|  |  |  |  |
| --- | --- | --- | --- |
| # | Item | Comment | Answer |
| 1 | C34-C49 | Adding these capacitors here will completely destroy your input impedance. It probably is there either as an EMI or as an ESD filter, but this is far from the best solution. 1/2/3.14/50/4.7e-9 = 678k, that will give an even worse eCMRR as a scope.  I recommend replacing the current capacitors to ground with a optional differential capacitor between the  + and the – ( see figure 80 in the datasheet). I would leave this capacitor NC and only add something if you really have issues.  Also maybe add low input capacitance (<10pF) TVS diodes to ground before the resistors. | Would it be useful to add spark gaps from IN to AGND?  Nope  Is 4.99k as safety resistor sufficient for BF rating?  Would ESD5V3U2U03FH6327XTSA1  work as TVS with common to AGND? |
| 2 | R11/R10 | Recommend adding a capacitor in parallel for stability (simulation or real live test needed to see what value is needed, probably in the pF range) | Done |
| 3 | Bias\_elec | Probably you want a capacitor (100pF or so) here to ground, to ensure high frequency response. | Done |
| 4 | C62 | See #1 | Set to NC and added 0603 to ground before the safety resistor |
| 5 | SJ1 | Since your using asymmetric supply nog AGND always is AVSS. I think you can delete this jumper and just call everything AGND (and make sure they are really nicely tied together). | Removed AVSS and linked all to AGND |
| 6 | KLBR4 | What kind of connector is this going to be? Is it shielded? | Nope ☹  Just a 3.5mm stereo jack to be able to use the available snap leads. |
| 7 | Pin 61 | To be able to make an average reference you need to buffer and low pass filter this pin. I now see there also is an option to use this chip as an average/common reference. You can use SRB 1 for that. Currently that is connected to the buffered common reference. I would make an jumper to switch between the buffered average reference and common reference. | Don’t want to do Average Reference. For common reference it should be usable as this. |
| 8 | REF\_ELEC\_CON | Not connected to anything. Connect this to an connecter and you can make it an common reference amplifier. | It is connected to the shield of X2 |
| 9 | SRB2 | Currently not connected to anything. Thin pin can be connected to any positive input. I think it is best to connect it to the BIAS\_ELEC (PGND) | Leave NC |
| 10 | BIAS\_ELEC | Confusing name. Please rename to PGND | Done |
| 11 | Stimulus-Respons Buffer | Confusing name. Suggestion: Common reference buffer. | Added note |
| 12 | C66, R6, SJ5 | Not needed, will break average reference. Leave BIASOUT NC | BiasIn is now hard connected to PGND, BIAS OUT is not connected and BiasInv is connected to J1 (for future use) |
| 13 | BIASIN | Can be muxed to both positive and negative inputs. Maybe connect to PGND, so any cable can be grounded. | BiasIn is now hard connected to PGND |
| 14 | R4/R9 | Add capacitor in parallel to get a more stable signal | Done |
| 15 | JP26 | Net labels not shown. If not connected use NC. Now they might be connected, but you won’t see it. | Removed the traces, Eagle doesn’t know an NC net class |
| 16 | CN7-CN10 | What’s this? | For size reference |
| 17 | IC1 pin 19,21,22,56,59,54 | Missing bypass capacitors. I would recommend one bypass capacitor for 19,21,22 and 1 for 56,59,54 | Are present |
| 18 | IC pin 48,50 | Missing bypass capacitor | Are present |
| 19 | C30, C29 | AGND and AVSS are the same now, so these are not needed. | Removed |
| 20 | ESD1 – ESD8 | This is not a TVS diode, but a zener diode. A TVS diode is a bidirectional zener (two zeners with reversed polarit in series). In the way it is put on the board now it will start to things at Vf is ~0.7V in one direction. That is too low and will limit the usefull input voltage range. | Planning to use ESD5V3U2U03FH6327XTSA1, symbol in the schematics is not correct. |
| 21 | ESD1 – ESD8 | The ESD5V3U2U03FH6327XTSA1 has the same issue. There still is a zener diode without another zener diode in reverse in series. By the way don't worry to much about having a super low input capacitance; you already have 20Pf input capacitance in your ads1299. |  |

Board:

|  |  |  |  |
| --- | --- | --- | --- |
| # | Item | Comment | Answer |
| 1 | Ground planes | Non existent. You probably want to add planes of copper that are connected to GND/VCC on the top and bottm in between all the other pads. | Added on top and bottom layer. |
| 2 | Silkscreen in Gerber | TOP: “Only one iso…” “BioRobotics… ” “Design..” “Based upon.. ”  BOTTOM: Isolator pin labels, “R2 = SPI…”  are too wide, and significantly different from the Eagle design | 17-6: Fixed:  Changed all fonts from Proportional to Vector. |
| 3 | Assembly editor fail J1, J10, J11  EuroCircuits  53398-1071 - Molex |  |  |
| 4 | Assembly editor fail J2-J9  SJ1-3535NG |  | 17-6: wrong variant, pin spacing is correct, and part will be soldered by hand. |
| 5 | L1 to L4 Footprint | Indicated as 0805, but pads seem rather small? | 17-6: Changed to 0805-WIDE packages |
| 6 | U1 orientation incorrect?  R1SE-0505-R |  | Rotation is 90, should be 0 |
| 7 | U2 orientation incorrect?  MAX14483AAP+ |  | Rotation is 270, should be 180.  Don’t know why this goes wrong. |
| 8 | U3 & U4 orientation incorrect? |  | Rotation is 180, should be 90 |
| 9 | IC1 orientation incorrect?  ADS1299IPAG |  |  |
| 10 | Via ESD2 & ESD3 too close to pad. |  | 17-6: fixed |
| 11 | Via C7, C9, C14, C30 too close to pad |  | 17-6: fixed |
| 12 | Exact part missing for JP20-JP25 | 2.54mm header, high enough for clearance with Nucleo |  |
| 13 | D4 & L1 overlap |  | 17-6: fixed |
| 14 | U2 & C69 & C67 overlap |  | 17-6: fixed |
| 15 | Layer 15 | Way too little copper in this layer. Copper balance might be off. | 17-6: added GND and PE planes to the unused parts |
| 16 | Layer 15 | Why 2 separate copper planes? I would use just one big plane like you have in layer 2 | 17-6: AVDD and DVDD have separate voltages regulators, I cannot combine them. |
| 17 | RISE-0505R | Check footprint, pads seem way too small |  |
| 18 | AM1DM-0505SH60-NZ | Placed on the bottom of the PCB, but are you sure you have space there to place this tall part? | Intended to be on top! Will check..  17-06: in Eagle it is located on TOP, where was indicated it is bottom? |
| 19 | MAX14483 | Keepout area not drawn | 17-06: bKeepout layer seems normal for this part? |
| 20 | general | Please check pcb using PCB visualizer (for example by uploading it to eurocircuits) | 17-06: I already did, those are review points 1 to 14, will do again after processing review feedback. |
|  | Gerbers | Gerbers don't match the view in eagle for some reason. The d of Shield overlaps with refp and the Muijzer lies outside of the board | 17:06: see point 2 above, and fixed by forcing Vector font. |