Schematic Capture and PCB Layout with Free Open Source KiCad

DATE: 11/15/19   TIME: 9:00 am to 4:00 pm

LOCATION: Microchip Technology Regional Training Center, 112 Turnpike Road, Suite 100, Westborough Executive Park, Westborough, Massachusetts 01581

COURSE DESCRIPTION: Schematic capture and PCB layout tools can be expensive. Sharing designs using commercial tools is a problem if everyone does not have the same tools. KiCad is a free open source tool set that runs on Windows Linux or Macintosh and is as feature rich as tools costing thousands of dollars per module.

OUTLINE:

- Projects
- Libraries
- Schematic Capture
- Symbol Creation
- Netlists
- BOMs
- PCB Layout
- Footprints
- Design Rules
- 3D Views of PCBs
- Gerber Files
- Ordering your PCBs

MATERIALS: A DVD containing a PDF of the presentation and copies of the FREE software tools.

PREREQUISITES: Basic familiarity of Schematic Capture and PCB Layout

REQUIRED: Students should bring their own computer. Payment by 11/8/19.

FEE: $299.00 per person.

ACCEPTED PAYMENT: Check, PayPal, or Credit Card.

TO REGISTER: Email Mchipguru@gmail.com. You will receive instructions on how to provide payment information and available discounts.