



DEMONSTRATION BOARD FOR SA305EX

DB62

HTTP://WWW.APEXMICROTECH.COM (800) 546-APEX (800) 546-2739

INTRODUCTION

The DB62 is designed to demonstrate the capabilities of the SA305 as a 3 phase brushless DC motor driver IC. The PWM inputs to the SA305 are controlled by an on-board microcontroller. The EVAL49 board is pre-wired for all required and recommended external components. Please refer to applications note 32 in the Apex catalog for guidance in filter component selection. The 1µF ceramic capacitors supplied with the kit are for high frequency bypassing of the VS and VDD supplies (C1, C11, C4 and C7 on the EVAL49 schematic). Two additional 100uF bypass capacitors are provided with the kit for the VS supplies (C2 and C12 on the EVAL49 schematic). Please refer to applications note 30 for help with power supply bypassing and other useful information.

PWM CONTROL IN DB62

BLDC motors are electronically commutated based on the rotor position. The stator windings are energized in a sequence based upon the combination of three Hall sensor signals which provide the rotor position. In the DB62 the SA305 input PWM control is achieved by means of by an on-board Microchip® PIC 18F2331 microcontroller. The sequence table for commutation (obtained from the motor data sheet) is entered in

the program memory of the microcontroller. Hall sensors A, B and C are connected to the IC pins of the Input Capture (IC) module in the PIC18F2331. This module interrupts on every transition on any of the IC pins. Upon interrupt, the interrupt service routine reads all the three inputs and loads the correct sequence for commutation. The following PWM control features are programmed in the MCU provided with the DB62:

1. User Controlled PWM frequency using external potentiometer (VFREQ in EVAL 49 board). The PWM frequency can be varied from 4 KHz to 118 KHz.
2. User Controlled PWM duty cycle using external potentiometer (VDUC in EVAL49 board). The voltage set by the potentiometer will act as a 'Speed Reference'. The duty cycle can be varied from 0 % to 98 %.

The Forward/Reverse button is provided to control the direction of the motor. Depending upon the direction, the corresponding hall sensor commutation sequence is used and the PWM signals are generated accordingly. The START/STOP button can be used to run or stop the motor.

As mentioned earlier, the MCU provided with the kit is pre-programmed for all of the features described above. The hall sensor commutation table that has been used in programming the MCU is shown in table 1.

| Sensor Output | | | Driver Output | | |
|------------------------------|-------|-------|---------------|-----------|-----------|
| HALLA | HALLB | HALLC | ØC (OUTC) | ØB (OUTB) | ØA (OUTA) |
| CCW Rotation Facing Lead End | | | | | |
| 0 | 0 | 1 | X | HI | LO |
| 0 | 1 | 1 | HI | X | LO |
| 0 | 1 | 0 | HI | LO | X |
| 1 | 1 | 0 | X | LO | HI |
| 1 | 0 | 0 | LO | X | HI |
| 1 | 0 | 1 | LO | HI | X |
| CW Rotation Lead End | | | | | |
| 0 | 1 | 0 | LO | HI | X |
| 0 | 1 | 1 | LO | X | HI |
| 0 | 0 | 1 | X | LO | HI |
| 1 | 0 | 1 | HI | LO | X |
| 1 | 0 | 0 | HI | X | LO |
| 1 | 1 | 0 | X | HI | LO |

Table 1. Commutation Table.

The algorithm for the microcontroller is shown in figure 1. The program for Microchip® PIC 18F2331 is available for download on www.apexmicrotech.com.



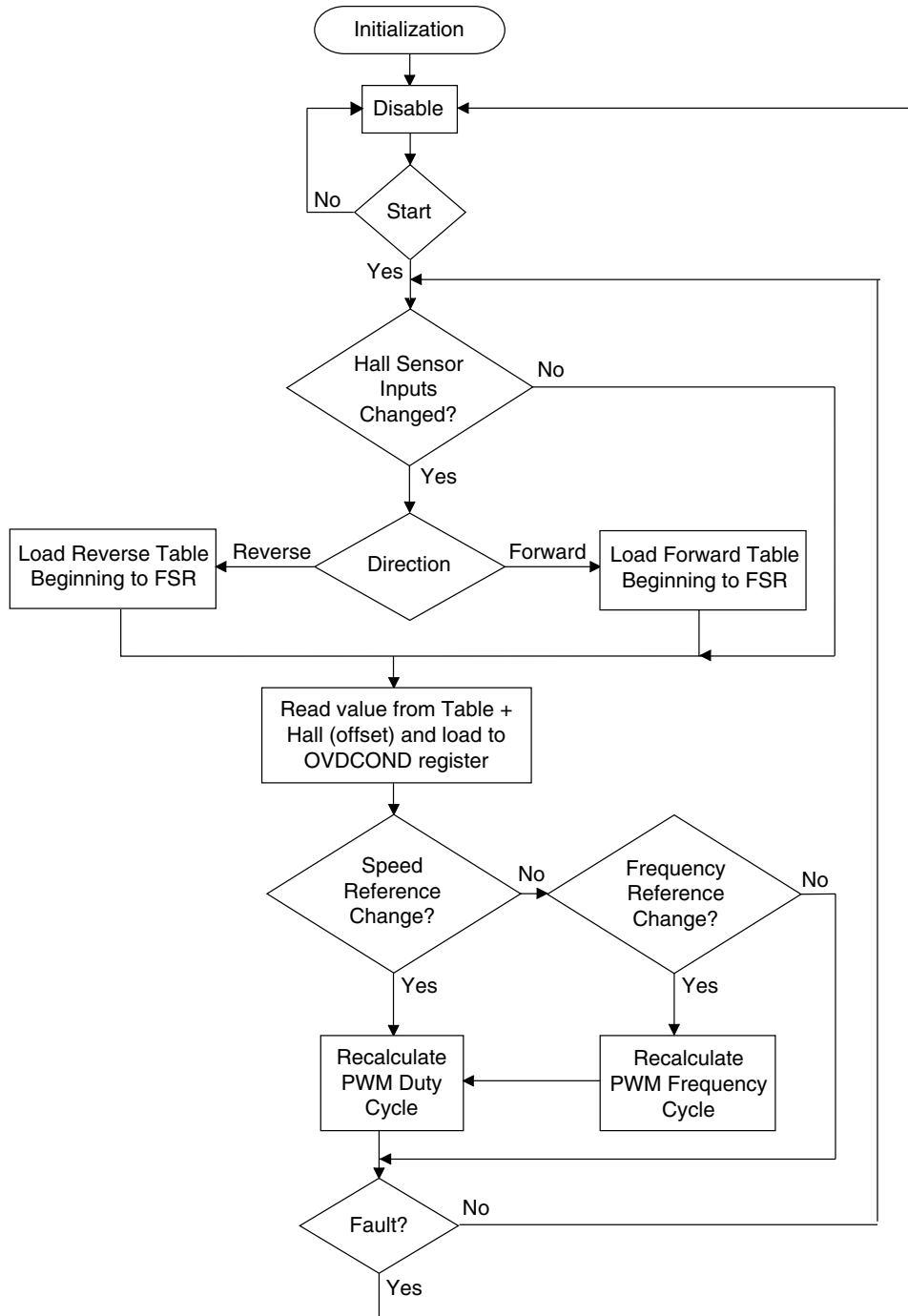


Figure 1: Algorithm for Microchip® PIC 18F2331 MCU as implemented for DB62.

SPECIAL CONSIDERATIONS

- a) 6 schottky diodes D1-D6 (2 for each output provided with DB62) are recommended for use as external flyback diodes because of superior reverse recovery characteristics. Please note that in the DB62, jumpers are needed in order to connect the external schottky diodes (see EVAL49 layout).
- b) External RC snubber circuits (R13 and C5, R16 and C8, R19 and C10 in EVAL49 schematic) can provide noise immunity especially in high load current scenarios. A 100 V 4.7nF capacitor in series with a 50 ohm 5W resistor (provided with kit) is recommended for the snubber to be connected between each output and PGND.

The power dissipated in the snubber components can be estimated by;

$$P = V2 \cdot C \cdot FSW$$

Where

P is the power dissipated

V is the VS supply voltage

C is the snubber capacitor value (15nF)

FSW is the switching frequency.

GROUNDING CONSIDERATIONS

The EVAL49 has three separate ground paths for signal ground (SGND), power ground (PGND) and digital ground (DGND). SGND and PGND are the ground paths for the SA305 while the DGND is for the MCU and related circuitry. These grounds need to be connected on the board by putting a jumper between SGND and DGND and another between PGND and DGND. These jumpers are located in the center of the board on the right of the SA305. Do not connect the grounds at any other points to avoid ground loops and unwanted voltage drops.

PARTS LIST

| Apex Part # | Description, Vendor | Qty |
|----------------------|----------------------------------|-------|
| EVAL49 | PC Board, Apex | 1 |
| OX7R105KWN | 1µF Cap, Novacap | 4 |
| MS06 | Pin Socket | 1 |
| MS11 | Socket Strip | 1 Bag |
| (Black) 151-203-RC | Test Point Mouser | 3 |
| (White) 151-201-RC | Test Point Mouser | 5 |
| 110-13-328-41-001000 | DIP28 Socket Mill-Max | 1 |
| (White) 571-0600 | 4mm single PCB socket Deltron | 16 |
| (Red) 571-0500 | 4mm single PCB socket Deltron | 3 |

| | | |
|------------------|--|---|
| (Black) 571-0100 | 4mm single PCB socket Deltron | 3 |
| ECG-EVQPAC07K | Push Button Switch, Panasonic | 3 |
| 3352W-1-102 | Potentiometer, Bourns | 2 |
| 50SQ100 | Schottky Diodes, International Rectifier | 6 |
| 140-ESRL100V100 | 100uF cap, Xicon | 2 |
| MCU-305 | PIC18F2331-1/SP-ND Microchip | 1 |
| CFRN60D1001FRE6 | 1K Ohm Resistor, (R23,R24.R25) 5% 0.25W | 3 |
| RN60D4701FB14 | 4.7K Ohm Resistor, (R1, R2, R3, R6) 5% 0.25W | 4 |
| CFI-4C1R0J | 1 Ohm Resistor, (R5, R10) 5% 0.25W | 2 |
| RN60D2000FB14 | 200 Ohm Resistor, (R17, R18, R20, R21, R22, R26) 5% 0.25W | 6 |
| WP63GD | Green LED's, (D8-D12) | 5 |
| WP631D | Red LED, (D7) | 1 |

(Optional Components: Refer to SA305U for details)

| Apex Part # | Description, Vendor | Qty |
|----------------|------------------------------|-------|
| HWRE06 | Includes the following: | |
| HS32 | Heatsink, Apex | 1 |
| CLAMP05 | Clip, Apex | 1 |
| TW12 | Thermal Washer, Apex | 1 Box |
| C062K472K2X5CA | 4.7 nF Capacitor, Kemet | 3 |
| 286-51-RC | 50 ohm 5W resistor, Xicon | 3 |

ASSEMBLY

Before starting the assembly, please review the special considerations section to be able to differentiate between required and recommended components. The assembly instructions below cover all the components in the board irrespective of whether they are required or optional. Use the bread boarding area for any additional circuitry (example: pull-up for hall sensors) that might be required.

1. Note that each side of the circuit board is identified as either the “component side” or “bottom side.” The component side is labeled on the silk screen side of the board. Cut the MS11 strip into strips of 12 and 11. Discard the remaining small strip of 7. Insert the two strips into the two rows of holes for the DUT from the component side of the PCB. From the bottom side, solder all cage jacks to the circuit board pads. Be sure that the cage jacks are fully seated before soldering. Be careful that the solder does not flow into the cage jack. Remove the plastic carrier strips.
2. From the component side, solder a jumper between SGND and DGND and another between PGND and DGND.
3. From the component side, solder the surface mount capacitors, C1, C4, C7 and C11 and resistors R5 and R10.
4. From the component side, solder jumpers into place for the following components: R12, R14, R15, R27, R28 and R29.
5. From the component side, solder pin sockets into place for the following components: C5, C8, C10, R8, J1A (2 pin sockets), J1B, J2A, J2B, J3A, J3B and DGND.
6. Study the DIP28 socket and notice the notch on one end. From the component side, insert the socket, making sure that the socket has the same orientation as the silk screen diagram. Solder in the socket from the bottom side.
7. Solder one end of an insulated jumper wire to Pin 11 of the DIP28 socket, and plug the other end into one of the two pin sockets at J1A.
8. From the component side, insert the potentiometers VDUC and VFREQ in the specified orientation and solder from the bottom side. Carefully bend the potentiometers down to a horizontal position.
9. From the component side, insert switches SW1 and SW2 and solder from the bottom side.
10. From the component side, insert White Test Points into TP1, TP2, PWM_HC1, PWM_HB1 and PWM_HA1. Insert Black Test Points into PWM_LC1, PWM_LB1 and PWM_LA1. Solder all test points from the bottom side.
11. From the component side, insert the following components: R1, R2, R3, R6, R7, R11, R17, R20, R26, R22, R21, R18, R23, R24 and R25, and solder from the bottom side.
12. From the component side, insert LED's D7, D9, D12, D11, D10 and D8 and solder from the bottom side.
13. If Hall Effect Sensor pull-up resistors are necessary, install resistors in bread board to the left of the HALL_A, HALL_B and HALL_C sockets.
14. Install pin socket in bread board directly under DGND pin socket.
15. From the component side, insert the 4MM PCB sockets. Note that there are 4 Black sockets, for SGND, DGND,

PGND and OFF. There are 5 Red sockets, for VPWR, VDD2, VDD, FAULT and WARNING. The remaining 10 White sockets are for the remaining Input/Output signals. Solder all sockets from the bottom side.

16. From the component side, insert Schottky diodes D1, D2, D3, D4, D5 and D6. Solder from the bottom side. Do NOT trim off excess lead length at this time.
17. If Hall Sensor pull-up resistors are to be used, at this time, on the bottom side, connect wires from HALL_A, HALL_B, HALL_C and VDD to the appropriate bread-board sockets as determined in Step 13.
18. On the bottom side, connect a wire from OFF PCB connector to pin socket directly under DGND pin socket (installed in Step 14).
19. On the bottom side, connect 16-gauge wire from D4 cathode to AOUT PCB connector. Connect another 16-gauge wire from D6 cathode to COUT PCB connector. Connect a third 16-gauge wire from D5 cathode to BOUT PCB connector. Now, trim off the excess lead length from the Schottky diodes.
20. From the component side, insert the snubber resistors, R13, R16 and R19, and solder from the bottom side. Insert snubber capacitors, C5, C8 and C10 into the previously installed pin sockets.
21. From the component side, insert the electrolytic capacitors, C12 and C2, and solder from the bottom side.
22. Install jumper as desired for J3A and/or J3B. Install jumper from J1A to J1B. If desired, jumper J2A and J2B.
23. Using appropriate hardware, install stand-offs at each corner hole.
24. Clip the SA305 into the HS32 heat sink with the CLAMP05 clip. Carefully insert the part with the heat sink into the MS11 sockets. Bolt the heat sink in from the bottom side of the board using the holes provided on the PCB.

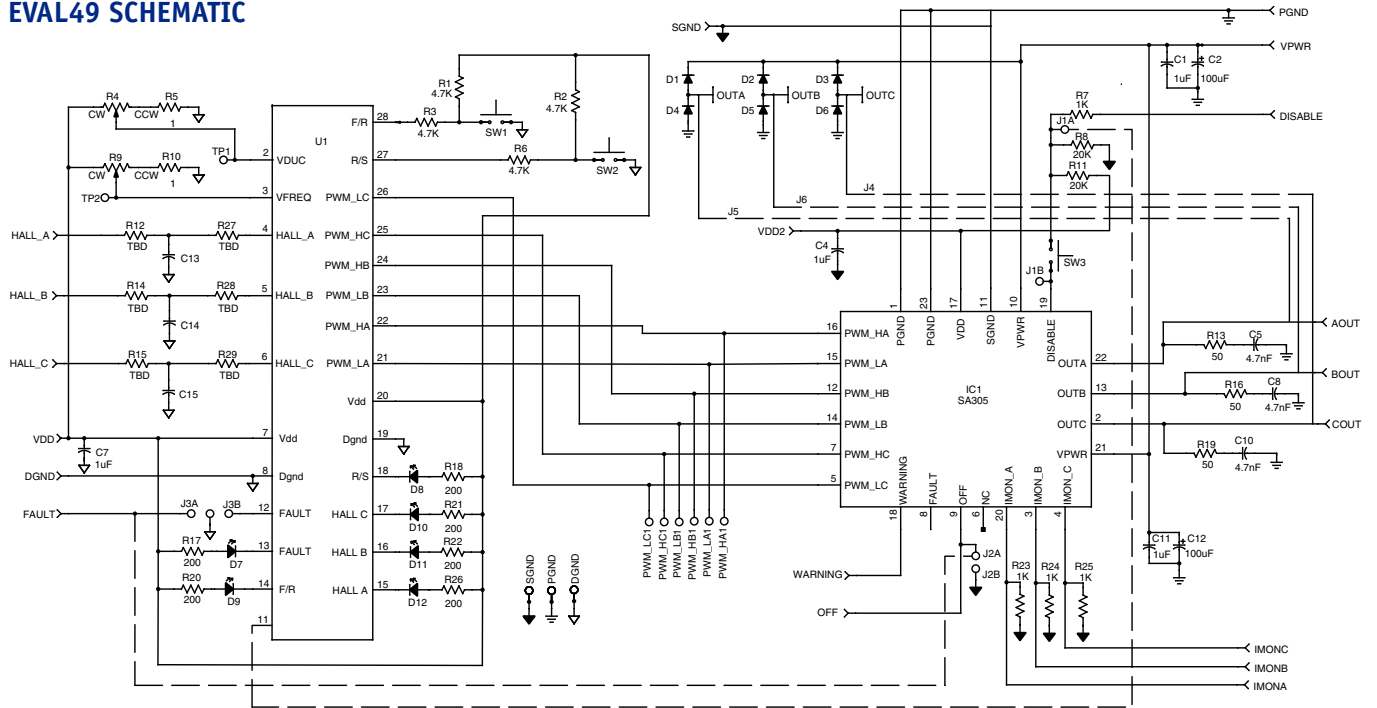
SOFT-START FEATURE:

A soft start algorithm has been developed so that the duty cycle at start-up is low and the motor ramps up to the specified speed reference. The startup duty cycle should be programmed so that the over current limit of the SA305 is not exceeded at start-up. The algorithm enables controlling the rate of acceleration and consequently the maximum current. When the motor is to be reversed the soft-start algorithm is once again initiated, causing the motor to gradually accelerate in the opposite direction. Please refer to application note 45 for details on the soft-start algorithm.

FAULT RESET:

The SA305 requires an external reset pulse at the disable pin to resume normal operation when the part is in a latched fault condition because of Short-circuit, Over Temperature (>160°C) or Over-Current (>10 A). In the DB62 board, there is a provision to reset the part automatically without human intervention. A jumper needs to be connected between J1A and P11 of MCU as shown in the schematic. When a fault occurs, an interrupt is generated which in turn starts generating pulses at P11 to reset the SA305.

EVAL49 SCHEMATIC



EVAL49 LAYOUT

