Hi, Thanks!

That's great. The Gryphon/Jackalope PCB for step one is ready to go to production. It is a 4 layer PCB with all the corrections from the original Gryphon and the follow-on board (Xagdin). Thanks to Dave (yoda) for all his hard work building and testing both boards, especially Xagdin, as the transition to KiCAD was even more painful than we expected. However I think the new Gryphon/Jackalope board is in good shape and just needs to go to production and build and test to shake out any remaining problems.

I will get a cost estimate for 5 PCBs assuming some people would like to join in the build and test. I an *guessing* each PCB will cost no more than $30 each. Also if you want to support the project but don't want to build and test or would like to wait until the design has improved/further stabilized you can also "buy a board" for the Gryphon team or just donate one to another person who will build and test.

Last year when I ordered the last batch of Gryphon/Xagdin boards (5) it cost about $70 plus shipping. I expect it will be higher this time due to 4 layer boards and may have to switch vendors.

Thanks to Dave's CPLD improvements and the addition of the "trampoline" the Gryphon/Jackalope board supports RAM at address 0 which means a Linux is at least theoretically possible. It might be time to start planning for Linux but it will still be a ways off for a full blown support. The board definitely needs more DRAM which comes in Step two of above.