

Náhľad nových vlastností obľúbeného EDA systému KiCAD, si môžete prezrieť v tomto 25 minútovom zázname:

mp4: [https://ftp.heanet.ie/mirrors/fosdem-video/2018/K.4.201/cad\\_kicad\\_v5.mp4](https://ftp.heanet.ie/mirrors/fosdem-video/2018/K.4.201/cad_kicad_v5.mp4)

webm: [https://ftp.fau.de/fosdem/2018/K.4.201/cad\\_kicad\\_v5.webm](https://ftp.fau.de/fosdem/2018/K.4.201/cad_kicad_v5.webm)

- Improved Eagle import
- Preliminary scaling support for High DPI monitors
- Integrated system variable editor
- Uses local symbols as default. Github plugin should be deprecated
- Rendering improvements (anti aliasing of fonts, translucency of layers, color wheel)
- Digikey made a large donation, largest in the project's history (undisclosed sum)
  
- net highlighting in eeschema + synchronized highlighting between eeschema and pcb\_new
- simpler tool for updating pcb from schematic
- symbol lib table added (symbol lib management is now similar to footprint management)
- reorganized libraries. Libs now have their own download section on the website. (The github plugin has been dropped for default setups)
- New symbol library editor interface
- automatic junction management + better connection algorithm (eeschema)
- Better handling of buses in eeschema (Not sure if everything that was proposed has been merged)
- Line styles for graphical polygons in eeschema
- Integrated interface for a spice simulator
- new component chooser. (adding symbols to a schematic will allow to select footprints at the same time.) This is sadly not enabled by default as it seems there are performance issues on some platforms.
- New pad types (polygon and rounded rectangle)
- Graphical polygons in pcb\_new and in the footprint editor
- Flip board in pcb\_new (look at it from the bottom)
- Graphical polygons in eeschema now support different line colors and line styles.
- Better copy paste support