when adding permeation for materials of non-uniform thickness. Since the reciprocal of the overall permeability is the sum of the reciprocals of the permeability at each different thickness, averaging the thickness will not give a true result. The area at each thickness should be added as a separate permeability item.

J

This parameter relates to diffusion and permeation of gases in metals. Diatomic gases such as nitrogen and oxygen show a square root dependence with pressure. J should be set to 0.5 for metals and one for all other materials.

kdes

This constant sets the one hour desorption rate and is measured as an outgassing rate - e.g. in units of Watts/(metre^2).

k

Sets the relationship between outgassing rate and temperature, the same constant is used for all three contributors.

kdif

This constant sets the one hour diffusion rate and is measured as an outgassing rate - e.g. in units of Watts/(metre^2).

permeability

This is the permeability constant and is measured in permeability units - e.g. square metres per second.

Permeation notes

Care must be exercised when adding permeation for materials of non-uniform thickness. Since the reciprocal of the overall permeabilty is the sum of the reciprocals of the permeability at each different thickness, averaging the thickness will not give a true result. The area at each thickness should be added as a separate permeability item.

Using published data

Very little consistent data is available on outgassing partly because of the strong effect of surface preparation and previous history. However, sensible use of what there is can be used to provide useful insight into the limiting effects of outgassing on vacuum system performance.

Most data is provided to fit an outgassing model giving outgassing rate as A/(t^a)

where A and a are constants specified at given time after the start of pumpdown.

The desorption part of the model gives an a of 1 while the diffusion part gives an a of 0.5.

At a time t given by (kdes/kdif)^2 the slope of the outgassing model will change from a=1 to a=0.5.

If the slope given for the one hour rate is best fitted by one then kdes should be set to the quoted constant with kdif being set to a lower value to move the breakpoint further out in time. If the slope is closer to 0.5 then kdif should be set to the quoted value and kdes set to a smaller value to move the breakpoint to a shorter time.

Permeation is time independent and is generally very small in metals, it is most important for "O" ring materials where it sets a limit to the decrease in outgassing rate with time.

Outgassing rates are strongly affected by temperature, data for diffusion and permeation is usually plotted in log form against 1000/T where T is the temperature in Kelvin. The parameter k used in the model is 7.69 times the number of decades per unit 1000/T. That is, if the rate drops 2 decades from 1000/T=1 to 1000/T=2 then k=2x7.69=15.38.

Data for desorption with temperature is more difficult to find but has a similar characteristic. Since the same parameter k is used for all three outgassing components a value appropriate to the main source of outgassing during bakeout should be used.

See Outgassing test schematic

Outgassing - fixed temp

This is a three part model giving an idealised model of the outgassing process for an inner surface with the outer surface at atmosphere.

The three components are surface desorption, diffusion and permeation.

Surface desorption gives a 1/T dependence with time and models both adsorption and desorption.

Diffusion gives a 1/sqrt(T) time dependence and is bi-directional while permeation gives the steady state diffusion of gas through the material from the outside.

The three components of the outgassing process are temperature dependent, the temperature for the material can be set for the simulation run.

Simulation parameters

Starting pressure
The simulation starts as though the inner surface had been at this pressure for an infinite time.

External Pressure
The pressure on the outside wall of the vacuum chamber. This will normally be 1 atmosphere. This is used by the diffusion and permeation models

Temperature
The temperature of the outgassing material.

Model parameters

Area
The surface area presented to the vacuum.

Thickness

The thickness of the vacuum wall. Care must be exercised when adding permeation for materials of non-uniform thickness. Since the reciprocal of the overall permeability is the sum of the reciprocals of the permeability at each different thickness, averaging the thickness will not give a true result. The area at each thickness should be added as a separate permeability item.

J

This parameter relates to diffusion and permeation of gases in metals. Diatomic gases such as nitrogen and oxygen show a square root dependence with pressure. J should be set to 0.5 for metals and 1 for all other materials.

kdes

This constant sets the one hour desorption rate and is measured as an outgassing rate - e.g. in units of Watts/(metre^2).

k

Sets the relationship between outgassing rate and temperature, the same constant is used for all three contributors.

kdif

This constant sets the one hour diffusion rate and is measured as an outgassing rate - e.g. in units of Watts/(metre^2).

permeability

This is the permeability constant and is measured in permeability units - e.g. square metres per second.

Using published data

Very little consistent data is available on outgassing partly because of the strong effect of surface preparation and previous history. However, sensible use of what there is can be used to provide useful insight into the limiting effects of outgassing on vacuum system performance.

Most data is provided to fit an outgassing model giving outgassing rate as A/(t^a)

where A and a are constants specified at given time after the start of pumpdown.

The desorption part of the model gives an a of 1 while the diffusion part gives an a of 0.5.

At a time t given by (kdes/kdif)^2 the slope of the outgassing model will change from a=1 to a=0.5.

If the slope given for the one hour rate is best fitted by one then kdes should be set to the quoted constant with kdif being set to a lower value to move the breakpoint further out in time. If the slope is closer to 0.5 then kdif should be set to the quoted value and kdes set to a smaller value to move the breakpoint to a shorter time.

Permeation is time independent and is generally very small in metals, it is most important for "O" ring materials where it sets a limit to the decrease in outgassing rate with time.

Outgassing rates are strongly affected by temperature, data for diffusion and permeation is usually plotted in log form against 1000/T where T is the temperature in Kelvin. The parameter k used in the model is 7.69 times the number of decades per unit 1000/T. That is, if the rate drops 2 decades from 1000/T=1 to 1000/T=2 then k=2x7.69=15.38.

Data for desorption with temperature is more difficult to find but has a similar characteristic. Since the same parameter k is used for all three outgassing components a value appropriate to the main source of outgassing during bakeout should be used.

See Outgassing test schematic

PIPEWORK

Pipework - aperture

Models a circular aperture in the viscous and molecular flow regions.

This component may be used with the "Generic" gas.

Model parameters (general)

Area

The area of the hole in the aperture disk.

Model parameters (gas specific)

Visc Cond
Flow constant in the viscous flow region

Mol Cond
Flow constant in the molecular region

Deriving Flow Constants

Mol Cond
The coefficient of Molecular Conductance is given by L/s, can be calculated by 0.1*v/4, where v is the Average velocity of the gas in m/s.

For Air, ApMolCond = 11.6

Visc Cond
For a gas mixture the viscous conductance coefficients are the same for each of the components.

The Viscous Conductance is given by (2y/(y+1))*(kT/m)^0.5 * (2/(y+1)) ^ (1/(y+1)), where y is the ratio of specific heats, m is the molecular mass, k Boltzmanns constant, and T is the Temperature for the gas mixture.

SPECIFIC HEAT RATIOS:

monatomic:	1.67
diatomic:	1.4
triatomic:	1.33

Pipework - elbow

Models an elbow in a circular section pipe in the viscous and molecular flow regions. As pipe length is reduced to zero the model defaults to a circular aperture. The elbow is modeled as a pipe of the same length and cross-section.

This component may be used with the "Generic" Gas.

Simulation Parameters

Pressure:
The pressure of the pipe internal volume at the start of simulation.

Model parameters (general)

Diameter
The internal diameter of the pipe.

Length
The overall length of the pipe.

Model parameters (gas specific)

Pressure
Initial Pressure

Visc Cond
Flow constant in the viscous flow region

Mol Cond
Flow constant in the molecular region

Deriving Flow Constants

See Pipework Straight

Pipework - straight

Models a straight circular section pipe in the viscous and molecular flow regions. As pipe length is reduced to zero the model defaults to a circular aperture. The model will give good results for curved pipes such as flexible vacuum hoses where the bend radius is large compared with the pipe internal diameter.

This component may be used with the "Generic" gas.

Simulation Parameters

Pressure:
The pressure of the pipe internal volume at the start of simulation.

Model parameters (general)

Diameter

The internal diameter of the pipe.

Length
The overall length of the pipe.

Model parameters (gas specific)

Pressure
Initial Pressure

Visc Cond
Flow constant in the viscous flow region.

Mol Cond
Flow constant in the molecular region

Deriving Flow Constants

Visc Cond
In the viscous regime of pressure the viscosity of a gas mixture is described by a single quantity – it does not make sense to speak of the individual gas components having different viscous conductance parameters.

The viscosity of a gas mixture is given approximately by the average of the individual viscosities weighted by the relative proportion of the gases.

The Viscous Conductance in L/s, can be calculated by 10e8*pi/128/n, where n is the Dynamic Viscosity in Pa-s, pi=3.1416. For Air, Visc Cond = 1.38e11.

Mol Cond
The Molecular Conductance is given by 1000*pi*v/12, where v is the Average Velocity of the gas component in m/s. E.g. for Air, v=445 M/s, so Mol Cond = 1.16e5. Note that the molecular conductance will be different for each gas component in the mixture.

VALVES

Valve

The valve components are modelled as circular apertures the size of which is reduced to zero when closed. This component may be used with the "Generic" Gas.

Controlled Valves (On/Off)
A choice of four controlled valve types are given allowing normally open or normally closed operation and with or without a valve closed signal indicator.

The normally open valve is open when the applied control signal is less than 0.5, the normally closed valve is open when the control signal is greater than 0.5. The closed indicator is 1 when the valve is closed.

Pressure controlled valves (Variable)
The *Pres Limit* valve opens whenever the pressure difference between the high and low pressure port is greater than the "Control" value. The amount the valve opens is proportional to the pressure difference.

The *needle valve* opens whenever the total absolute pressure, at the "Low Pres" port, is less than the control value. The amount the valve opens is proportional to the pressure difference.

The Needle and Pres Limit valves do not have a control input, but do have a analogue signalling output. (0=closed, 1=open)

Model parameters

Area
The effective area of the valve aperture when fully open.

Visc Cond
The viscous conductance of a circular aperture for each gas. see Pipework Aperture

Mol Conductance
The molecular conductance of a circular aperture for each gas. see Pipework Aperture.

CONTROL

Heater

The heater is intended for use with the outgassing model and allows a temperature profile against time to be defined.

This component may be used with the "Generic" Gas.

Simulation Parameters

Initial temp
The base temperature for the profile.

Temp
The temperature to which the profile moves, this may be above or below the base temperature.

Rise time
The time taken to move from Initial temp to temp

Fall time
The time taken to revert to Initial temp.

On time
The time spent at temp.

Timer

Two timers are provided, one simply provides a delay while the other gives a delayed pulse.

This component may be used with the "Generic" Gas.

Simulation Parameters

Delay
The time at which the timer output goes from zero to one.

On time
The pulse duration for the delayed pulse timer.

Trip

The trip gives an output of "1" when the quantity being sensed (typically a pressure) falls below the set level; the output will return to "0" when the quantity rises above the set level plus the hysteresis.

If the quantity being sensed is a group quantity (eg pressure), then the combined value (ie total pressure) will be monitored. The trip output will be a single quantity, even if multiple gases are being sensed.

Trip
Trip_level
Trip works in pressure units whereas Trip_level works in system units.

Trip_level may be used with the temperature output from a cryo pump, for example, where system units are degrees Kelvin.

The trip gives an output of "1" when the input being sensed falls below the set level, the output will return to "0" when the input rises above the set level plus the hysteresis.

This component may be used with the "Generic" Gas.

Simulation Parameters

Level
The input level below which the trip signals "1"

Hysteresis
This is the fraction of Level which determines the point at which rising pressure causes the trip output to return to "0". If set to .01 then the input at which the trip output will fall to "0" is 1.01 times Level.

Current Pulse

The Pulse current source generates a single current pulse, with a time delay, a rise time, a fall time and an on time. This is intended for use with the Sublimation pump model, to control the filament heater.

Note that the sublimation pump assumes that a current of 1 Amp is equivalent to a sublimation rate of 1 g/hour. Typical values in using a sublimation pump are in the 0.01 – 0.1 g/hr region.

This component may be used with the "Generic" Gas.

Simulation Parameters

Initial Current
The base current for the profile.

Current
The current to which the profile moves, this may be above or below the base current.

Rise time
The time taken to move from Initial current to current

Fall time
The time taken to revert to Initial current.

On time
The time spent at current.

See also

Sublimation pump

Current PWL

Current PWL

The piecewise linear current source generates a user specified current waveform over time. The waveform consists of up to 9 linear segments. It is important that the time points are in asscending order, otherwise the simulator will generate an error "Break point in the past".

This is intended for use with the Sublimation pump model, to control the filament heater. Note that the sublimation pump assumes that a current of 1 Amp is equivalent to a sublimation rate of 1 g/hour. Typical values in using a sublimation pump are in the 0.01 – 0.1 g/hr region.

This component may be used with the "Generic" Gas.

Simulation Parameters

Current 0
The initial current at time 0.

Current 1

The current to which the profile moves, during the first segment.

Time 1

The time at which the current reaches Current 1.

Current2, Time2 .... Current9, Time9

Current and Time data points for segments 2 to 9. If not all line segments are needed then leave the parameter entries blank.

See also

Sublimation pump

Current Pulse

MISCELLANEOUS

Logic

A selection of simple gates is provided, the logic levels are 0 and 1 with 1 representing "true". The component "Set Level" can be used to provide a constant logic level of 1 or 0.

These components may be used with the "Generic" Gas.

Log_Ramp , LogRamp

The purpose of this model is to assist in the measurement of pump characteristics. It provides a pressure which decreases logarithmically with time. The rate is set such that if the simulation is performed with the timescale set in days the pressure at any time reduces by a factor of ten each day with pressure at time zero equal to one atmosphere.

The days timescale is used to mask the transient effects of pump start up times etc.

The component LogRamp differs from Log_ramp, as it uses a user specified time scale, and pressure values. The parameters "Start Pres" and "Stop Pres", specifiy the range of pressures to be simulated in powers of 10. The default range is from 1e-7 atm to

1e+0 atm (ie has parameters of -7 and 0 respectively). You may use either a rising or a falling pressure with time.

This component may be used with the Generic gas.

Use of the model is shown in Pump test schematic

Simulation Parameters

Initial Pressure
Limits the pressure at the start of simulation to a value below atmosphere where higher pressures might be inappropriate to the pump type.

Gauges

Three types of gauge are provided. These are, speed, throughput, and compression ratio. These can be used in monitoring system performance and measurement of pump characteristics. If multiple gases are used, then the "combined", or "total", plot will have no meaning.

This component may be used with the "Generic" Gas.

Speed
The speed gauge measures speed into a pump, if gas flow is out of the pump it will give zero indication.

Throughput
The throughput gauge measures gas mass flow in both directions, if flow is in the negative direction only linear plotting of the output is possible.

Compression ratio
Gives the ratio of output pressure to input pressure across a pump.

Plotters

Plotters display their inputs as a function of the simulation time. Single pen and dual input plotters are supported. A plotter can be used to plot any quantity in the circuit. The plotter will automatically select the correct units, and decide whether logarithmic or linear scaling would be appropriate.

To change the scale, titles, or plot style, use the Plot View Set Axes command.. From the plot window, individual gases, or inputs, can be enabled or disabled.

Simulation Parameters

Min time:
The minimum value of the plotter time axis. The units of time selected for this quantity determine the units used on the time axis of the plot.

Max time:
The maximum value of the plotter time axis. If this is less than or equal to the Min time value then the time range is set to that specified in the Simulation Setup Dialog

Axis Style
Select Logarithmic or Linear

AutoScale(1):
Select autoscale or manual with the checkbox

Min Value:
The minimum value on the plotter axis. For the Pressure plotter, the units selected for this quantity determine the units used on the vertical axis of the pressure plot.

Max Value:
The maximum value of the plotter vertical axis. If this is less than or equal to the Min Value then autoscaling is enabled.

Set to 1 for a logarithmic vertical axis and 0 for a linear one.

Y_AutoScale(1):
Set to 1 for autoscaling of the axis to fit the displayed data points. 0 for no autoscaling.

To choose the gases to be plotted select Plot View Select Gases window. The default is to plot all the gases being simulated and also the combined pressure. Click in the check boxes to de-select or select a gas.

Leak - constant

The constant leak model, simulates a constant mass flow into a chamber. Note that the leak rate is independent of the pressure.

Simulation Parameters

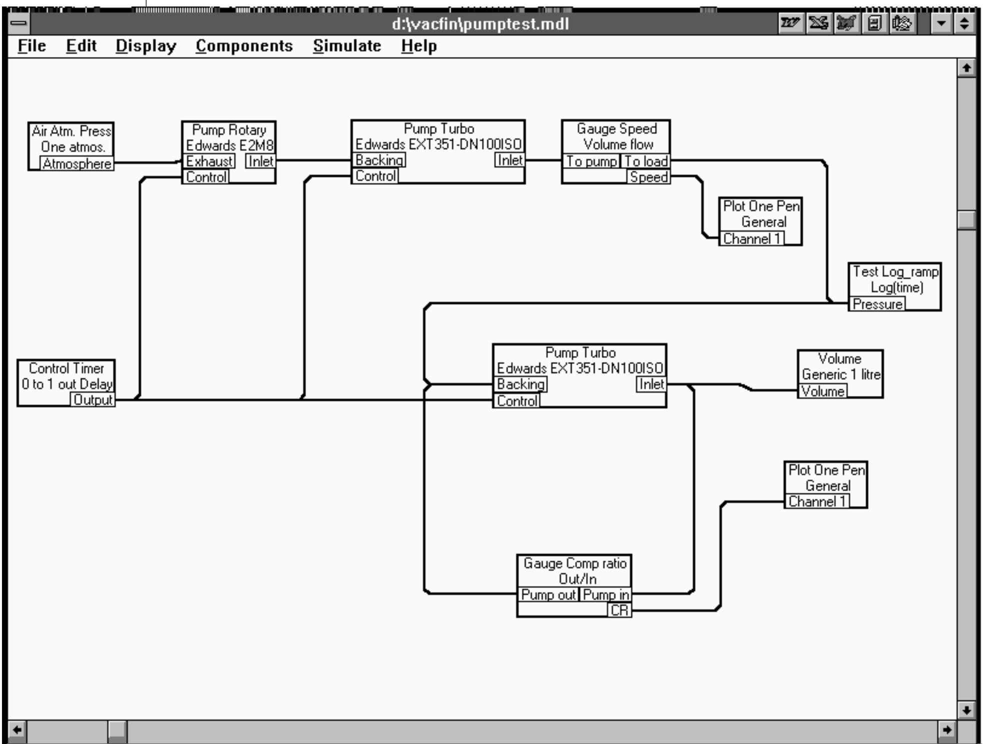
Rate (Throughput)

Specifies the leak rate as a mass flow rate. (std litres per sec)

TEST SCHEMATICS

Pump test schematic

When setting parameters in Generic pump models it may be useful to be able to plot the pump characteristics to match

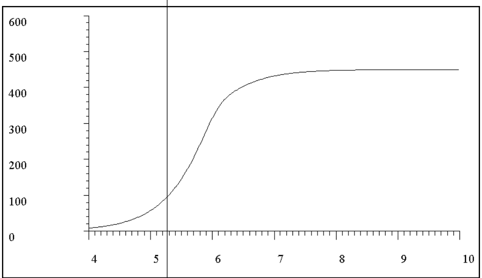


against the Manufacturer's published curves.

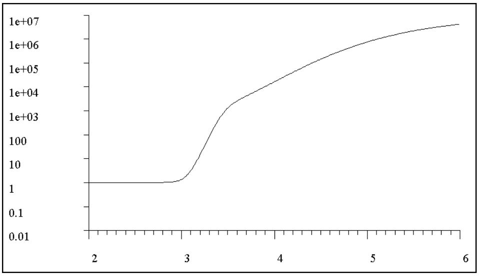
The schematic diagram PUMPTEST.MDL gives an illustration of how this may be done.

The example taken is for a Turbo pump and gives speed against inlet pressure and compression ratio against backing pressure. Published speed curves are usually given for operation with a specified rotary pump as shown in the diagram. The Log_ramp model is used to give log of pressure on the horizontal axis, one of these models being used to control the pump inlet pressure for speed measurement and another to control backing pressure for compression ratio plotting.

The plot axes can be manipulated to match those in the published curves to produce plots of Speed against inlet pressure and



Speed against Inlet pressure



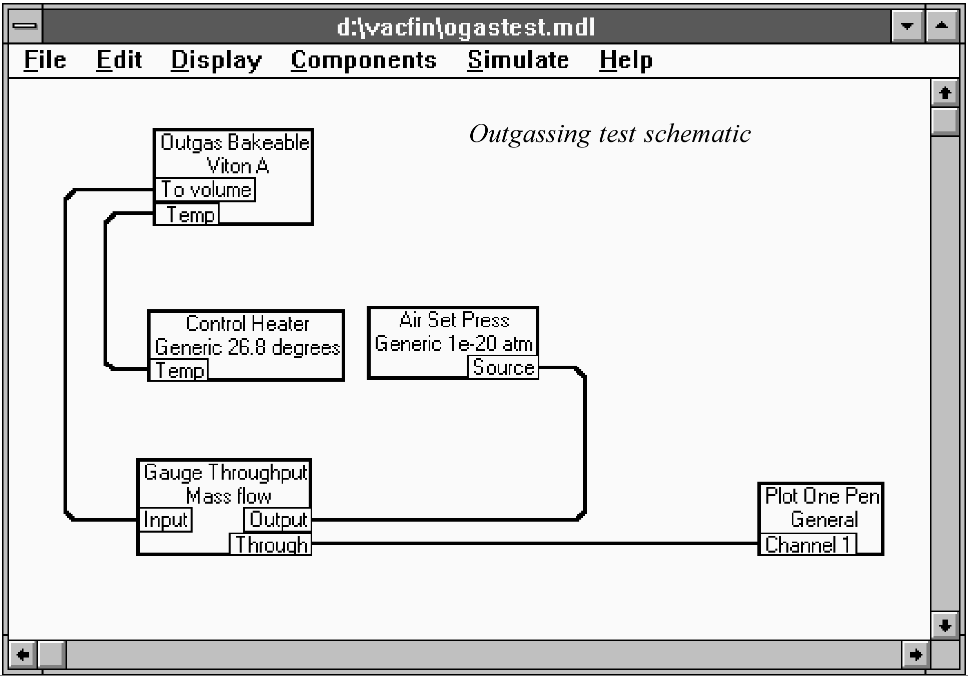
Compression Ratio against Backing pressure

Compression Ratio against backing pressure.

Outgassing test schematic

The schematic diagram OUTGAS.MDL can be used to plot the outgassing rate of a material against time.

The volume initial pressure should be set to a value below the range of interest and the result is best displayed on a logarithmic time scale. The area of material can be set to correspond to the units of the given data so that, with the throughput gauge, the result is in consistent units.



The ripple that can be observed in the plots is a result of the simulation method and gives a small discrepancy when compared with the reliability of the published data.

An example of the effect of baking a material can be found in BAKEOUT.MDL.

DETAILED PROGRAM REFERENCE

Starting the Simulator

You start the simulator in the normal way by clicking on Start, then moving the mouse to "Programs" and selecting "VacSim Multi", then "vacsim32". The main drawing window appears with the title “Untitled”.

From the Windows file manager (NT3.51), Windows Explorer (Windows 95 or NT 4), or from "My Computer" (Windows 95 or NT 4.0) you can drag a file with the .MDL extension onto VACSIM.EXE. This will start the simulator and load the .MDL file.

You can use the Program Manager’s *File/Run* command, specifying the VACSIM.EXE file and an optional .MDL file.

Menu Commands for the Drawing Window

The available menu bar commands in the drawing window are:

File

Edit

Display

Components

Simulate

Database

Help

File

The available File menu commands are:

New

Open

Save

Save As

Import

Export Selection

Set Units Library

Set Model Library

Set Components Library

Print

Print Setup

Clip

Exit

New

Clears the existing drawing and prepares for entry of a new vacuum system schematic model.

If the currently loaded drawing has changed since it was last saved to disk, you are offered the option to save it before it is cleared.

Open

Presents a file dialog listing the vacuum system model files (.MDL) in the current directory.

You can select a file from the list by clicking the mouse button on the file name in the Files list box or type a name into the Filename box directly. You can also change the directory which is searched by clicking in the Directories list box.

To confirm the selection, click on the *OK* button. To cancel the operation, click on *Cancel*.

Double clicking on a file in the list box selects the file and confirms the operation.

If the program cannot find the filename you entered, an error message is displayed.

Save

Saves the schematic model drawing to the current .MDL filename.

The current filename is shown in the window title.

If the drawing is “Untitled”, the *File/Save As* dialog is executed.

This menu item is disabled (greyed) if the program is operating in demonstration mode.

Save As

Presents a file dialog listing the .vacuum system model files in the current directory.

You can select a file from the list by clicking the mouse button on the file name in the Files list box or type a name into the Filename box directly. You can also change the directory which is searched by clicking in the Directories list box.

To confirm the selection, click on the *OK* button. To cancel the operation, click on *Cancel*.

Double clicking on a file in the list box selects the file and confirms the operation.

If the filename you entered is that of an existing file, you are prompted to confirm that you wish to overwrite it.

This menu item is disabled (greyed) if the program is operating in demonstration mode.

Import

The same as Open except that the chosen .MDL file is appended to the current schematic drawing. The imported components will be centred at the position of the most recent left mouse button click.

This menu item is disabled (greyed) if the program is operating in demonstration mode.

Export Selection

The same as Save As except that only the currently highlighted selection of components is saved and not the whole drawing. Only link lines running between selected components are saved.

This menu item is disabled (greyed) if the program is operating in demonstration mode.

Set Units Library

Presents a file dialog listing the units library files (.ULB) in the current directory.

You can select a file from the list by clicking the mouse button on the file name in the Files list box or type a name into the Filename box directly. You can also change the directory which is searched by clicking in the Directories list box.

To confirm the selection, click on the *OK* button. To cancel the operation, click on *Cancel*.

Double clicking on a file in the list box selects the file and confirms the operation.

If the program cannot find the filename you entered, an error message is displayed.

Set Model Library

Presents a file dialog listing the model library files (.MLB) in the current directory.

You can select a file from the list by clicking the mouse button on the file name in the Files list box or type a name into the Filename box directly. You can also change the directory which is searched by clicking in the Directories list box.

To confirm the selection, click on the *OK* button. To cancel the operation, click on *Cancel*.

Double clicking on a file in the list box selects the file and confirms the operation.

If the program cannot find the filename you entered, an error message is displayed.

Set Components Library

Presents a file dialog listing the components library files (.CLB) in the current directory.

You can select a file from the list by clicking the mouse button on the file name in the Files list box or type a name into the Filename box directly. You can also change the directory which is searched by clicking in the Directories list box.

To confirm the selection, click on the *OK* button. To cancel the operation, click on *Cancel*.

Double clicking on a file in the list box selects the file and confirms the operation.

If the program cannot find the filename you entered, an error message is displayed. This allows different component libraries to be selected, for instance for different manufacturers' pumps.

Print

Prints the current system schematic diagram on the currently selected printer. The schematic drawing is scaled to fit on a piece of A4 paper in landscape orientation. You are prompted to enter the number of copies required; the default is 1.

The current printer is selected and configured using *File/Print Setup*.

This menu item is disabled (greyed) if the program is operating in demonstration mode.

Print Setup

Presents a list box of the currently available printers. If you select a printer, the dialog to set up that particular printer is executed. VACSIM takes a local copy of the printer configuration information and does not change the settings of the default Windows printer.

Clip

Copies the current system schematic diagram to the Windows clipboard.

The formats supported are Object Graphics CF_OGC and Bitmap. Most graphics applications support the Bitmap format.

This menu item is disabled (greyed) if the program is operating in demonstration mode.

Exit

Closes the program and any associated plot windows. If the simulator's SPICE Engine is running, this is also closed.

If the currently loaded drawing has changed since it was last saved to disk, you are offered the option to save it before it is cleared.

Edit

The available Edit menu commands are:

Undo

Cut

Copy

Paste

Delete

Select All

Undo

Undoes the last *Edit* action, restoring the schematic drawing to its previous state.

Cut

Deletes the currently selected vacuum component and its associated link lines (See Select a Component or Link, Link Two Components) from the schematic. A copy of the item(s) is stored; this can be recalled in subsequent *Paste* operations to place copies of the item(s) in the schematic drawing. Only links connecting items within a selection are saved with the *Cut* object.

Copy

A copy of the currently selected vacuum component(s) is stored; this can be recalled in subsequent *Paste* operations to place copies of the item(s) in the schematic drawing. Only links connecting items within a selection are saved with the *Copied* object.

This action can be undone using the *Undo* command. The buffer containing the copy is cleared.

Paste

Places a copy of the last *Cut* or *Copied* object at the current insertion point (See Position Insertion Point) in the schematic drawing.

This action can be undone using the *Undo* command. The pasted object is removed from the drawing.

Delete

Deletes the currently selected component(s) or link line (See Select Components or Links).

This action can be undone using the *Undo* command. The deleted object is restored.

Select All

Selects or de-selects all the components in the schematic.

Display

The available Display menu commands are:

Colour Schematic

Mono Schematic

System Bitmap

Component Bitmap

Colour Schematic

Vacuum components are displayed as rectangular boxes which contain text describing the type of component, the manufacturer and the component part name. Within each box are one or more rectangular boxes which represent the connection ports of the device.

Typically, the ports will represent vacuum system connections or control logic links. For example, a rotary pump will have three ports: inlet; exhaust and on/off control.

When a component is first added to the schematic drawing (See *Components/Add Component*), the port rectangles are coloured magenta and the rest of the box is red. When a port is connected to at least one other port in the system, the port colour changes to green (See Link Two Components).

Under normal circumstances, when all ports of an item have been connected, the red component box also turns green to indicate that the component is correctly connected. The exception to this occurs when the system has been unable to associate a valid mathematical model with that component (See *Components/ Associate Model*, Library Files).

When all components in the drawing are green, and at least one plotter is included in the schematic, the system diagram is ready to be passed to the SPICE Engine for simulation.

The schematic diagram supports the notion of a currently selected item (See Select a Component or Link) which can be either a vacuum component or a link line. The currently selected vacuum component has a yellow, as opposed to black, outline. The currently selected link line is red, as opposed to blue.

If the currently selected item is a link line, it can be deleted using *Edit/Delete*. If it is a vacuum component it can be *Cut*, *Copied* and *Pasted* as well. The currently selected vacuum component can be manipulated using commands from the *Components* menu option.

The schematic display supports the full range of schematic editing commands. This is in contrast with the *System Bitmap* and *Component Bitmap* displays in which the *Edit* and *Components* menu items are disabled.

Mono Schematic

This display mode is identical in behaviour to the Colour Schematic display but presents the schematic in black and white. This mode is useful when generating drawings on black and white printers.

System Bitmap

In the mathematical models library database (See Mathematical Model Library) each model has associated with it a bitmap file. *Display/System Bitmap* displays this.

The bitmaps can be moved around the drawing window by clicking and dragging with the mouse (See Move Components), but the *Edit* and *Components* menu items are disabled.

If the bitmap file called for is not found, the simulator displays the colour schematic representation of the component (*Display/Colour Schematic*).

Component Bitmap

In the component models library database (See Component Library) each component has associated with it a bitmap file. *Display/Component Bitmap* displays this.

The bitmaps can be moved around the drawing window by clicking and dragging with the mouse (See Move Components), but the *Edit* and *Components* menu items are disabled.

If the bitmap file called for is not found, the simulator displays the corresponding system bitmap representation of the mathematical model associated with the component (*Display/System Bitmap, Components/Associate Model*).

If the system bitmap file is not found, the simulator displays the colour schematic representation of the component (*Display/Colour Schematic*).

Components

The available Components menu commands are:

Add Component

Edit Parameters

Substitute Component

Associate Model

These commands are only available if *Display/Colour Schematic* or *Display/Mono Schematic* is selected.

Add Component

Adds a component from the currently defined (*File/Set Components Library*) Component Library.

The component description contains a reference to a corresponding mathematical model. The current Model Library is scanned and the corresponding model is associated with the component.

If no corresponding Model is found, a warning message box is displayed. The rectangular box representation of the component will remain red, even when all connection ports are linked (See *Display/Colour Schematic*) to indicate that the component description is not complete. This situation may occur if incompatible Component and Model libraries are used. See *Components/Associate Model* for how to correct this.

You are presented with a list box of available component types: Pumps, Pipework, Control Components etc. Selecting an item from this may produce a list of sub-types (e.g. For Pumps: Rotary, Turbo, Ion etc.) from which a similar selection is made.

Once the component type (and sub-type) have been selected, you are presented with a list of available items of that type. Typically these will be a description of the component or the manufacturer's part number. Selecting an item from this list completes the selection.

For components with user-editable parameters you are prompted to enter the parameter values. See the entry under *Components/ Edit Parameters* for details.

The newly added component becomes the currently selected component in the context of editing the drawing (See *Edit*).

Edit Parameters

This menu item is enabled if there is only one currently selected item (see Select Components or Links) and it is a component with editable parameters.

A dialog is displayed, asking for each parameter showing the parameter name, its current value and the units of measure of that value. The parameters are grouped into a number of sections. The parameters in the "General" section, are for quantities that are the same for all gases (such as volume and length) All other parameters are duplicated for all gases selected for simulation. (see Simulation/Setup) This allows for components to have different characteristics, for the different gases. The default parameters are for a system where the gas being pumped is "air", and that the system is vented to the atmosphere at 1 atmosphere STP, with the normal gas concentrations.

If you change the units of measure by clicking on the drop down listbox, the parameter value displayed will change to the new units. You may edit the parameter value or take the default supplied. If you select the Cancel option for the dialog, the parameter value and units of measure remain unchanged, previously entered parameters for the component are accepted.

Certain component types have one component called GENERIC (See Generic Models). This component is the most general available for that component type and **all** its parameters are editable. In order to distinguish different instances of the same generic model, you are prompted to enter a name for each generic component you add. The schematic component box is scaled to fit this name within its boundary, so if you want to keep the schematic component box as small as possible, you should keep this name short.

Substitute Component

This menu item is enabled if the currently selected item (see Select a Component or Link) is a component.

You are presented with a list of available items of the same type (and sub-type) as the current item. This allows you to substitute different components without having to delete (*Edit/Delete*) the current one, add another of the same type (*Components/*Add Component) and re-link it (Link Two Components).

For components with user-editable parameters you are prompted to enter the parameter values. See the entry under *Components/ Edit Parameters* for details.

Associate Model

This associates a mathematical model from the currently selected Model Library with the currently selected component.

This operation may be necessary if your Component (Set Components Library) and Model Libraries are not compatible. If this has occurred, a warning message box will be displayed and the schematic rectangle for the component (*Display/Colour Schematic*) will remain red even when the component is fully linked up (Link Two Components).

You should select a model library (*File/Set Model Library*) which is compatible with the Components library from which the component was extracted, select the non-associated component (Select Components or Links) then execute *Components/Associate Model*.

Simulate

The available Simulate menu commands are:

Go

Pause

Continue

Stop

Setup

Go

Starts the simulation.

If you have not previously set up the simulation, the *Setup* dialog is presented.

The schematic drawing is checked to make sure that all components are connected (Link Two Components) and have a valid model associated with them (*Components/Associate Model*). If the check fails an error message is displayed.

The drawing is then checked to make sure it has at least one Plotter component to display the results of the simulation. If the check fails an error message is displayed.

If the drawing has changed since it was last saved, you are asked whether you want to save it before simulating. *Yes* saves it (*File/Save*) and then simulates; *No* simulates without saving; *Cancel* does not save or simulate.

To simulate the system, VACSIM builds a SPICE Circuit description of the Vacuum system and saves it in a file with the .CKT extension(SPICE Circuit File). It then launches the SPICE Engine process to read this file and to carry out the simulation calculations.

Pause

Pauses a running simulation.

This action must be taken before the simulation can be *Stopped*.

If you make changes to the drawing whilst the simulation is paused, these do not affect the simulation when you *Continue* it.

Continue

Continues a simulation run from where it was *paused*.

Stop

Aborts a *Paused* simulation.

The SPICE Engine is stopped and the process closed down.

Setup

Presents the simulation setup dialog.

Stop Time is the time up to which the simulation runs.

Simulator Options is provided for diagnostic purposes and is normally left blank.

The Time Units drop down list box shows the current units of measure for the Stop Time. If you activate the list box and select new units, the time value displayed is automatically converted to the new units.

The available gases for simulation are displayed in the right hand box, marked "Available Gases". Gases can be selected by clicking on the appropriate check boxes. A gas will be used in the simulation if an "X" appears in the box. If a reduced number of gases is selected for simulation, and the parameters of a component are edited, then you will only be asked for parameters for those gases which are selected for simulation.

To accept the options, choose the *OK* button. If you choose *Cancel*, the parameter values which were presented as defaults when the dialog was opened will be used.

Database

The Database menu is used to update the user database files. You can add, modify and remove components from the user databases. All files with a .USR extension in the VACSIM directory, will be automatically linked into the component database.

You may save your new components into any file with a .USR extension. This flexibility allows you to group your components by pump type, for example you could use ROTARY.USR and TURBO.USR for turbo & rotary pumps. Alternatively you could group the new components by manufacturer. eg LEYBOLD.USR, or VARIAN.USR

Add Component

To add a component to the user database, you must first place a generic component onto the schematic. You must specify values for all of the parameters, and test the component to ensure the parameters are correct. To add the component to the database, select the generic component, and choose the Database|Add Component menu option.

A file save dialog will be displayed. Enter, or choose, the name of the database file to which the component is to be added.

You will now be presented with a database parameter dialog. You must specify names for the Model and the Manufacturer of the component. The model may not be Generic, and the model must be unique. You also have the option of hiding parameters from the user. When the user places a component on the schematic from the database, hidden parameters will not be displayed in the component parameter dialog.

Edit Component

To edit a component in one of the user databases, select this menu option, and you will be asked to choose a component in the database. You will only be allowed to modify a component in one of the user database files. You can modify the component parameters, name , and the parameters that are hidden. Changes to the database do not effect components already placed on a schematic. The new parameters will only effect components that are added to the schematic after the database has been modified.

Remove Component

This command removes a component from the user database. Any instances of this component in a schematic file will still work, after the component is deleted, but no new instances may be added.

Help

The available Help menu commands are:

Contents

Search

About

Contents

Starts the online help, displaying the CONTENTS page.

Search

Starts the online help, displaying the topic search dialog.

About

Displays a panel showing:

The Version of the software.

The Copyright Notice.

The Licence ID and expiry date (See Dongle and Codeword).

The amount of free memory available on the system (the pool for this is the physical RAM and the Windows swap file).

The percentage of free system resources.

Menu Commands for the Plot Window

The available menu bar commands for the plot window are:

File

View

Help

The file menu has two commands.

Print

To Clipboard

Print

Prints the contents of the plot window using the printer setup configuration of the main drawing window. See File/Print Setup for the drawing window.

To Clipboard

Copies the current plot to the Windows clipboard. This feature allows you to import the results of a VACSIM simulation into other documents.

1. To use this feature, load up the other application, for example a word processor, spreadsheet, or graphics program.

2. Enter a circuit and run the simulator.

3. Select File/Clip from the plot window's File menu.

4. Position the cursor in your other application, and select File/Paste.

The formats supported are Object Graphics CF_OGC, Bitmap, Text and OEM Text. Most graphics applications support the Bitmap format. The Text output contains the plotted data in a tab delimitted text format which may be pasted into a suitable external plotting program or spreadsheet such as Microsoft Excel.

The View Menu (Plot Window)

The view menu allows you to modify the graph in a number of ways. You can set the scale, plot style, and plot labels, as well as being able to zoom and unzoom the plot window. The selection of traces being viewed in the plot window may be modified.

Position

This menu option is used to inspect the current cursor position within a plot window. After selecting Position, left click in the plot window to display the cursor coordinates. This feature is useful to measure the pressure accurately from a pressure plot.

Unzoom

Undoes any Zooming and displays the entire plot region. See Zoom Selected Area

and Set Axes.

Select Inputs

The Select Inputs menu options, displays a dialog box, which allows you to select which of the inputs connected to the plotter component, are to be displayed in the plot window.

You must have at least one input selected, and at least one gas selected for this command to have any effect. (see View/Select Gases)

Select Gases

The Select Gases menu options, displays a dialog box, which allows you to select which of the simulated gases are to be displayed in the plot window. (see Simulate/Setup)

You must have at least one input selected for this command to have any effect. (see View/Select Inputs)

Set Axes

Allows you to specify how the plot should be displayed.

The Plot Title specifies the title to be displayed at the head of the plot.

All subsequent options apply to the horizontal (time) axis and the vertical axis.

The Axis Title specifies the title of each axis.

The Axis Style group contains two radio buttons which allow you to choose a logarithmic or linear axis.

The Auto Scale check box enables auto scaling when checked. If auto scaling is checked, you cannot specify values for the next 4 parameters listed.

Maximum - upper limit of the axis (must be greater than

Minimum).

Minimum - lower limit (must be less than Maximum).

Major Ticks - number of tick marks which have coordinate values displayed besides them.

Minor Ticks - number of ticks between Major Ticks (applies to linear axes only).

Units - specifies the units of measure for the axes. If you change this by selecting a new unit of measure from the drop down list box, the Maximum and Minimum values are changed to reflect the new units.

Units Type - For the horizontal axis, this is always Time. For the vertical axis, you can choose any units type from those available in the drop down list box. The plotter does not know whether the quantity you are plotting is a pressure, a throughput etc., so you must choose the units type to reflect the type of quantity you are plotting. Refer to VACUUM SYSTEM COMPONENT MODELS for a discussion of how VACSIM models represents values.

Help (Plot Window)

Starts the online help, displaying the CONTENTS page.

Mouse Operations in the Drawing Window

The mouse is used to perform the following operations in the drawing window:

Position Insertion Point

Select a Component or Link

Select a Component and Edit its Parameters

Link Two Components

Move a Component

Move a Link Line

Position Insertion Point

Move the cursor to the point in the drawing window where you would like to insert the next component (*Components/Add Component*) then click the left mouse button. The next component or group of components you add will be centred on this point.

Select Components or Links

To select a single component, click the left mouse button when the cursor is over a component (if the component is displayed as a schematic, the cursor must not be over a link port). To select a group of components, press the left mouse button outside of an object and then extend the chooser rectangle to encompass the components desired. Releasing the left button will now cause all components inside the rectangle to become selected. A component can be added to or removed from the current selection by left clicking it with the Control key pressed. If Display/Colour Schematic is selected, the outline of components changes from black to orange (for a single component) or yellow (for a multiple selection). If Display/Colour Schematic or Display/Mono Schematic is selected, all commands from the Edit menu option can be activated.

To select a link line, as opposed to a component, click the mouse button when the cursor is over the line. The selected line changes from blue to red. If Display/Colour Schematic or Display/Mono Schematic is selected, Edit/Delete deletes the link and Edit/Undo restores it. There are no Cut, Copy and Paste operations available for the link lines.

Select a Component and Edit its Parameters

Double click the left mouse button when the cursor is over a component (if the component is displayed as a schematic, the cursor must not be over a link port). The component becomes the currently selected one. If *Display/Colour Schematic* is selected, the outline of the component changes from black to yellow. If *Display/Colour Schematic* or *Display/Mono Schematic* is selected, all commands from the *Edit* menu option can be activated.

If the component has any user-editable parameters, the parameters are presented for editing as if *Components/Edit Parameters* had been selected.

Link Two Components

This operation may only be performed if *Display/Colour Schematic* or *Display/Mono Schematic* is selected.

Place the cursor over the link port of a component and down click with the left mouse button. Drag the cursor to the link port of another component (not the same one) and release the mouse button. A link line is drawn between the ends of the ports and becomes the currently selected item in the context of editing the drawing (See *Edit*).

The form of the link line created will be chosen by the program depending upon the relative positions of the connection ports. The simpler form of link line comprises two horizontal lines joined by a vertical line: the horizontal position of the vertical line can be changed by clicking the left mouse button when the cursor is over the link line and then dragging the mouse. The more complex link line has three horizontal line segments and two vertical segments. The vertical lines and the middle horizontal line can be re-positioned by clicking and dragging. (See Move a Link Line).

Move a Component

Click the left mouse button down when the cursor is over a component or selection of components (if the component is displayed a schematic, the cursor must not be over a link port) (See also Select Components or Links). Drag the mouse to a new position on the display and release the mouse button. The component or selection of components is moved to the new location and all its link lines are re-drawn.

Move a Link Line

A link line is drawn between two connection ports on components as a horizontal section, a vertical section and another horizontal section. By default the vertical section is at the mid- point of the horizontal positions of the end points. The horizontal position of the vertical section can be moved to make the schematic diagram clearer. If the program considers that the connection would be shown more clearly if the link line had more bends in it, it will form the line as three horizontal sections and two vertical sections. (See Link Two Components)

Click the left mouse button down when the cursor is over a link line; the link becomes the currently selected one (See also Select Components or Links). Drag the mouse to a new position on the display and release the mouse button. The vertical section of the link line is moved to the new location provided it does not overlap certain other link lines. The ends of the link line are defined by the positions of the two connecting ports which it links.

To avoid ambiguity in the schematic drawing, there are several restrictions on the positioning of the vertical section of a link line. The vertical sections of two link lines may overlap if the two link lines share a common connection port. However, if they do not share a common port the schematic drawing package will attempt to move the horizontal position of the vertical part of the dragged link so that its horizontal and vertical sections do not overlap the horizontal or vertical sections of the other line. In a complicated drawing, it may not be possible to route a line to avoid overlap with others - in this case a horizontal overlap is generated. This overlap can then be removed by repositioning one or more components.

The horizontal position of the vertical line section moves in steps of the line thickness so that it is relatively easy to overlap lines which are permitted to be overlaid.

If you hold down the **Control** button when clicking the left mouse button over the single vertical part of a link line, the vertical section can be dragged horizontally into two parts connected by a horizontal line. You can then adjust the horizontal positions of the two vertical parts and the vertical position of the horizontal line joining them by clicking and dragging on the appropriate part of the link. This allows you to route the link lines in a way which will reduce the number of overlapped links and hence improve the legibility of the drawing. If you drag the vertical sections so that they have the same horizontal position, they will automatically re-merge as a single vertical section.

Mouse Operations in the Plot Window

The mouse is used to perform the following operations in the plot window:

Zoom Selected Area

Find Cursor Coordinates

Zoom Selected Area

Position the cursor cross at one corner of the region you wish to display in more detail and click the left button down. You then drag the mouse to the opposite corner of the region and release it. The plot is re-scaled to display the new region.

The limits for the axes are rounded up and down to give reasonable axis bounds. This may prevent you from displaying details in very small areas. If this is the case, you should use the manual range setting available in the View/Set Axes dialog.

To restore the view to the entire plot, use the View/Unzoom menu item.

Find Cursor Coordinates

This action is carried out in conjunction with the View/Position menu item to inspect the cursor coordinates.

Having selected Position from the menu, you place the cross-hair cursor over a point on the plot and click the left mouse button. The coordinates of the point (in the current units of measure for the two axes) are displayed in a message box.

Details of Simulator Operation

This section gives a brief description of some of the internal workings of the simulator system. You should not need to refer to this information routinely, but you may find it useful in developing an understanding of the VACSIM program.

How the Simulator Works

The main VACSIM drawing window and associated interface allows the user to draw a connected set of items representing vacuum system components. Each component is associated with a mathematical model describing its general behaviour, and a set of parameters which specify the detailed behaviour.

The mathematical model is written in a format compatible with the SPICE electronic circuit simulator produced by the University of California at Berkeley. Traditionally, SPICE simulation is used to solve for Voltages and Currents in electronic circuits. However, there is an exact parallel between Pressures and Throughputs in vacuum systems; VACSIM exploits this isomorphism to simulate pumpdown performance.

When a schematic drawing of a vacuum system is completely linked up, VACSIM exports a set of SPICE Subcircuits, connected up to form a SPICE Circuit. The entire file is passed over to a SPICE simulator program (the "SPICE Engine"), based on release 3F4 of Berkeley SPICE). The SPICE Engine solves for the pressures and throughputs in a series of time steps and passes back the results of each time step to a set of Plotting windows which VACSIM has generated. The data values are plotted in these windows "on-the-fly" so you can monitor the simulation as it proceeds.

Adding a Component from the Components Library

When you take a component from the Component Library and add it to a schematic drawing, the component description contains the following:

- A list of parameters specific to the component (e.g. the Pump Speed), a flag to indicate whether they are editable and the units in which they are measured.
- Lists of names of, and positions for, the connection ports of the component.
- A reference to a mathematical model which the simulator

expects to find in the Model Library.

- A reference to a bitmap file specific to that component.

The Model Library is scanned to find the model which has the same type and sub-type as the component (See *Components/ Associate Model*) and a copy of the model is added to the component.

If a component has a parameter which may be edited by the user, the simulator displays a dialog showing the component value (in the units of measure stored with the parameter). The user may change the value and the program stores the changed value with the component's data.

In order to display the editable parameters in appropriate units, the Units Library is scanned until the units type (e.g. Pressure, Temperature etc.) and units of measure (e.g. Torr, atmospheres, mbar etc.) are matched. The Units Library contains conversion factors to change from one unit of measure to another. The parameter for the component is stored in an agreed internal format (system units), and presented to the user in converted format.

Linking Components Together

When you link the ports of two components you add to each connection port a pointer to the other port. Each port can have an arbitrary number of such links.

Saving a Schematic Drawing to a File

Each component, in turn is saved to an output stream associated with the file selected as the output (See *File/Save*). The current parameter values (in system units) and their current units of measure are also saved so that if you subsequently retrieve the component, it will present the parameter in the same units of measure as you last used.

The copy of the mathematical model referenced by the component is not saved (see Adding a Component from the Components Library), but the reference type and sub-type is.

To save the currently highlighted selection of components along with any internal connections use the File/Export Selection menu command.

Retrieving a Schematic Drawing from a File

When you open a previously saved schematic drawing, all items are read in and reconstructed in memory.

For each component, the reference to the corresponding mathematical model is used to extract a copy of the model from the model database (see Adding a Component from the Components Library for a description of the process).

To append a previously saved drawing to the current schematic use the File/Import menu command.

Running a Simulation

Checking the Schematic

SPICE can only carry out a simulation if:

- Each component has a valid mathematical model associated with it (see *Components/Associate Model*).
- All connection ports are linked up.
- There is at least one plotter component to extract the results.

When all checks are successful, the SPICE file is built. See Building the SPICE file.

Building the SPICE file

If the schematic is correctly set up, the simulator builds the SPICE file. The following stages are involved:

- For each component, the parameter names present in its mathematical model are replaced with the parameter values carried by the component. Selected parts of the resulting model are passed through an algebraic expression parser to covert simple formulae to numbers.
- The connectivity defined by the links between connection ports, is used to assign unique node numbers to points in the

circuit. Any connection ports which are connected, either directly or indirectly, by links will have the same node number.

- The simulator creates the SPICE circuit file (see SPICE Circuit File) and writes out the SPICE run header. The header contains the simulation parameters (see Simulate/Setup Parameters).
- For each component, a SPICE subcircuit definition, containing the parsed mathematical model, is written to the SPICE circuit file.
- For each component, a SPICE subcircuit reference line is created, using the previously assigned node numbers to define the connectivity.
- If multiple gases are being simulated, then the SPICE circuit file is created with one copy of the component model, for each of the gases being simulated.

Launching the SPICE simulator

The SPICE Engine process (SPICE32.EXE) is run with the name of the SPICE Circuit File being passed as an argument. The SPICE Engine, based on release 3F4 of Berkeley SPICE.

The Spice Engine uses the Spice circuit file name to register unique Windows messages for communication with the main VACSIM program (VACSIM.EXE) and then reads in the SPICE file.

Note that all of this happens automatically once a VACSIM simulation is started.

Creating Plot Windows

When it receives a startup message form the SPICE Engine, VACSIM creates a new window for each plotter in the system. The plot window is positioned on the corner of the plotter component in the schematic and moves with it when the main drawing window is scrolled or repositioned.

The plot windows can be sized, iconised or maximised, but can only be deleted by VACSIM. The iconised window is also positioned on the corner of the plotter component when first iconised. It may be repositioned by dragging.

Drawing Plots

Each time the SPICE Engine completes a timestep in its simulation, it sends a Windows message to the main VACSIM program and a pointer to some globally accessible data. The data block contains the elapsed time through the simulation and the plotted data values at that time.

VACSIM extracts the time and data values, stores them internally, and notifies all the plotter windows that new data has arrived and that they should plot it.

Each plotter reads the latest data point it must plot, optionally rescales the plot axes (*SetAxes*) and converts the data values to fit within the plot's axes. The point is then plotted.

Files Used by the Simulator

The files used by the simulator may be broken down into the following categories.

Executables and DLLs

Library Files

Bitmap Files

Codeword File

System Model Files

Simulator Status Files

You should not attempt to alter any of these files directly except for the Simulator Status Files which may be listed or viewed with a text editor.

The Executables and DLLs, Library Files and Bitmap Files may be restored by re-installing the software.

The Codeword File may be typed in as text.

The System Model Files are created by you and should be backed up as part of a routine backup procedure.

The Simulator Status Files are temporary files created by the SPICE Engine and may be discarded.

Executables and DLLs

VACSIM.EXE is the main simulator file. It handles the user interface and controls the SPICE Engine.

SPICE32.EXE is the SPICE Engine, based on release 3F4 of Berkeley SPICE. It is started and controlled by VACSIM.EXE. Its role is to do the SPICE simulation and to pass back the data values to VACSIM.EXE which plots them.

DK12WN16.DLL and DK12WN32.DLL are the dynamic link libraries which handle the dongle functions under the 16 bit or 32 bit Windows operating systems. See Dongle and Codeword.

In addition, there are some operating system specific device drivers through which DK12WN16 and DK12WN32 communicate with the parallel port on which the dongle is placed. On Windows 3.1x operating systems (Windows 3.1, Windows for Workgroups 3.11 etc.) the driver file is DK12.386; on the Windows 95 operating system it is DK12WN95.386 and under Windows NT there are two files: DK12DRV.DLL and DK12DRV.SYS. These device drivers are installed in the correct places by the driver installation software. They are not required if the software in running in demonstration mode.

VACSIM.HLP is the help source file for VACSIM. The Windows function WINHELP.EXE uses this to display online help for VACSIM.

Library Files

Mathematical Model Library

Mathematical model libraries have the default file extension **.MLB**.

The default library is VACSIM.MLB and is installed in the same directory as VACSIM.EXE.

For future expansion, there are facilities to change the name of the library using the command *File/Set Model Library*.

The model library contains a series of generic SPICE models, one for each type of component supported. These models provide a general description for the behaviour of a class of components, such as a rotary pump. This is turned into a model of a specific instance of a pump (say an Edwards E2M5) by substituting information from the components database. The components in the Component Library cross refer to entries in the Model Library. See Building the SPICE file for a description of the association between components and their SPICE model.

Each model entry in the library also contains a reference to a bitmap file which is displayed by the menu command *Display/ System Bitmap*. See also Bitmap Files.

Component Library

Component libraries have the default file extension **.CLB**. The default library is VACSIM.CLB and is installed in the same directory as VACSIM.EXE.

For future expansion, there are facilities to change the name of the library using the command *File/Set Components Library*.

The components library contains a series of components descriptions including the parameters to be substituted into the corresponding SPICE model, one for each component supported. The components in the Components Library cross refer to entries for the component type in the Model Library. See Building the SPICE file for a description of the association between components and their SPICE model.

Each model entry in the library also contains a reference to a bitmap file which is displayed by the menu command *Display/ Component Bitmap*. See also Bitmap Files.

Units Library

Units libraries have the default file extension **.ULB**. The default library is VACSIM.ULB and is installed in the same directory as VACSIM.EXE.

For future expansion, there are facilities to change the name of the library using the command *File/Set Units Library*.

Each record in the Units Library contains the following information:

- The units type (Pressure, Temperature, Volume, Area etc.)
- The units of measure for that units type. (torr, atmosphere, mbar, Pascal, dynes/cm^2 etc.).
- Conversion factors to convert the units of measure to the system units of measure for that units type. The conversion factors allow transformations of the form y = a*x + b.

The system units of measure are the units of measure used in the SPICE models. For example, a voltage of 1 volt in the SPICE electronic simulation is used to represent a pressure of 1 atmosphere: the system units of measure for pressure is atmospheres.

The information in the Units Library is used throughout the simulator wherever the user enters a numerical quantity. VACSIM stores all its data internally in system units, but the user is always presented with the data in the units of measure last used.

Bitmap Files

Bitmap files for the simulator all have the **.BMP** extension and are installed in the same directory as VACSIM.EXE.

The bitmap files are displayed when *Display/System Bitmap* or *Display/Component Bitmap* is selected from the drawing window menu. The records in the Model Library and the Component Library contain references to their corresponding bitmap files.

Codeword File

The file VACSIM.CWD is the codeword file which authorises the use of the software. It is an ASCII file containing 16 hexadecimal digits, and is installed in the same directory as VACSIM.EXE.

If the file is absent, or the codeword is invalid or out of date, VACSIM will run in demonstration mode.

System Model Files

VACSIM Drawing Model File

Drawing model files have the file extension **.MDL** and store the schematic drawing of the vacuum system. The *File/Open*, *File/Save* and *File/Save As* menu commands allow you to place these files wherever you wish - the default directory is that in which VACSIM.EXE is installed.

VACSIM Model Report File

Report files have the .TXT extension and are generated by VACSIM each time a simulation is run. If the VACSIM Drawing Model File is called MYMODEL.MDL, then the report file will be MYMODEL.TXT and will be in the same directory as MYMODEL.MDL.

The report lists the vacuum components in the model together with the parameters which specify their behaviour.

You may delete the .TXT files after a simulation is run, but this will rarely be necessary because they occupy about a quarter of the space of their corresponding .MDL file.

SPICE Circuit File

SPICE circuit files have the **.CKT** extension and are generated by VACSIM each time a simulation is run. If the VACSIM Drawing Model File is called MYMODEL.MDL, then the SPICE circuit file will be MYMODEL.CKT and will be in the same directory as MYMODEL.MDL.

You may delete the **.CKT** files after a simulation is run, but this will rarely be necessary because they occupy about a quarter of the space of their corresponding **.MDL** file.

Simulator Status Files

Each time the SPICE Engine is run, it produces two output files, SPICESTD.OUT and SPICESTD.ERR, in the same directory as the VACSIM Drawing Model File. You should not need to inspect these unless you get a run-time error from the SPICE Engine.

Dongle and Codeword

Overview

The dongle and codeword together provide authorisation for you to use all features of the simulator. The dongle affords copy protection and the codeword provides the authority to use the software up to an agreed licence date.

If the dongle or codeword is absent, or the codeword out of date, the software will run in <u>Demonstration Mode</u>.

The Dongle

The dongle is a Data Encryption Systems DK12 parallel port copy protection device. It can be placed on any parallel port on the computer; the DK12WN16 and DK12WN32 dynamic link libraries and the appropriate parallel port device driver (see Executables and DLLs) will automatically detect its presence and establish contact with it as required.

The dongle is designed to be stackable and is completely transparent when not in use. If you do have problems with interactions between dongles, place the DK12 first in the chain.

Codeword

The codeword is contained in the file VACSIM.CWD. It is an ASCII file containing 16 hexadecimal digits, and is installed in the same directory as VACSIM.EXE. When you receive the dongle, it will be shipped with a codeword.

To create the codeword file you should open the Windows Notepad editor and type in the 16 character codeword without any blank lines or spaces at the start of the file; all text characters should be entered in upper case. You can then use the *File/Save As* command to save it as VACSIM.CWD in the same directory as VACSIM.EXE.

The codeword determines the duration of your licence to run

VACSIM. If you have a limited duration licence VACSIM will notify you at startup when the codeword is within one month of expiry. You should then contact Technology Sources for a new codeword (see Getting Technical Support).

When the codeword has expired, or if it is absent, the software will run in Demonstration Mode.

Demonstration Mode

Demonstration mode allows you to use all the simulation features of the simulator, but disables the following actions from the drawing window and plotting windows:

- *File/Save*
- *File/Save As*
- *File/Print*
- *File/Clip*
- *Print*
- *Clip*

The GENERIC vacuum components, which allow the user access to all the parameters of a component, are also disabled.

Demonstration mode allows you to construct a system from a restricted range of components and to simulate its performance. It does not allow you to construct arbitrary systems or save the results from the simulation.

Trouble Shooting

If VACSIM encounters an unexpected condition, it produces an informative message box describing the error and the remedial action required.

Messages from VACSIM are grouped into 3 categories; the severity level is indicated by the heading in the message box:

- Errors.
- Warnings
- Information

A fourth category - Windows system errors are produced by Windows. These include hardware errors, floating point exceptions etc.

VACSIM Errors are internal system errors. If these occur, there is a high probability that the simulation will not give correct results. If you generate any of these errors contact Technology Sources. (See Getting Technical Support).

VACSIM Warnings indicate that there is a fault which needs to be corrected by the user. The Warning message should contain sufficient information to allow you to rectify the fault. If you receive an "Error detected in SPICE Simulator" Warning message, refer to Avoiding Convergence Problems.

VACSIM Information messages are produced to tell you that certain conditions exist. You may be presented with a choice of actions, any of which is acceptable.

END