

Circuit Design, Simulation, And Implementation Using Tina

Prof. Dr. Dogan Ibrahim

Near East University, Department of Biomedical Engineering, Lefkosa, TRNC
E-mail: dogan@neu.edu.tr,
Tel: 90392 2236464



Keywords: *Circuit simulation, Spice, circuit design, PCB design*

Abstract

Circuit simulation is very important in all branches of electrical and electronic engineering. In circuit theory electronic laboratory experiments, it is required to purchase electronic components and use them to build circuits and then carry out experiments using these circuits. Usually, it may be difficult to obtain the required parts, and also physical parts can easily be damaged if used incorrectly. This paper describes the use of a very popular circuit simulation software tool called TINA. With the help of this tool students can easily design and simulate their circuits, and then implement their designs on a PCB.

1. INTRODUCTION

Laboratories are very important part of every engineering course. Students learn the theory in classes and apply their knowledge into practise by using real equipments in laboratories. For example, electronic engineering students learn the theory of transistor amplifiers in the classroom. Then, they are given assignments in laboratories where they obtain the required components and construct a transistor amplifier circuit. Students then use real physical voltmeters, multimeters, or oscilloscopes to analyze the behaviour of the amplifier circuit.

Although laboratory experiments are very useful they have some problems:

- It is not always easy to find the required components.
- It is easy to damage real components, for example by passing very large currents through them.
- The characteristics of electronic components can change with aging and temperature.
- An instructor should be present in the

laboratory to make sure that students connect the components correctly.

- Real laboratory measuring equipment (e.g. storage oscilloscopes) are expensive to purchase and maintain.
- A large number of measuring equipment may be required for a class and this can be very costly.
- Real laboratory measuring equipment usually need calibration from time to time and this can be costly.
- Students can get electric shock using mains operated equipment in laboratories.

Computer simulation is an alternative to real experiments. Circuit simulation is basically a computer program predicting the behavior of a real circuit. It replaces real components with idealized electrical models (e.g. Spice models). In a circuit simulation students run a computer program and design their circuit using model components from a library of the simulator. They can then add virtual instruments such as voltmeters, oscilloscopes, multimeters and so on. The simulation can then be activated and the A.C. and D.C responses of the circuit, transient response, Fourier analysis and other analysis can easily be performed and the circuit behaviour can be analyzed in detail.

Although simulation can be an invaluable tool in building and analyzing the behaviour of circuits, it has the following advantages and disadvantages:

- Any component whatever the cost or the model can be simulated with a simulator. In a real laboratory experiment it may be necessary to order a component and wait considerable amount of time before it arrives.
- Simulation may not take into account the component tolerances, aging, or temperature effects. Students may think that all components are ideal at all times.
- The virtual instruments used in simulation are only computer programs and thus there are no cost issues. It is easy to add additional virtual instruments to a simulation by simply clicking a button.
- Real laboratory instruments can easily be damaged if connected wrongly. There are no such problems when virtual instruments are used in simulations.

- Laboratory instruments require calibration. There are no such problems with virtual instruments used in simulations.
- The components used in a simulation model are just computer programs and therefore there is no possibility of damaging a component.
- Simulation allows measurements of internal currents, voltages and power that in many cases are virtually impossible to do using real instruments.

The aim of this paper is to describe, with examples, the main features of one of the most popular educational and industrial circuit simulation suites called TINA.

2. WHAT IS TINA ?

TINA¹ is a powerful integrated software design suite which includes schematic circuit design, circuit simulation, PCB design, and many other useful electronic design tools (e.g. Filter design). TINA is developed by *DesignSoft*, and the software package is also called “*The Complete Electronic Lab*”.

TINA can be used in technical colleges and universities for teaching the principles of circuit design and simulation. In addition, it can be used by hobbyists to simulate electronic circuits. TINA can also be used in commercial and industrial applications for designing and simulating highly complex analog and digital circuits, including microcontroller based circuits. For example, the operation of a complete RF communication receiver system can be simulated using TINA. Similarly, the operation of a microcontroller based system can be simulated by writing programs for the microcontroller and then loading these programs into the microcontroller.

TINA is an integrated software design package. Once a circuit has been simulated and we are happy about its behaviour, we can easily transfer the circuit to a PCB with the click of a button. We can then carry out auto-placement and auto-routing of the circuit components and connections, and have a complete PCB design of the circuit in a few seconds. In addition, we can generate Gerber plotter and CNC drilling files for our design so that the designed PCB can be manufactured.

TINA is currently at Version 9 and it can perform the following operations:

- Schematic capture
- Live 3D breadboard
- PCB design
- Electrical rule check
- Schematic symbol editor

- Library manager
- Parameter extractor
- Test and equation editor
- DC analysis
- Transient analysis
- Fourier analysis
- Digital simulation
- VHDL simulation
- Microcontroller simulation
- Flowchart editor
- AC analysis
- Network analysis
- Noise analysis
- Symbolic analysis
- Monte-Carlo analysis
- Design tool
- Optimization
- Post-processor
- Presentation
- Interactive mode
- Virtual instruments
- Real-time test & measurements
- Training and examination

In this paper, because of the limited space, we shall be looking at some of the important features of TINA by giving a circuit simulation and a PCB design example.

There are several versions of TINA, tailored to meet the needs of various individuals and companies. The available versions are:

- Industrial version
- Network version
- Educational version
- Classic version
- Student version
- Basic version

The *industrial version* includes all of TINA's features and is the most expensive version. The *basic version* has limited functionality with the circuit size limited to 100 nodes, and the number of pads on the PCB limited to 100. Other versions have some limited functionalities and further details can be obtained from *Tina Users Manual*.

In addition to the circuit simulation and PCB design, TINA supports a real multifunction PC instrument called TINALab II. With the help of TINALab II students can interface the hardware to TINA simulation package and observe the results on the oscilloscopes, signal analyzers, and multimeters, in real-time. In addition, a function generator is provided which can generate sine, square, ramp, triangle, and arbitrary waveforms from DC to 4 MHz, with logarithmic and linear sweeps. Arbitrary waveforms can be programmed using TINA's easy to use interpreter language.

In the next section we shall be looking at a

circuit simulation example, and demonstrate some of the features of TINA.

3. EXAMPLE CIRCUIT SIMULATION

In this section we shall be looking at the simulation of a circuit with the aim of showing some of the features of TINA. TINA has so many features and because of the limited space in this paper, only some important features will be demonstrated here.

Example Circuit Simulation

In this example we shall be simulating an oscillator circuit built using a LM555 type monostable/astable chip². The transient response of the circuit will be displayed on the screen.

The required components are connected together on the TINA screen using a mouse, and the schematic circuit diagram of the oscillator is shown in Figure 3.1.

Power to the circuit is provided using a voltmeter, set to 10V. The frequency of oscillation of an astable designed from LM555 chip is set by resistors R1 and R2 and capacitor C and is given by:

$$f = \frac{1.44}{C(2R_1 + R_2)}$$

The component values in the circuit are: R1 = R2 = 1k and C = 1μF. Therefore, the frequency of oscillation is given by:

$$f = \frac{1.44}{10^{-6}(2 \times 10^{-3} + 10^{-3})}$$

or, f = 480 Hz, and the period is T = 1/f = 2.083 ms

The transient analysis of the circuit is shown in Figure 3.2. It is clear that a square wave is generated with the period of the oscillation around 2.08 ms.

555 AS ASTABLE OSCILLATOR

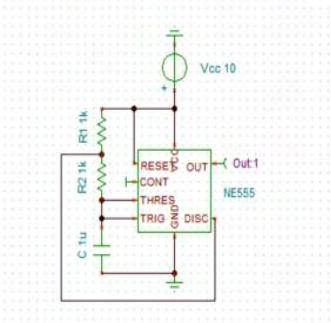


Figure 3.1 Schematic of the LM555 based astable oscillator

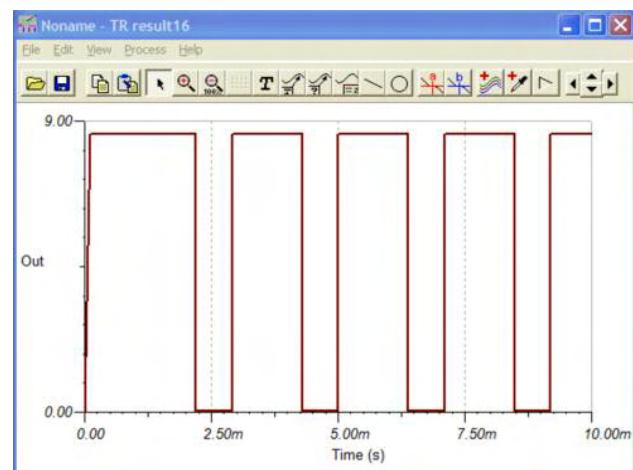


Figure 3.2 Transient analysis of the circuit

We can easily display the Fourier spectrum of the output signal and this is shown in Figure 3.3.

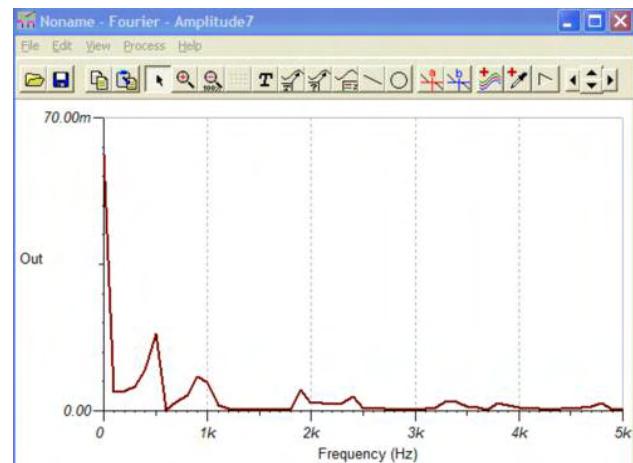


Figure 3.3 Fourier spectrum of the output signal

We can now add a **virtual oscilloscope** to the design and observe the output waveform on the oscilloscope. This is shown in Figure 3.4.

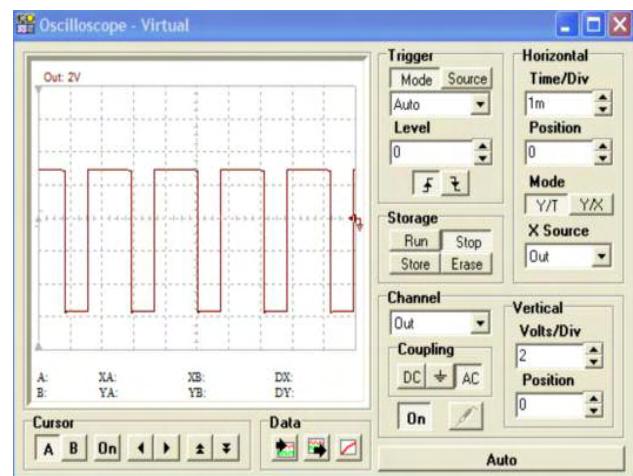


Figure 3.4 Output waveform on an oscilloscope

We can see the pictures of the components used in the design by clicking the **2D/3D** button. This is shown in Figure 3.5.

555 AS ASTABLE OSCILLATOR

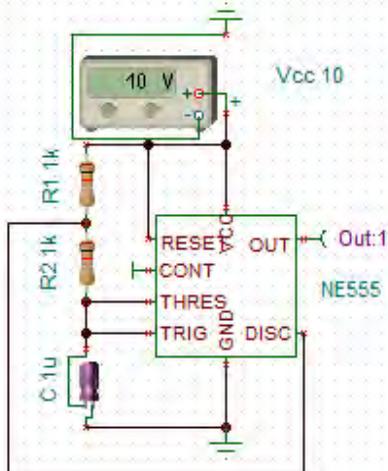


Figure 3.5 Showing the components used

We can also see the circuit built on a breadboard by selecting the option **Live 3D Breadboard**. This is shown in Figure 3.6.

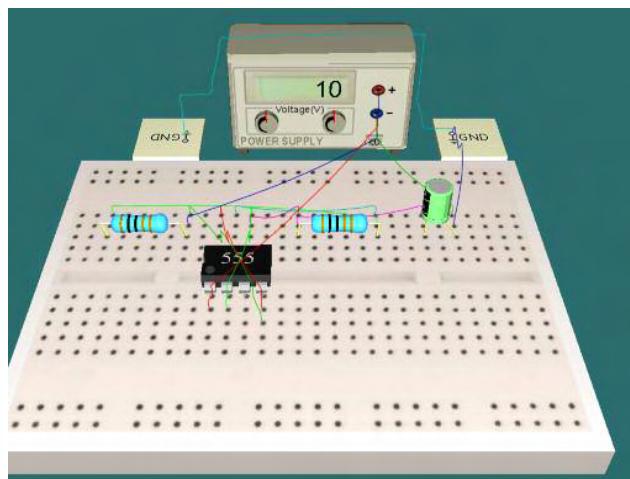


Figure 3.6 Displaying the circuit on a breadboard

The linear amplitude spectrum of the output waveform can be displayed by selecting the **Spectrum Analyzer**. This is shown in Figure 3.7.

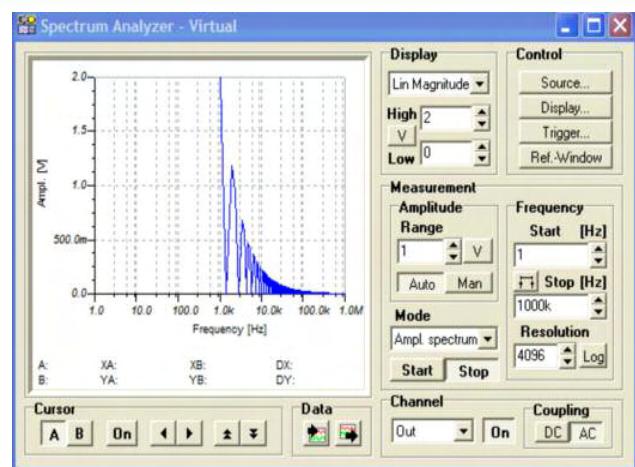


Figure 3.7 Linear amplitude spectrum of the output

4. THE PCB DESIGN

PCB design is integrated into TINA design suite. Pressing the **PCB Design** button starts the PCB software and gives user the option to transfer the circuit to a PCB and place the components on a blank board of specified dimensions. The user can change the position of the components on the PCB. A 2-way header is added to the output of the circuit and the PCB layout with the components placed manually is shown in Figure 3.8.

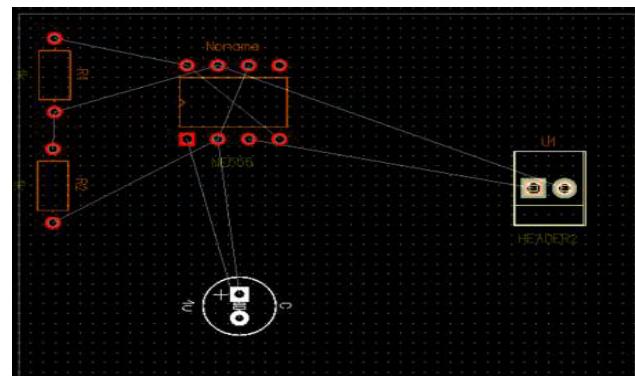


Figure 3.8 Components placed manually

Choosing the option **Autoroute board** routes the board and the final routed PCB layout is shown in Figure 3.9.

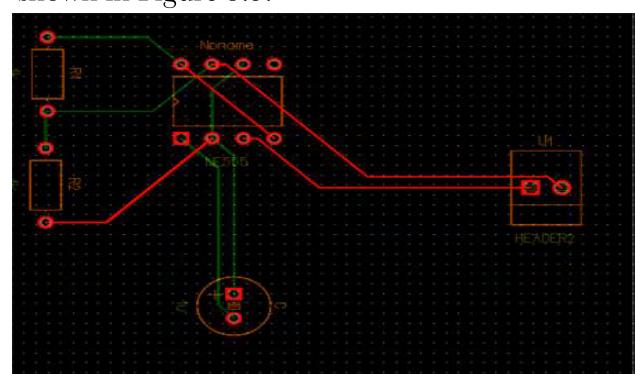


Figure 3.9 Routed board

After the board has been routed we can view the placement of the components in 3-dimensional mode by clicking the **3D View** button, and this is shown in Figure 3.10.

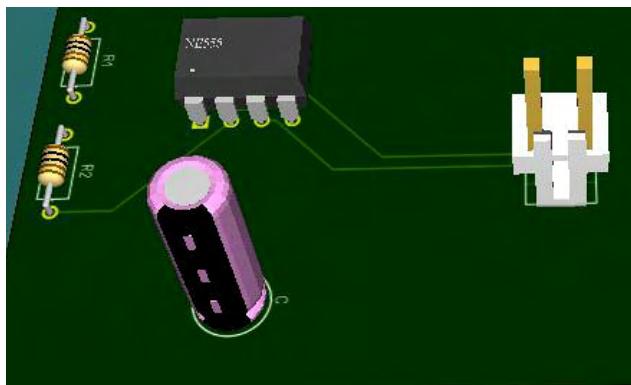


Figure 3.10 Displaying the components on the routed board

Once we are happy with the design we can generate the Gerber output file for the photo-

plotter, and the drilling file for the CNC machine so that the PCB can easily be manufactured.

In addition to the schematic design and PCB design features, TINA also has the following built-in features:

- Equation editor
- Interpreter
- Netlist editor
- Logic designer
- Filter designer
- Flowchart editor
- And many more features

As an example, using the **Filter designer** option we can easily design low-pass, high-pass, band-pass and band-stop filters. Figure 3.11 shows the Filter design screen. The designed filter circuit diagram using operational amplifiers is automatically generated by TINA and this is shown in Figure 3.12.

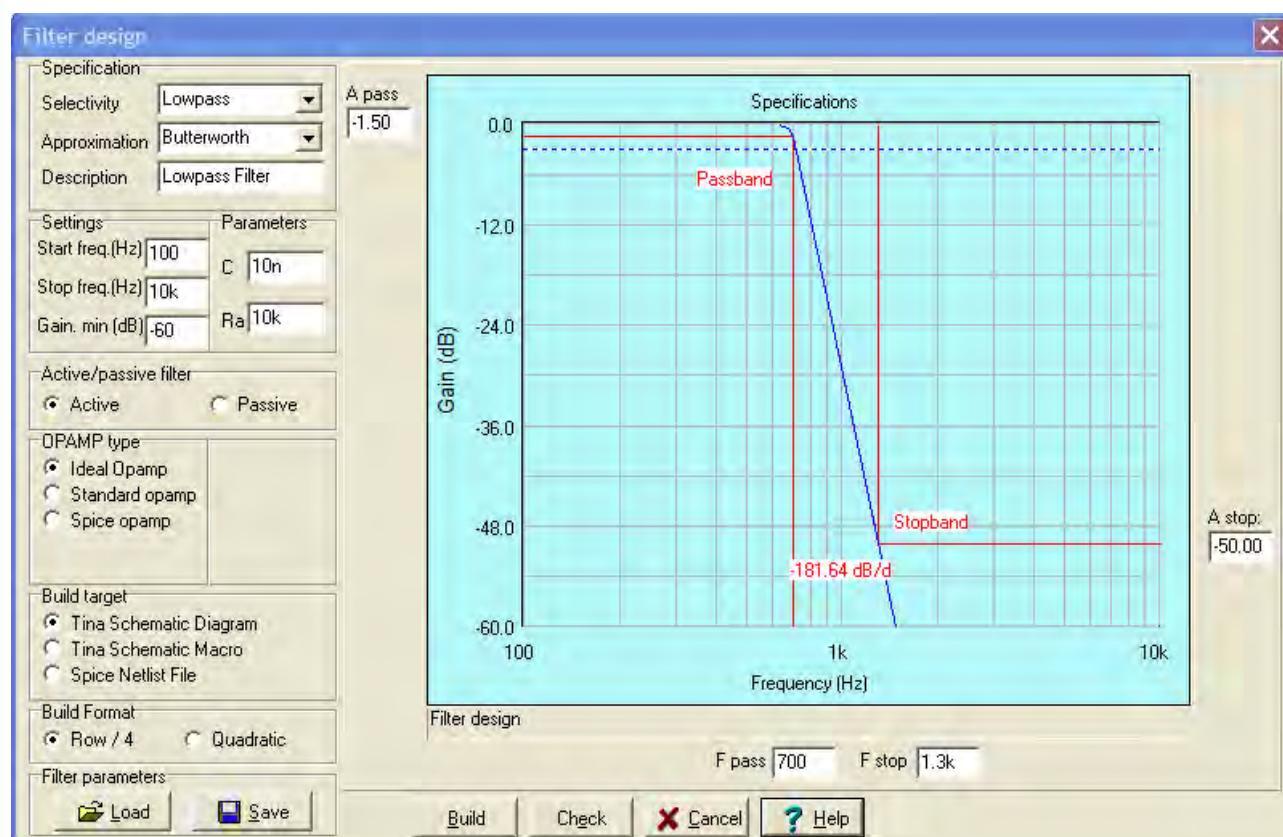


Figure 3.11 TINA Filter design screen

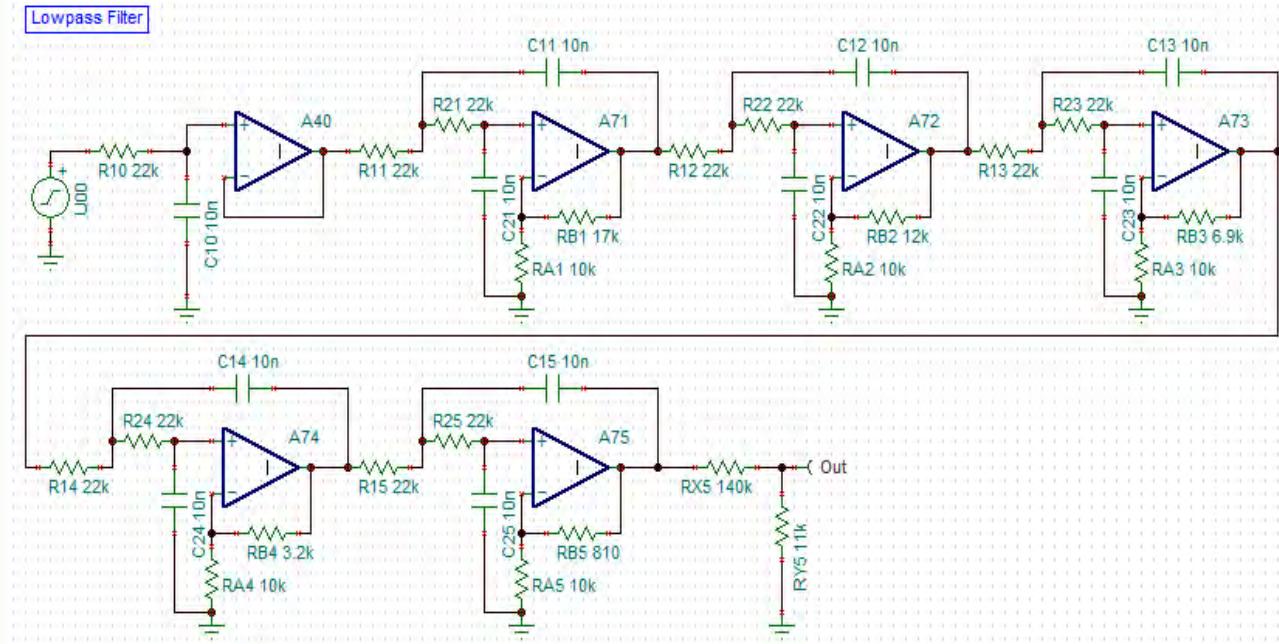


Figure 3.12 Designed filter circuit diagram

5. CONCLUSION

TINA is a great tool for simulating electrical and electronic circuits. In addition, the simulated circuit can be transferred to a PCB and the circuit can be implemented on a PCB.

It is recommended by the author that every electrical or electronic engineering student should own a personal copy of the *student version* of TINA and practise to learn how to design and simulate complex circuits. DesignSoft offers special reduced price for the *student version* of TINA and students should take advantage of such offers.

An example is given in the paper on circuit

simulation and PCB design. Because of the limited space in this paper it has not been possible to describe all the features of TINA. Interested readers should contact the DesignSoft web-site for further information.

6. REFERENCES

- [1]. “TINA Users Manual”, *DesignSoft* web site: <http://www.designsoftware.com>
- [2]. “National Semiconductors Analog Databook”, web site: <http://www.ni.com>

Balomuza Davet

Odamızın düzenlediği Geleneksel EMO Balosu bu yıl 16 Nisan 2011, Cumartesi günü düzenlenecektir.

Geleneksel EMO Balosu'nun bu yılı konuğu Türkiyeli ünlü ses sanatçısı Suavi ve özel orkestrası olacaktır.

Sınırsız yerli ve yabancı içki dâhil (Her şey dahil) olan Geleneksel EMO Balosu'nun bilet ederi, kişi başı 70-TL olup, arzu eden üyelerimiz biletlerini odamız sekreterliğinden temin edebilir.

Konaklama isteyen üyelerimiz için çift kişilik odada (Bungalow) kişi başı konaklama (kahvaltı dâhil) 80- TL; tek kişilik konaklama ücreti 120- TL'dir.

Çocuklar: 0-4 yaş arası ücretsiz; 4-12 yaş arası %50 indirimli; 3'üncü kişi % 25 indirimli olacaktır.

Tüm üyelerimizin aileleri ile birlikte katılımı özlenir.

YER: Acapulco Hotel

Saat: 19.30