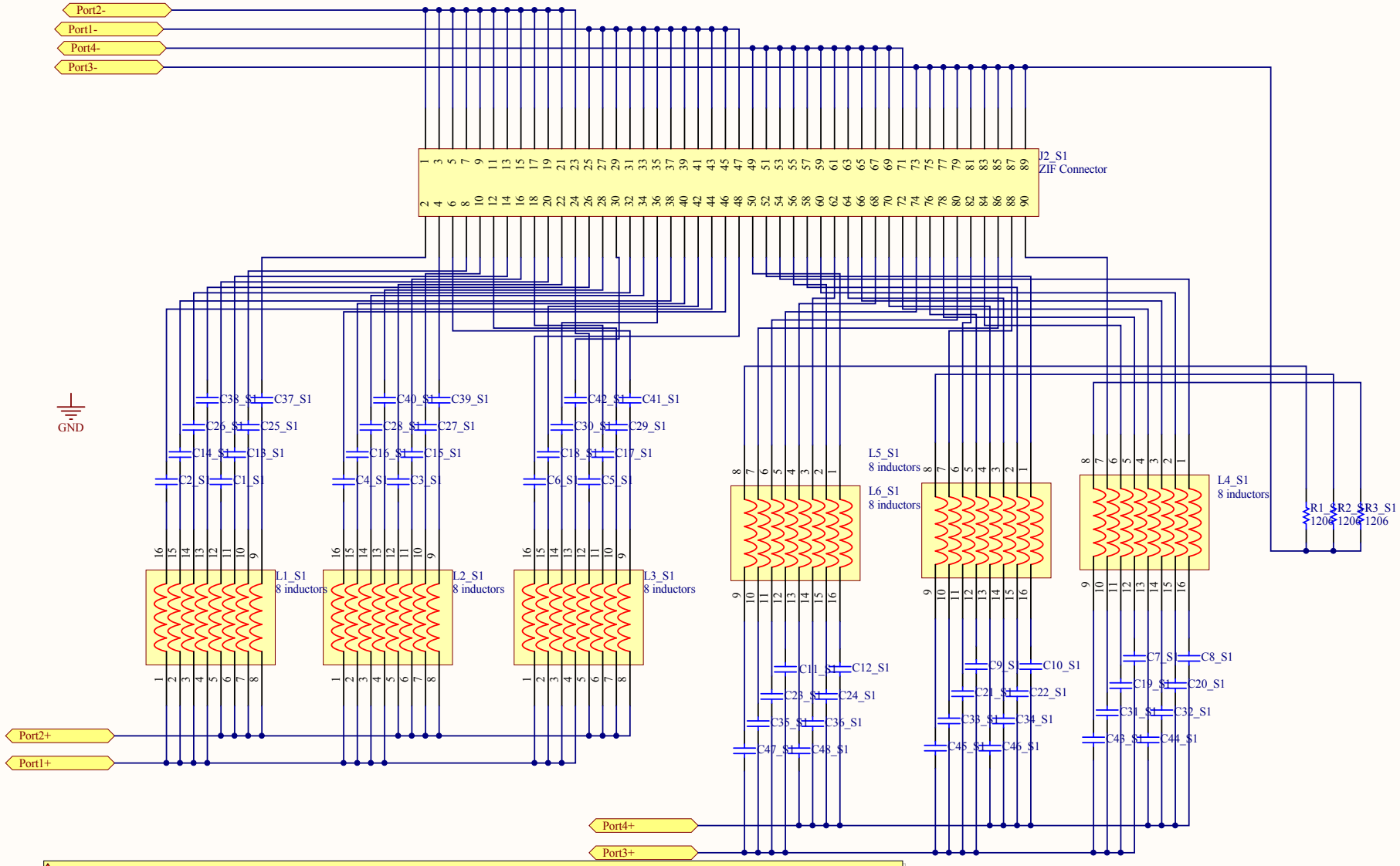
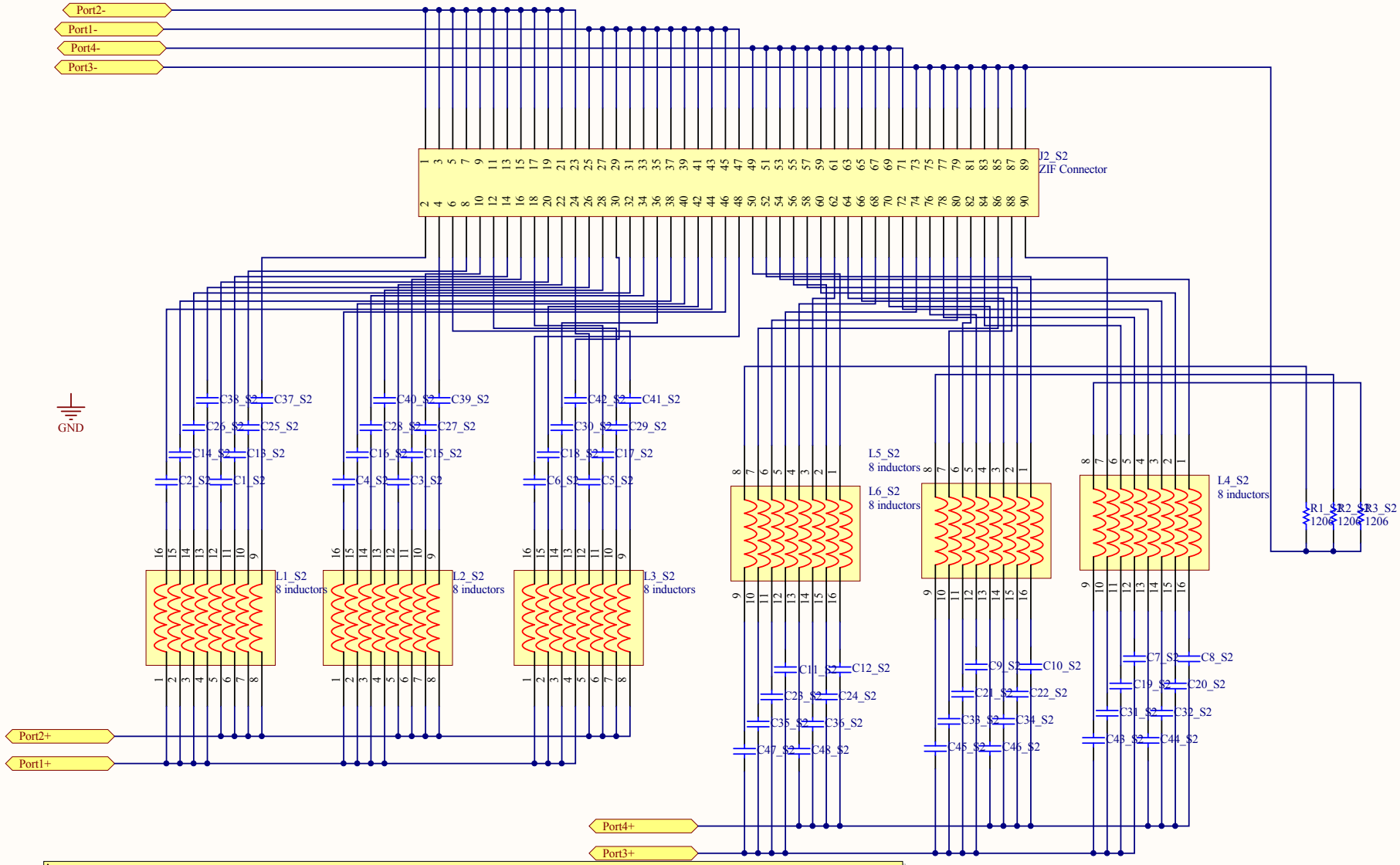


This will be the schematic for the majority of either side of the board. We will need to add ports to this page so signals can go in and out of this schematic page to the top level. On the top level we will have two coppies of this page and the one MicroD connector. Given that the MicroD has 21 pins, at most 10 single signal ports are needed on this page

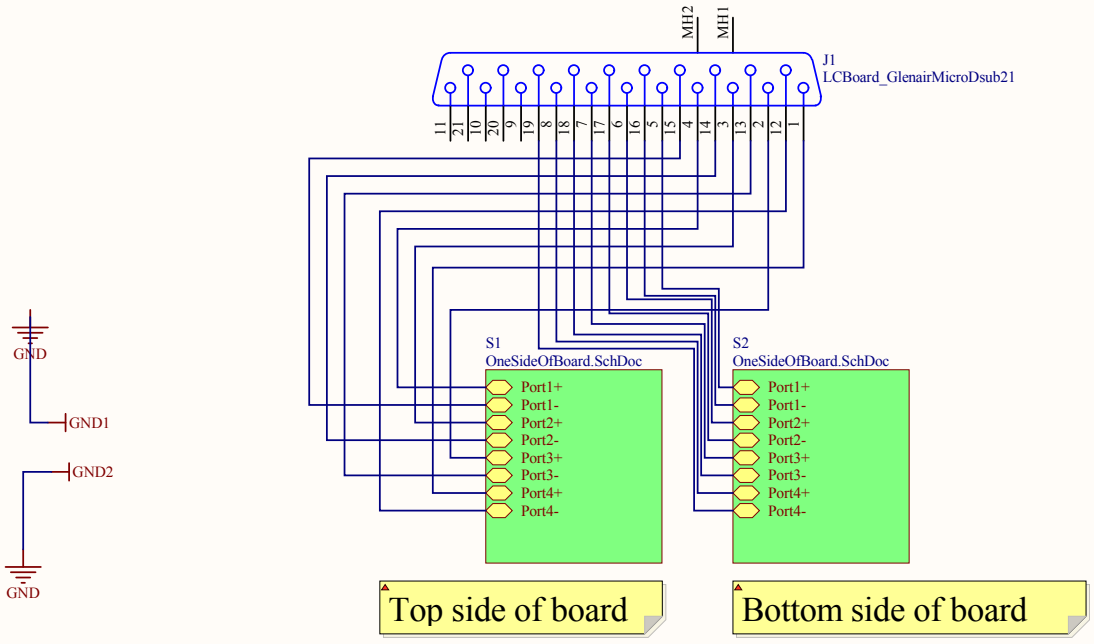
Title		
Size: Letter	Number	Revision
Date: 27/06/2011		Sheet of
File: C:\Users\..\OneSideOfBoard.SchDoc		Drawn By:



This will be the schematic for the majority of either side of the board. We will need to add ports to this page so signals can go in and out of this schematic page to the top level. On the top level we will have two coppies of this page and the one MicroD connector. Given that the MicroD has 21 pins, at most 10 single signal ports are needed on this page

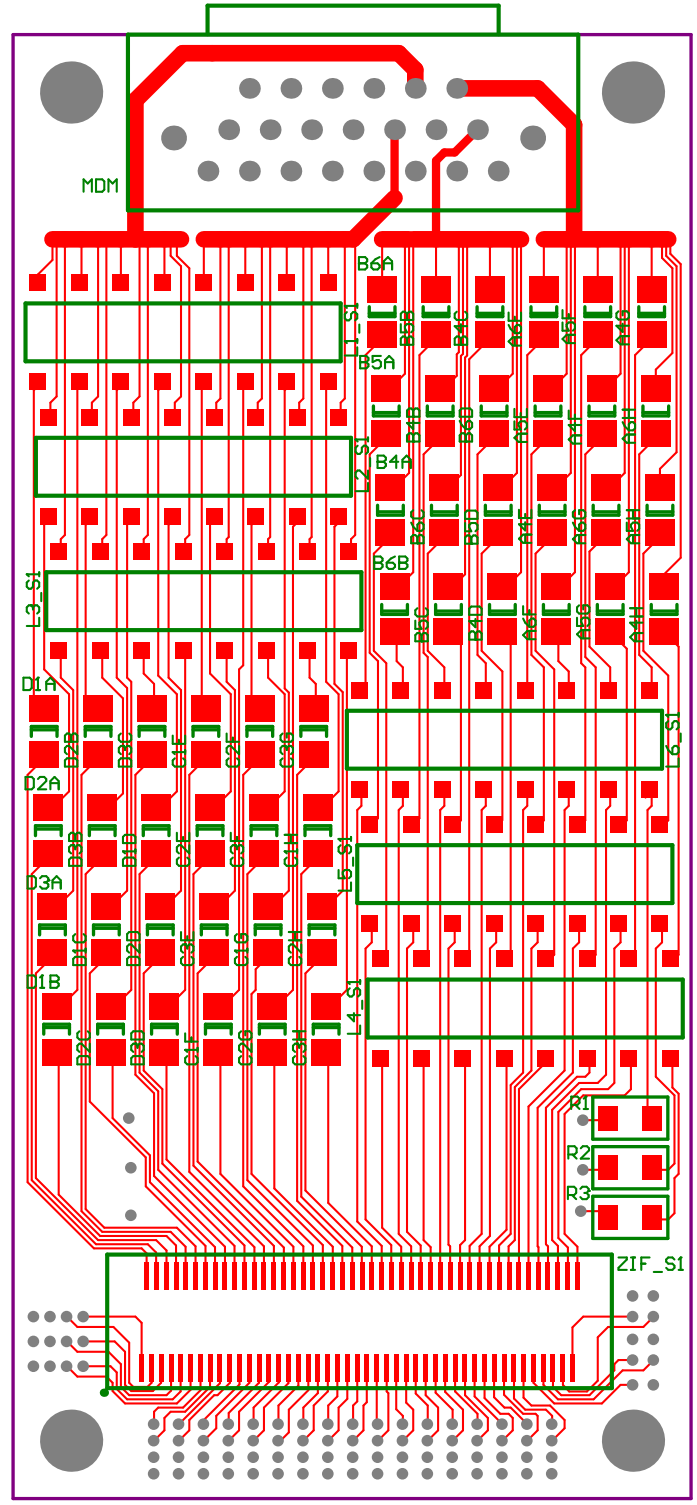
Title		
Size: Letter	Number	Revision
Date: 27/06/2011		Sheet of
File: C:\Users\..\OneSideOfBoard.SchDoc		Drawn By:

6 layer board (2 signal layers each side, and 2 ground planes)



Partioning the design like this into two sides, allows us to use Altiums rooms to save us some time. Only do the PCB layout for one of rooms, we can then copy the room formating and apply it to the other room. You then have two completely identical rooms. Its possible to then flip the other room onto the opposite side of the board.

Title		
Size: Letter	Number	Revision
Date: 27/06/2011		Sheet of
File: C:\Users\..\LCboardRev2.SchDoc		Drawn By:


MDM
B6A
B5A
B4A
B6B
L1_S1
L2_S1
L3_S1
L4_S1
L5_S1
L6_S1
D1A
D2A
D3A
D1B
R1
R2
R3
ZIF_S1