Náhľad nových vlastnosti 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
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 webside. (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