Contribution ID: **2**   Type: **not specified**

KiCad: a Free and Open Source Software (FOSS) tool for Printed Circuit Board (PCB) design

Friday 9 May 2025 14:10 (15 minutes)

This presentation introduces KiCad, a powerful, free and open-source software suite for Printed Circuit Board (PCB) design. With robust features, active development, and growing community support, KiCad is a viable alternative to proprietary tools. We'll highlight key capabilities, recent improvements, and examples of successful designs to show why it's well-suited for use at CERN.

Presenter: FERNANDEZ BAUTISTA, Roberto (CERN)