PLEASE READ

Yes I know it's a lot of stuff but please read, it's important I promise

USAGE COMMENTS

This PCB file was generated by the Acheron Setup script Joker template, an automated tool to generate design–ready KiCAD schematic and PCB files for keyboard projects.
It is offered under a no–liability, as–is clause. Please visit http://github.com/AcheronProject/AcheronSetup/joker_template for more information.
The files generated are compatible with KiCAD developmental ("nightly") versions up to sptember 9, 2021 version.
These files should also be accompanied of a librires folder where the used libraries are added as submodules which last commit should point to the remote's HEAD.
Please note that the files need such libraries to fully work.
You can safely delete this message, as long as you keep these comments in mind while using these files.
For requests/issues please submit a issue in the github folder. Do not attempt to contact the developers directly.
We ask that bugs/problems be reported through the issues page too.

NOTES ON TOLERANCES

The manufacturing tolerances used for PCBs in the Acheron Project are a way to uniformize the design settings of the projects. These settings were obtained using mainly three manufacturers: PCBWay, Elecrow and JLCPCB (see their capabilities pages on references [1–3]). These are big players in the asian PCB manufacturing market and the tolerances practiced by them are accepted industry–wide; hence, the values used here should be enough for most manufacturers.
It must also be kept in mind that, since the Acheron Project is heavily DIY–oriented, the tolerance settings used were the easiest or cheapest available. For instance, Elecrow offers "standard PCB" and "advanced PCB" services ([2]), while PCBWay offers "normal difficulty", "medium difficulty" and "high difficulty" processes ([3]) In both manufacturers, the advanced settings have smaller tolerances and better manufacturing features but are significantly more expensive, specially when paired with PCB assembly services. Such settings are meant for more complex projects (high–speed or analog signal processing, finer electronic components, high–density designs etc) and are completely unnecessary for keyboard projects.
Therefore, it must always be understood that a single body of uniform tolerances throughout a wide spectrum of projects is unfeasible due to the very nature and requirements of the different PCB projects. The values suggested here used in these files are only *recommendations* based on experience, having been used in a myriad of projects throughout the Acheron Project and in Gondolindrim's personal and professional projects.
The tolerances used might seem too loose or big, but these are the bare large–scale manufacturable values found through experience using multiple factories.
Adjust the values at your own discretion, but be very mindful of these tolerances as they are imperative for the manufacturing process and feasibility of the final PCB.
The values used divide into two groups: the "factory minimums " and the "recommended minimum" ones.

– "Factory" are the values minimally feasible needed by factories. Different ones will inevitably have different numbers, but through using multiple a common denominator was found. These values should not be used often, but seldomly at the designer's discretion and in ultimate case. Their use will incurr in higher manufacturing fees, larger lead times, more quality check issues (PCBs failing after large production).
– Recommended are the smalles tolerances doable with no fabrication or quality control issues. These can be safely used without incurring in higher costs or major large–scale production issues. There is no particular reason for any of these but experience through usage of many factories. This KiCAD file was set to use the recommended values in its Design Rule Checks and routines; these values were used successfully throughout the Acheron Project and are proven to work.

Keep in mind that these are, after all, minimum values. Always try to stray away from them when there is chance, so as to give you and the factory headroom to work with.
The actually used values can very well vary according to your specifications and the capabilities of the factory being used.
For the actual values, check references [1–3].

TLDR
You should read it all though

– The values used are based on experience but should be manufacturable at low cost and accepted industry–wide
– Such values *should* be enough for keyboard PCBs
– Manufacturing parameters might change in which case the tolerance values will change accordingly
– You can freely change these parameters around but make sure what those changes entail to, cost and production–wise
– If you are beggining on PCB design or are using a new fab, it might be wise to just use the values here for now

NOTES ON COPPER POURS

Many DIY designers will state that the usage of copper pours is perfeccionism; in some cases, designers will argue that the pours are actually detrimental to the design while I (gondolindrim) disagree with the former I agree with the latter in some respects. Ground pours are an integral part of digital high–speed signal design; since most (if not all) modern keyboards work under USB communication which uses differential pair topology, a ground copper pour is absolutely needed to ensure proper return currents paths, low ground impedance, EMI resistance, efficiency in ESD protection, protection from overheating, and so on. Particularly in keyboard PCBs, however, the copper pours make the PCBs stiffer, reducing what is known as "flex". The way to countermeasure that is by deploying flex cuts (also known as relief cuts) or leaf–spring mounting points.
Use copper pours are your discretion but I (Gondolindrim) recommend always using them. My designs make liberal use of such pours even for other signals.
At the bottom–right of the page there is a copper polygon with the settings generally used for the Acheron Project. The values are tuned for general use, but adjustments can be made. Thermal spoke width and relief gap can be adjusted according to the thermal needs –– for THT components or connectors, it is recommended to use higher gaps (0.5mm will do the trick) to avoid bad solder joints.
Avoid using hatched–pattern copper pours. They have their reason to be, but not on keyboard PCBs. They are ugly (yeah, fight me) and don't do what we want them to do here. Always use solid fills. The drawback is that they can make the PCB stiffer, so use flex cuts on the PCB.

REFERENCES

[1] PCBWay manufacturing capabilities. Link: <https://www.pcbway.com/capabilities.html>. Last access august 21, 2021.
[2] JLCPCB manufacturing capabilities. Link: <https://jlcpcb.com/capabilities/Capabilities>. Last accessed august 21, 2021.
[3] Elecrow manufacturing capabilities. Link: <https://www.elecrow.com/download/quote/PCB_Specification_FAQ.pdf> . Last access august 21, 2021.
[4] Sanjeeb Mishra, Neeraj Kumar Singh, Vijayakrishnan Rousseau. System on Chip Interfaces for Low Power Design. Morgan Kaufmann, 2016. ISBN 9780128016305. Accessible via https://www.sciencedirect.com/science/article/pii/B9780128016305000104)
[5] Seeedstudio manufacturing capabilities. Link: <https://support.seeedstudio.com/knowledgebase/articles/447362–fusion–pcb–specification> . Last accessed august 21, 2021.

IMPORTANT

The values and observations here listed consider a two–layer, 1
A change in manufacturing parameters (layers and copper weigh
Values also pertain to the cheapest setting on the referenced m
Therefore even for a 2–layer 1oz/ft2 PCB tolerances can be dif
DO NOT use these values in other settings; always coordinate pr
For clarification, read the NOTES ON TOLERANCES.

USED TOLERANC

FACTORY MINIMUMS

TRACK WIDTH: 0.15
COPPER CLEARANCE: 0.15 (see [i])
VIA HOLE: 0.2
VIA ANNULAR WIDTH: 0.13
VIA DIAMETER: 0.4
COPPER–TO–HOLE CLEARANCE: 0.3 (see [iii])
COPPER–TO–EDGE CLEARANCE: 0.2
MINIMUM THROUGH HOLE DRILL: 0.2
HOLE–TO–HOLE CLEARANCE: 0.5
SILKSCREEN CHARACTER HEIGHT: 0.8
SILKSCREEN TRACE WIDTH: 0.15
SILKSCREEN CHARACTER TRACE–TO–HEIGHT RATIO: 1:6 (see [ii])
PAD–TO–SILKSCREEN CLEARANCE: 0.15
SOLDERMASK EXPANSION: 0.05
MINIMUM SOLDERMASK BRIDGE: 0.2

ALL VALUES IN MM

REC

TRACK WIDTH:
COPPER CLEARA
VIA HOLE: 0.3
VIA ANNULAR WI
VIA DIAMETER:
COPPER–TO–HO
COPPER–TO–ED
MINIMUM THROU
HOLE–TO–HOLE
SILKSCREEN CH
SILKSCREEN TR
SILKSCREEN CH
PAD–TO–SILKSC
SOLDERMASK EX
MINIMUM SOLDE

NOTES

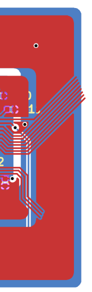
[i] Official copper–copper clearances are 0.2mm but not exactly "all copper". Pad–to–pad minimums are 0.5
[ii] The recommended ratio between silkscreen character height and its trace width so they are clearly legible
[iii] The hole–to–copper clearance changes on occasion. For instance, via–to–track and NPTH–to–track clea
[iv] The distance of copper–to–edge is a big problem for fabs in designs where traces need to be close to c
where the PCB traces need to be routed close to the flex cuts for lack of real–estate. This is why this va
[v] The 1:6 ratio for silkscreen is OK for large characters but can become unreadable to the naked eye on a

OVERALL DESIGN RECOMM

The first experience with PCB design is always very frustrating and overwhelming. There will be a lot o
good PCB designer, and there is always something to learn. Since PCB design is a highly techni
some sorte of black magic. Do not stray from resilience. The best way to learn PCB design is de
electronics and embedded systems is also recommended, but higher education in maths and eng
Sharing your design and knowledge is one of the most fruitful and effective ways to learn design and
talk to other designers. This ensures a collective knowledge is achieved both in hard and soft sk
PCB design involves a lot of creativity and not everyone understands your codes and conventions. Take
(1) Make liberal use of the F.Fab and B.Fab layers. Those should be used for documentation whi
and polarity, values, designators, everything goes to make the fab's job easier –– and yo
(2) Treat the silkscreen layers with very good care. Make sure that the component designators a
visible. Also make sure that the silkscreen indicates the polarity of components either thr
does not accompany its files; when someone looks at the PCB, component designators an
(3) KiCAD allows custom user layers. Use those to indicate screw hole anchors/sizes, plate mar
in the eventuality you want to make a revision or correction, all the information is in the
or Discord conversations.
(4) If you want to share or open–source your design, be kind and make sure all your thought p
a PCB, much like a computer code or a maths test, requires knowledge that is either a c
(5) Yes, documenting is boring. Everyone knows that, and yet everyone takes the time to do it b
Even though keyboard PCB design is very forgiving in terms of the mistakes and errors you can make,
The most common problems for first–timers in keyboard PCB design are (1) USB data traces routi
and routing. In this order, they are the most problematic parts of designing a keyboard PCB and a
(1) USB DATA TRACES: make sure to route them as short as possible, parallel to one another. Also
breaking and VIA'ing them as much as possible. If those are needed, do so in a simmetrical ma
communication (see [4], pages 331–344).
(2) CRYSTAL TRACES: these are the highest frequency traces on the PCB and should be handled wit
the MCU (in its vicinities) and if possible sharing the same ground (ideally islanded).
(3) BYPASS CAPACITORS: the precise function and nature of bypass (also called decoupling) capacit
and engineering. In layman's terms, bypass capacitors are placed to mitigate noise inherently p
of traces and the switching characteristic of digital circuitry. Failure to properly place and rout
to work properly. In the specific case of keyboard PCBs, there are plenty of digital circuits tha
LEDs and level shifters. Treat bypass capacitors like you would a crystal: as close to the main
datasheets, reference manuals and application notes which will generally inform how many cap

TATION.

an to use it for actually selling PCBs, you MUST
e notes, the documentation and understand it fully.



EXAMPLE COPPER POUR