Demonstration Circuit Board for the HPMX-3002 Driver Amplifier IC

1/03/95 M. R.

Applications Bulletin

Introduction

This board is designed for use with the HPMX-3002 IC which comes in a plastic SO8 package.

There are a few simple guidelines to follow to get maximum performance from the board. When using the board **it is strongly recommended that you do not exceed the maximum IC voltage ratings shown on the data sheets**. For the HPMX-3002, Vcc1 should be set to 4.5 Volts and Vcc2 should be set to 6 Volts.

Assembly Notes:

Table 1 lists the parts required to assemble the circuit board for 900 MHz operation. Use bypass capacitors on both Vcc lines and the power control line. Mount the capacitors as close to the IC as possible. Chip capacitors of 1000 pF or more should be used to ensure adequate low frequency bypassing. Oscillation problems are usually the result of inadequate bypassing.

Table 1: parts list for 900 MHz operation

Qty.	Part description
1	HPMX-3002 circuit board
2	PC mount SMA connectors
3	1000 pF or more chip capacitors
1	27 pF chip capacitor for output block
1	8 nH spring coil inductor (Coilcraft XXX)
1	HPMX-3002 IC

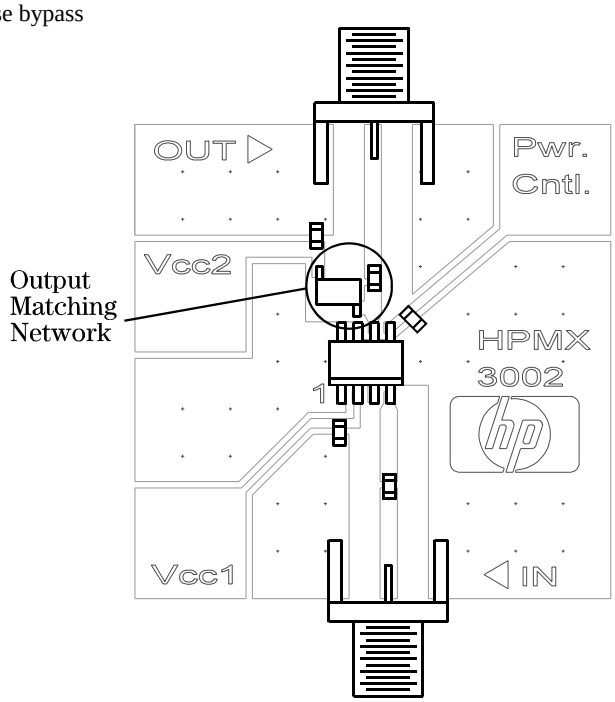

Figure 1. Assembled HPMX-3002 demonstration circuit board

Both the input and output lines require the use of blocking capacitors to isolate the IC from external DC voltages that might cause bias shifts (and resulting performance irregularities) in the IC. The input blocking capacitor should be large enough to have a reactance under 5 ohms at the operating frequency. At 900 MHz, 100 pF is adequate. The output blocking capacitor is part of the circuit that tunes the output to 50 Ω, so its value will depend upon the frequency of operation. At 900 MHz, the optimum output network consists of an 8 nH shunt inductor and a 27 pF series capacitor. Mount them in the positions shown in figure 1.

The board has been designed to accommodate EF Johnson model 142-0701-801 SMA connectors. These connectors are readily available from Newark, Digi-Key and others for about $7 each. The connectors will just slip on to the edge of the board without any drilling. Be sure to solder the ground pins on the connector to the ground-planes on both sides of the board to ensure low ground inductance.

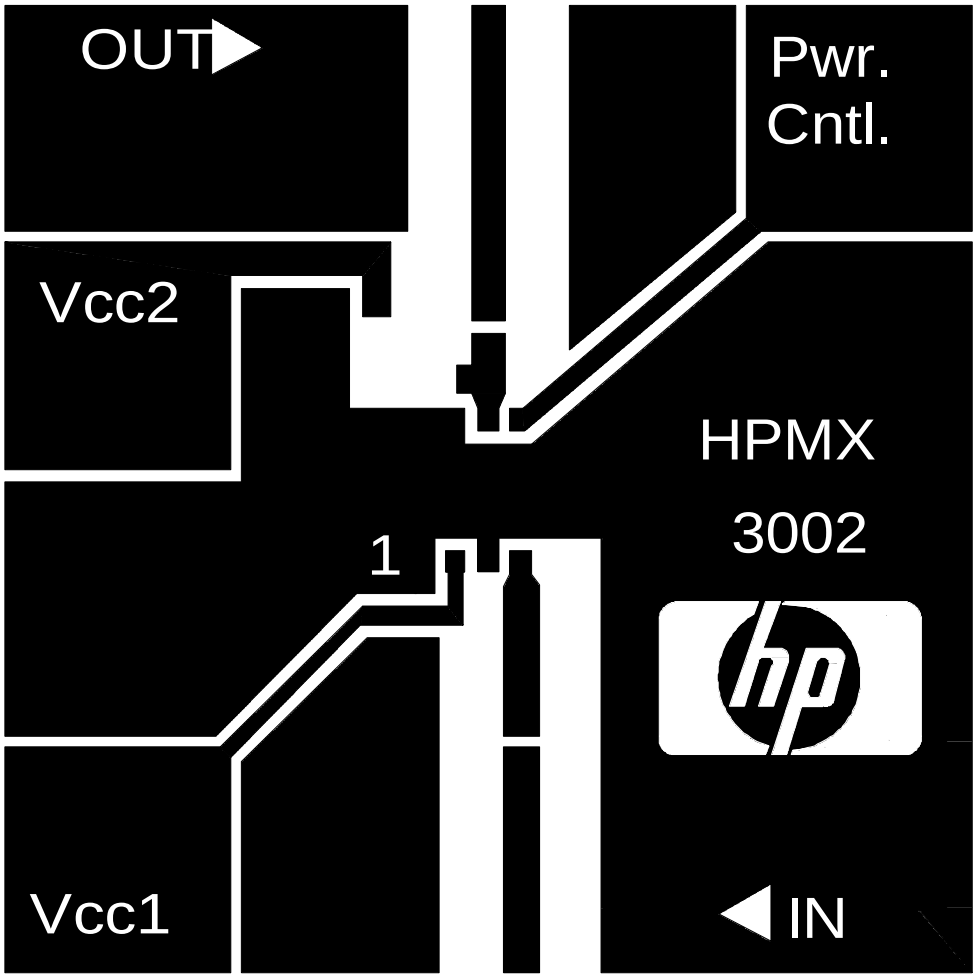

Fig. 2 Scale layout of demonstration circuit board.

Final size of the board: 1.5" x 1.5" x 1/32", FR-4 (dielectric constant ≈ 4.8), copper clad and tin/lead coated on both sides. The back side is an unetched ground plane. Use many plated through via holes in the ground areas on the top side of the board.