ENGR3499-2: Real World Microcontrollers
Assignment-1: Your first PCB

In this class you have one assignment and one project, this is the assignment. Its goals are to give you many of the skills needed to complete your project, most importantly how to draw schematics and layout printed circuit boards using KiCad.

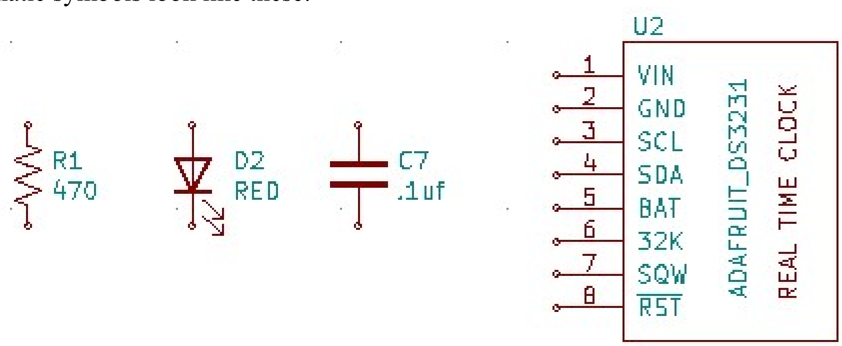
You have 3 weeks to complete this assignment. Feel free to finish faster if you want more time for your project. Grading will be done once you have completed all steps, however be prepared to discuss your progress each week during our Zoom sessions.

In this assignment, you will design your first printed circuit board. Below you will find a simple schematic of a circuit that goes Beep Beep Beep when you press its button. Your job is to recreate that schematic in KiCad, then layout a PCB for it.

The steps below detail what you are to do for this assignment, but talk less about how to do it. For instruction using KiCad, you will watch a series of videos that take you through a similar project. And always, please feel free to ask questions if anything is unclear.

- The first video talks about what KiCad is and how to install it.

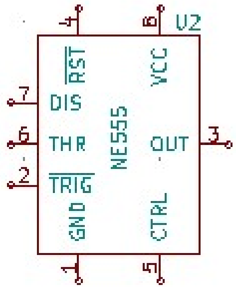
  Watch: *Getting To Blinky 5.0 – Introduction* (6 min)
  - https://www.youtube.com/watch?v=BVhWh3AsXQs&list=PLy2022BX6EspFAKBCgRuEuzapuz_4aJCn&index=2&t=0s
- Install KiCad on your computer. Download it here: https://kicad-pcb.org/download/
- Schematic symbols look like these:

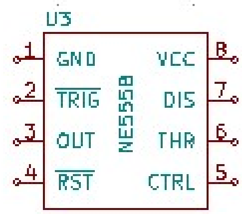
  

  They are used to represent each component in your circuit. The KiCad library has many to choose from, but you will need to draw some yourself (such as the Adafruit module above).

  Watch: *Getting To Blinky 5.0 - Schematic Symbol Creation* (15 min)
  - https://www.youtube.com/watch?v=LaUd8WfFooU&list=PLy2022BX6EspFAKBCgRuEuzapuz_4aJCn&index=3&t=0s
- In the steps that follow you will use KiCad to redraw the schematic that I have attached below. Start by running KiCad, then create a New Project, call it *RealWorldBeepy.*
- Before drawing a schematic, make sure you have all the symbols that your circuit uses. As mentioned, some symbols you will need to create yourself. The first step is making an empty *Symbol Library* to put your custom symbols in. Using the *Library Editor*, create a new library called *RealWorldBeepyLib*. When asked to choose between *Global* and *Project*, pick *Project*.

- Create a schematic symbol for the NE555 chip, I've included the datasheet below. Arrange the order of the pins in a way that makes drawing the schematic intuitive (as was done in the video). For example, when drawing my circuit, the symbol on the left is better than the one on the right.

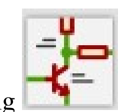




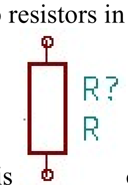
  Note: When defining a pin in the *Pin Properties* dialog box, you can set the *Electrical Type* to a value that best fits that pin, or simply set all pins to *bidirectional*.

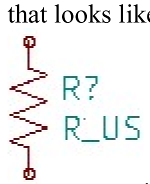
- Now that you have all the custom symbols ready to go, figuring out how to draw a schematic is next.

  Watch: *Getting To Blinky 5.0 - Making The Schematic* (14 min)
  - https://www.youtube.com/watch?v=01OYONrYtNQ&list=PLy2022BX6EspFAKBCgRuEuzapuz_4aJCn&index=4&t=0s

- Start drawing your schematic by clicking  in the main KiCad window. It will open an empty schematic, just showing the border and title block. Setup your schematic by clicking  (found on the top toolbar). Set the page size to *8.5 x 11* and *Title Block* values:
  - Issue date = today's date
  - Revision = Rev 1.0
  - Title = Real World Beepy
  - Company = (You can pick your own company name. I often use *Stanley Reifel & Co.* as I can use it without filing a fictitious business name.)

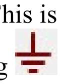
- Redraw the Beepy schematic by copying my version found below. Be sure to use the NE555 symbol that you created earlier. Here are a few notes as you draw yours:

  - When looking up resistors in the *Choose Symbol* dialog box, you can type *R* to select the symbol that looks like this 

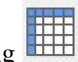
 commonly used in Europe and Asia, or you can type *R_US* to select 

. Either one is OK.

  - Note that in my example schematic, *C1* and *C3* are non-polarized capacitors, so search for the symbol *C*. *C2* and *C4* are polarized, use symbol *CP1* for these.

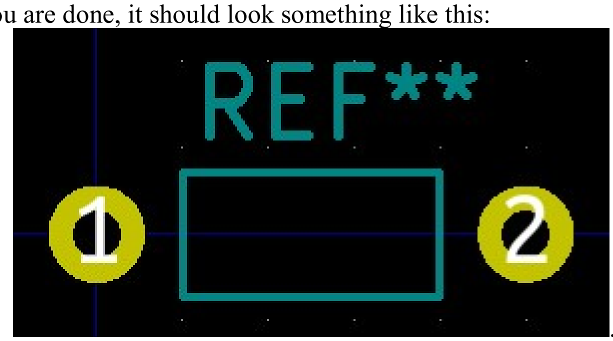
  - Also notice that my schematic does not show wires connecting the 9 volt and the ground pins. Instead, I used power and ground symbols. This is done to reduce clutter, making the schematic easier to read. Find these symbols by clicking  on the right.

  - Instead of having a battery, my schematic uses a 2 pin power connector (a battery plugs into that connector). On your schematic, I suggest using the generic connector symbol *Conn_01x02*. Note the *01x02* indicates a connector having pins in 1 row with 2 columns.

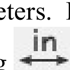
- After placing all the symbols and wired them up, be sure to run the *Annotate Schematic Symbols* command by clicking . This will assign each component a part reference number (i.e. R1, C3, J4…).
- An important subject not covered in the videos is creating the BOM, or *Bill of Materials*. When done correctly, the schematic can automatically generate a parts list for your project, complete with quantities and order numbers. Below I've included the BOM made by KiCad for my example schematic.
- KiCad already knows the reference number (i.e. R1, C3) and value (i.e. 47K, 0.1uf) for each component in your design. What needs to be added now are the Digikey order numbers. Start by clicking  found on the top toolbar. In the dialog box, press the *Add Field* button to create a new column, label it *Digikey*. For each component displayed in the list, copy the order numbers from my BOM into your *Digikey* fields. Later when designing your board, knowing these numbers will help you choose the right PCB footprint.
- Watch: *Getting To Blinky 5.0 - Associating Symbol and Footprint* (11 min)
  - https://www.youtube.com/watch?v=IIPKGoW0VBY&list=PLy2022BX6EspFAKBCgRuEuzapuz_4aJCn&index=5&t=0s
- For many parts in your circuit, you will find a footprint in KiCad's library that works nicely. However, there are times when you need to design your own.

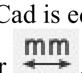
  Watch: *Getting To Blinky 5.0 - Creating A Custom Footprint* (15 min)
  - https://www.youtube.com/watch?v=-7yFlz6wdUA&list=PLy2022BX6EspFAKBCgRuEuzapuz_4aJCn&index=6&t=0s
- For this assignment, you will create two custom PCB footprints.
  - The first step is to make an empty library to put new footprints in. The video above talks about how to do it. When asked to choose between *Global* and *Project*, pick *Project*.
  - The Beepy circuit uses resistors that are through-hole and rated for 1/8 watt. Surprisingly the KiCad library does not have a good footprint for these. Draw your own with two round pads (56mil diameters and 29mil holes). Space the two pads 250mils apart. On the Front Silkscreen Layer (F.SilkS) draw the part's outline using a rectangle.

    When you are done, it should look something like this:

  - The circuit has a tiny speaker that is soldered directly to the board. KiCad does not have a speaker footprint so create one. Start by looking at the speaker's datasheet to determine it's dimensions. Here's how to find the datasheet:
    1. Find the speaker's Digikey part number from your Bill of Materials (Fun fact: All Digikey part numbers end in "-ND").
    2. Go to www.digikey.com then paste the part number in the search box.
    3. Look on the speaker's page for "Datasheets" then click the link.

    Note all dimensions in speaker's datasheet are in millimeters. KiCad is equally happy to work in inches or millimeters. Switch back-and-forth by clicking 

    or 

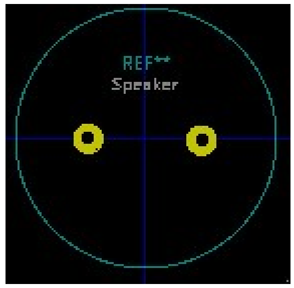
    .

    From the datasheet, the speaker's pins have a diameter of "0.8 +/- 0.05 mm", let's call it 0.85mm. When choosing the pad's hole size, be sure to make it bigger than the pin. In most cases adding 8mils or 0.2mm should be enough.

The diameter of the pad obviously needs to be bigger than the hole. I suggest making the pad twice the diameter of the hole.

It's time to make a footprint for the speaker. Start by placing two round pads. After adding them, be sure to set their diameter and hole size. Also draw the component outline on the Front Silkscreen Layer (F.SilkS), the same size as the speaker.

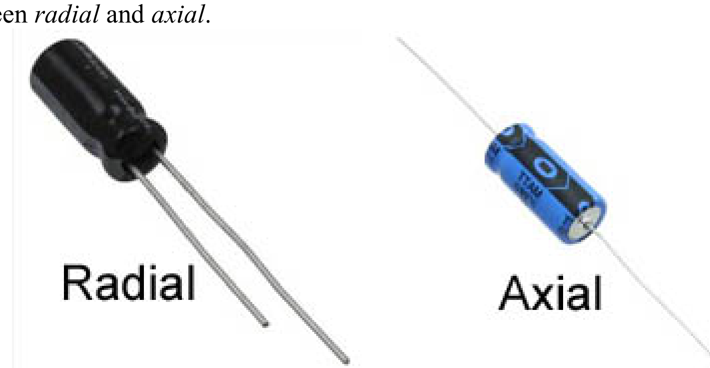
Mine looks like this:



- Now go through your schematic, one symbol at a time, adding the footprints to each. For the resistors and speaker, use the footprints you just made. For the others, use a matching footprint from KiCad's library.

  Follow the steps in the video for assigning the footprints. Note: An important difference between the video and your project is that they used surface mount components, whereas you are using through-hole. Here are some tips for choosing the right footprints to match the parts in your BOM:

  - For C1 & C3, start in the library by selecting *Capacitor_THT* (THT indicates through-hole). These capacitors are non-polarized so choose a footprint that begins with *C_*, not *CP_*. I suggest picking one that begins with *C_Rect*. The most important thing is getting the pin spacing right. Digikey's website is a great resource for figuring this out. Lookup the part on Digikey, then scroll down to *Product Attributes* to find the *Lead Spacing*, *Package*, and *Size* (Note: *Lead spacing*, *Pin spacing* and *Pitch* all mean the same thing). C1 & C3 have a pin spacing of 2.5mm, so choose a footprint that ends in *_P2.50mm*. Lastly, pick the footprint that is close in length *L* and width *W* to your part.
  - For C2 & C4, also start with *Capacitor_THT*. C2 & C4 are polarized so use *CP_*. Next pick between *radial* and *axial*.

    Again, look up the part on Digikey to find its *Lead spacing, Size* and *Package*. Now choose the footprint from the library with the right diameter (D) and lead spacing (P).
  - My favorite connectors for PCBs are *Molex KK* connectors with 0.1" pin spacing. For J1, find it in the footprint library by starting with *Connector_Molex*. Note that all dimensions in the KiCad libraries are in millimeters, so 0.1" is represented as 254 (2.54mm). J1 has 2 pins, organized in 1 row with 2 columns. For J1, find the footprint whose name includes:

    Molex, KK, 254, 1x02, Vertical
  - Use these same techniques to choose the footprints for SW1 and U1.

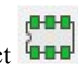
- Your schematic is now done, congratulations! The next step is learning about PCB layout.

  Watch: Getting To Blinky 5.0 - Finishing The Layout (25 min)
  - https://www.youtube.com/watch?v=D_7Y33AoagY&list=PLy2022BX6EspFAKBCgRuEuzapuz_4aJCn&index=7&t=0s

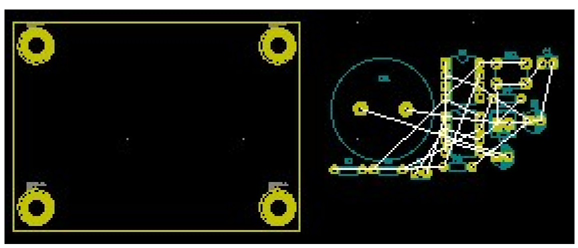
- Before designing your PCB, be sure to generate the schematic's netlist by clicking .

- Now start your layout. Follow the instructions in the video *Getting To Blinky 5.0 - Creating A Custom Footprint*. They talk about the first steps beginning at 12 minutes, 40 seconds.

- Define the *Board Outline* as a rectangle, 2.5" x 1.8". This is easiest if you set the grid (on the top toolbar) to 100.00 mils. As you draw the outline, it's also helpful to keep an eye on the status line's *dx* & *dy*.

- Add 4 mounting holes to your board, one in each corner. They do not cover this in the video, so here's how:
  - From the right toolbar select , then click in the workspace.
  - In the listbox, expand *MountingHole*.
  - Choose a mounting hole with a *_Pad*. Here are some useful sizes:
    - For 4-40 screws: MountingHole_3.2mm_M3_Pad
    - For 6-32 screws: MountingHole_3.7mm_Pad
  - Double click on the pad of your choice, then click to place it in your layout.
  - Clean up a mounting hole's display by hover over it then pressing *E*. Uncheck all the *Show* checkboxes.

- If you have followed the steps in the video, your board should include the footprints, looking something like this:



- Perhaps the most important step in laying out a PCB is positioning each component. I spend a great deal of time on this step, about an hour for a board with 30 parts.

  First I place parts that I want in specific locations (perhaps connectors along one side, push buttons that line up with holes in the board's enclosure, …).

  Then I think about organizing the layout so the parts wired to each other are close to each other. Often rotating a part by 90° or 180° can be a big help. Basically, you are trying to minimize trace lengths and layer changes.

- Now that you have positioned each component, the fun part begins! Using the techniques in the last video, route the traces on your board to wire everything up.

- Watch: Getting To Blinky 5.0 - Output Files For Manufacturing (20 min)
  - https://www.youtube.com/watch?v=3oL6ZdCUZvA&list=PLy2022BX6EspFAKBCgRuEuzapuz_4aJCn&index=8&t=0s

- When the board is "electrically" finished, I add additional text on the front silkscreen layer. For this board, I suggest adding:
  - The name of your board: "REAL WORLD BEEPY"
  - The version number: "V1.0"
  - Label the power connector with "9V" and mark the pins with a "+" & "-"
  - Be sure to add your name. I often copyright my designs by adding:

    (c) 2020 - STANLEY REIFEL & CO.

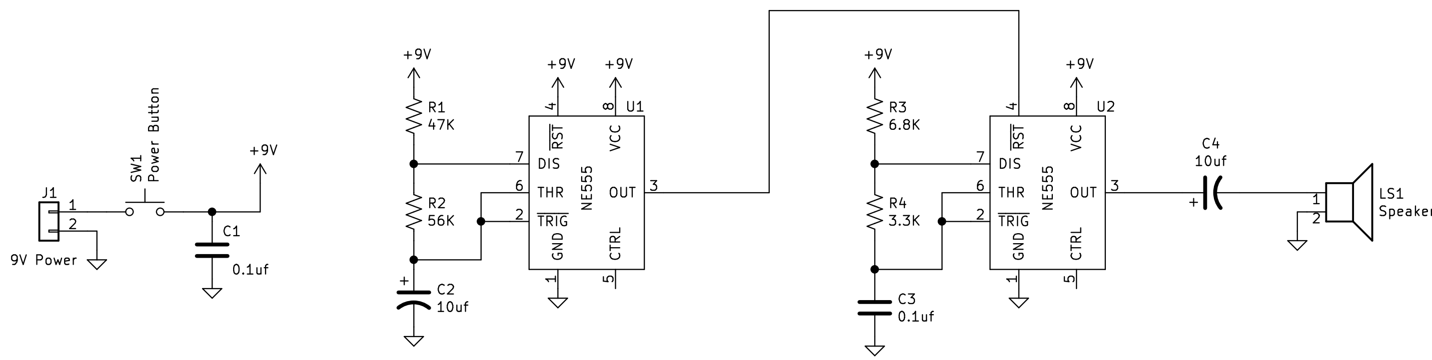
- Now make the files needed to manufacture your PCB. First create a folding inside your project named *Gerbers*, then generate the files for these layers, saving them in the *Gerbers* folder.
  - F.Cu
  - B.Cu
  - F.Silk
  - B.Silk
  - F.Mask
  - B.Mask
  - Edge.Cuts
  - The Drill file

- Congratulations!!! You have designed your first printed circuit board! The only thing left to do is turn it in. Copy all of the project's files into a single ZIP file, then submit the ZIP by email to:

  stanley.reifel@olin.edu

  with the email's subject line formatted as follows:

  RWM - <Your Name> - Assignment 1



S. Reifel & Co.

Sheet: /

File: Beepy.sch

Title: Real World Beepy

Size: A	Date: 2020-08-22	**Rev: Rev 1.0**
KiCad E.D.A. kicad (5.1.6)-1		Id: 1/1

Beepy Bill Of Materials

Item	Qty	Reference(s)	Value	Digikey
1	2	C1, C3	0.1uf	BC1084CT-ND
2	2	C2, C4	10uf	493-14344-1-ND
3	1	J1	9V Power	900-0022232021-ND
4	1	LS1	Speaker	2104-SM231508-1-ND
5	1	R1	47K	CF18JT47K0CT-ND
6	1	R2	56K	CF18JT56K0CT-ND
7	1	R3	6.8K	CF18JT6K80CT-ND
8	1	R4	3.3K	CF18JT3K30CT-ND
9	1	SW1	Power Button	450-1647-ND
10	2	U1, U2	NE555	296-NE555P-ND

FAIRCHILD
SEMICONDUCTOR®

January 2013

LM555
Single Timer

Features

- High-Current Drive Capability: 200 mA
- Adjustable Duty Cycle
- Temperature Stability of 0.005%/°C
- Timing From µs to Hours
- Turn off Time Less Than 2 µs

Applications

- Precision Timing
- Pulse Generation
- Delay Generation
- Sequential Timing

Description

The LM555 is a highly stable controller capable of producing accurate timing pulses. With a monostable operation, the delay is controlled by one external resistor and one capacitor. With astable operation, the frequency and duty cycle are accurately controlled by two external resistors and one capacitor.

8-DIP

1

8-SOIC

1

Ordering Information

Part Number	Operating Temperature Range	Top Mark	Package	Packing Method
LM555CN	0 ~ +70°C	LM555CN	DIP 8L	Rail
LM555CM		LM555CM	SOIC 8L	Rail
LM555CMX		LM555CM	SOIC 8L	Tape & Reel

Block Diagram

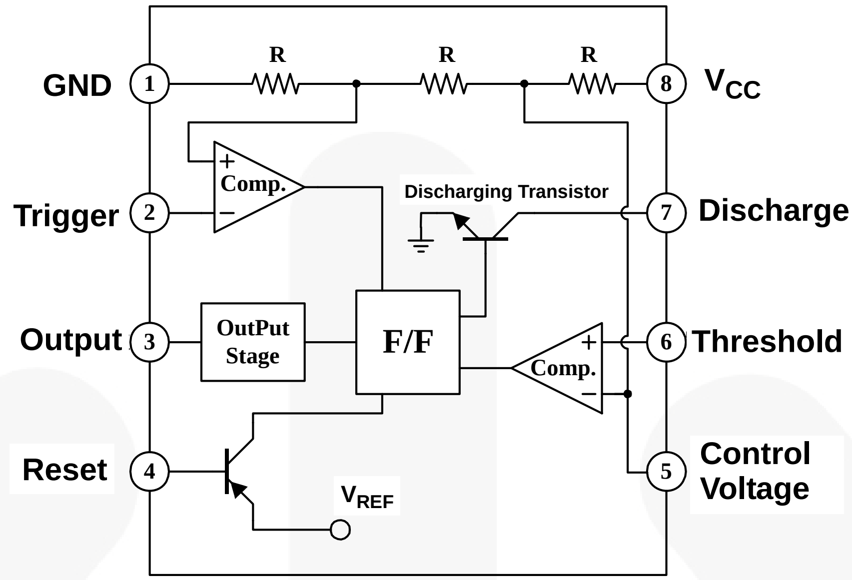


Figure 1. Block Diagram

Absolute Maximum Ratings

Stresses exceeding the absolute maximum ratings may damage the device. The device may not function or be operable above the recommended operating conditions and stressing the parts to these levels is not recommended. In addition, extended exposure to stresses above the recommended operating conditions may affect device reliability. The absolute maximum ratings are stress ratings only. Values are at T_A = 25°C unless otherwise noted.

Symbol	Parameter	Value	Unit
V_{CC}	Supply Voltage	16	V
T_{LEAD}	Lead Temperature (Soldering 10s)	300	°C
P_D	Power Dissipation	600	mW
T_{OPR}	Operating Temperature Range	0 ~ +70	°C
T_{STG}	Storage Temperature Range	-65 ~ +150	°C

Electrical Characteristics

Values are at T_A = 25°C, V_{CC} = 5 ~ 15 V unless otherwise specified.

Parameter	Symbol	Conditions		Min.	Typ.	Max.	Unit
Supply Voltage	V_{CC}			4.5		16.0	V
Supply Current (Low Stable) (1)	I_{CC}	V_{CC} = 5 V, $R_L = \infty$			3	6	mA
		V_{CC} = 15 V, $R_L = \infty$			7.5	15.0	mA
Timing Error (Monostable) Initial Accuracy (2)	ACCUR	R_A = 1 kΩ to100 kΩ C = 0.1 µF			1.0	3.0	%
Drift with Temperature (3)	$\Delta t / \Delta T$				50		ppm / °C
Drift with Supply Voltage (3)	$\Delta t / \Delta V_{CC}$				0.1	0.5	% / V
Timing Error (Astable) InItial Accuracy (2)	ACCUR	R_A = 1 kΩ to 100kΩ C = 0.1 µF			2.25		%
Drift with Temperature (3)	$\Delta t / \Delta T$				150		ppm / °C
Drift with Supply Voltage (3)	$\Delta t / \Delta V_{CC}$				0.3		% / V
Control Voltage	V_C	V_{CC} = 15 V		9.0	10.0	11.0	V
		V_{CC} = 5 V		2.60	3.33	4.00	V
Threshold Voltage	V_{TH}	V_{CC} = 15 V			10.0		V
		V_{CC} = 5V			3.33		V
Threshold Current (4)	I_{TH}				0.10	0.25	µA
Trigger Voltage	V_{TR}	V_{CC} = 5 V		1.10	1.67	2.20	V
		V_{CC} = 15 V		4.5	5.0	5.6	V
Trigger Current	I_{TR}	V_{TR} = 0 V			0.01	2.00	µA
Reset Voltage	V_{RST}			0.4	0.7	1.0	V
Reset Current	I_{RST}				0.1	0.4	mA
Low Output Voltage	V_{OL}	V_{CC} = 15 V	I_{SINK} = 10 mA		0.06	0.25	V
			I_{SINK} = 50 mA		0.30	0.75	V
		V_{CC} = 5 V, I_{SINK} = 5 mA			0.05	0.35	V
High Output Voltage	V_{OH}	V_{CC} = 15 V	I_{SOURCE} = 200 mA		12.5		V
			I_{SOURCE} = 100 mA	12.75	13.30		V
		V_{CC} = 5 V, I_{SOURCE} = 100 mA		2.75	3.30		V
Rise Time of Output(3)	t_R				100		ns
Fall Time of Output(3)	t_F				100		ns
Discharge Leakage Current	I_{LKG}				20	100	nA

Notes:

1. When the output is high, the supply current is typically 1 mA less than at V_{CC} = 5 V.
2. Tested at V_{CC} = 5.0 V and V_{CC} = 15 V.
3. These parameters, although guaranteed, are not 100% tested in production.
4. This determines the maximum value of $R_A + R_B$ for 15 V operation, the maximum total R = 20 MΩ, and for 5 V operation, the maximum total R = 6.7 MΩ.

Application Information

Table 1 below is the basic operating table of 555 timer.

Table 1. Basic Operating Table

Reset (PIN 4)	V_{TR} (PIN 2)	V_{TH} (PIN 6)	Output (PIN 3)	Discharging Transistor (PIN 7)
Low	X	X	Low	ON
High	< 1/3 V_{CC}	X	High	OFF
High	> 1/3 V_{CC}	> 2/3 V_{CC}	Low	ON
High	> 1/3 V_{CC}	< 2/3 V_{CC}	Previous State	

When the low signal input is applied to the reset terminal, the timer output remains low regardless of the threshold voltage or the trigger voltage. Only when the high signal is applied to the reset terminal, the timer's output changes according to threshold voltage and trigger voltage.

When the threshold voltage exceeds 2/3 of the supply voltage while the timer output is high, the timer's internal discharge transistor turns on, lowering the threshold voltage to below 1/3 of the supply voltage. During this time, the timer output is maintained low. Later, if a low signal is applied to the trigger voltage so that it becomes 1/3 of the supply voltage, the timer's internal discharge transistor turns off, increasing the threshold voltage and driving the timer output again at high.

1. Monostable Operation

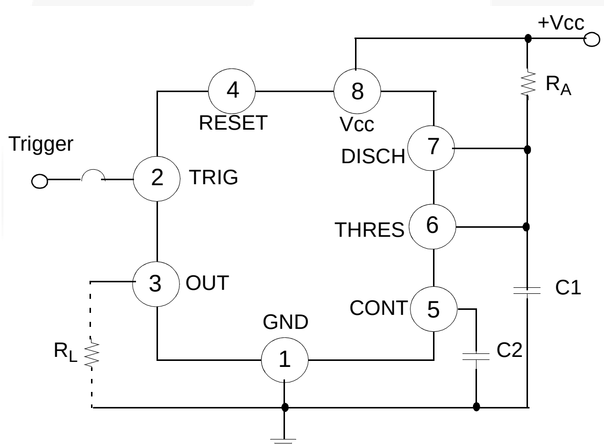


Figure2. Monostable Circuit

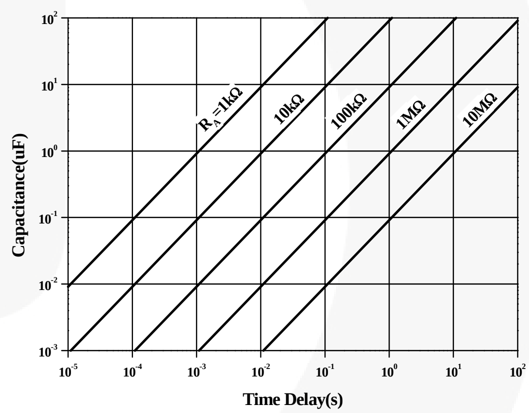


Figure 3. Resistance and Capacitance vs. Time Delay (t_D)

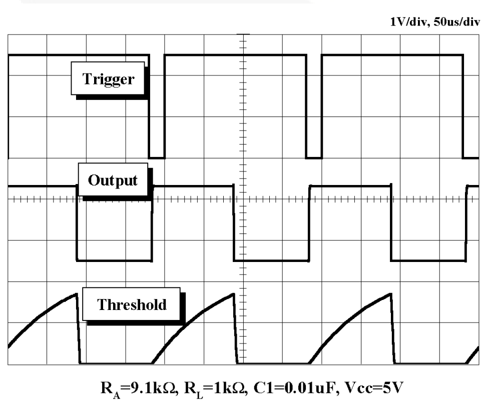


Figure 4. Waveforms of Monostable Operation

1. Monostable Operation

Figure 2 illustrates a monostable circuit. In this mode, the timer generates a fixed pulse whenever the trigger voltage falls below $V_{CC}/3$. When the trigger pulse voltage applied to the #2 pin falls below $V_{CC}/3$ while the timer output is low, the timer's internal flip-flop turns the discharging transistor off and causes the timer output to become high by charging the external capacitor C1 and setting the flip-flop output at the same time.

The voltage across the external capacitor C1, V_{C1} increases exponentially with the time constant $t = R_A{*}C$ and reaches $2\,V_{CC}/3$ at $t_D = 1.1\,R_A{*}C$. Hence, capacitor C1 is charged through resistor R_A. The greater the time constant R_AC, the longer it takes for the V_{C1} to reach $2\,V_{CC}/3$. In other words, the time constant R_AC controls the output pulse width.

When the applied voltage to the capacitor C1 reaches $2\,V_{CC}/3$, the comparator on the trigger terminal resets the flip-flop, turning the discharging transistor on. At this time, C1 begins to discharge and the timer output converts to low.
In this way, the timer operating in the monostable repeats the above process. Figure 3 shows the time constant relationship based on R_A and C. Figure 4 shows the general waveforms during the monostable operation.

It must be noted that, for a normal operation, the trigger pulse voltage needs to maintain a minimum of $V_{CC}/3$ before the timer output turns low. That is, although the output remains unaffected even if a different trigger pulse is applied while the output is high, it may be affected and the waveform does not operate properly if the trigger pulse voltage at the end of the output pulse remains at below $V_{CC}/3$. Figure 5 shows such a timer output abnormality.

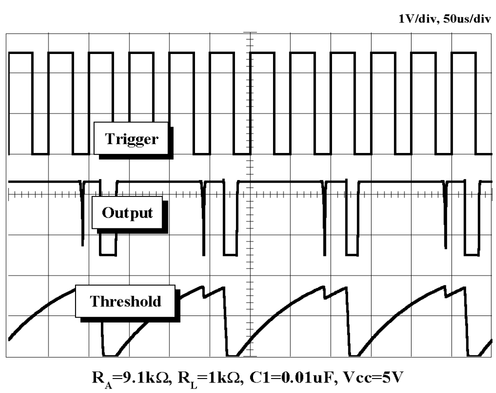


Figure 5. Waveforms of Monostable Operation (abnormal)

2. Astable Operation

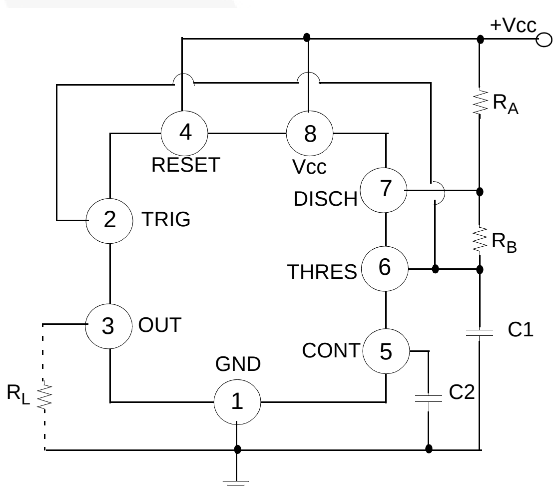


Figure 6. A Stable Circuit

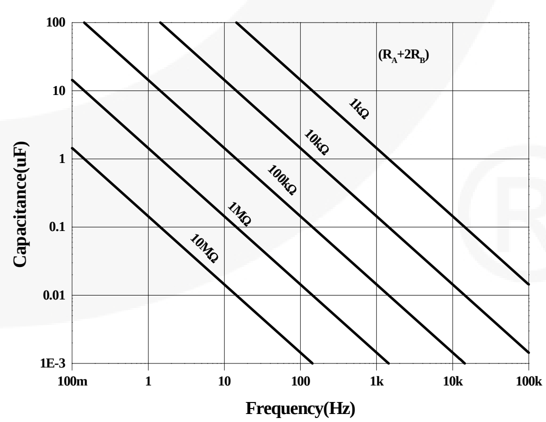
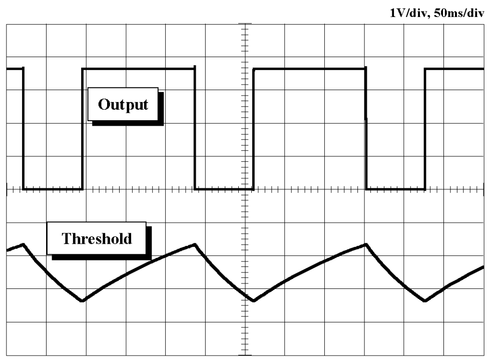


Figure 7. Capacitance and Resistance vs. Frequency



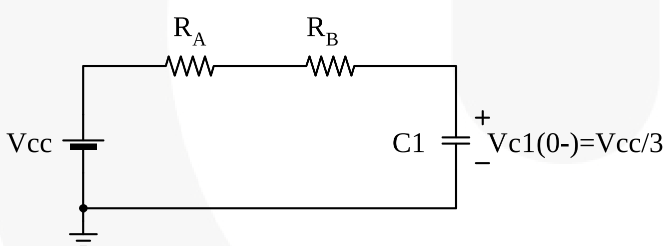
R_A=1kΩ, R_B=1kΩ, R_L=1kW, C1=1uF, Vcc=5V

Figure 8. Waveforms of Astable Operation

An astable timer operation is achieved by adding resistor R_B to Figure 2 and configuring as shown on Figure 6. In the astable operation, the trigger terminal and the threshold terminal are connected so that a self-trigger is formed, operating as a multi-vibrator. When the timer output is high, its internal discharging transistor. turns off and the V_{C1} increases by exponential function with the time constant (R_A+R_B)*C.

When the V_{C1}, or the threshold voltage, reaches 2 V_{CC}/3; the comparator output on the trigger terminal becomes high, resetting the F/F and causing the timer output to become low. This turns on the discharging transistor and the C1 discharges through the discharging channel formed by R_B and the discharging transistor. When the V_{C1} falls below V_{CC}/3, the comparator output on the trigger terminal becomes high and the timer output becomes high again. The discharging transistor turns off and the V_{C1} rises again.

In the above process, the section where the timer output is high is the time it takes for the V_{C1} to rise from V_{CC}/3 to 2 V_{CC}/3, and the section where the timer output is low is the time it takes for the VC1 to drop from 2 V_{CC}/3 to V_{CC}/3. When timer output is high, the equivalent circuit for charging capacitor C1 is as follows:



$$C_1\frac{dv_{c1}}{dt} = \frac{V_{cc} - V(0\text{-})}{R_A + R_B} \qquad (1)$$

$$V_{C1}(0+) = V_{CC}/3 \qquad (2)$$

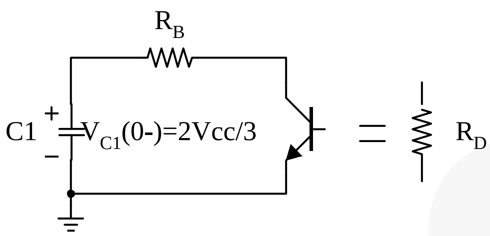
$$V_{C1}(t) = V_{CC}\left(1 - \frac{2}{3}e^{-\left(-\frac{t}{(R_A + R_B)C1}\right)}\right) \qquad (3)$$

Since the duration of the timer output high state (t_L) is the amount of time it takes for the V_{C1}(t) to reach 2 V_{CC}/3,

$$V_{C1}(t) = \frac{2}{3}V_{CC} = V_{CC}\left(1 - \frac{2}{3}e^{-\left(-\frac{t_H}{(R_A + R_B)C1}\right)}\right) \qquad (4)$$

$$t_H = C_1(R_A + R_B)\ln 2 = 0.693(R_A + R_B)C_1 \qquad (5)$$

The equivalent circuit for discharging capacitor C1, when timer output is low is, as follows:



$$C_1 \frac{dv_{C1}}{dt} + \frac{1}{R_A + R_B} V_{C1} = 0 \qquad (6)$$

$$V_{C1}(t) = \frac{2}{3} V_{CC} e^{-\frac{t}{(R_A + R_D)C1}} \qquad (7)$$

Since the duration of the timer output low state (t_L) is the amount of time it takes for the VC1(t) to reach $V_{CC}/3$,

$$\frac{1}{3} V_{CC} = \frac{2}{3} V_{CC} e^{-\frac{t_L}{(R_A + R_D)C1}} \qquad (8)$$

$$t_L = C_1 (R_B + R_D) \mathrm{In} 2 = 0.693 (R_B + R_D) C_1 \qquad (9)$$

Since R_D is normally $R_B >> R_D$ although related to the size of discharging transistor,

$t_L = 0.693 R_B C_1$ (10)

Consquently, if the timer operates in astable, the period is the same with
'$t = t_H + t_L = 0.693(RA + R_B)C_1 + 0.693 R_B C_1 = 0.693(R_A + 2R_B)C_1$'
because the period is the sum of the charge time and discharge time. Since frequency is the reciprocal of the period, the following applies:

$$\text{frequency,} \quad f = \frac{1}{t} = \frac{1.44}{(R_A + 2R_B)C_1} \qquad (11)$$

3. Frequency Divider

By adjusting the length of the timing cycle, the basic circuit of Figure 1 can be made to operate as a frequency divider. Figure 9. illustrates a divide-by-three circuit that makes use of the fact that retriggering cannot occur during the timing cycle.

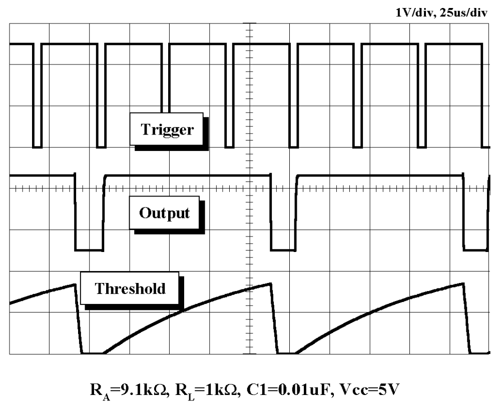


Figure 9. Waveforms of Frequency Divider Operation

4. Pulse Width Modulation

The timer output waveform may be changed by modulating the control voltage applied to the timer's pin 5 and changing the reference of the timer's internal comparators. Figure 10 illustrates the pulse width modulation circuit.
When the continuous trigger pulse train is applied in the monostable mode, the timer output width is modulated according to the signal applied to the control terminal. Sine wave, as well as other waveforms, may be applied as a signal to the control terminal. Figure 11 shows the example of pulse width modulation waveform.

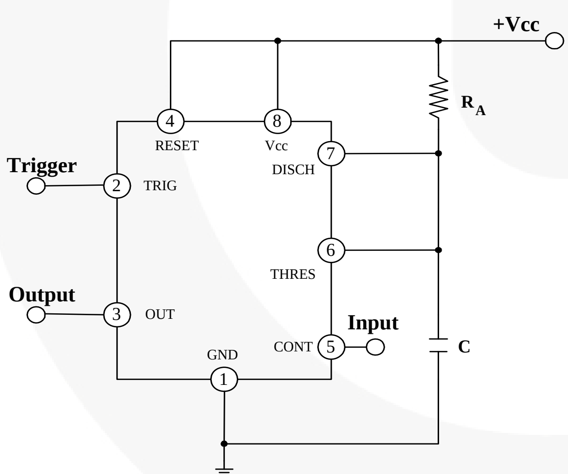


Figure 10. Circuit for Pulse Width Modulation

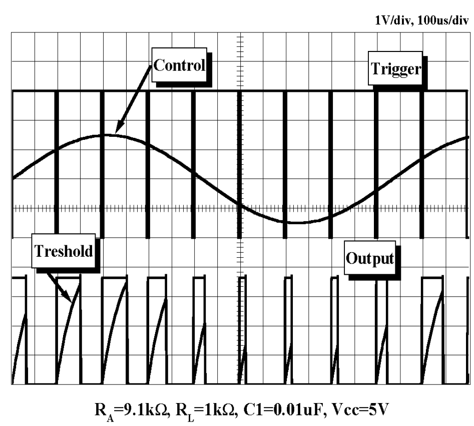


Figure 11. Waveforms of Pulse Width Modulation

5. Pulse Position Modulation

If the modulating signal is applied to the control terminal while the timer is connected for the astable operation, as in Figure 12, the timer becomes a pulse position modulator.
In the pulse position modulator, the reference of the timer's internal comparators is modulated, which modulates the timer output according to the modulation signal applied to the control terminal.
Figure 13 illustrates a sine wave for modulation signal and the resulting output pulse position modulation; however, any wave shape be used.

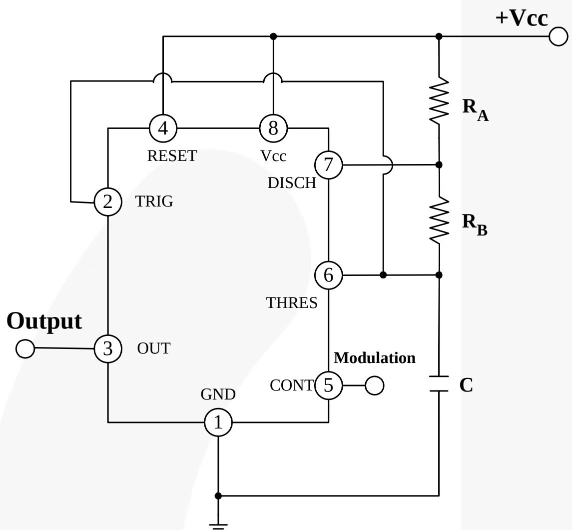


Figure 12. Circuit for Pulse Position Modluation

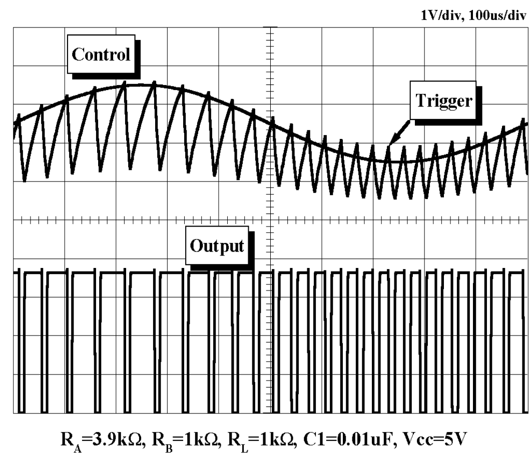


Figure 13. Wafeforms of pulse position modulation

6. Linear Ramp

When the pull-up resistor RA in the monostable circuit shown in Figure 2 is replaced with constant current source, the V_{C1} increases linearly, generating a linear ramp. Figure 14 shows the linear ramp generating circuit and Figure 15 illustrates the generated linear ramp waveforms.

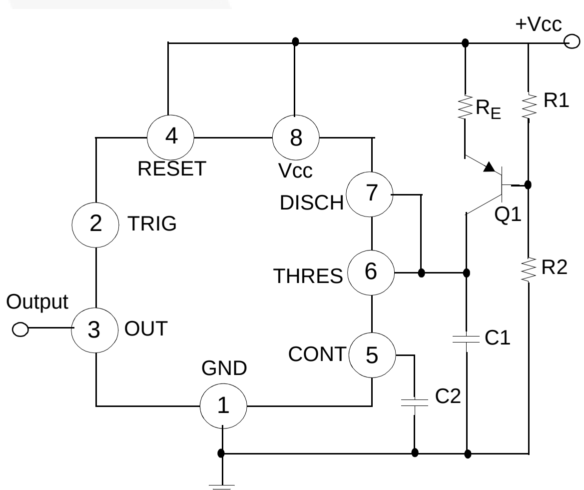


Figure 14. Circuit for Linear Ramp

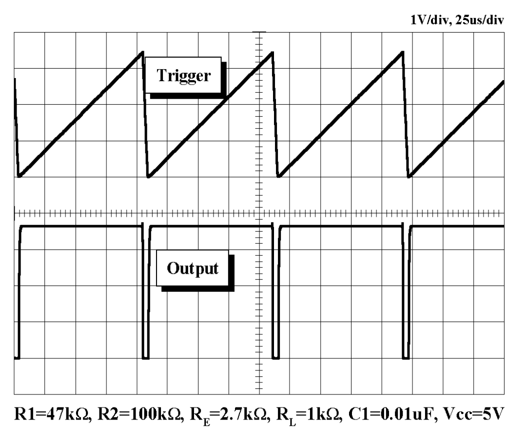


Figure 15. Waveforms of Linear Ramp

In Figure 14, current source is created by PNP transistor Q1 and resistor R1, R2, and R_E.

$$I_C = \frac{V_{CC} - V_E}{R_E} \qquad (12)$$

Here, V_E is

$$V_E = V_{BE} + \frac{R_2}{R_1 + R_2} V_{CC} \qquad (13)$$

For example, if V_{CC} = 15 V, R_E = 20 kΩ, R1 = 5 kΩ, R2 = 10 kΩ, and V_{BE} = 0.7 V,
V_E=0.7 V+10 V=10.7 V, and
I_C=(15-10.7) / 20 k=0.215 mA.

When the trigger starts in a timer configured as shown in Figure 14, the current flowing through capacitor C1 becomes a constant current generated by PNP transistor and resistors.
Hence, the V_C is a linear ramp function as shown in Figure 15. The gradient S of the linear ramp function is defined as follows:

$$S = \frac{V_{p-p}}{t} \qquad (14)$$

Here the Vp-p is the peak-to-peak voltage.
If the electric charge amount accumulated in the capacitor is divided by the capacitance, the V_C comes out as follows:

V = Q/C (15)

The above equation divided on both sides by t gives:

$$\frac{V}{t} = \frac{Q \S t}{C} \qquad (16)$$

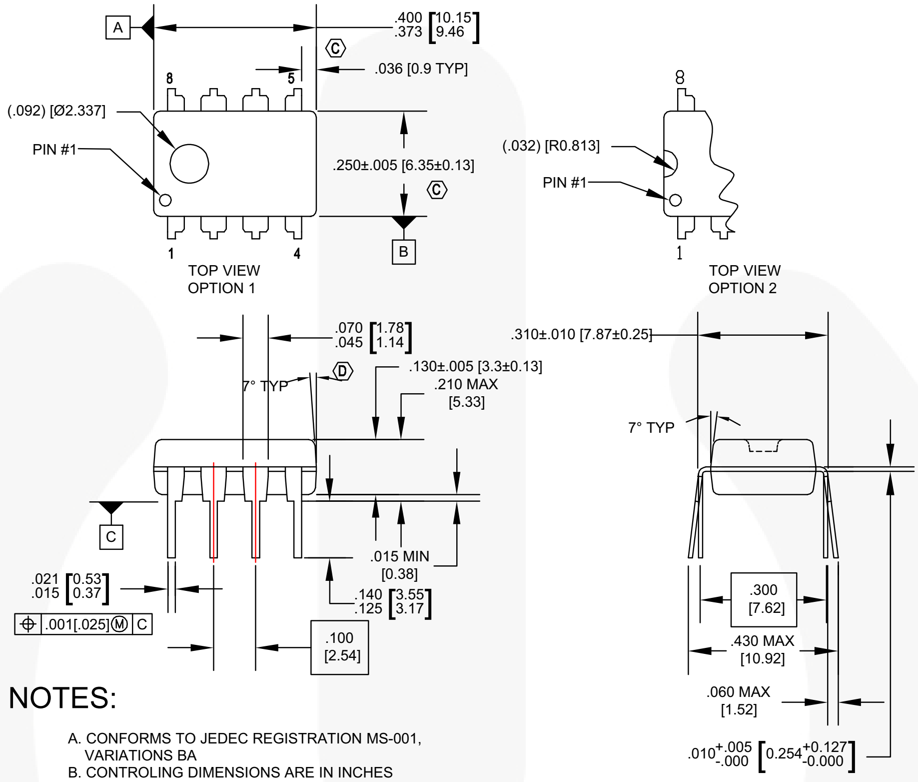
and may be simplified into the following equation:

S = I/C (17)

In other words, the gradient of the linear ramp function appearing across the capacitor can be obtained by using the constant current flowing through the capacitor.
If the constant current flow through the capacitor is 0.215 mA and the capacitance is 0.02 µF, the gradient of the ramp function at both ends of the capacitor is S = 0.215 m / 0.022 µ = 9.77 V/ms.

Physical Dimensions

8-DIP



NOTES:

A. CONFORMS TO JEDEC REGISTRATION MS-001, VARIATIONS BA
B. CONTROLING DIMENSIONS ARE IN INCHES REFERENCE DIMENSIONS ARE IN MILLIMETERS
C. DOES NOT INCLUDE MOLD FLASH OR PROTRUSIONS. MOLD FLASH OR PROTRUSIONS SHALL NOT EXCEED .010 INCHES OR 0.25MM.
D. DOES NOT INCLUDE DAMBAR PROTRUSIONS. DAMBAR PROTRUSIONS SHALL NOT EXCEED .010 INCHES OR 0.25MM.
E. DIMENSIONING AND TOLERANCING PER ASME Y14.5M-1994.

N08EREVG

Figure 16. 8-Lead, DIP, JEDEC MS-001, 300" WIDE

Package drawings are provided as a service to customers considering Fairchild components. Drawings may change in any manner without notice. Please note the revision and/or date on the drawing and contact a Fairchild Semiconductor representative to verify or obtain the most recent revision. Package specifications do not expand the terms of Fairchild's worldwide terms and conditions, specifically the warranty therein, which covers Fairchild products.

Always visit Fairchild Semiconductor's online packaging area for the most recent package drawings: *http://www.fairchildsemi.com/packaging/*.

For current tape and reel specifications, visit Fairchild Semiconductor's online packaging area: *http://www.fairchildsemi.com/products/discrete/pdf/8dip_tr.pdf*.

Physical Dimensions (continued)

8-SOIC

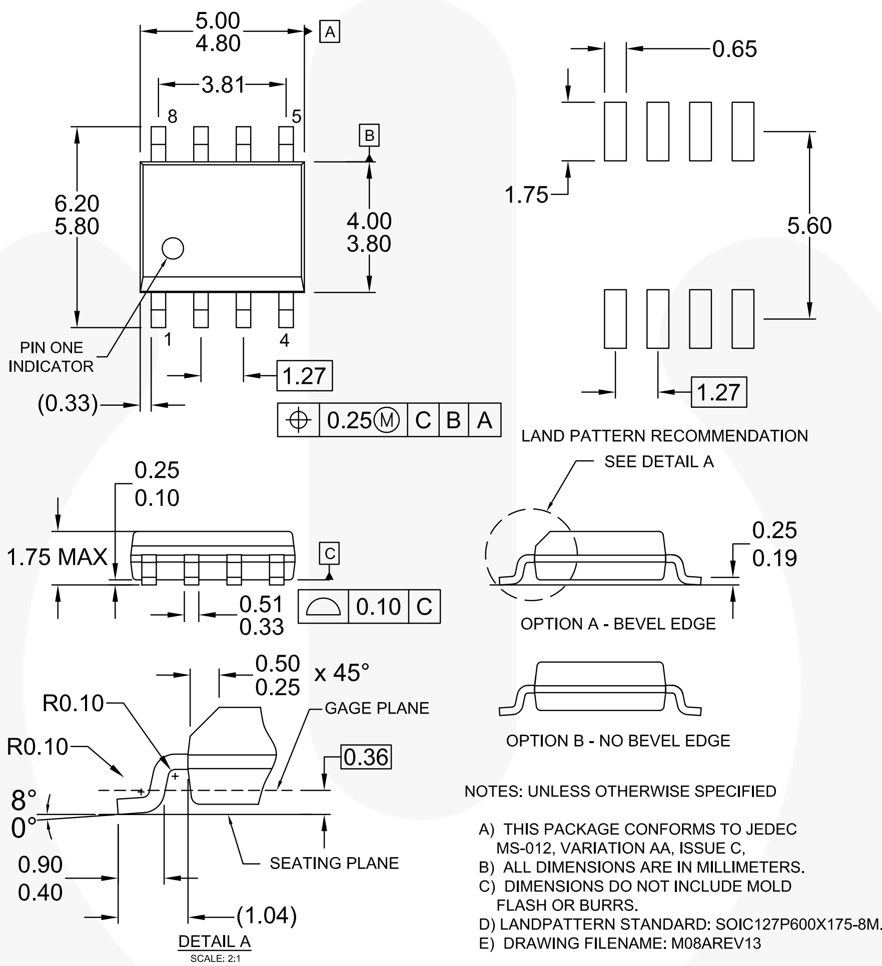


Figure 17. 8-Lead, SOIC,JEDEC MS-012, 150" NARROW BODY

Package drawings are provided as a service to customers considering Fairchild components. Drawings may change in any manner without notice. Please note the revision and/or date on the drawing and contact a Fairchild Semiconductor representative to verify or obtain the most recent revision. Package specifications do not expand the terms of Fairchild's worldwide terms and conditions, specifically the warranty therein, which covers Fairchild products.

Always visit Fairchild Semiconductor's online packaging area for the most recent package drawings:
http://www.fairchildsemi.com/packaging/.

For current tape and reel specifications, visit Fairchild Semiconductor's online packaging area:
http://www.fairchildsemi.com/dwg/M0/M08A.pdf.

FAIRCHILD SEMICONDUCTOR®

TRADEMARKS

The following includes registered and unregistered trademarks and service marks, owned by Fairchild Semiconductor and/or its global subsidiaries, and is not intended to be an exhaustive list of all such trademarks.

2Cool™
AccuPower™
AX-CAP™*
BitSiC™
Build it Now™
CorePLUS™
CorePOWER™
CROSSVOLT™
CTL™
Current Transfer Logic™
DEUXPEED®
Dual Cool™
EcoSPARK®
EfficientMax™
ESBC™
®
Fairchild®
Fairchild Semiconductor®
FACT Quiet Series™
FACT®
FAST®
FastvCore™
FETBench™
FPS™
F-PFS™
FRFET®
Global Power Resource℠
GreenBridge™
Green FPS™
Green FPS™ e-Series™
G*max*™
GTO™
IntelliMAX™
ISOPLANAR™
Making Small Speakers Sound Louder and Better™
MegaBuck™
MICROCOUPLER™
MicroFET™
MicroPak™
MicroPak2™
MillerDrive™
MotionMax™
mWSaver™
OptoHiT™
OPTOLOGIC®
OPTOPLANAR®
®
PowerTrench®
PowerXS™
Programmable Active Droop™
QFET®
QS™
Quiet Series™
RapidConfigure™
™
Saving our world, 1mW/W/kW at a time™
SignalWise™
SmartMax™
SMART START™
Solutions for Your Success™
SPM®
STEALTH™
SuperFET®
SuperSOT™-3
SuperSOT™-6
SuperSOT™-8
SupreMOS®
SyncFET™
Sync-Lock™
SYSTEM GENERAL®*
The Power Franchise®
the power franchise
TinyBoost™
TinyBuck™
TinyCalc™
TinyLogic®
TINYOPTO™
TinyPower™
TinyPWM™
TinyWire™
TranSiC™
TriFault Detect™
TRUECURRENT®*
μSerDes™
μSerDes™
UHC®
Ultra FRFET™
UniFET™
VCX™
VisualMax™
VoltagePlus™
XS™

* Trademarks of System General Corporation, used under license by Fairchild Semiconductor.

DISCLAIMER

FAIRCHILD SEMICONDUCTOR RESERVES THE RIGHT TO MAKE CHANGES WITHOUT FURTHER NOTICE TO ANY PRODUCTS HEREIN TO IMPROVE RELIABILITY, FUNCTION, OR DESIGN. FAIRCHILD DOES NOT ASSUME ANY LIABILITY ARISING OUT OF THE APPLICATION OR USE OF ANY PRODUCT OR CIRCUIT DESCRIBED HEREIN; NEITHER DOES IT CONVEY ANY LICENSE UNDER ITS PATENT RIGHTS, NOR THE RIGHTS OF OTHERS. THESE SPECIFICATIONS DO NOT EXPAND THE TERMS OF FAIRCHILD'S WORLDWIDE TERMS AND CONDITIONS, SPECIFICALLY THE WARRANTY THEREIN, WHICH COVERS THESE PRODUCTS.

LIFE SUPPORT POLICY

FAIRCHILD'S PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS WITHOUT THE EXPRESS WRITTEN APPROVAL OF FAIRCHILD SEMICONDUCTOR CORPORATION.

As used herein:

1. Life support devices or systems are devices or systems which, (a) are intended for surgical implant into the body or (b) support or sustain life, and (c) whose failure to perform when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in a significant injury of the user.
2. A critical component in any component of a life support, device, or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.

ANTI-COUNTERFEITING POLICY

Fairchild Semiconductor Corporation's Anti-Counterfeiting Policy. Fairchild's Anti-Counterfeiting Policy is also stated on our external website, www.fairchildsemi.com, under Sales Support.

Counterfeiting of semiconductor parts is a growing problem in the industry. All manufacturers of semiconductor products are experiencing counterfeiting of their parts. Customers who inadvertently purchase counterfeit parts experience many problems such as loss of brand reputation, substandard performance, failed applications, and increased cost of production and manufacturing delays. Fairchild is taking strong measures to protect ourselves and our customers from the proliferation of counterfeit parts. Fairchild strongly encourages customers to purchase Fairchild parts either directly from Fairchild or from Authorized Fairchild Distributors who are listed by country on our web page cited above. Products customers buy either from Fairchild directly or from Authorized Fairchild Distributors are genuine parts, have full traceability, meet Fairchild's quality standards for handling and storage and provide access to Fairchild's full range of up-to-date technical and product information. Fairchild and our Authorized Distributors will stand behind all warranties and will appropriately address any warranty issues that may arise. Fairchild will not provide any warranty coverage or other assistance for parts bought from Unauthorized Sources. Fairchild is committed to combat this global problem and encourage our customers to do their part in stopping this practice by buying direct or from authorized distributors.

PRODUCT STATUS DEFINITIONS

Definition of Terms

Datasheet Identification	Product Status	Definition
Advance Information	Formative / In Design	Datasheet contains the design specifications for product development. Specifications may change in any manner without notice.
Preliminary	First Production	Datasheet contains preliminary data; supplementary data will be published at a later date. Fairchild Semiconductor reserves the right to make changes at any time without notice to improve design.
No Identification Needed	Full Production	Datasheet contains final specifications. Fairchild Semiconductor reserves the right to make changes at any time without notice to improve the design.
Obsolete	Not In Production	Datasheet contains specifications on a product that is discontinued by Fairchild Semiconductor. The datasheet is for reference information only.

Rev. I63

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
19521 E. 32nd Pkwy, Aurora, Colorado 80011 USA
Phone: 303−675−2175 or 800−344−3860 Toll Free USA/Canada
Fax: 303−675−2176 or 800−344−3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800−282−9855 Toll Free USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910
Japan Customer Focus Center
Phone: 81−3−5817−1050

ON Semiconductor Website: **www.onsemi.com**

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative