

TINA - TI

Version 9

**Special complimentary version of the
TINA circuit simulation program
distributed by Texas Instruments**

QUICK START

DesignSoft

www.designsoftware.com

COPYRIGHTS

© *Copyright 1990-2011 DesignSoft, Inc.* All rights reserved.

All programs recorded on the original release CD of *TINA* and the accompanying documentation are copyrighted. *TINA* is provided under a License Agreement and may be used or copied only in accordance with its terms and conditions.

LIMITED LIABILITY

TINA, together with all accompanying materials, is provided on an “as is” basis, without warranty of any kind.

DesignSoft, Inc., its distributors, and dealers make no warranty, either expressed, implied, or statutory, including but not limited to any implied warranties of merchantability or fitness for any purpose.

In no event will DesignSoft Inc., its distributor or dealer be liable to anyone for direct, indirect, incidental or consequential damages or losses arising from the purchase of *TINA* or from use or inability to use *TINA*.

TRADEMARKS

IBM PC/AT, PS/2 are registered trademarks of *International Business Machines Corporation*

Windows, Windows 9x/ME/NT/2000/XP/Vista / Windows 7 are trademarks of *Microsoft Corporation*

PSpice is a registered trademark of *MicroSim Corporation*

Corel Draw is a registered trademark of *Corel Inc.*

TINA is a registered trademark of *DesignSoft, Inc.*

** English version*

TABLE OF CONTENTS

1. INTRODUCTION	7
1.1 What is TINA and TINA Design Suite?	7
1.1.1 What is TINA-TI?	13
1.1.2 New features in TINA-TI v9	14
1.1.3 Extra features in TINA DESIGN SUITE 9 compared to TINA-TI	15
2. NEW FEATURES IN TINA	17
2.1 New features in TINA-TI v9	17
2.2 List of New features in TINA v9	18
2.3 List of new features in TINA Design Suite 8	20
3. INSTALLATION AND START-UP	23
3.1 Installation Procedure	23
3.1.1 Minimum hardware and software requirements	23
3.1.2 Installation from TI website	23
3.1.3 Installation from CD-ROM	24
3.1.4 Following the Installation Steps	24

3.1.5	Welcome and Software License Agreement	24
3.1.6	Entering User Information	25
3.1.7	Choose Destination Location	24
3.1.8	Selecting the Program Folder	25
3.1.9	Selecting the Symbol Set	25
3.1.10	Final check and copying the files	25
3.1.11	Completing the Setup	26
3.2	Uninstalling TINA	26
3.3	Starting Up	26
3.4	Experimenting with Example Circuits, avoiding common problems	27

4. GETTING STARTED 29

4.1	Schematic Editing Using the Mouse	29
4.1.1	Using the right mouse button	29
4.1.2	Using the left mouse button	30
4.2	Measurement Units	32
4.3	The Basic Screen Format	32
4.4	Placing the Circuit Components	40
4.4.1	Wire	41
4.4.2	Input and Output	42
4.5	Exercises	43
4.5.1	Creating and editing Circuit Schematic	43
4.6	Analyses	48
4.6.1	Analysing the circuit	50
4.6.2	Testing Your Circuit with Virtual and Real Time Instruments	57
4.6.3	Analysis of SMPS circuits	62

5. CREATING SUBCIRCUITS FROM SPICE MACROS AND MODELS 73

5.1 Making a Macro from a schematic..... 73

5.2 Making a Macro from a Spice subcircuit 80

5.2.1 Creating Spice Macros in TINA 80

5.2.1.1 Creating macros from downloaded files..... 80

5.2.1.2 Creating macros on-the-fly by browsing the web 84

5.2.2 Adding Parameters to Spice Macros 90

6. MAKING YOUR OWN SCHEMATIC SYMBOLS 93

7. ADVANCED TOPICS 97

7.1 Introduction 97

7.2 Table of Contents 98

INTRODUCTION

1.1 What is TINA and TINA Design Suite?

TINA Design Suite is a powerful yet affordable software package for analyzing, designing and real time testing of analog, digital, VHDL, and mixed electronic circuits and their layouts. You can also analyze RF, communication, and optoelectronic circuits, and test and debug microprocessor & microcontroller applications. Every year, electronic circuits become faster and more complex, and therefore require more and more computational power to analyze their operation. To meet this requirement DesignSoft engineers have included the ability to utilize the increasingly popular scalable multi-thread CPUs. Computers that incorporate dual or quad core CPUs can deliver up to 20 times faster execution time for TINA's analysis engine—while maintaining the accuracy of prior single-threading computers.

A unique feature of TINA permits you to bring your circuit to life with the optional USB controlled TINALab II and LogiXplorer hardware, turning your computer into a powerful, multifunction T&M instrument. Electrical engineers will find TINA an easy to use, high performance tool, while educators will welcome its unique features for the training environment.

TINA is distributed in two major versions – TINA and TINA Design Suite. **TINA** includes circuit simulation only, while **TINA Design Suite** also includes the advanced PCB designer. This fully integrated layout module has all the features you need for advanced PCB design, including Multilayer flexible PCB's with split power planes, powerful autoplacement & autorouting, rip-up and reroute, manual and "follow-me" trace placement, DRC, forward and back annotation, pin

and gate swapping, keep-in and keep-out areas, thermal relief, fanout, plane layers, Gerber file output and much more.

TINA can also be used in the training environment. It includes unique tools for testing students' knowledge, monitoring progress and introducing troubleshooting techniques. With optional hardware it can be used to test real circuits for comparison with the results obtained from simulation. Of great importance to educators, the package includes all the tools needed to prepare educational materials.

Schematic Capture. Circuit diagrams are entered using an easy to use schematic editor. Component symbols chosen from the Component bar are positioned, moved, rotated and/or mirrored on the screen by the mouse. TINA's semiconductor catalog allows the user to select components from a user-extendible library. An advanced "rubber wire" tool is provided allowing easy modification of the schematic diagrams. You can open any number of circuit files or subcircuits, cut, copy and paste circuit segments from one circuit into another, and, of course analyze any of the currently open circuits. TINA gives you tools to enhance your schematic by adding graphics elements such as lines, arcs, arrows, frames around the schematic, and title blocks. You can also draw non-orthogonal (diagonal) components such as bridges and 3-phase networks.

Live 3D Breadboard Tool. You can take your design for a solder-less breadboard (sometimes called a "whiteboard") and automatically build a life-like 3D picture of the breadboard. Now when you run TINA in interactive mode, virtual components such as switches, LEDs, instruments, etc. become "live" and will function with satisfying realism. Students will use the Live 3D Breadboard tool to prepare and document eye-catching lab experiments.

PCB Design. TINA includes only circuit simulation, while TINA Design Suite includes TINA's advanced PCB designer. This fully integrated layout module has all the features you need for advanced PCB design, including Multilayer PCB's with split power planes, powerful autoplacement & autorouting, rip-up and reroute, manual and "follow-me" trace placement, DRC, forward and back annotation, pin and gate swapping, keep-in and keep-out areas, thermal relief, fanout, plane layers, 3D view from any angle, and much more. With TINA Design Suite you can prepare a PCB in at least two ways: using the G-Code control files to make in-house prototypes with milling machines using the G-Code control files provided by TINA; or sending Gerber files to PCB manufacturers.

Electrical Rules Check (ERC) will examine the circuit for questionable connections between components and display the results in the Electrical Rules Check window. ERC is invoked automatically, so missing connections will be brought to your attention before analysis begins.

Schematic Symbol Editor. In *TINA*, you can simplify a schematic by turning portions of it into a subcircuit. In addition, you can create new *TINA* components from any Spice subcircuit, whether created by yourself, downloaded from the Internet, or obtained from a manufacturer's CD. *TINA* automatically represents these subcircuits as a rectangular block on your schematic, but you can create any shape you like with *TINA*'s Schematic Symbol Editor.

Library Manager. *TINA* has large libraries containing Spice- and S-parameter models provided by semiconductor manufacturers such as Analog Devices, Texas Instruments, National Semiconductor, and others. You can add more models to these libraries or create your own Spice- and S-parameter library using *TINA*'s Library Manager (LM).

Parameter Extractor. Using *TINA*'s Parameter Extractor you can also create component models that more closely represent actual real world devices by converting measurement or catalog data into model parameters.

Text and Equation Editor. *TINA* includes a Text and Equation Editor for annotating schematics, calculations, includes graphic output, and measurement results. It is an invaluable aid to teachers preparing problems and examples.

The circuit diagrams and the calculated or measured results can be printed or saved to files in standard Windows BMP, JPG and WMF format. These output files can be processed by a number of well known software packages (Microsoft Word, Corel Draw etc.). Netlists can be exported and imported in Pspice format and also to drive popular PCB packages such as ORCAD, TANGO, PCAD, PROTEL, REDAC and other programs.

DC analysis calculates the DC operating point and the transfer characteristic of analog circuits. The user can display the calculated and/or measured nodal voltages at any node by selecting the node with the cursor. For digital circuits, the program solves the logic state equation and displays the results at each node step-by-step.

Transient analysis. In the transient and mixed mode of *TINA* you can calculate the circuit response to the input waveforms that can be

selected from several options (pulse, unit step, sinusoidal, triangular wave, square wave, general trapezoidal waveform, and user-defined excitation) and parameterized as required. For digital circuits, programmable clocks and digital signal generators are available.

Fourier analysis. In addition to the calculation and display of the response, the coefficients of the **Fourier series**, the **harmonic distortion** for periodic signals, and the **Fourier spectrum** of non-periodic signals can also be calculated.

Digital Simulation. TINA also includes a very fast and powerful simulator for digital circuits. You can trace circuit operation step-by-step, forward and backward, or view the complete time diagram in a special logic analyzer window. In addition to logic gates, there are ICs and other digital parts from TINA's large component library.

VHDL simulation. TINA now includes an integrated VHDL simulator to verify VHDL designs both in digital and mixed-signal analog-digital environments. It supports the IEEE 1076-1987 and 1076-1993 language standards and IEEE standards 1164 (standard logic). Your circuits can contain editable VHDL blocks from TINA's library, FPGAs & CPLDs, or VHDL components created by yourself or downloaded from the Internet. You can edit the VHDL source of any VHDL component and see the result instantly. With the optional external VHDL simulator, you can develop and debug your VHDL code both externally and inside TINA. The VHDL simulator includes Waveform Display, Project Management and Hierarchy Browser, and 64-bit time.

Microcontroller (MCU) simulation. TINA includes a wide range of microcontrollers (PIC, AVR, 8051, HCS, ARM) which you can test, debug and run interactively. The built in MCU assembler allows you to modify your assembler code and see the result promptly.

Flowchart Editor and Debugger. Writing MCU assembly code is often a hard and tedious task. You can simplify software development and gain more time to design the electronics hardware if, instead of manual coding, you use TINA's Flowchart editor and debugger to generate and debug the MCU code. This easy-to-use tool works with symbols and flow control lines with which you can represent the algorithm you want.

AC analysis calculates, complex voltage, current, impedance, and power can be calculated. In addition, Nyquist and Bode diagrams of the amplitude, phase and group delay characteristics of analog circuits can

be plotted. You can also draw the complex **phasor diagram**. For non-linear networks, the operating point linearization is done automatically.

Network analysis determines the two-port parameters of networks (S, Z, Y, H). This is especially useful if you work with **RF circuits**. Results can be displayed in Smith, Polar, or other diagrams. The network analysis is carried out with the help of TINA's network analyzer. The RF models of the circuit elements can be defined as SPICE subcircuits (SPICE macros) which contain parasitic components (inductors, capacitors) or as an S-parameter model defined by its S (frequency) function. S functions are normally provided by the component manufacturers (based on their measurements) and can be downloaded from the Internet and inserted into TINA either manually or by using TINA's library manager.

Noise analysis determines the noise spectrum with respect to either the input or the output. The noise power and the signal-to-noise ratio (SNR) can also be calculated.

Symbolic analysis produces the transfer function and the closed form expression of the response of analog linear networks in DC, AC, and transient modes. The exact solution, calculated through the symbolic analysis, can also be plotted and compared to the numerically calculated or measured results. The built-in **interpreter** can evaluate and plot arbitrary functions.

Monte-Carlo and Worst-case analysis. Tolerances can be assigned to the circuit elements for use in Monte-Carlo and/or worst-case analyses. The results can be obtained statistically, and their expected means, standard deviations and yields can also be calculated.

Design Tool This powerful tool works with the design equations of your circuit to ensure that the specified inputs result in the specified output response. The tool offers you a solution engine that you can use to solve repetitively and accurately for various scenarios. The calculated component values are automatically set in place in the companion TINA schematic and you can check the result by simulation. This feature is also very useful for semiconductor and other electronics component manufacturers to provide application circuits along with the design procedure.

Optimization. TINA's enhanced optimization tool can tweak one or more unknown circuit parameters to achieve a predefined target

response. The target circuit response (voltage, current, impedance, or power) must be “monitored” by meters. For example, you can specify several working point DC voltages or AC transfer function parameters and have TINA determine the values of the selected components.

Post-processor. Another great new tool of TINA is its post-processor. With the post-processor, you can add new curves of virtually any node and component voltage or current to existing diagrams. In addition, you can post-process existing curves by adding or subtracting curves, or by applying mathematical functions to them. You can also draw **trajectories**; i.e., draw any voltage or current as a function of another voltage or current.

Presentation. With TINA you can make quality documents incorporating **Bode plots, Nyquist, Phasor, Polar and Smith diagrams, transient responses, digital waveforms** and other data using linear or logarithmic scales. Customize presentations easily using TINA’s advanced drawing tools—you can print your plots directly from TINA, cut and paste them into your favorite word processing package, or export them to popular standard formats. Customization includes complete control over texts, axes, and plot style; e.g., setting line width and color, fonts in all sizes and color, and automatic or manual scaling for each axis.

Interactive mode. When everything is in order, the ultimate test of your circuit is to try it in a “real life” situation using its interactive controls (such as keypads and switches) and watching its displays or other indicators. You can carry out such a test using TINA’s interactive mode. You can not only play with the controls, but you can also change component values while the analysis is in progress. In addition, you can assign hotkeys to component values and switches to change them simply by pressing a key. You will immediately see the effect of the change. You can also test MCU applications in TINA’s interactive mode. You can not only run and test them using the several lifelike interactive controls e.g., keyboards. but you can also debug them while the MCU executes ASM code step by step, And displays the register contents and TINA’s outputs in each step. If necessary you can modify the ASM code on the fly and test your circuit again without using any other tool.

Virtual instruments. In addition to standard analysis presentations such as Bode and Nyquist plots, *TINA* can present its simulation results on a wide range of high-tech virtual instruments. For example,

you can simulate the time response of your circuit using a virtual square wave generator and a virtual oscilloscope. Using TINA's virtual instruments is a good way to prepare for the use of real test and measurement equipment. Of course it is important to remember that the “measurement results” obtained with virtual instruments are still simulated.

Real-time Test & Measurements. *TINA* can go beyond simulation when **supplementary hardware** is installed on the host computer. With this hardware, TINA's powerful tools can make **real-time measurements** on real circuits and display the results on its virtual instruments.

Training and Examination. *TINA* has special operating modes for training and for examination. In these modes, under TINA's control, the students solve problems assigned by the teacher. The solution format depends on the types of problems: they can be selected from a list, calculated numerically, or given in symbolic form. The interpreter - providing a number of solution tools - can also be used for problem solving. If the student cannot solve the problem, he/she can turn to the multilevel Advisor. The package includes all the tools needed to produce educational materials. A collection of examples and problems worked out by teachers is also part of the package. Another special educational function of *TINA* is the **software or hardware simulation of circuit faults** to practice troubleshooting. Using *TINA*, you can transform existing PC classrooms into contemporary electronics training labs at low cost.

1.1.1 What is TINA-TI?

TINA-TI is a complimentary, limited version of TINA Design Suite circuit simulation software that is distributed by Texas Instruments (TI) in cooperation with DesignSoft, the developer of TINA.

It includes macromodels and application examples for TI integrated circuits now including not only analog Operational Amplifiers but also SMPS controllers and is intended to demonstrate commercial circuit applications for TI's products.

TINA-TI is not authorized for educational use, and should not be used by educators for the purpose of educating pupils, or by students

or others for educational purposes. For educational use special versions of TINA are available which were developed by DesignSoft especially for teachers and students. For more information see www.tina.com under “Education”.

The information provided and derived herein is believed to be reliable; however Texas Instruments Inc. (TI) assumes no responsibility for inaccuracies or omissions. TI assumes no responsibility for the use of this information, and the use of all such information shall be entirely at the user’s own risk. No patent rights or licenses of any kind are implied or granted to any third party. TI does not authorize or warrant any TI product for use in life-support devices and/or systems.

TINA-TI is derived from TINA Design Suite, the Complete Electronics Lab program by DesignSoft for analyzing, designing and real time testing of analog, digital, VHDL, and mixed electronic circuits and their PCB layouts.

From TINA-TI you can upgrade to the full version of TINA Design Suite at a special price at

www.tina.com/ti-upgrade.htm

1.1.2 New features in TINA-TI v9

- Multi core support
- Over 100 SMPS models from TI
- Up to 10 times faster on 1 core, 15x on dual core, 20x on quad core
- Fast diagram drawing and processing speed, even for very large circuits, scrollbar for moving multiple diagrams on the same page
- Combinational and parallel stepping, separate cases in diagram
- Schematic Symbol Editor Included
- Addition of new components from other manufacturers
- The nodes list in post processor shows the names and node names in the relevant macros
- Gain margin, Phase margin
- Linear circuits allowed
- Ideal Opamp added

- Minimizing TINA while analysis is running
- Running several TINAs on the same computer simultaneously
- Running second TINA without extra license allowed on the same computer
- Position cursor both by entering x or y value, 0 db Cross over frequency
- ERC in Spice subcircuits, you can make and ERC either for the whole circuit or for (Spice or schematic) subcircuits
- ERC detects floating nodes in spice subcircuits
- Open Examples command in File menu
- Open .cir files from Open command in the File menu
- My Places button in Open dialog
- Conversion to v7, for compatibility with v7 and v8
- Possibility to place hyperlink on the schematic diagram
- Detailed run-time statistics
- Links to TI Design Center, FAQ and Sample part request
- XML format export and import of circuit schematic diagrams
- TINA is now digitally signed by DesignSoft

1.1.3 Extra features in TINA DESIGN SUITE 9 compared to TINA-TI

- Digital and mixed-mode analysis
- Monte Carlo and Worst-case analysis, statistics
- Symbolic analysis
- Interactive circuit analysis, with switches, buttons, displays, etc.
- Multiparameter optimization
- Network analysis, S-parameters
- Large extendible component library containing more than 20,000 parts and models
- User defined excitation
- Parameter stepping with multiple parameters

- Analog control components and simulation
- Fourier spectrum (continuous spectrum)
- Phasor-,Nyquist- and group-delay diagrams
- Interpreter for post processing the results
- Optional real-time test and measurement
- Library manager for extending component libraries
- Parameter extractor for creating new component models
- More components: Optoelectronics, bus, VCO, BSIM, communication circuits
- Built-in VHDL analysis also in interactive mode
- User defined VHDL components, modifiable on-the-fly VHDL, Xilinx - - UCF generation of digital circuits for synthesis, place-and-route and presentations with the TINA FPGA board
- Microcontroller simulation including PIC, AVR, 8051 and ARM 9 processors
- ASM debugging
- Stress (smoke) Analysis
- Active and passive filter design
- Logic design (Truth table, Karnaugh table, Quine-McClusky optimization, schematic diagram generation)
- Single click 3D component preview

TINA Design Suite v9:

- Includes all TINA 9 features plus TINA's advanced PCB designer
 - Autoplacement and autorouting, rip-up-and-retry
 - Manual and "follow-me" trace placement
 - Forward and backward annotation
 - Pin and gate swapping
 - Keep-in and keep-out areas
 - Thermal relief, fanout
 - Plane layers for multilayer designs
 - Gerber file output
 - Footprint Editor

Much more, described at

www.tina.com/ti-upgrade.htm

NEW FEATURES IN TINA

This chapter describes the new features and changes in the latest TINA v9 version and also in the previous 8.0 and 7.0 of TINA. Many of them were suggested by TINA users, while others were created by DesignSoft's team of software and electrical engineers. We are sure you will share our excitement about these new features. Note that not all new features of TINA 9 are included in TINA-TI.

2.1 New features in TINA-TI v9

- Multi core support
- Over 100 SMPS models from TI
- Up to 10 times faster on 1 core, 15x on dual core, 20x on quad core
- Fast diagram drawing and processing speed, even for very large circuits, scrollbar for moving multiple diagrams on the same page
- Combinational and parallel stepping, separate cases in diagram
- Schematic Symbol Editor Included
- Addition of new components from other manufacturers
- The nodes list in post processor shows the names and node names in the relevant macros
- Gain margin, Phase margin
- Linear circuits allowed
- Ideal Opamp added
- Minimizing TINA while analysis is running
- Running several TINAs on the same computer simultaneously
- Running second TINA without extra license allowed on the same computer

- Position cursor both by entering x or y value, 0 db Cross over frequency
- ERC in Spice subcircuits, you can make and ERC either for the whole circuit or for (Spice or schematic) subcircuits
- ERC detects floating nodes in spice subcircuits
- Open Examples command in File menu
- Open .cir files from Open command in the File menu
- My Places button in Open dialog
- Conversion to v7, for compatibility with v7 and v8
- Possibility to place hyperlink on the schematic diagram
- Detailed run-time statistics
- Links to TI Design Center, FAQ and Sample part request
- XML format export and import of circuit schematic diagrams
- TINA is now digitally signed by DesignSoft

2.2 List of New features in TINA v9

- Full Vista and Windows 7 compatibility
- Multi core support for dual, quad, i7 and higher multi-core processors
- Enhanced analysis speed up to 10 times higher speed on 1 core, 15x on dual core, 20x on quad core
- Improved convergence performance based on the latest state of the art algorithms
- Advanced powerful Spice-VHDL mixed mode analysis
- Extended catalog with ARM 7, ARM 9 and HCS08 microcontrollers, more SMPS ICs models, realistic ADC and DAC models, LCD display, Bi-color LEDs
- Extended MCU simulation, USB and other modules
- Programmable Design Tool where users can implement design procedures for calculating and setting circuit parameters so that the circuits can produce predefined target output values.
- Extended flowchart tool: PIC, AVR, 8051, code box, USART, external interrupt handling
- Open and save TINA designs, models and libraries directly from the Web

- Import Spice .CIR and .LIB files directly from the Web
- Import Libraries, Examples and Designs from any earlier versions of TINA v7 and later.
- Show analysis results on diagrams during analysis
- Fast diagram drawing and processing speed, even for very large circuits
- New tool in the View menu of the diagrams for easier hiding and showing curves on diagrams
- New more effective storing of analysis results, save all analysis results for all circuits without speed compromise
- Parametrizable output curve filter for even more effective storage of results
- Scrollbars for browsing multiple diagrams on the same page
- New diagram cursor tools to find and annotate
 - local and global maximum, minimum
 - gain and phase margins
 - zero crossover frequency
 - horizontal and vertical coordinates
- Combinational and parallel stepping, show stepping cases (test benches) in separate diagrams
- More advanced tools for addition of parts to TINA from semiconductor manufacturers both from downloaded files and directly from the internet
- Node list in post processor now displays macro names and local node names in the relevant macros
- Minimize TINA main window while analysis is running
- Run two TINA instances on the same computer
- Multi-level, hierarchical ERC. Checks the main or any subcircuits, or the whole circuit in full depth
- ERC detects and shows floating nodes in Spice subcircuits
- Open Examples command in File menu, for easier return to TINA's Examples folder
- Open .CIR files from Open command in the File menu
- New arrow button in Open and Save dialog, for easier finding the TINA EXAMPLES or User folders

- Updated and extended examples under the EXAMPLES folder
- Conversion of circuits to TINA v7 format, for compatibility with TINA v7, v8 and v9
- Detailed Transient Statistics from the View menu
- TINA v9 is now digitally signed by DesignSoft

PCB Design

- G-Code export of PCB designs for creating control files for milling machines
- Revised and largely extended PCB footprint catalog

2.3 List of new features in TINA Design Suite 8

- Vista style installation and folder scheme
- Behavioral building blocks, nonlinear controlled sources
- Powerful Spice-VHDL co-simulation including MCUs
- Finite State Machine (FSM) editor with VHDL generation
- Flowchart editor and debugger for controlling MCUs
- Any number of MCUs in one circuit
- Extended MCU catalog including PIC18, CAN and more
- Execution time measurement and statistics for Transient Analysis
- Hyperlinks can be added to schematics and to the diagram window
- Extended semiconductor catalog
- Application examples from Texas Instruments
- Labview based virtual instruments
- Interface to build LabVIEW based virtual instruments
- Wave (.wav) files can be used as input
- Wave (.wav) export output file
- New Open Examples command in file menu to open built in examples
- Autosave. Save your current schematic or PCB design at adjustable time intervals.
- Parameter adding possibility to Spice subcircuits
- On line update possibility for libraries, program or both

- Post-processing formulas are stored with schematics, editable later
- Live 3D Breadboard (displaying and animating circuits with 3D parts on a virtual 3D breadboard)
- 3D virtual instruments to prepare and document lab experiments
- Integrated electronic design textbook with “live” circuits (optional)
- SMPS design templates from Christophe Basso (optional)
- Detection of components or nodes linked with convergence or irregular circuit problems.

PCB Design (Only in TINA Design Suite v8 Edition, additional to TINA v8)

- Creating “flex” PCBs including 3D display
- Creating PCBs of any shape including round edges
- Buried and blind vias
- Extended catalog
- Improved optimizing autorouter
- Display of complete 3D circuits including parts connected externally to the PCB

INSTALLATION AND START-UP

3.1 Installation Procedure

3.1.1 Minimum hardware and software requirements

- Intel Pentium or equivalent processor
- 256 MB of RAM
- 200 MB of available hard disk space
- CD-ROM
- Mouse
- VGA adapter card and monitor
- Microsoft Windows 9x / ME / NT / 2000/ XP / Vista / Windows 7

3.1.2 Installation from TI website

TINA-TI can be downloaded as a zip file from Texas Instruments web site at www.ti.com/tina-ti. To download and start installation click the Download button, select Open, and after the download

double-click on the TINA9-TI.exe file inside the zip file. You can also download and save the zip file. Open the saved copy and doubleclick TINA9-TI.exe inside the zip file. The setup program will start.

3.1.3 Installation from CD-ROM

To begin the installation simply insert the CD into your CD-ROM drive. The Setup Program will start automatically if the Auto-Run function of your CD-ROM has been enabled (Windows-Default).

If not, Select Start/Run and type:

D:SETUP (Enter) (where D represents your CD-ROM drive).

The setup program will start.

3.1.4 Following the Installation Steps

TINA's Setup Procedure follows the steps standard with most Windows Programs. There are several screen pages where you can enter or change important installation choices, such as Type of Installation, Destination Directory, etc. To continue installation, click on . You can always step back, using the -Button. If you do not want to continue installation for any reason, click on . If you elect to cancel installation, the program will ask you if you really want to exit. At this point you can either resume or exit Setup.

3.1.5 Welcome and Software License Agreement

To begin the Procedure click on Next on the Welcome Page. The first step is the Software License Agreement.

NOTE:

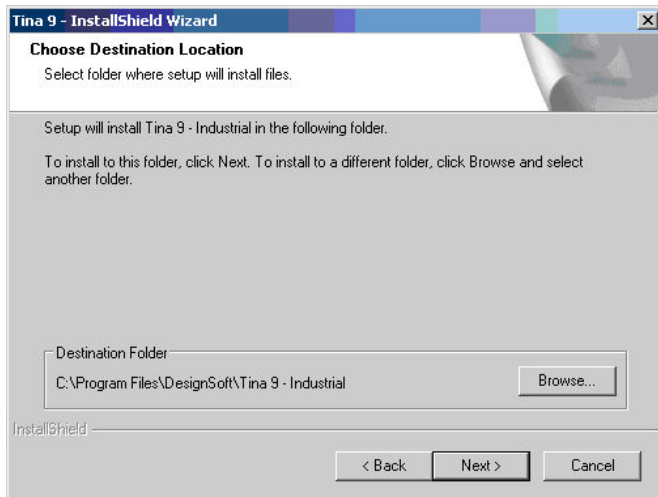
By clicking on "Yes" you are agreeing fully with DesignSoft's Terms and Conditions for using this software.

3.1.6 Entering User Information

This data is used to personalize your copy of the software. By default, the installation program picks up the data entered when you set up Windows. You accept these names as defaults by clicking on Next or you can change them.

3.1.7 Choose Destination Location

Here you can select an Installation Directory other than the one suggested as a default. The default is the Windows Standard Directory for Programs. To change the directory, click on Browse and select a different drive and/or directory from the Choose Folder Dialog.



IMPORTANT NOTE:

If you are installing TINA for Windows to a hard disk that already has an earlier version of TINA, you must be sure to use a new directory name for TINA for Windows, such as the suggested directory, C:\Program Files\DesignSoft\Tina, or the working files you have already created will be overwritten and lost. If uncertain, exit setup, copy your TINA files safely to another hard disk directory or to floppy disks, then resume setup.

3.1.8 Selecting the Program Folder

Here you can choose where in the Programs Section of your Windows Start Menu the Program Icons will appear. The default is a new sub menu called for example TINA 9 – Industrial. You can change this name or select an existing Program Folder from the list.

3.1.9 Selecting the Symbol Set

TINA can display its component schematic symbols according to the US (ANSI) or the European (DIN) conventions. Select the one appropriate for you.

3.1.10 Final check and copying the files

This page lists the settings you have made, giving you an opportunity to check and alter them and step back if changes are necessary. After you click on Next, the Setup Program starts copying the files automatically.

3.1.11 Completing the Setup

After all the selected files have been copied and the Start Menu entries created, you are asked if you want to place a Shortcut to the TINA program file on your Desktop. The last page indicates a successful installation and invites you to open and read a file with the latest information about TINA. We urge you to take a moment and review that file. Click on finish when you're ready.

3.2 Uninstalling TINA

You can uninstall *TINA* at any time. Note that this will not delete files you have created.

1. To begin Uninstallation choose **Tina** from the **TINA Start Menu Entries**.
2. In the window that appears, double-click on **Uninstall Tina**.
3. Click on *Yes* if you are positive you want to uninstall *TINA*.

After all the files have been removed successfully, an *OK* Button appears. Click on it and uninstallation is complete.

3.3 Starting Up

After successfully installing *TINA*, you can start the program by simply double-clicking the *TINA* icon on your Desktop or by choosing *Tina* from the Tina Start Menu Entries.

3.4 Experimenting with Example Circuits, avoiding common problems

Start the program and click the *File* menu item in the top line of the screen to drop down the *File* menu. Select the *Open* command and the standard open file dialog box appears with **.TSC*, indicating that a file name with *.TSC* extension is sought. Select the *EXAMPLES* folder, and a list of files with *.TSC* extensions will appear.

After selecting a file, the circuit schematic will appear. Now you can execute an analysis, modify or expand the circuit, and evaluate the results.

GETTING STARTED

In this chapter, we present TINA's screen format and menu structure. A step by step introduction is given using examples.

4.1 Schematic Editing Using the Mouse

Here are some basic mouse techniques to help you edit schematics:

4.1.1 Using the right mouse button

If you press the right button of the mouse at any time, a popup menu appears. Using this menu you can:

- **Cancel Mode:** Exit from the last operation (e.g. moving a component, drawing a wire).
- **Last Component:** Return to the last component and reposition it.
- **Wire:** Switch to wire-drawing mode. In this mode, the cursor turns into a pen and you can draw a wire. For more details, see the **Wire** paragraph below.
- **Delete:** Delete selected component(s).
- **Rotate Left, Rotate Right, Mirror:** Rotate or mirror the component currently selected or being moved. You can also rotate a selected component by pressing the Ctrl-L or Ctrl-R keys.

- **Properties:** Use this command to edit the properties (value, label) of the component currently selected or being moved. From the Properties menu, you can set all parameters of a component (before it is placed). This lets you place multiple copies of the component, all with the properties just entered. While you are in the component property editor, the right mouse button has another function. When you are editing the field of any component parameter other than the label field, you can copy that field to the label field by pressing the right mouse button and then selecting the *Copy to Label* command. You can accomplish the same thing by pressing [F9]. TINA PRO v6 can be extended with an optional integrated VHDL simulator that verifies VHDL designs both in digital and mixed-signal analog-digital environments. The VHDL simulator includes Waveform Display, Project Management and Hierarchy Browser, and 64-bit time. It supports the IEEE 1076-1987 and 1076-1993 language standards and IEEE standards 1164 (standard logic), 1076.3 (numeric standard), and 1076.4 (VITAL standard).

4.1.2 Using the left mouse button

In the descriptions below, ‘clicking’ always refers to the left mouse button.

- **Selection:** Clicking on an object will select the desired object and deselect all other objects.
- **Multiple selection:** Clicking while holding down the [Shift] key will add the object under the cursor to the group of currently selected objects. If the object under the cursor is already in the currently selected group, clicking will remove it from the group.
- **Block selection:** To select a block of objects all at once, first make sure there is no object under the cursor. Then press and hold down the left button while moving the mouse (dragging). This will create a rectangular block, and all objects within the block will be selected.
- **Selection of all objects:** Press Ctrl+A to select all objects.

- **Moving objects:** A single object can be moved by dragging it (Position the cursor on the object, press and hold the left button, and move the mouse.) Multiple objects can be moved by first selecting them (see above), then clicking the left button while the cursor is over one of the selected objects, holding the left button down, and dragging.
- **Parameter modification:** Double-clicking on an object will bring up its parameter menu so that you can modify its parameters (if it has any).
- **Crossing wires:** The crossing of two wires does not result in a connection at the crossing unless you deliberately choose there to be one. Use Edit.Hide/Reconnect to place or remove a connecting dot. However, it is better drafting practice to never make a connection at a wire crossing, as this avoids ambiguity about the presence or absence of a dot.
- **Block or symbol copying:** After a block or symbol has been selected, you may copy it by pressing Ctrl+C. Then click outside the block or symbol to release it, and press Ctrl+V. You will see a copy of the block which you can place as you wish. If the schematic window doesn't show enough room for the copy, press Alt - to zoom out. Once you've located the block, click the left mouse button once to anchor it and a second time to deselect the moved block.

4.2 Measurement Units

When setting parameters for electronic components or specifying numerical values, you may use standard electronic abbreviations. For example, you can enter 1k (ohm) for 1000 (ohm). The multiplier abbreviations should follow the numeric value, e.g., 2.7k, 3.0M, 1u, etc.

The following characters indicate multiplier factors:

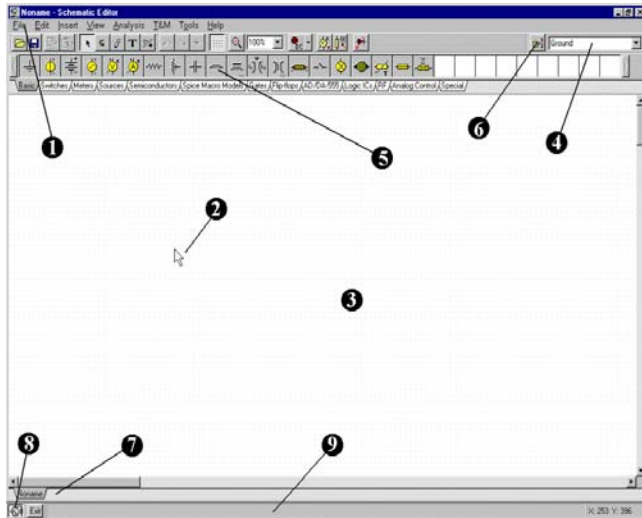
p = pico = 10^{-12}	T = tera = 10^{12}
n = nano = 10^{-9}	G = giga = 10^9
u = micro = 10^{-6}	M = mega = 10^6
m = milli = 10^{-3}	k = kilo = 10^3

NOTE:

Upper and lower cases must be carefully distinguished (e.g., M = m), and the selected letter must follow the numeric characters without a space (e.g., 1k or 5.1G), or TINA will indicate an error.

4.3 The Basic Screen Format

After start-up, the following screen appears on your monitor:



- ❶ **The Menu bar**
- ❷ **The Cursor or pointer:** This is used to select commands and to edit schematics. You can move the cursor only with the mouse. Depending on the mode of operation, the cursor assumes one of the following forms:
 - An **arrow**, when a command selection is required in the edit window.
 - A **component symbol** (accompanied by an arrow and small box), when inserting that component onto the circuit in the schematic window. Until the position of the component on the schematic is chosen, its movement is controlled by the mouse.
 - A **pen**, when defining the endpoint of a wire.
 - An **elastic line**, when defining the endpoint of a wire or the second node of an input or output.
 - An **elastic box**, when defining a block after fixing its first corner.
 - A **dashed line box**, when positioning a component label or a text block.
 - A **magnifying glass**, when defining a zoom window.

- 3 The Schematic window:** This shows the circuit schematic currently being edited or analyzed. The schematic window is actually a window onto a larger drawing area. You can move the screen window over the full drawing area using the scroll bars at the right and bottom of the screen. When selecting the New command on the File menu, the system automatically aligns the origin of the editor window with the center of the entire editor drawing area. The same is true when an existing circuit file is loaded, as this is the default window position.

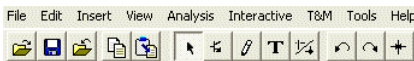
You can think of *TINA3*'s schematic as existing on several "layers." In addition to the primary layer that holds components, wires, and text, there are two other drawing layers, which you can turn on or off individually. It is generally convenient to have these two layers on.


View | Pin Markers On/Off: Displays/hides component pin ends.

A grid of closely spaced dots covering the entire drawing area may be made visible or invisible in the schematic window, depending on the current state of the *grid button* of the *Grid On/Off* switch on the *View* menu. At some schematic zoom levels, you will not see the dots of the grid; nevertheless, all component pins and connecting wires will be on the grid. These dots represent the only available interconnecting points. Component symbols are positioned on the drawing area horizontally and vertically. These symbols are rigid patterns with predefined pin positions and are handled as single units. This permits the software to unambiguously recognize the network nodes.

View | Grid On/Off: Displays/hides the grid.

- 4 The Tool bar:** You can select most of the editor commands (e.g., select, zoom, wire etc.) from this tool bar. Let's summarize the most important commands on the Toolbar. You can find more detailed information in *TINA*'s Help system. Note that most commands on the toolbar can also be found under the drop down menus, and can often be activated by Hotkeys. We show the menu name separated from the command name by a dot (Menu name.Command name).



 (*File.Open*) Opens a schematic circuit file (.TSC or .SCH), *TINA* macro files (.TSM) or Spice netlist files (.CIR).

.TSC extension is the current schematic file extension used in *TINA* v6 and above. The .SCH extension was also used for schematics in *TINA* v4 and v5.

.TSM is the extension of TINA macros, which can contain subcircuit either as a TINA schematic, Spice netlist, or VHDL code.

.CIR files must be circuit files or subcircuits in Spice netlist format. The files will appear in the Netlist Editor, where you can execute most TINA analyses, and edit or complete the netlist.



(File.Open from the web) This command starts the built in web browser that allows you to navigate to any website and then directly open TINA files with TSC, SCH or CIR extension by simply clicking a link. It will also save TSM, LIB, and TLD files into their proper place in the user area. Finally, TINA will recognize the.ZIP file extension and will help you select, copy, and extract files.

By default the built in web browser in TINA opens the [TINA circuits on Web](http://www.tina.com/English/tina/circuits) page on the www.tina.com website (at the time of writing this manual www.tina.com/English/tina/circuits) where you will find interesting electronic circuit files which you can download or open directly from the web and then simulate with TINA.



(File.Save) Saves the actual circuit or subcircuit into its original storage location. It is wise to frequently save the circuit that you are working on to avoid loss of data in case of a computer crash.



(File.Close) Closes the actual circuit or an open subcircuit on the screen. It is very useful for closing an open subcircuit.



(Edit.Copy) Copy a selected part of the circuit or text to the clipboard.




(Edit.Paste) Paste Clipboard contents into the schematic editor. Note that the content may come from the schematic editor itself, TINA's diagram window, or any other Windows program.





Selection mode. If this button is pressed you can select and drag components with the cursor. To select a component (part), wire or text, just click on it with the cursor. You can also select several objects by holding down the Ctrl key and clicking on the objects one-by-one; or by clicking at one corner of the area, holding down the left mouse button, moving to the opposite corner and then


releasing the mouse button. Selected objects will turn red. You can drag the selected objects by dragging one of them. Click and hold the left mouse button when the cursor is over one of the selected objects and move them with the mouse. You can unselect all selected object(s) by clicking on an empty area. One or more selected objects can be deleted while leaving the others still selected by holding the Ctrl key down and left-clicking the mouse.



You can select all the wire segments, connected by the same ID on Jumpers, by holding down the Shift Key and clicking one of the wire segments.


 (*Insert.Last component*) Retrieves the last component inserted, for a new insertion of another copy, with the same parameters as the previous insertion.

 (*Insert.Wire*) Use this icon for inserting (adding) wires to the schematic design.

 (*Insert.Text*) Add comments into schematics and analysis results.


 (*Edit.Hide/Reconnect*) Use Hide/Reconnect to place or remove a connecting dot between crossing wire or wire-component connection.


  (*Edit.Rotate Left (Ctrl L)*, (*Edit.Rotate Right (Ctrl R)*), Rotates the selected component.

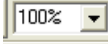
 (*Edit.Mirror*) Mirrors the selected component.

Hotkeys Ctrl L, Ctrl H



 (*Switches*) On/Off the grid. I.e.makes the grid visible or invisible.

 (*Explicitly zoom*) in on a selected portion of the current view. It will also zoom out a portion if you just click with the icon into the middle of the area you want to zoom out.

 Select *zoom ratio* from a list from 10% to 200%. You can also select Zoom All which will zoom the effective drawing to full screen.

Interactive modes, see also on the Interactive menu:



DC mode



AC mode



Continuous transient mode




Single shot transient mode, the time is the same as set at Analysis Transient




Digital mode





VHDL mode


 With this listbox you can select the analysis and the Interactive Mode Options dialog.

 (*Analysis.Optimization Target*) Select Optimization Target to set up Optimization mode or to change settings.

 (*Analysis.Control Object*) Select Control Object for Parameter Stepping or Optimization.

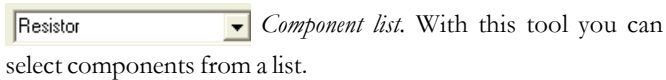
 (*Analysis.Faults enabled*) If this button is pressed it enables component Faults, set by Fault property of components. You can set component faults with the Property Editor by double-clicking on components.

 (*View.3D view/2D view*) Hotkey F6. 2D/3D view. If this button is pressed, TINA's Schematic Editor displays circuit components as 3D pictures of the real component assigned to the schematic symbol. This is a simple but useful check before starting the PCB design.

 (*Tools.PCB Design*) Invokes the dialog which initiates the PCB design module of TINA, if available.



Component tool. Invokes a component searching and placement tool. This tool helps you find by name any component in the TINA catalog. The search string that you enter will be found wherever it occurs at the beginning, at the end, or anywhere within the component's name. This tool is useful when you don't know where a particular component is located, or if you want a list of all the components that match particular search criteria. A component found in a search can be placed immediately into the schematic by selecting it and pressing the Insert button of this tool.




- 5 The Component bar:** Components are arranged in groups, named by the tabs on the Component bar. Once you have selected a group, the available component symbols appear above the tabs. When you click on the desired component (and release the button), the cursor changes to that component symbol and you can move it anywhere in the drawing area. You can also rotate the component by pressing the + or - keys (on your computer's numeric keypad) or mirror it by pressing the asterisk (*) key (also on your computer's numeric keypad). Once you have selected the part's position and orientation, press the left button of the mouse to lock the symbol in place.
- 6 Find component tool:** This tool helps you find by name any component in the TINA catalog. The search string that you enter will be found wherever it occurs—at the beginning, at the end, or anywhere within the component's name.

This tool is useful when you don't know where a particular component is located, or if you want a list of all the components that match particular search criteria. A component found in a search can be placed immediately into the schematic by selecting it and pressing the Insert button of this tool.
- 7 Open files tab:** You can have several different circuit files or different parts (macros) of a circuit open in the schematic editor at the same time. Clicking on a tab brings that circuit page up in the editor.

- ⑧ **The TINA Task bar:** *TINA's* Task bar appears at the bottom of the screen and provides speed buttons for the various tools or T&M instruments currently in use. Each tool or instrument operates in its own window and can be made active by clicking on its speed button (icon of the tool). Once the cursor is over the speed button, a brief hint appears. Note that the first button (furthest to the left), the Lock schematic button, has a special function. When the Lock schematic button is pressed, the schematic window is locked in place as a background behind other windows, so that it can never cover a diagram or virtual instruments. When the schematic window is not locked and it is currently selected, you will always see the entire schematic window with any other windows hidden behind.
- ⑨ **The Help line:** The Help line, at the bottom of the screen, provides short explanations of items pointed to by the cursor.

4.4 Placing the Circuit Components

Components are selected from the Component bar and their symbols are moved by the mouse to the required position. When you click the left mouse button, the program locks the pins of the component symbol to the nearest grid dots.

Components can be positioned vertically or horizontally and rotated by 90-degree steps in a clockwise direction by pressing the [+] or [Ctrl-R] keys, or in a counterclockwise direction with the [-] or [Ctrl-L] key. In addition, some components (like transistors) can also be mirrored around their vertical axis by using the [*/] key on the numeric keypad. You can also use the  buttons or the popup menu (right mouse button) to position components.

After a component symbol has been selected and positioned, you may double click on it to enable a dialog window where you can enter parameter values and a label. When entering numeric values, abbreviations of integral powers from 10^{-12} to 10^{12} can be used. For example, 1k is understood as 1,000.

NOTE:

Press the Help button on the component dialog box to invoke TINA's HTML-based help resource. You will find the parameters and mathematical model of the selected component. You can also access Component help from the help menu.

TINA will automatically assign a label for each component you place on the schematic. It will also display the numerical value of the main component parameter (for example: R4 10k). Note that the value is shown only if the Values option of the View menu is checked. For files from the older versions of *TINA*, the Values option is turned off by default. The first part of the label, e.g., R4, is required for symbolic analysis modes. You can also display the units of the capacitors and inductors (for example: C1 3nF) if both the Values and the Units options of the View menu are checked.

4.4.1 Wire

A wire establishes a simple short (zero ohm connection) between two component pins.

To place a wire, move the cursor to the component terminal point where you want to begin. The cursor will change into a drawing pen. Depending on View.Options settings you can draw a wire in two different ways:

- 1) Select the starting point of the wire with a left mouse click, then move the pen with the mouse while TINA draws the wire along the path. While drawing the wire, you can move in any direction and the wire follows. At the end point of the wire, click the left button of the mouse again. This drawing mode is the default setting in version 6 of TINA.
- 2) Hold down the left mouse button while positioning the pen; release it at the end point.

The actual wire-drawing mode depends on the editor options settings in View.Options. TINA by default uses the first method, but the program will remember that you have changed wire-drawing mode.

While drawing a wire, you can delete previous sections by moving backwards on the same track. By pressing the Ctrl key while drawing you can move the last horizontal or vertical section.

You can easily modify existing wires by selecting and dragging sections or edges.

For short wire sections, you may need to hold down the shift key while drawing.

You can also invoke the Wire-drawing tool by the Insert | Wire command (hotkey: [Space]). You can start drawing the wire at any place by clicking the left button of the mouse. When you have completed wiring, use the popup menu, press the right mouse button, or press the Esc key to terminate the wiring mode.

Be sure not to leave any component nodes unconnected. If there are unconnected components or terminals, TINA's Electric Rule Check tool (ERC) will issue a warning (unless you have disabled it).

Wire segments made by the Wire tool are always vertical or horizontal. However, you can add angled wire segments using the components made for bridges, Y and D circuits under the Special component toolbar.

4.4.2 Input and Output

Certain types of analysis (DC Transfer characteristic, Bode diagram, Nyquist diagram, Group delay, Transfer function) cannot be executed until both input and output have been selected. These establish where the excitation is applied and where the circuit response is taken. The output(s) chosen also determine which curve(s) will be displayed in the chosen analysis mode. Sources and generators can be configured as inputs, while meters can be configured as outputs. However, meters can also serve to determine the location of the input quantity that will be used when computing AC Transfer curves and functions. For even greater flexibility, inputs or outputs can be established at nearly any location by using the **Insert | Input** and **Insert | Output** commands. Note that you can define the input parameter for parameter sweeping only through the **Insert | Input** command.

To insert an input or output, move the cursor over the icon on the Tool bar, click, release, and drag the input or output to the schematic node that it should define. Then click the mouse button and hold it, while you drag the input or output in the direction of the other node, (usually a ground reference node). When the input or output has been extended to the second node, release the mouse button.

Since an input reference can be established in so many ways, it is important to remember that only one input at a time can be defined within a circuit.

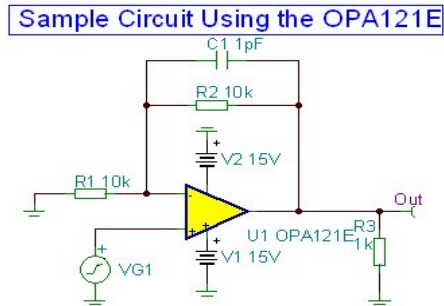
Similarly, in some of *TINA's* analysis methods (e.g., Symbolic Analysis) only one output can be defined within a circuit.

4.5 Exercises

These exercises will help you build upon and integrate what you've learned from the manual so far.

4.5.1 Creating and editing a Circuit Schematic

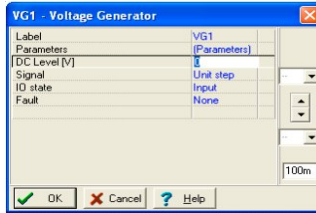
Create the circuit diagram using an OPA121E operational amplifier from TI as shown in the following figure. Note that you can skip creating this circuit; simply load this pre-drawn circuit from the Examples folder as “OPA121E Test Circuit.TSC” and go to the section below to analyze it. You can also create the circuit yourself—see the next section.



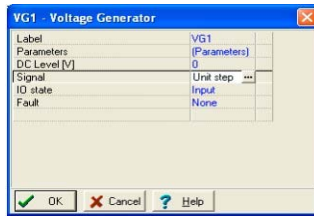
If you just opened TINA and wish to create your own circuit, you can start adding components right away. The circuit file name in the top line is set to Nname by default, indicating that a new circuit file Nname.TSC is being edited. If you already have a circuit loaded in the editor you can start a new one with the File | New command. You can switch between multiple circuits by clicking the Tabs at the bottom of the screen.


Now start adding components. Left click on the voltage generator icon then release the mouse button. The cursor will change into the generator symbol. Position it using the mouse (or by pressing the [+]/[Ctrl-R] or [-]/[Ctrl-L] key for rotation) or the [*] key for mirroring) somewhere in the middle of the screen, then press the left mouse button to drop the component into the schematic. We still need to set the prop-

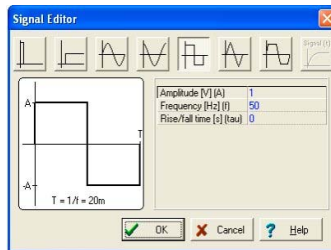
erties of this generator. Double-click on the generator and the following dialog box will appear:



Leave the DC level, Signal, and the IO state parameters unchanged. Note that by accepting Input for the IO state parameter you have selected the output of this generator to be the input for this analysis (a Bode diagram in this example). Click on the Signal menu line. The dialog box will change as shown below:

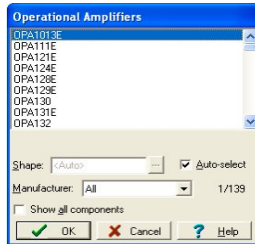


Press the  button. A new dialog box with the graphic icons of available voltage generator signals will appear. When you select one of them (in our case, click on the Square Wave button), the associated curve comes up with some default parameters. In the case of the Square Wave signal, these are:




Change the Amplitude to 500m (this represents 500mV peak), the frequency to 100k (100kHz), and the Rise/Fall time to 1p (1ps). Click on OK and return to the previous dialog box and click on OK again. The program will automatically place the label (VG1) near the component and you will be able to position and place the component and the label together. If the default label position is not satisfactory, you'll be able to drag the label to the desired position later on.

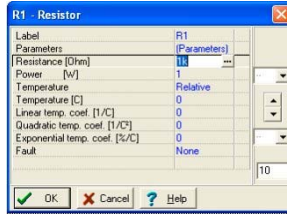
Now click on the Spice Macros tab and press the leftmost Operational Amplifiers button. The following dialog box will appear:



Click on the 3rd line (OPA121E) and press the OK button. (Alternatively you can double-click on that line). The schematic symbol of this opamp will appear and be attached to the cursor. By moving the mouse, position the opamp as shown on the schematic at the beginning of the section and then press the left mouse button to place the op amp into your schematic.

Note that other types of ICs are available under the buttons next to the Operational Amplifiers: Difference Amplifiers, Fully-Differential Amplifiers, Comparators, Voltage Regulators, Buffers, Current Shunt Monitors, and Other Components). You can bring all of these various components into the dialog box for any of the buttons if you set the Show All Components checkbox. In addition to selecting an IC on the list you can also find it by clicking on any item on the list and the typing in the name of the IC. Another way of finding an IC is the  Find Component tool at the top-right corner of the screen.

Now click on the Basic tab on the Component bar and click the Resistor icon. The resistor symbol will be attached to the cursor. Move the resistor to the position of the R1 resistor on the sample schematic diagram at the beginning of this section and press the left mouse button to place this resistor into the schematic. Double-click on the resistor and the following dialog box will appear.





Change the value in the Resistance field to 10k, and press OK.

Note that you can set a component value before placement while you are moving it. To do that press the right button of the mouse and select Properties on the popup menu. The above dialog will appear and you can set the properties of that component. After pressing OK you can return to the component placement.

Now let's place R2 at the top of the circuit. Click on the resistor symbol on the component toolbar, move and place the resistor. When you drop the resistor you will see that its value is already 10k since the program remembers the previous value.

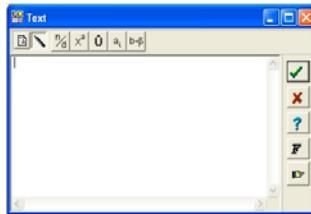
Now let's place R3 which needs to be turned by 90 degrees. Click on the resistor symbol on the toolbar and turn the component by 90° by


clicking the  or  buttons or pushing the Ctrl L or Ctrl R keys. (+ and – on the numeric keyboard have the same effect). Place the component on the right side of the screen and set its value to 1k with the technique described above.


Continue circuit entry with the Capacitor, Battery, and Ground components as indicated in the figure above. Set the parameters to C = 1 p, V1=15, and V2=15. Place a Voltage Pin Out (chosen from the Meters component group) at the right side of the new schematic. Pay attention to battery polarities and rotate the symbols if necessary.

Note that even though all the computed voltages, currents and signals are available after running an analysis (see below in this chapter and also in the Post-processing analysis results section), you still need to define at least one output. We have placed the parts into the schematic but they are still unconnected. To connect devices, move the cursor over an appropriate pin node until a small drawing pen icon appears. When this pen appears, click the left button of the mouse, draw the wire, and left click again at its endpoint.

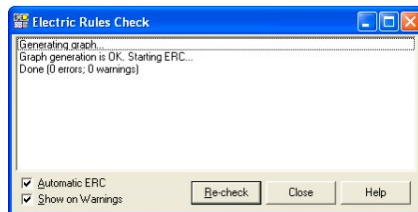
Finally, add a title to the schematic. Click the  icon on the toolbar and the text editor will appear.



Type in: “Sample Circuit Using the OPA121E”. Click on the  icon and set the font size 12. The editor also lets you choose another font style, color, etc.

Now click on the  icon, and then position and drop the text into the schematic.

Let’s check the circuit we have just built and run ERC from the Analysis menu. If everything is OK the following dialog will appear:



If there is problem with the circuit, a list of warning or error messages will appear in the dialog box. If you click on an error message, the related part or wire will be highlighted in the circuit diagram.

Before going on to the next step, save your circuit using the File | Save As command. Name the circuit “OPA121.TSC” (the .TSC extension is added automatically). If you wish, you can still edit the circuit in many ways:

- Add new components.
- Delete, copy, or move selected objects using the Edit | Cut, Copy, Paste, and Delete commands.
- Move, rotate, or mirror groups of components. Select the components one by one, holding down the shift key as you click on them.

You may also use a window selection to identify the group by dragging a box around the desired components. When you've selected the last component, release the left mouse button, then move the cursor over one of the selected components. Next press and hold down the left mouse button, and drag the selected parts with the mouse. While dragging, you can use the [+]/[Ctrl-L]/[Ctrl-R], [-] and [*] keys to rotate and mirror components.

- Move any component label separately by clicking on it with the mouse button and dragging it to the desired location before dropping it by releasing the mouse button.
- Modify component parameter value(s) and labels of the component by double-clicking on it.

Of course, if you want to keep these changes you must save the circuit again.

4.6 Analyses

TINA has a variety of analysis modes and options:

The analysis method is analog when a circuit contains only analog components; then the components are modeled with their analog models.

The analysis method is mixed when a circuit contains both analog and digital components. *TINA* will analyze the analog parts in analog, the digital parts in digital, and will automatically create the interfaces among the components. This ensures synchronization and fast convergence.

The analysis method is digital when a circuit contains only digital components; then the components are modeled with their fast digital models.

Analysis Options

This dialog, with which you can set analysis parameters, is on the Analysis menu. Examine the screenshot that follows to see the parameters that you can adjust.

In this manual we will present only the most important options, those which you might want to change. The complete explanation of all the options is included in the on-line help, which you can display pressing the Help button on the dialog.

Note: If you change any settings in this dialog and close the dialog by pressing the OK button, the program will remember the changed settings.

Performance: TINA v9 and later support multi-core processors and run analysis in parallel threads. This results in a significant speed increase. By default the number of threads is equal to the number of cores in your CPU. However you can control this with the Number of threads parameter.

Number of threads: This parameter is set to Max by default and results in one thread per core. You can, however, set it to any number between 1 and Max, or to Dynamic. If you have a quad (4) core machine, it would be good to set this parameter to 3, to leave some computing capacity for other programs and processes. You can also use the Dynamic parameter which controls the number of threads automatically, depending on other processes running on your computer.

Matrix compilation: By default this parameter is enabled and results in compilation of an extremely fast code for some matrix operations. The only reasons to disable this parameter are to perform speed comparisons or to debug software. For maximum performance, it should normally be enabled.

Enable run-time statistics: If this option is set, Tina displays the simulation time of the transient analysis in the status bar and creates a file which contains detailed information on the last transient run. You can load this file by selecting the View.Transient statistics menu item.

Enable instant diagram drawing: If this option is selected TINA will draw the diagram during transient analysis, refreshing the diagram every 1-2 seconds. This is very useful for observing the progress of long calculations.

Save all analysis results: Check this box if you want to save the result of all nodal voltages, resistor, capacitor and inductor voltages and currents, to make the later post processing more convenient. However this option may slow down the analysis by 30-50%.

Disable warning for large site analysis results If you run a transient analysis, a warning will appear if the number of points is greater than 1,000,000 (1M). Select this checkbox if you want to disable this warning.

4.6.1 Analysing the circuit

TINA has a variety of analysis modes and options, making it a very versatile simulation tool.

The analysis method is analog when a circuit contains only analog components; in this case the components are modeled with their analog models.

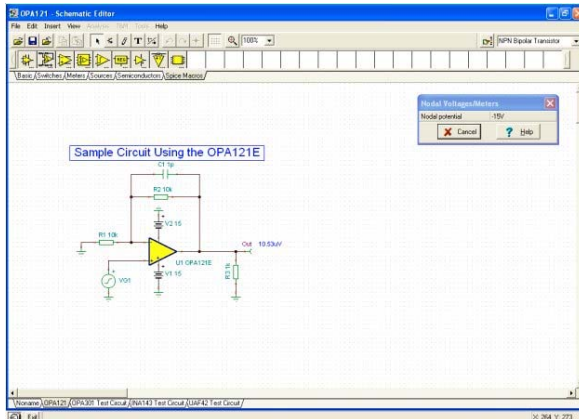
The analysis method is mixed when a circuit contains both analog and digital components; in this case the components are modeled with their analog models.

The analysis method is a digital mode when a circuit contains only digital components; in this case the components are modeled with their fast digital models. (TINA-TI does not include this digital mode.)

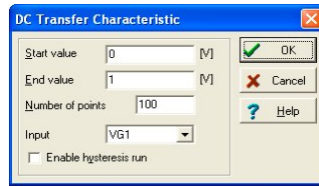
Now let's first check the operating point of our circuit. Select **DC Analysis** from the Analysis menu and select **Calculate Nodal Voltages**.

At the Out pin, the voltage of this point, 10.53uV, will appear—which is practically zero, as expected. In addition a probe shape cursor will appear allowing you to test the voltages of all nodes. In the figure below, the probe was placed on the op amp's negative supply pin.

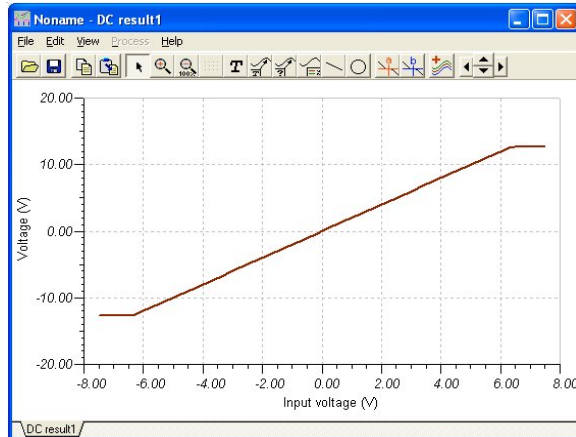
Let's do another DC analysis by calculating the DC transfer characteristic of this circuit.



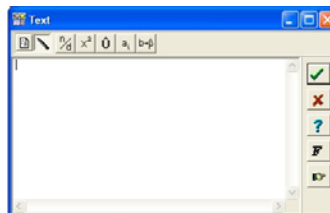
Select the DC Analysis | DC Transfer Characteristic... from the Analysis menu. The following dialog box will appear:




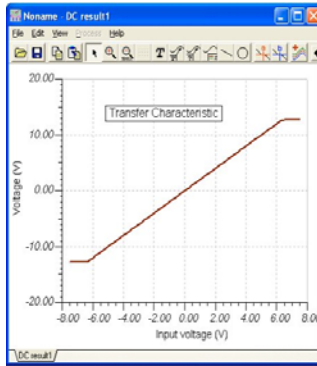
Set the Start value to -7.5 , the End value to 7.5 , and then press OK. After a short running time, a Diagram Window will appear as shown below. This displays the circuit's transfer curve—output voltage vs. input voltage.





As in the Schematic Editor, you can add a title or any text to the diagram by clicking the **T** icon on the Diagram Window toolbar. The text Editor will appear:



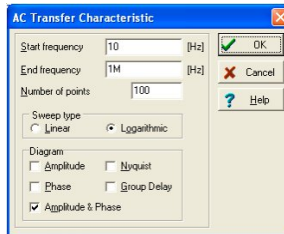
Type “Transfer Characteristic” and click the  icon. Now you can position and drop the text into the diagram.



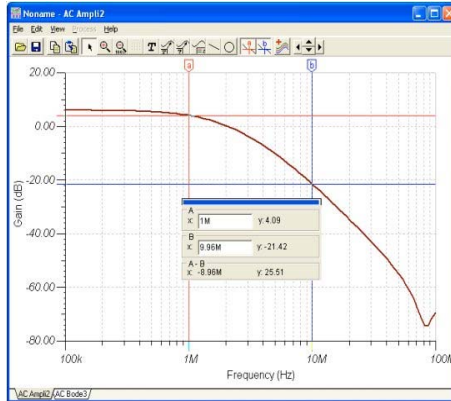
If you have already placed text into a Diagram, you can always bring it back to the Text editor and edit it further by double-clicking on it. You can change the font size with the  icon and there are also plenty of graphic features under the  icon, described later in this section.



Now try executing an AC and Transient Analyses on your circuit.




First select AC Analysis | AC Transfer Characteristic... from the main menu. The following dialog box appears:





The circuit’s Amplitude & Phase will be calculated by default. In addition, set the Amplitude checkbox on. Modify the Start frequency to 10k and 100M respectively and press OK. A progress bar will appear while the program is calculating. After the calculations are finished, the Bode amplitude characteristic will appear in the Diagram Window. You can easily switch to the Amplitude & Phase diagram by using the Tabs at the bottom on the Diagram Window.



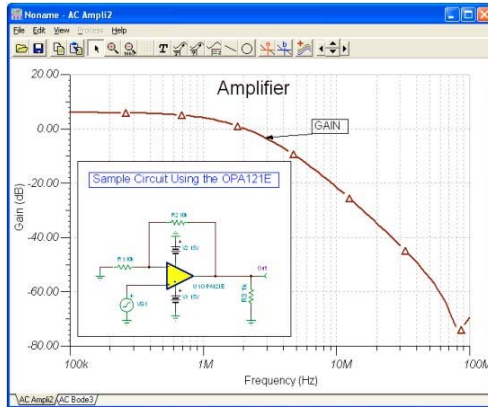
You can read exact input/output values by enabling one or more of the cursors, pressing the  and/or  and clicking on the curve where you want to run the cursor(s). If you run two cursors you can see the difference of the frequencies and values measured by the two cursors as shown in the figure below.

Using the graphic facilities of TINA, you can add more useful information to your diagram if desired. As an example, let's add markers, a special annotation, and the circuit schematic itself to the diagram. To add markers to a curve move the cursor over the curve, find a position when the cursor changes into a + shape and click on the curve at this position. When the curve becomes selected, it turns red. Now you can either double-click on it or press the right button of the mouse and select Properties on the popup menu. A dialog box will appear and you can set the curve's parameters: Color, Line width, Marker. To select Marker Type: Square and click on OK. To add some text, click on the  text icon. The Text editor will appear. Enter "Amplifier" and by clicking on the  button change the font size to 20. After returning to the text editor press the  button and place the text at the top of the Diagram Window.

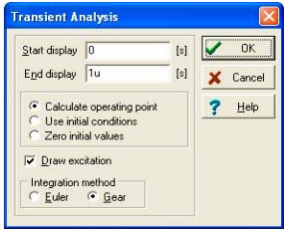
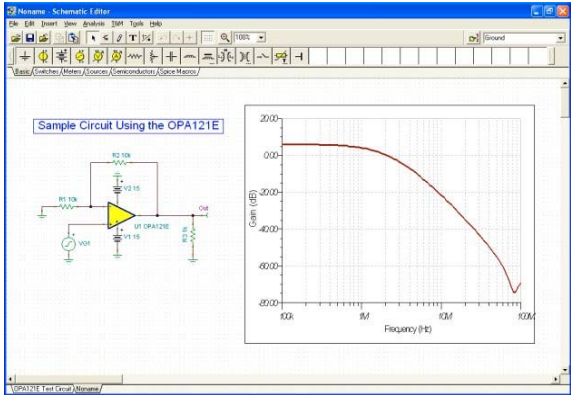
When the Text editor appears, type in "OPA121E". Note that using the  font icon of the editor you can select any font, style, size and color. Click on OK and place the text in the neighborhood of the curve. Now click on the  pointer icon, then on the text, and finally on the curve. Note that the cursor turns into a + when you are at the right position.

You have just entered a line and arrow that will always point from the text to the curve, even if you drag the text into another position or make other changes.

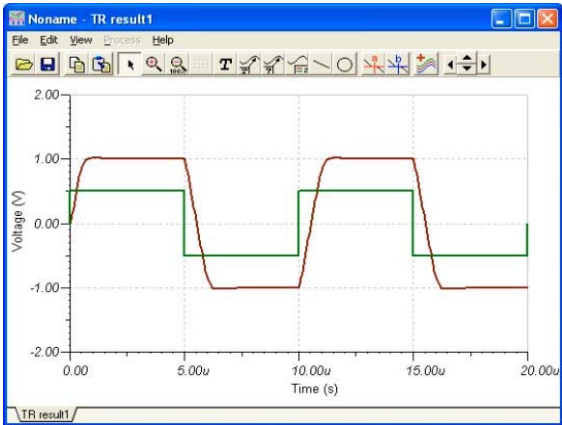
Now place the schematic itself on your diagram. Click the schematic editor Window and select Edit | Select All. Copy this selection into the clipboard by selecting Edit | Copy or clicking on the Copy icon or using the Ctrl C hotkey. Click on the Diagram Window and use Edit (Paste, click on the Paste icon, or use the Ctrl V hotkey). The frame of the circuit diagram will appear. Position and drop it at the left corner of your diagram. Now you can still modify this picture by dragging or double-clicking on it and changing its size, frame or background.





In a similar way you can do just the opposite and place the Bode diagram into the Schematic diagram. To do that, click on the top caption part (or any empty place) on the Diagram window and press the Ctrl C keys or use the Edit | Copy command **in the Diagram window menu** to copy the diagram to the clipboard. Now select the Schematic diagram window by clicking at an empty place or the top caption of the window, and press the Ctrl V keys or the Edit | Paste command of the **Schematic diagram window**. Now you can move and place the diagram on the schematic and then resize it by its corner handles or numerically by double-clicking on the picture.

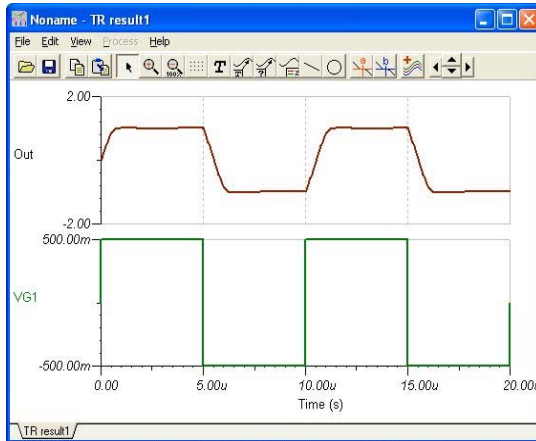


Now perform a Transient Analysis. First, make sure your cursor is the selection arrow, then double-click on the voltage generator and change the waveform to the default unit step. After selecting Analysis | Transient Analysis, the following dialog box appears:



Change the End Display parameter to 20 u then press OK. In a separate window the transient response waveforms will appear. Exact input/output data points can be read by enabling the  and/or  graphic cursors.

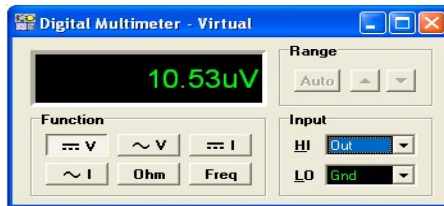
You may want to see these two different curves in separate coordinate systems. To do that, select the View Separate curves command in the Diagram window. As a result, each curve will be displayed in a separate diagram as shown in the next figure.






4.6.2 Testing Your Circuit with Virtual and Real Time Instruments

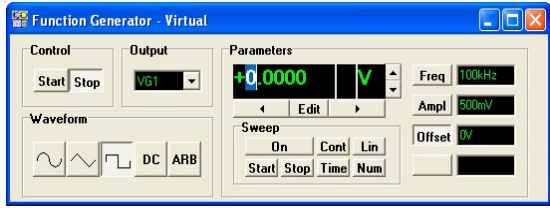
TINA lets you test and tune your circuit not only with the generators and analysis windows that you have used so far, but also with virtual and real-time measurements. Using the T&M menu you can place realistic resizable virtual instruments on screen which will automatically replace the generator and analysis windows. You can control the settings of these instruments and immediately see the results, just as you would in a real lab. TINA normally simulates measurements with its analysis engine, but if you have TINA's TINALab II optional Multi-function PC Instrument, you can simply switch to the Real Measurement mode. Now you can work with the same on-screen instruments and settings and you will be making real measurements on a real circuit. Note that even though circuit simulation driven virtual instruments are very convenient to use, this is true for relatively small circuits only. When the circuits are large and simulation of a change in the circuit takes a long time, it is easier to use the classic analysis methods shown in 4.6 section of this manual.

To begin working with virtual measurements, load the circuit, select the T&M menu and place a Function Generator and a Multimeter on your screen.



Click on the Multimeter's  button. The Multimeter will show only about 10u volts. Now click on the Offset button of Function Generator and move focus to the first 0 after the + sign by clicking on the  button or by using the Left and Right cursor arrows. Now you can change the first digit of the offset voltage by clicking on

Up and Down cursor arrows or the  arrows on the Function Generator and see the result on the Multimeter instantly.



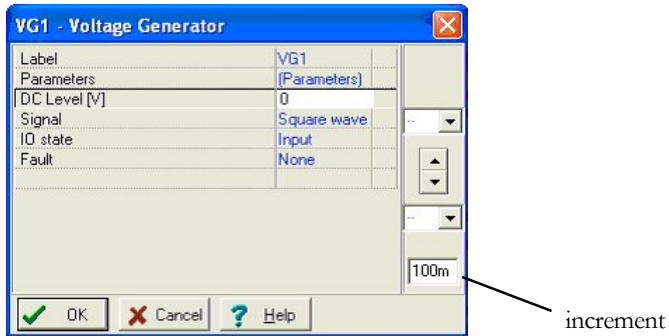
For example if you set 1V Offset on the Function Generator, the Multimeter will show 2V. If you raise the Offset voltage over 6V you will see that the output voltage will go into saturation at around 12.7V as we have already seen at the DC Transfer Characteristics.

You can also set the Parameters of the Function Generator by pressing the **Edit** button on the Function Generator, typing in the required number, and pressing Enter.


You can also see the effect of changing component parameters through their Property windows.


To demonstrate make sure that the **V** of the Multimeter is On and then double-click on the Voltage Generator symbol on the schematic diagram.

The property window of the generator will appear.



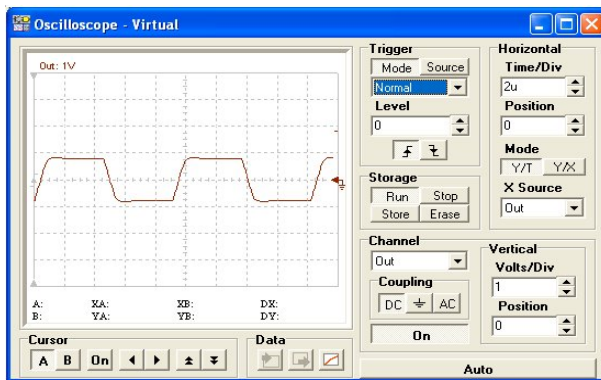
As the focus is already on the DC Level parameter, which is the same as the Offset on the Function Generator you can change it immediately



with the  buttons on the right side of the dialog box. The increment is shown at the bottom field on the right side. By default the increment is 100mV but, if desired, you can change the step size of the buttons by entering it into the edit box.


Click on the  buttons and see effect instantly on the Multimeter.

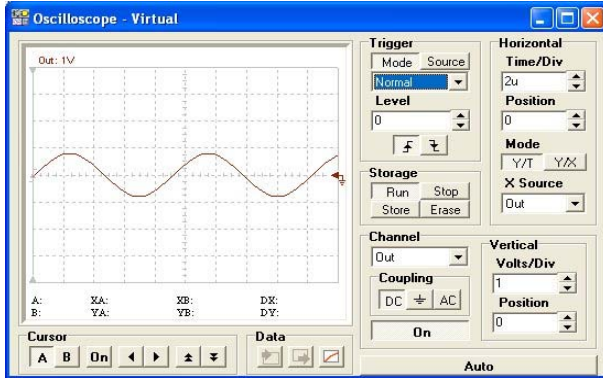
Set back the DC level to 0 and click on the OK button of the Voltage Generator dialog box.





Press Run in the Oscilloscope panel and a square wave will appear on the oscilloscope screen.



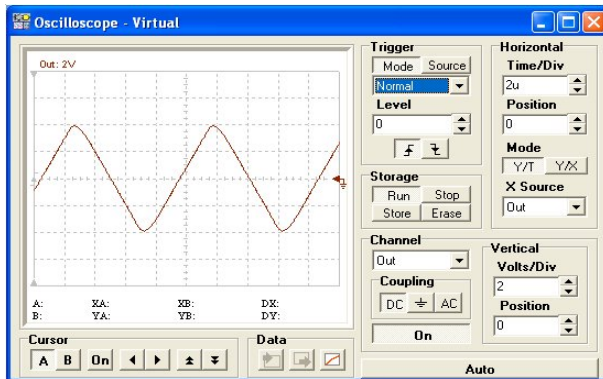
The wave is moving on the screen because by default the Oscilloscope is Auto (free running) mode. To synchronize the waveform select Normal mode with the list box below the Trigger field. The waveform will stop moving on the Oscilloscope. You can trigger the waveform with either a rising or falling edge by clicking on the  or the  buttons.

Now click on  button on the Function Generator. The waveform on the oscilloscope will change into a sine wave as shown in the figure below.

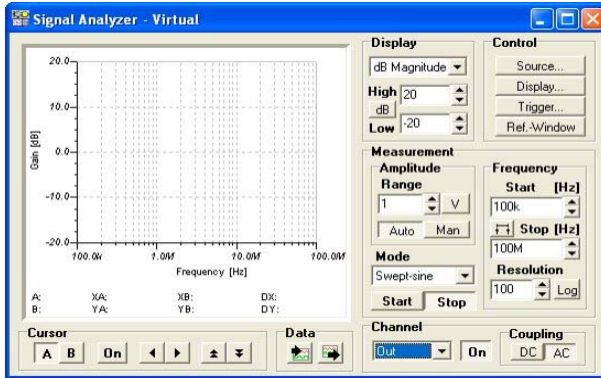


Now let change the amplitude of the sine wave provided by the Function Generator. Click on the **Ampl** button on the generator, Move the focus with the  and  buttons to the first digit of the default 500mV, then change the amplitude with the   buttons.

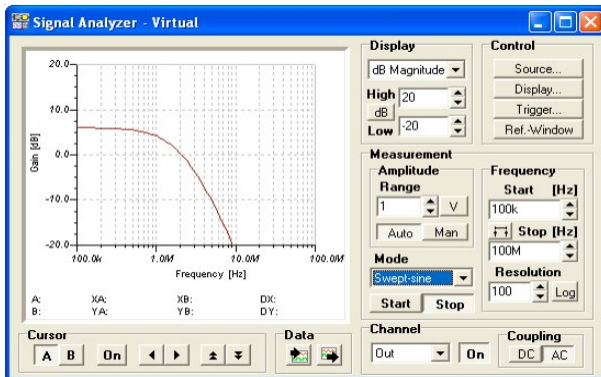
You can observe how the signal is distorted into a triangular waveform by slew rate limiting at around 2V of input voltage as shown below:




Now let's invoke the Signal Analyzer from the T&M menu to study the AC Transfer Characteristic of our circuit. The following window will appear.



A useful feature of virtual instruments of TINA is that they take over the settings of the Analysis menu. Therefore, in our case, press the Start button of the Signal Analyzer and the amplitude Bode diagram will appear as shown in the figure below:



You can export the result into a conventional diagram pressing the  button under the **Data** field at the bottom of the instrument.

You can make several other measurements (Phase, Two-panel Amplitude & Phase etc. by selecting the required mode from the list box under Display on the top of the instrument.

You can use the other controls on this instrument to set the Frequency Domain (**Start** and **Stop** under **Frequency**), vertical (Gain) axis settings (**High** and **Low** under **Display**) and more, which you can find in the help by clicking on the Oscilloscope and then pressing F1.

4.6.3 Analysis of SMPS circuits

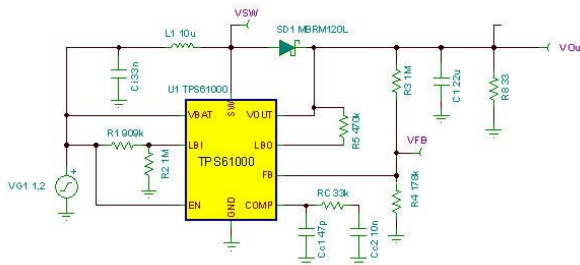
SMPS or Switching-Mode Power Supply circuits are an important part of modern electronics. The heavy transient analysis needed to simulate such a circuit may take a lot of time and computer storage. In order to support the analysis of such circuits TINA provides powerful tools and analysis modes. In this Chapter we will demonstrate these through examples.

Using the Steady State Solver

The most time consuming part of an analysis of an SMPS circuit is to reach its steady state, when the DC level of the output voltage does not change and the output waveform has only a small periodic ripple.

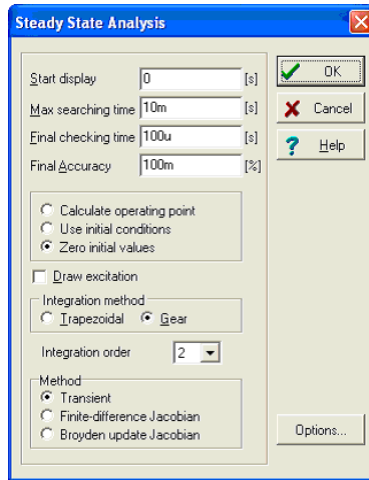
To find this state automatically, TINA has a Steady State Solver under the Analysis menu.

To demonstrate this tool, let's load the Startup Transient TPS61000.TSC Boost Converter circuit file from the EXAMPLES\SMPS\QS Manual Circuits folder.



In this Boost Converter circuit the VG1 input line voltage is 1,2V. This is converted by the SMPS circuit to 3,3V.

Select the Steady State Solver from the Analysis menu. The following dialog will appear:



The new parameters compared to the Transient Analysis dialog box areas follows -

Max searching time: The solver will try to find the steady state solution for max 20ms. After this, the analysis will discontinued whether or not a solution was found..

Final checking time: After the steady state search is done, there is a final check for the length specified here. You should have a stationary waveform for this time interval.

Final accuracy: the maximum DC level change allowed. When the change is below this, the analysis will end. Note that the 100m in the example above means 0.1%

Method: You can select the method used for finding the steady state:

Transient: The steady state is searched by using transient analysis.

Finite-Difference Jacobian, Broyden update Jacobian: The steady state is searched by the methods described in the paper, *Automated steady-state analysis of switching power converters* by Dragan Maksimovic.

Note that these last two methods may get to the steady state faster, but they do not go through the normal transient states, so the resulting waveform between the initial and final state does not reflect the real process (but rather the mathematical path of the methods to get there).

The new parameters compared to the Transient Analysis dialog box areas follows -

Max. searching time: The solver will try to find the steady state solution for max 20ms. After this, the analysis will discontinued whether or not a solution was found..

Final checking time: After the steady state search is done, there is a final check for the length specified here. You should have a stationary waveform for this time interval.

Final accuracy: the maximum DC level change allowed. When the change is below this, the analysis will end. Note that the 100m in the example above means 0.1%

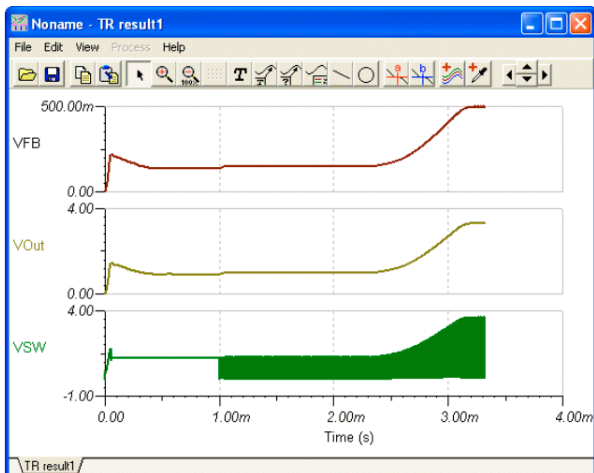
Method: You can select the method used for finding the steady state:

Transient: The steady state is searched by using transient analysis.

Finite-Difference Jacobian, Broyden update Jacobian: The steady state is searched by the methods described in the paper, *Automated steady-state analysis of switching power converters* by Dragan Maksimovic.

Note that these last two methods may get to the steady state faster, but they do not go through the normal transient states, so the resulting waveform between the initial and final state does not reflect the real process (but rather the mathematical path of the methods to get there).

Now lets run the Solver. After a few minutes of running (approximately 2 minutes on a 2GHz Pentium computer) we'll get the following resulting waveform:



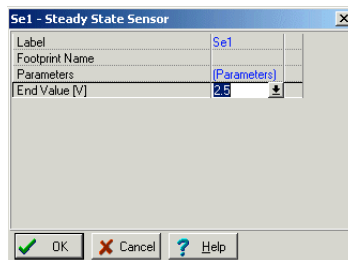
These waveforms show the detailed transient from switching on until reaching the steady output voltage. If you zoom out on the waveform, you can see that the period of the switching is around 500kHz and the time needed to arrive at the steady state is 4 milliseconds. Therefore, we need to calculate at least hundreds or sometimes thousands of periods if we want to see the whole transient waveform. This is why finding the steady state is such a time consuming process. The reason for this problem is the long start-up time of SMPS circuit compared with their switching frequency. The start-up time is basically determined by the filter capacitors on the output. The larger these capacitors are, the longer the start-up time.

NOTE:

In some cases you can accelerate the Steady State Search using the Finite-Difference Jacobian and Broyden update Jacobian methods, however they do not always converge and the intermediate waveforms provided by these methods do not reflect the real waveforms of the transient process.

Sensor

The purpose of this component is to set the target voltage(s) to be watched during the steady state search. You can add more than one sensor to a circuit. By adding sensors, you can significantly accelerate the steady state search. You can make the search even faster if you can give the final voltage at a certain node.



Using the “Max. no. of saved TR. points” parameter in the Analysis/Analysis parameters dialog, you can limit the maximum number of points placed in the Diagram. This is useful for large analyses to accelerate diagram drawing. By increasing this parameter, you can refine the diagrams but the drawing time will be greater.

The only electric parameter of the Sensor component is -

End Value: Voltage | Not Used

Trigger

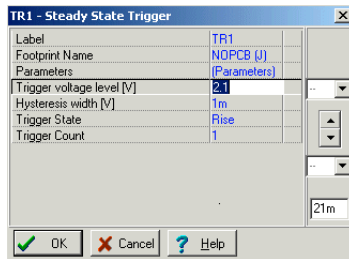
Use this to determine the starting and ending times of the switching period.

You can find this component on the Meters toolbar of TINA. You should connect it to the oscillator frequency control pin of the SMPS/PWM controller IC, but any node where the oscillator waveform of the IC is present will do.

NOTE:

If the SMPS IC in the circuit includes the oscillator, like 61000 in our example, this component is not needed.

If you double-click on the Trigger component you can set its parameters.



Trigger voltage level: the threshold voltage for the trigger event

Hysteresis width: hysteresis value for the trigger event. This value defines a region within which the trigger voltage is allowed to oscillate without generating a trigger event.

Trigger State: Rise/Fall The direction of the voltage change required for a trigger event

Trigger Count: you can take several periods for the waveform analysis.

This is useful in case of very slowly changing output signals. The only electric parameter of the Sensor component is -

End Value: Voltage | Not Used

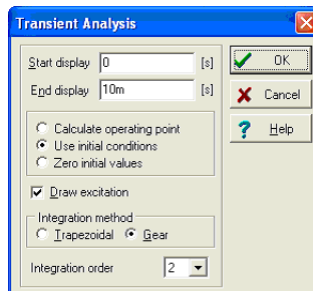
NOTE:

Using the "Max. no. of saved TR points" parameter in the Analysis/Analysis parameters dialog you can limit the maximum number of point placed in the Diagram. This is useful for large analyses to accelerate diagram drawing. By increasing this parameter you can refine the diagrams but the drawing time will be slower.

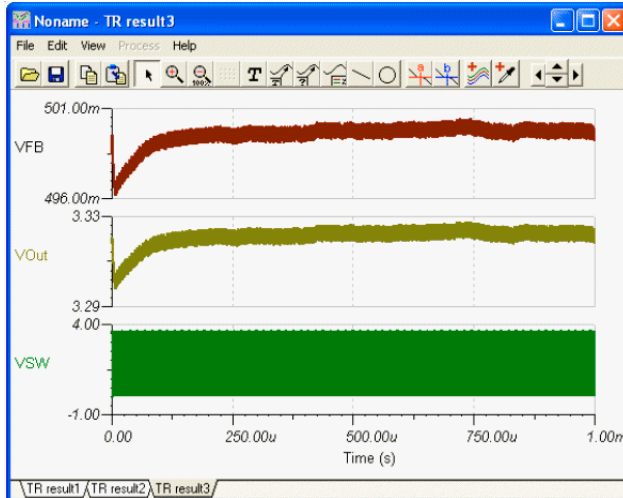
Accelerating SMPS simulation using initial values

As we mentioned in the previous section, the long analysis time needed for reaching the steady state of SMPS circuits is mostly used for charging the output filter and some capacitors. If we start the analysis using initial values for larger capacitors and inductors, the analysis time can be significantly reduced. In TINA, the Steady State Solver will automatically place initial values into the model of larger capacitors and inductors and so the following Transient Analysis can be run significantly faster (assuming that we do not make changes which will need significantly different initial values). For example, if you want to study the effect of changing the output filter capacitor, it will not significantly change the output voltage DC level. Therefore, starting the new analysis with an initial value calculated by the steady state solver for another output capacitor, will result in a much faster analysis. You can accelerate the analysis of input and load changes in the same way.

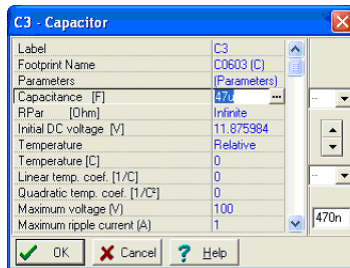
To demonstrate this feature, let's run a transient analysis for our example. Selecting the Transient command from the Analysis menu, the following dialog box appears.



Note that Use initial conditions is set in the dialog. Press OK to start Transient analysis. You should see that analysis runs very fast compared to the previous steady state analysis. The output waveform is shown in the picture below.



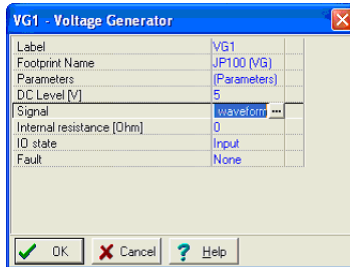
Why did the analysis run faster? Transient analysis was already preceded by the Steady State Analysis and the main capacitor's initial values (called Initial DC voltage in the Capacitor property box) was already set to the final DC voltage. For example, if you double-click on the C3 capacitor, you will see that the Initial DC voltage is already set to 3.31 V. Similarly, all the larger capacitor's initial values are set.



Once you have checked the initial transient, the steady state waveform, and SMPS circuit, the next thing you normally want to know is how it behaves when the input voltage or the load changes. This is realized by the **Input step** and the **Load step** analyses.

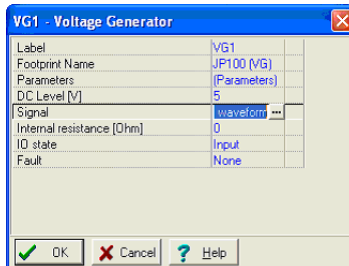
Input step analysis

One of the standard analyses for SMPS circuits is the calculation of the response to an input change to test the capability of the SMPS design to regulate the output with step changes in the input line. This can be accomplished by adding a pulse to the input voltage and checking the output and other voltages. Since the input change is relative to the steady state, we can start it from the steady state initial values calculated by TINA's steady state solver.




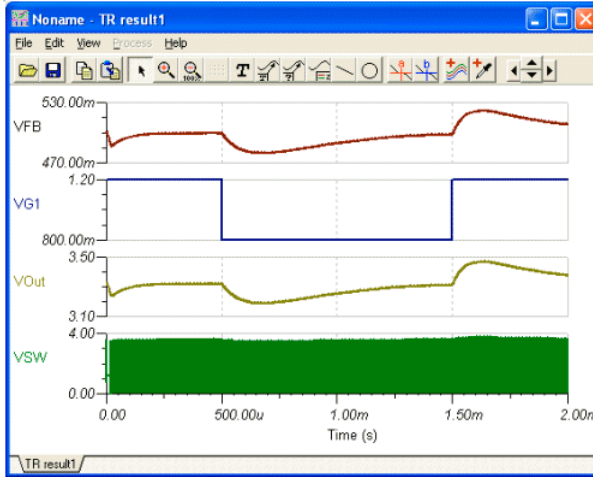
Load the Startup Transient TPS61000.TSC Boost Converter circuit file from the EXAMPLES\SMPS\QS Manual Circuits folder. The schematic design is the same as above.

To see the input step waveform, double-click on the VG1 voltagegenerator on the left. The following dialog box will appear:



According this, the input voltage is 1.2V. This is converted by the SMPS circuit to 3.3V.

Now click on the Signal line of the above dialog and then the  button. The following signal in the Signal Editor will appear:



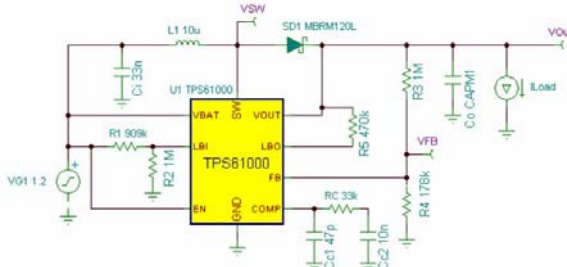
According to the waveforms, the input voltage will decrease from 1.2V to 0.8V for a $T_2=1\text{ms}$ time; and the starting edge (T_1) and the ending edge (T_3) of the pulse are 10us.

To see the response of the circuit, let's invoke and run the Transient analysis from the Analysis menu.

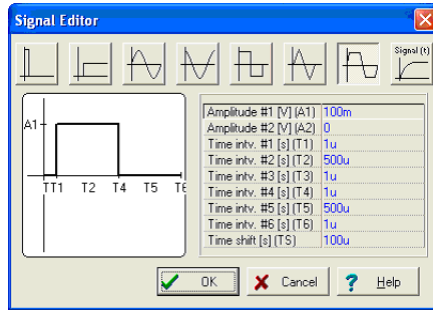
Load step analysis

Another standard analysis is to determine the SMPS response to a fast load change. Using simulation, the response to load changes is obtained by adding a current pulse to the load and analyzing the output and other voltages. Since the load change is relative to the steady state we can start it from the steady state initial values calculated by TINA's steady state solver.

Now load the example Load Step Transient TPS61000.TSC. The schematic design is the same as above, except for the ILoad Current generator on the output.



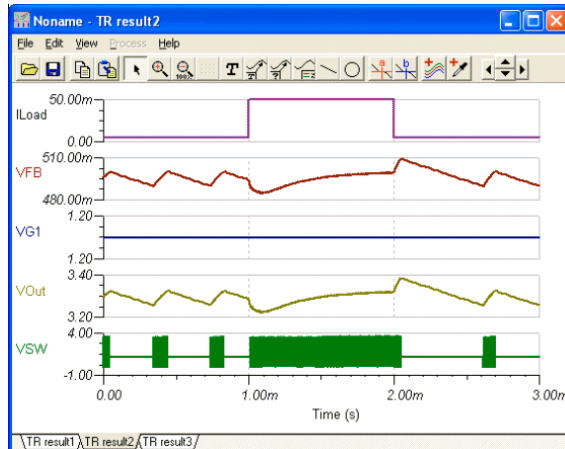
If you double-click on the ILoad generator and check its waveform, you will see that the DC part is 5mA while the pulse is 45mA in amplitude and 1ms in width.



Accordingly, the 5mA load current will rise to 50mA and then decrease to 5mA again.

Note that the output current is shown through the ILoad generator where the IO state parameter is set to Input.

Let's run Transient from the Analysis menu and see the result.

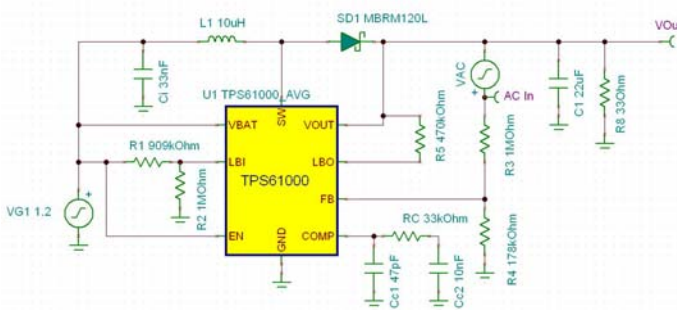


AC analysis

For AC analysis and stability analyses you can use the so called Average models provided in TINA. The average models represent a method, based on averaging the effects during the switching process. The

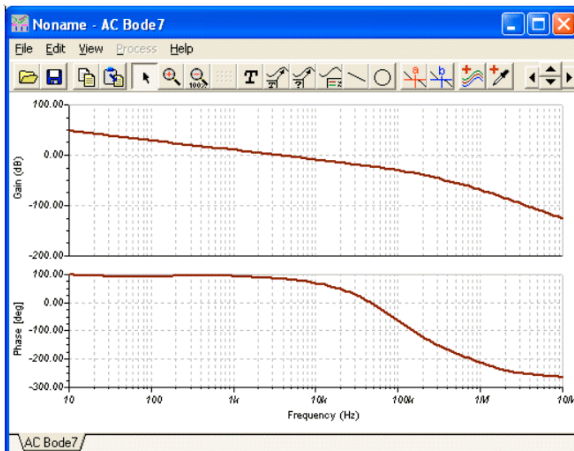
resulting equations are linear therefore the method is extremely fast in order to draw Bode and Nyquist plots needed for stability analysis. Note that for using the AC analysis function of TINA you need an average model, the transient models are not applicable and will give improper results.

To demonstrate this tool, let's load the Average model TPS61000.TSC circuit file from the EXAMPLES\SMPS\QS Manual Circuits folder.



Note the VAC generator which is providing signal for the AC analysis., and the AC In Voltage pin which is the Input of the AC analysis (its IO state parameter is set to Input).

Let's run AC Analysis/AC Transfer Characteristic... from the Analysis menu and see the result.



CREATING SUBCIRCUITS FROM SPICE MACROS AND MODELS

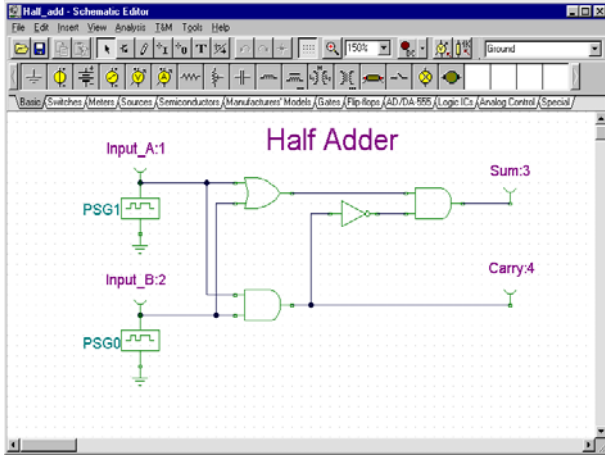
In *TINA-TI*, you can simplify a schematic by turning portions of it into a subcircuit. In addition, you can create new *TINA-TI* components from any Spice subcircuit, whether created by yourself, downloaded from the Internet, or obtained from a manufacturer's CD. In this chapter, we show through text and examples how easy it is to do this in *TINA-TI*.

5.1 Making a Macro from a schematic

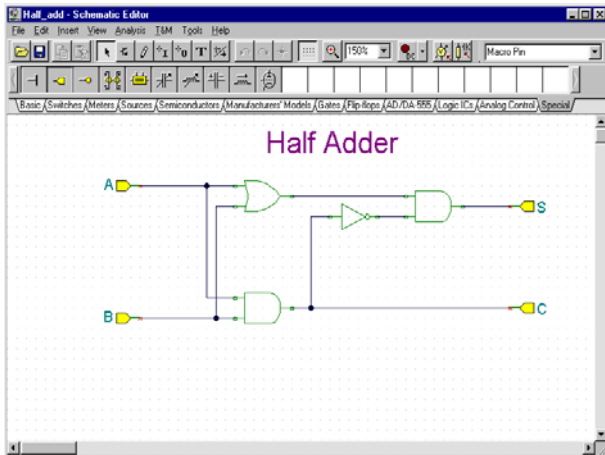
Using *TINA*'s macro facility, you can simplify schematics and hide clutter by turning portions of the schematic into a subcircuit. *TINA* automatically represents these subcircuits as a rectangular block on your schematic, but you can create any shape you like with *TINA*'s *Schematic Symbol Editor*.


You can convert any schematic diagram into a subcircuit - called a Macro in *TINA* - simply by adding the terminals and saving the new circuit in the special (*.tsm) format.

Now let's see how to create a macro in *TINA* through an example. Load the Half Adder example (*Half_add.tsv*) from the Examples folder of *TINA* and convert it into a macro.



Delete the old terminals and replace them with subcircuit terminals, called Macro Pins in *TINA*. You can find and select the Macro Pins under the Special component toolbar.

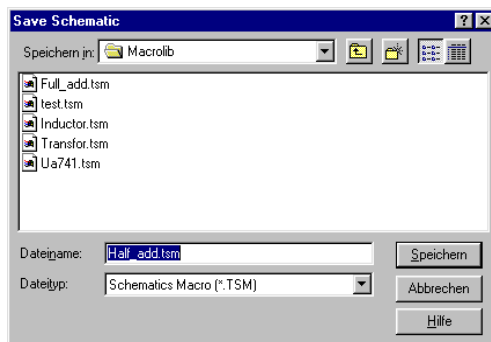


When you place Macro Pins, labels (such as Pin1, Pin2 etc.) are pre-filled in. Double click the Macro Pin and type in the new name in the label field. You can also drag the component with the mouse, or rotate it with the $[+]$ and $[-]$ keys or the  buttons.

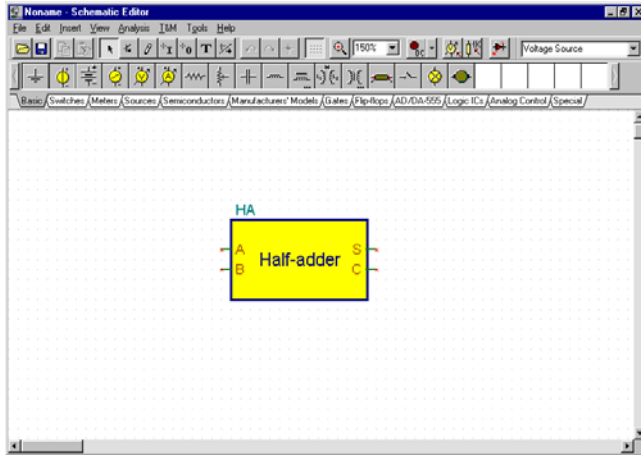
Next, create and save the new macro. Select the *New Macro Wizard* from the Tools menu. Set the Name to Half Adder (this will be displayed in the macro box that opens automatically), and set the Label to HA. This label will be displayed as the component label above the component. Note that you can leave this field blank if you don't want a component label.



When done, press OK. A Save dialog box will appear. Set Half Adder as the File name and press Save. Note that there is already a macro with a similar name (Half_add.tsm). This has the same content as the one we just created, and is included for reference. You can also use it in the next section.



Now let's see how to insert a macro into a schematic and use it. Clear the circuit with **File|New** or by restarting *TINA*. Select **Insert|Macro**, then our newly created *Half adder.tsm*, and click Open.



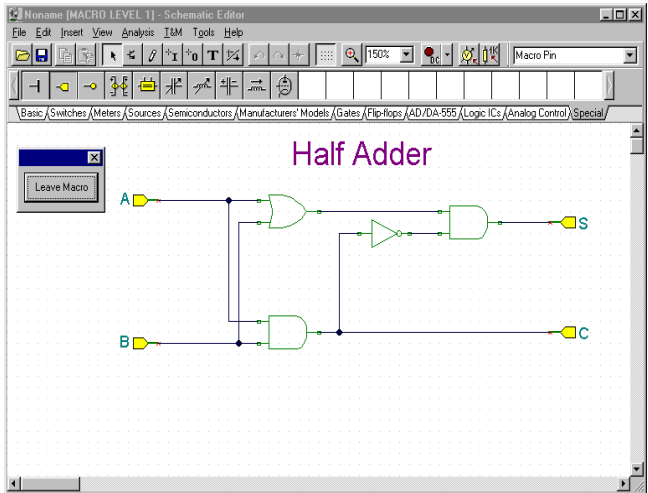
Our new macro will appear as a cursor. Move it to the center of the screen and click the left mouse button. The full symbol of the new macro will appear. Note that a rectangular schematic symbol has been automatically created, the macro name we specified is inside the rectangle, and the label name is above it.

Now you can add more components to the circuit, connecting them to the newly created macro, and start analysis as with any other circuit.

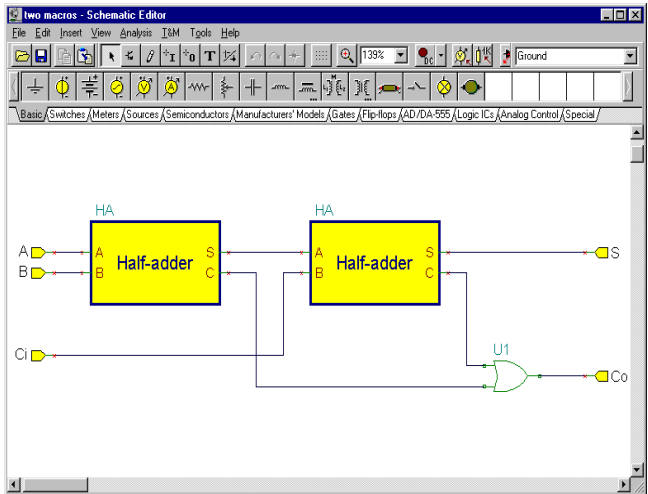
To check the content of the macro, double click on the symbol and *TINA* will display the model.

To return to the main circuit, click the Leave Macro button on the screen at the upper left, use the File/Leave Macro command, or the right click popup menu.


TINA allows a hierarchical macro structure; that is, macros can contain other macros inside, and so on. Let us use our half adder macro to create a full adder macro containing two half adder macros.

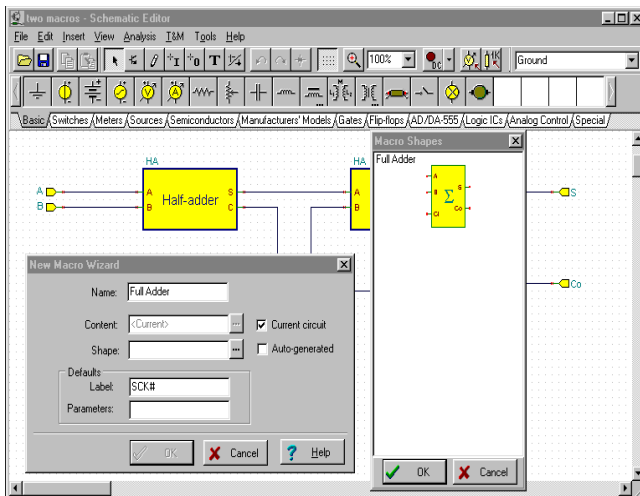


To do this, insert the newly created Half Adder twice into a new circuit and then add the additional components and wires as shown in the following picture.



Now create and save the new macro with the *New Macro Wizard* from the *Tools* menu. At this point let us note that although the automatic symbol creation is very convenient, you can also create your own schematic symbols with *TINA's* Schematic Symbol Editor and assign macros to them. Let us use this feature with an existing symbol. The creation of such a symbol will be described later in detail.

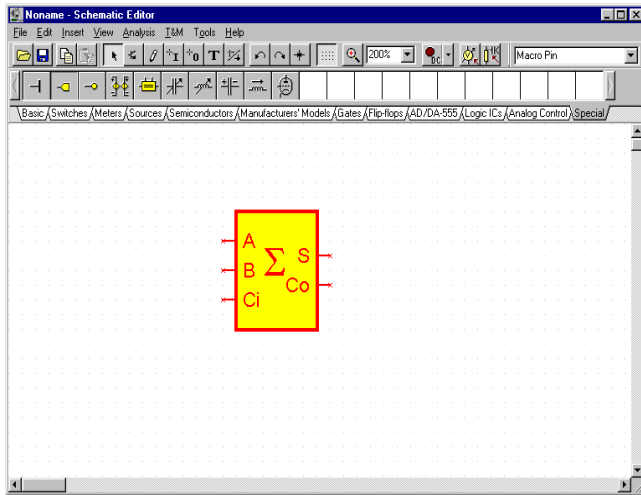
Set the Name to Full Adder and set the Macro Label to FA (this will be displayed as the component label above the shape). Uncheck the Auto generated checkbox and press the  button next to it. The list of available symbols will appear as shown below.



Note that in order to see the predefined symbols, the Macro Pin Label names must exactly match the names in the symbol. In our example, they must be (A, B, Ci, Co, S). If you do not see the symbol shown in the figure above, check the terminal names or try to recreate the symbol as shown later at “Making your own schematic symbols”.

Click the schematic symbol with the large summation sign and press OK. The name of the schematic symbol will appear in the shape field of the New Macro Wizard dialog box. Finally, click OK and save the macro under the name of Full adder.tsm.

To see our newly created full adder macro, select Macro from the Insert menu and then select Full adder.tsm and click Open.



Double click on the symbol and the schematic inside the circuit will appear, showing two of our previous half adder macros. You can then double click on either of the macros to see the schematic inside. Return to the Full Adder and the main circuit by clicking the Leave Macro button.

5.2 Making a Macro from a Spice subcircuit

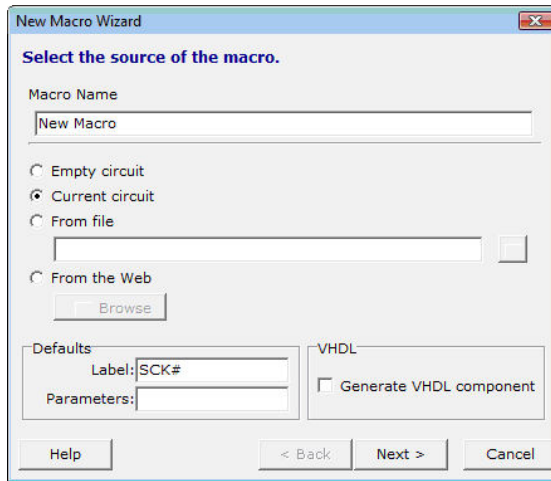
5.2.1 Creating Spice Macros in TINA


5.2.1.1 Creating macros from downloaded files

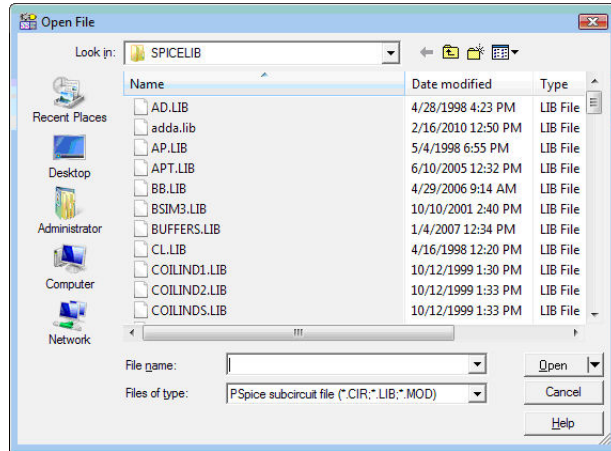
In *TINA*, you can create your own components from any Spice subcircuit that you have made or downloaded from the Internet. Note that there are already many Spice component models in the large and extensible manufacturers' model library provided with *TINA*. The extension of those libraries is described later.

Let's create a UA741 operational amplifier using a Spice subcircuit.

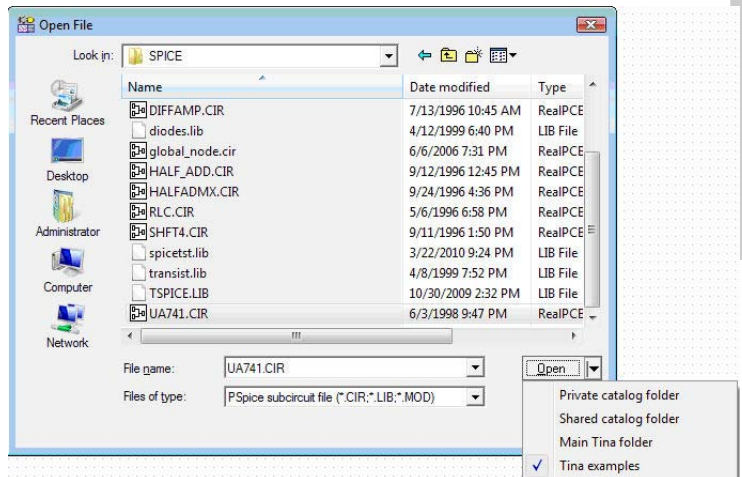
To do this, select the New Macro Wizard from the Tool menu. The following dialog box will appear:



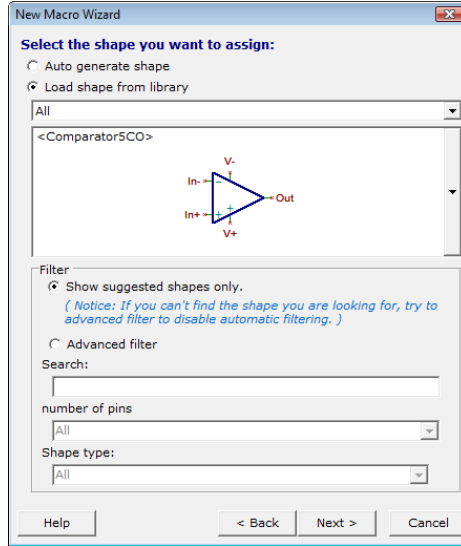
Change the settings from Current Circuit to From file and press the  button. An Open dialog box will appear.



Now lets navigate to the EXAMPLES\SPICE folder of TINA using the small arrow next to the Open button.

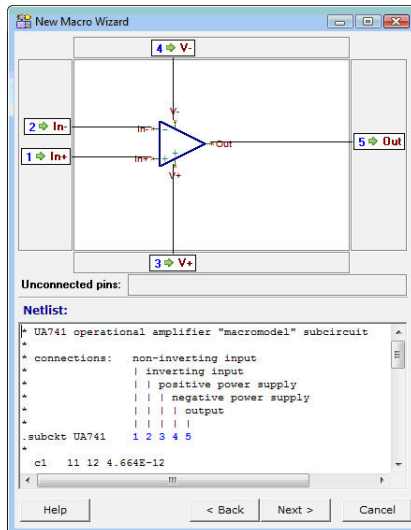


Select the UA741.CIR file and press the Open button. The New macro Wizard dialog will appear again with the path and name of the selected file. Now press the Next > button. The following dialog will appear:



The wizard has already automatically selected the appropriate symbol. If you want something different, you can view and select a symbol by pressing the long vertical button on the right.

Press the Next button. The following dialog will appear:



The dialog shows how the pin names on the graphic symbol are associated with the Spice node names in the macro. It also shows the text of the macro so you can check that the connections are properly made. If not, you can drag the macro node names to any terminal. However if all the pins are connected with a node number, then most likely the association is correct.

You've checked the connections; now press the Next button again. A Save dialog will appear and you can save the macro into the User macros area under **Documents\Designsoft\TINA_Industrial_install date_id_number\Macrolib** or to the TINA macros area, under Program Files. For easier file selection, use the small arrow next to the Save button. Note that under Vista and Windows 7, you cannot normally write into the TINA program area. After saving your macro, you will see a dialog with which you can test the macro or close the wizard altogether.

Now we will see how to insert the new (or any other) subcircuit into a schematic and check its contents. Select the Macro command of the Insert menu.

NOTE:

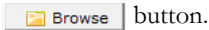
You may need to select the User Macro or TINA Macro area using the selection arrow next to the Open button, or navigate there using the selection list on the top of the Open dialog.

Click the UA741.TSM file and then press Open. Now the new macro will be attached to the cursor. Position it on the screen and drop it by clicking the left mouse button. Double click the symbol to see its content. The netlist editor will appear, showing the macro in detail. Note that you can modify this netlist, and the modified netlist will be saved with your circuit. However this will have no effect on the original macro; it remains unchanged.

5.2.1.2 Creating macros on-the-fly by browsing the web

A more convenient way to add new models to TINA is to browse manufacturer websites and add the Spice models of interest from the web site. Of course it also possible first to download the models and use the technique described in the previous section. Note that even if you prefer the latter you may find useful hints in the following section.

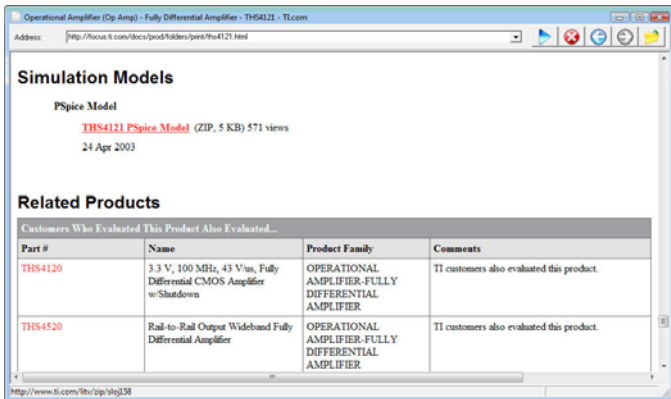
Now let's select the From the web option in the wizard and press the



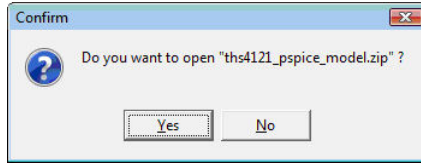
TINA's built-in internet browser will appear. Use this browser to find and select the THS4121 differential opamp from Texas Instruments. Enter www.ti.com and find the Spice macro on the TI web site using TI's Search option , or just enter following URL directly (use copy and paste if you like)

<http://focus.ti.com/docs/prod/folders/print/ths4121.html> (Note that the direct link above may change).

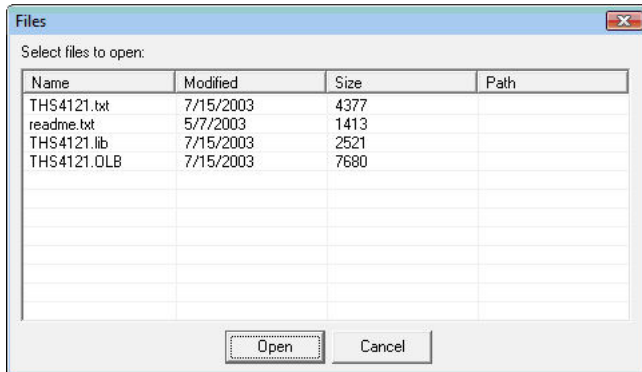
The THS4121 product page will appear. Scroll down the screen and find the link to the Spice model of this product shown below in red.



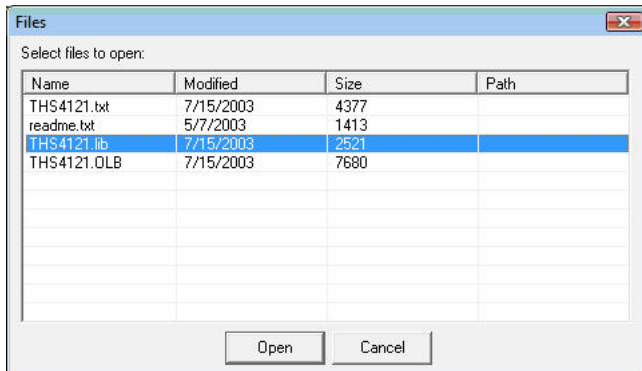
Click on the link. The following message appears in TINA:



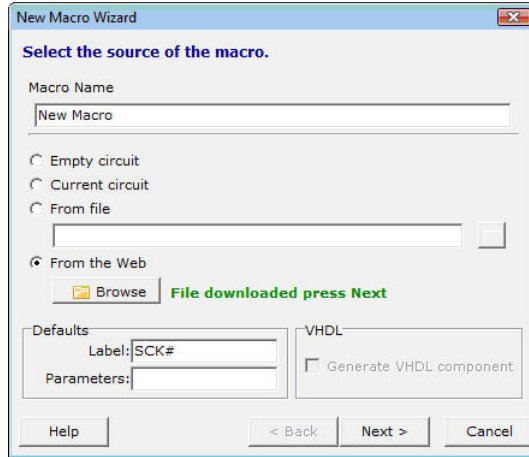
The file is compressed (or “zipped”), but TINA can open and download the files you need directly. Click the Yes button, and observe the next screen.



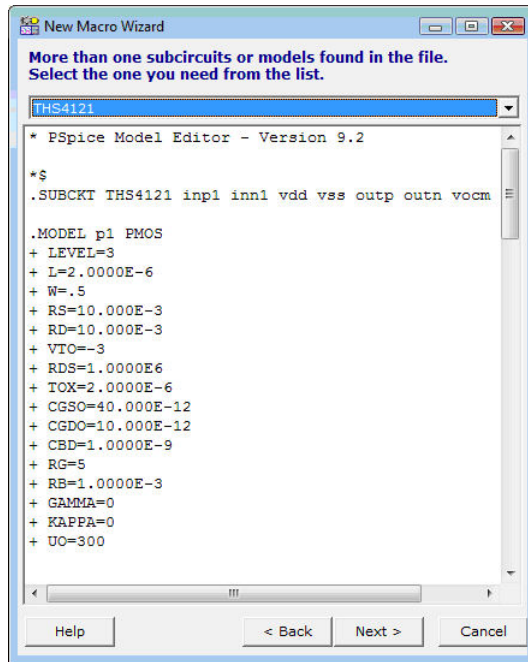
Click to select the THS4121.lib file



and press Open. The New Macro Wizard will appear again, with a (green) message (File downloaded press Next) confirming the successful download



Press the Next button.



If there is more than one macro in the file, TINA presents them in a list.

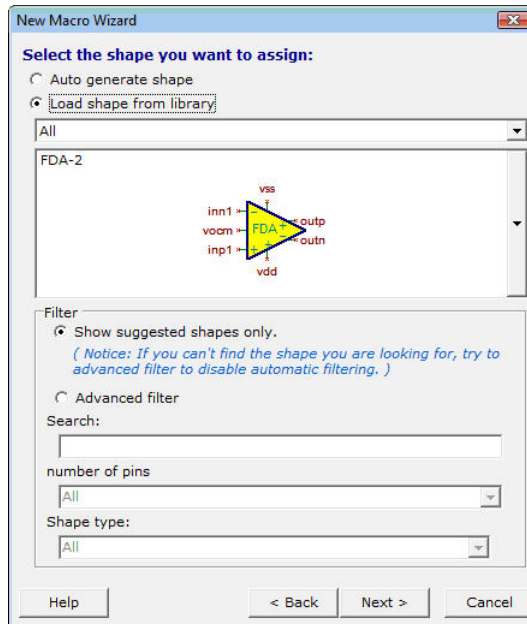
NOTE:

Some manufacturers place several device models in the same file.

You can bring these device models into TINA macros with this tool by selecting them one-by-one.

If there are a lot of models, you might want to use the Library Manager tool which allows you to add all the models into the TINA catalog in one step. Learn about it in the next section.

Now press the Next button. The wizard will show the schematic symbol (shape) suggested:



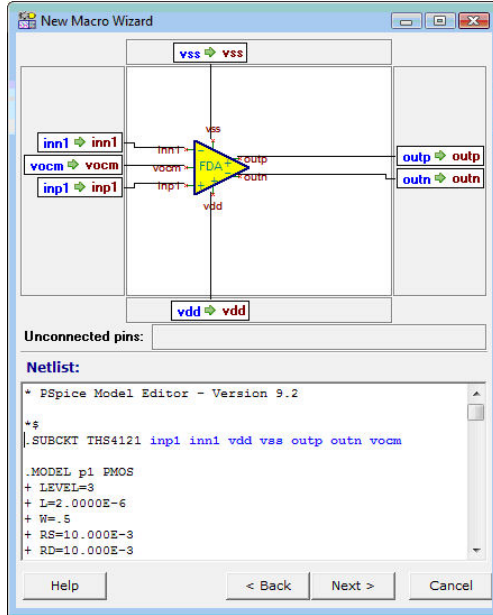
You can choose other shapes, if available, by pressing the long vertical button to the right of the symbol.

NOTE:

You may need to uncheck the "Show suggested shapes only" filter to see more shapes.

In this mode, you can also search by name, by the number of pins, and by function (op amps, comparators etc.) You must select the Shape type option before searching by function.

In our example, TINA appears to have automatically selected an appropriate shape, so you may now press the Next button. The selected shape, pin connections, and Spice macro text will appear.



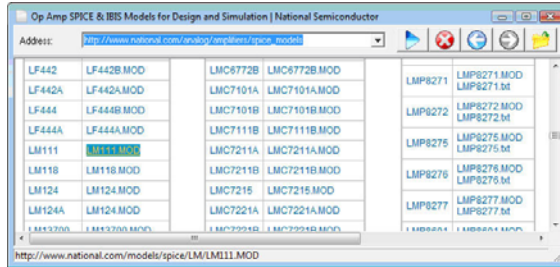
Take a moment to check the connections and if necessary correct them by dragging the connection labels.

If everything is correct, press the Next button again. The Save dialog will appear and you can save the macro into the User or TINA folder. Do so immediately, or do it later using the Insert menu.

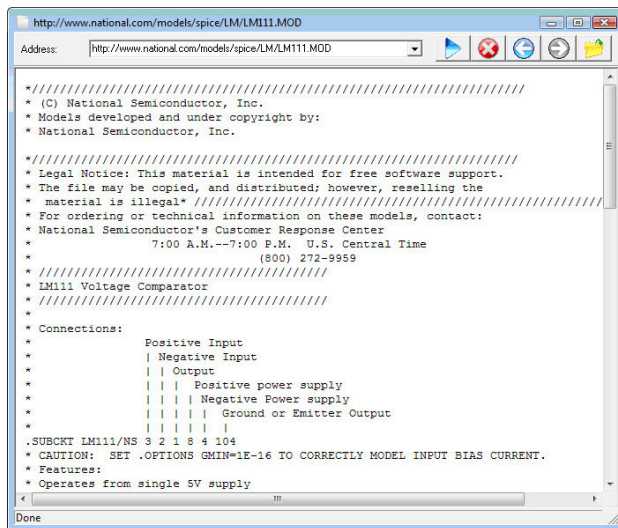
Now let's insert a model from another manufacturer. Our choice is the LM111 comparator from National Semiconductors.


After invoking the internet browser in TINA find the Spice model page at <http://www.national.com> which is, at the time of writing this manual, http://www.national.com/analog/amplifiers/spice_models.

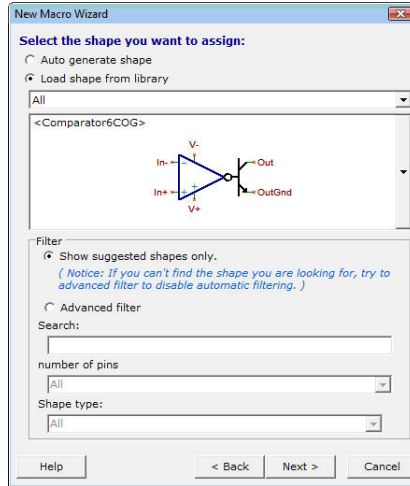
Scrolling down the page you will find the LM111 model under LM111.MOD name as shown below.



Click on the link and the text of the Spice model will appear in TINA.



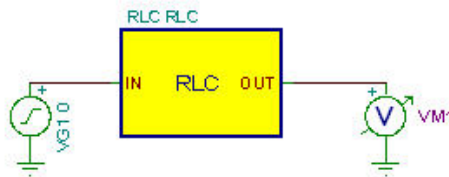
Now click on the  Open icon at the top-right corner of the browser window, and the Macro Wizard will appear again confirming the successful load. Press Next. TINA will automatically present the selected symbol.



The remaining steps are the same as described in the previous section.

5.2.2 Adding Parameters to Spice Macros

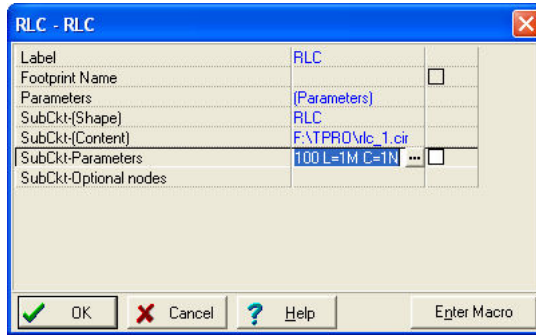
TINA lets you add parameters to Spice subcircuits and set them from TINA. The parameters in the subcircuit are defined by the standard Spice syntax using the PARAMS keyword. For example look at the subcircuit in the circuit given in the MAC_RLC.TSC file in the EXAMPLES\SUBCIRC folder.




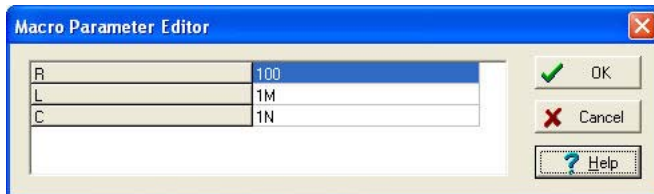
If you double-click on the RLC subcircuit and press the Enter Macro Button, the content of the subcircuit will appear:

```
.SUBCKT RLC In Out PARAMS: R=100 L=1M C=1N C1 Out
0 {C}
L1 1 Out {L}
R1 In 1 {R}
.ENDS
```

The parameters are R, L and C. You can set the parameters in the property dialog of the subcircuit in TINA created as described in this chapter. In our example

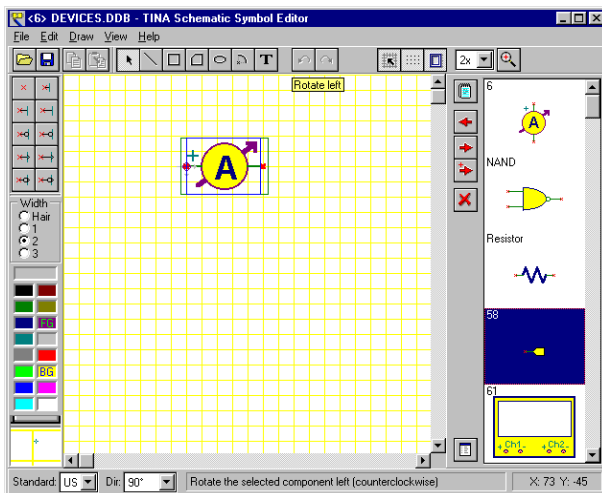



Edit the parameters either in the SubCkt-Parameters line or click the  button and the Macro Parameter Editor dialog will appear. Enter or edit the parameters you want to change and press the OK



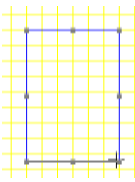
MAKING YOUR OWN SCHEMATIC SYMBOLS



Using *TINA*'s Schematic Symbol Editor, you can create new schematic symbols so that you can add your own circuit components to *TINA*. To create new symbols, you place lines, arcs, rectangles, and arbitrary characters with any fonts, specifying line-width, color, and area color fills. After drawing the symbol, you add and define connections to it. To get acquainted with some features of the editor, read in the list of existing symbols. Use the Windows 95 Start menu to locate the *TINA PRO* folder. Start the Schematic Symbol Editor by clicking its icon, then Select *File | Open* and double click on the *devices.ddb* file. On the right hand side of the Editor Window, the list of current schematic symbols will appear.

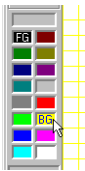


The first symbol on the list (an Ammeter) will appear in the editor window. Try the **Dir:** control at the bottom of the screen. Using this control, you can provide different shapes for symbols at each rotational orientation by designing each of them individually. Now click on the NAND symbol at the right side of the screen and press the  button. The NAND gate symbol will appear in the editor window. Try the **Standard:** control to see the US and European versions of the part shape. You can design symbol versions for each standard, if necessary. If the symbols are identical in the two standards, you need create only one version.

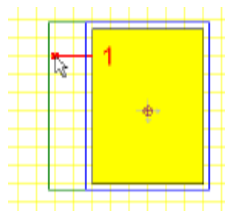
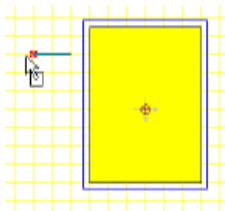
Now let's create a symbol for the full-adder circuit that was used above in our example of creating a half adder macro.



First clear the editor window by pressing the  button or by selecting the actual symbol in the editor window and pressing the Del button. Now draw a rectangle as the body of the component. First press the  button then click on any point in the drawing area, hold the mouse button, and move the mouse until the rectangle is properly sized.



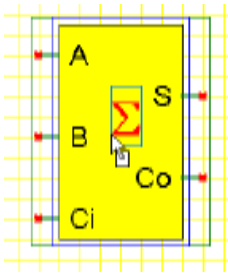
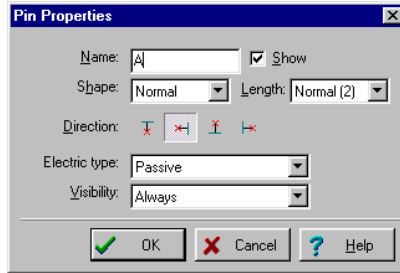
Fill the rectangle with a color by right-clicking on the palette at the lower left corner of the window. Note that a left-click will change the foreground color (FG), in our case the border of the rectangle.



Now add the terminals. Select the desired terminal type from the Terminal-Toolbar in the upper left corner of the window and move the cursor into the rectangle you just drew. Position it using the mouse

or pressing the $[+]$ or $[-]$ key for rotation and click to locate the terminal. Be sure the small red x, indicating the pin end, is outside the body. Continue this process until every terminal is positioned.

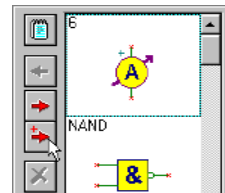
After you have positioned all the terminals, you can establish their properties by double clicking on each of them.



You should assign terminal names as shown in the picture below.

Next, add a large summation sign. Click on the **T** -Text-Editor button on the Toolbar, enter an S in the window, and select a font. To get the special Greek summation sign, select the Symbol-Font.

Press the **D** -Device Properties button, set the Name of the symbol to Full Adder, and press OK.



Finally, copy the new symbol into the symbol library with the **R** button (it appears now at the end of the list), and use the File|Save command to save the now extended devices.ddb file in the *TINA* main directory.

ADVANCED TOPICS^{*}

7.1 Introduction

In the previous chapters, we have introduced TINA's main features and how they can be accessed. Our demonstration, however, is far from complete, for TINA contains many other useful and advanced features for designing, testing and teaching electronics and electronic circuits. These include S-parameter models, network analysis, details of Fourier series and Fourier spectrum analysis, symbolic analysis, post-processing of analysis results, creation of phasor diagrams, Nyquist diagrams, the built-in interpreter, multiparameter optimization, creation of multilayer PCBs and other topics.

The detailed description of these topics is not included in the printed Quick Start manual; they are published only in electronic form. They can be found on the TINA install CD and on the web at www.tina.com in the Documentation section

www.tina.com/support.htm.

* Not all features described here are available in TINA-TI

7.2 Table of Contents of Advanced Topics

- Parameter Stepping
- DC Transfer Characteristic and Parameter Sweeping
- Phasor Diagram
- Nyquist Diagram
- Noise Analysis
- Network Analysis and S-parameters
- Symbolic Analysis
- Post-processing Analysis Results
- Design Tool
- Optimization
- Design Tool vs. Optimization
- Fourier Series and Fourier Spectrum
- Interpreter manual
- TINA PCB Design manual

We are continuously adding new topics and examples to this chapter, so please check back regularly to the Documentation section of our website www.tina.com for the latest version.