Altium Designer – Introduction

On the Schedule
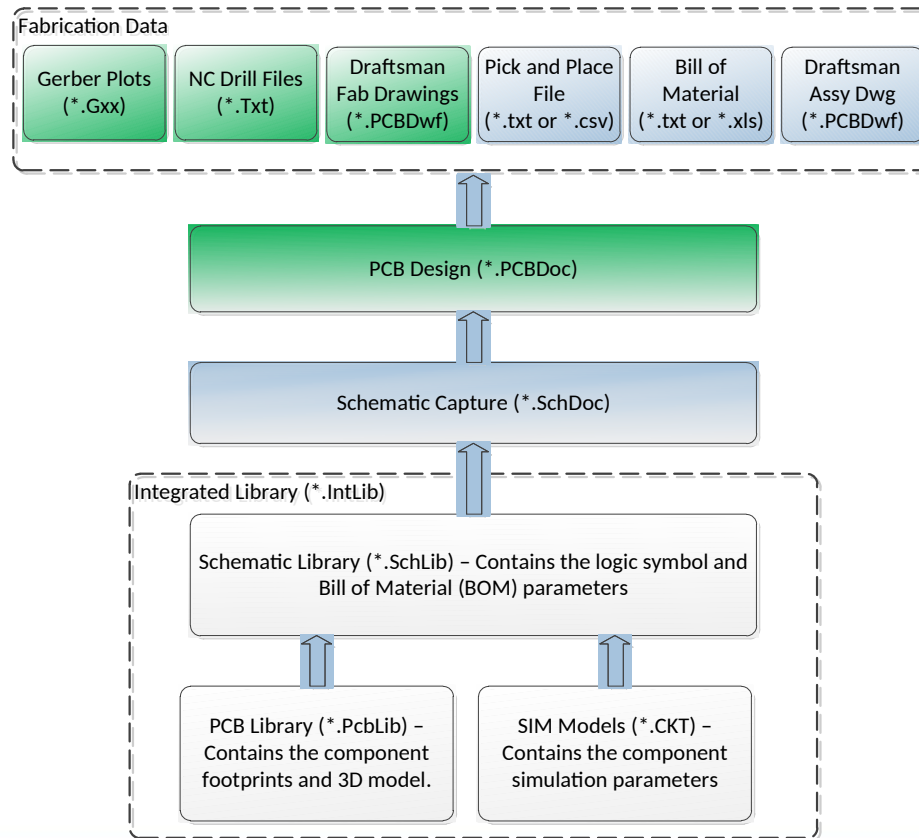
- Review Resources – Altium, PCB Fab, Other Free PCB ECAD packages
- Altium ECAD Package Breakdown
- Example Project
- Competition

Altium Designer Introduction

Resources

- Altium Resources:
  - Design Content - https://designcontent.live.altium.com/#UnifiedComponents
  - Video Guides - https://altiumvideos.live.altium.com/#Index/0/0
  - Installed Examples - C:\Users\Public\Documents\Altium\AD17\Examples

- PCB Manufacturing Resources:
  - PCB - http://www.4pcb.com
  - PCB - http://www.sunstone.com/
  - Stencils - https://www.oshstencils.com/#
  - PCB Assy - https://macrofab.com/

- Component Suppliers:
  - Digikey – https://www.digikey.com/
  - Mouser – http://www.mouser.com/

- Free PCB Design Tools:
  - https://circuitmaker.com/#why_circuitmaker
  - http://www.4pcb.com/free-pcb-layout-software/
 - https://www.sunstone.com/pcb-products/pcb123

Unified ECAD System

- Schematic Symbols and PCB Footprints (Libraries)
- Schematic Capture
- PCB Layout
- Fabrication Documentation

Fabrication Data

| Gerber Plots (*.Gxx) | NC Drill Files (*.Txt) | Draftsman Fab Drawings (*.PCBDwf) | Pick and Place File (*.txt or *.csv) | Bill of Material (*.txt or *.xls) | Draftsman Assy Dwg (*.PCBDwf) |

⇧

PCB Design (*.PCBDoc)

⇧

Schematic Capture (*.SchDoc)

⇧

Integrated Library (*.IntLib)

Schematic Library (*.SchLib) – Contains the logic symbol and Bill of Material (BOM) parameters

⇧ ⇧

PCB Library (*.PcbLib) – Contains the component footprints and 3D model.

SIM Models (*.CKT) – Contains the component simulation parameters

BLUR

Example Project – Getting Started

- Setup master library and project folder under root (Location doesn't really matter. I like to minimize the path.)
- Install the libraries:
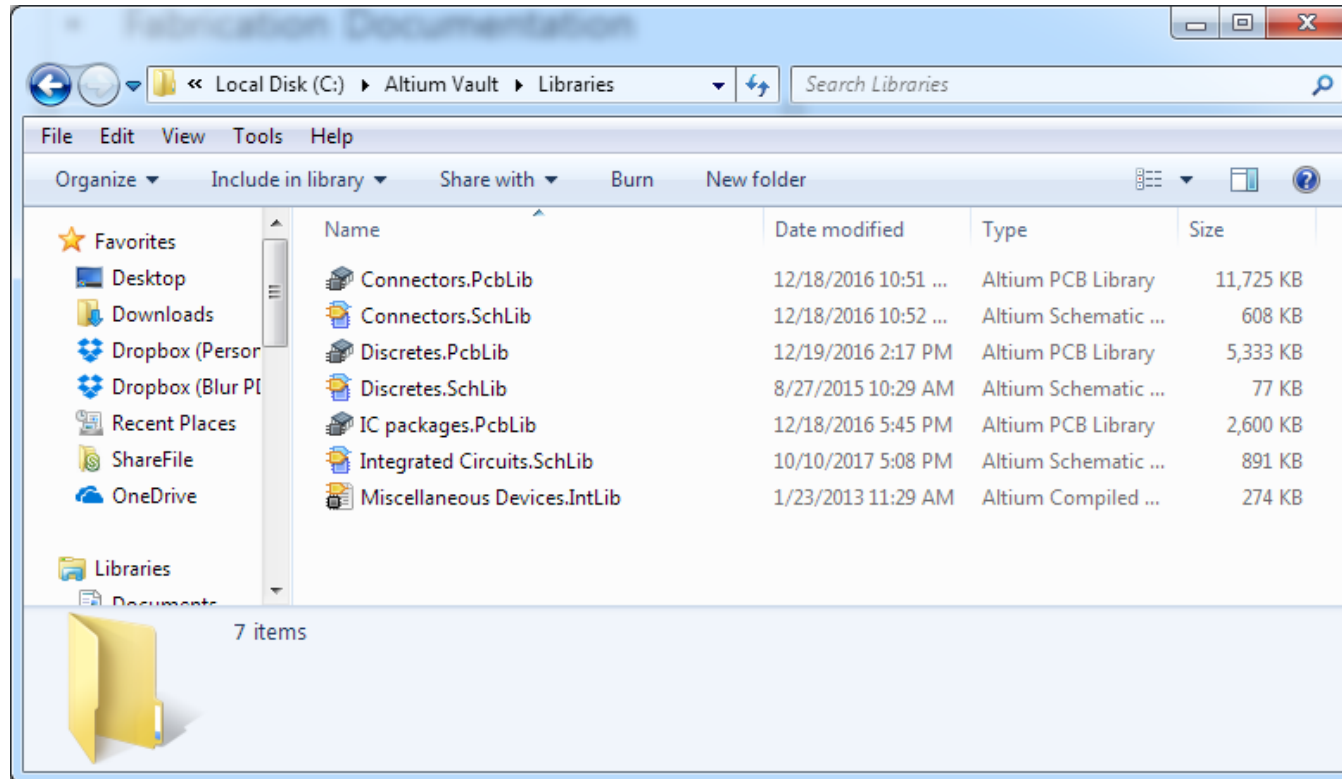
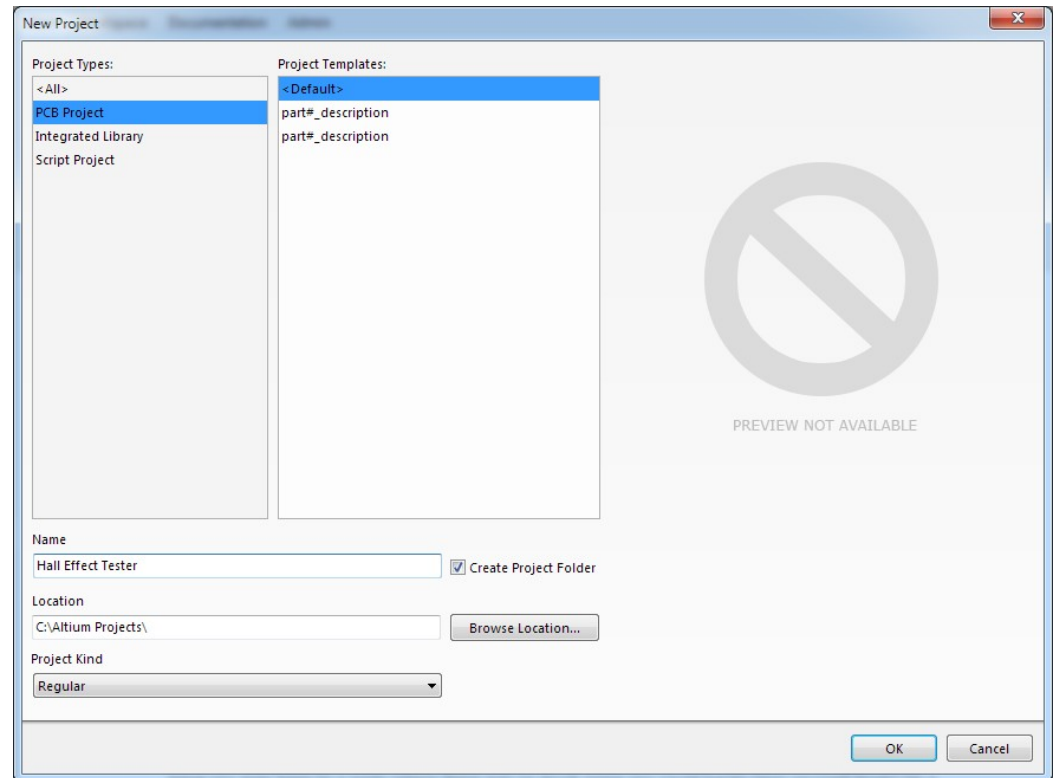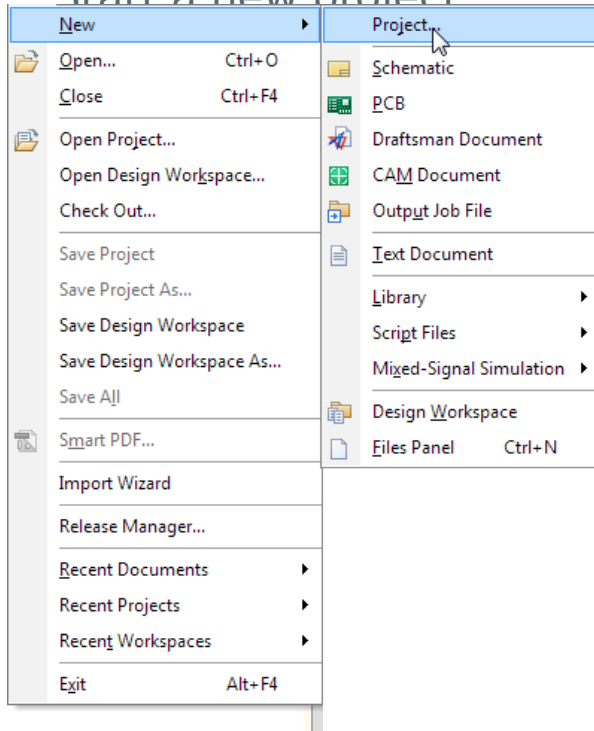# Example Project – Getting Started

- Altium Preferences
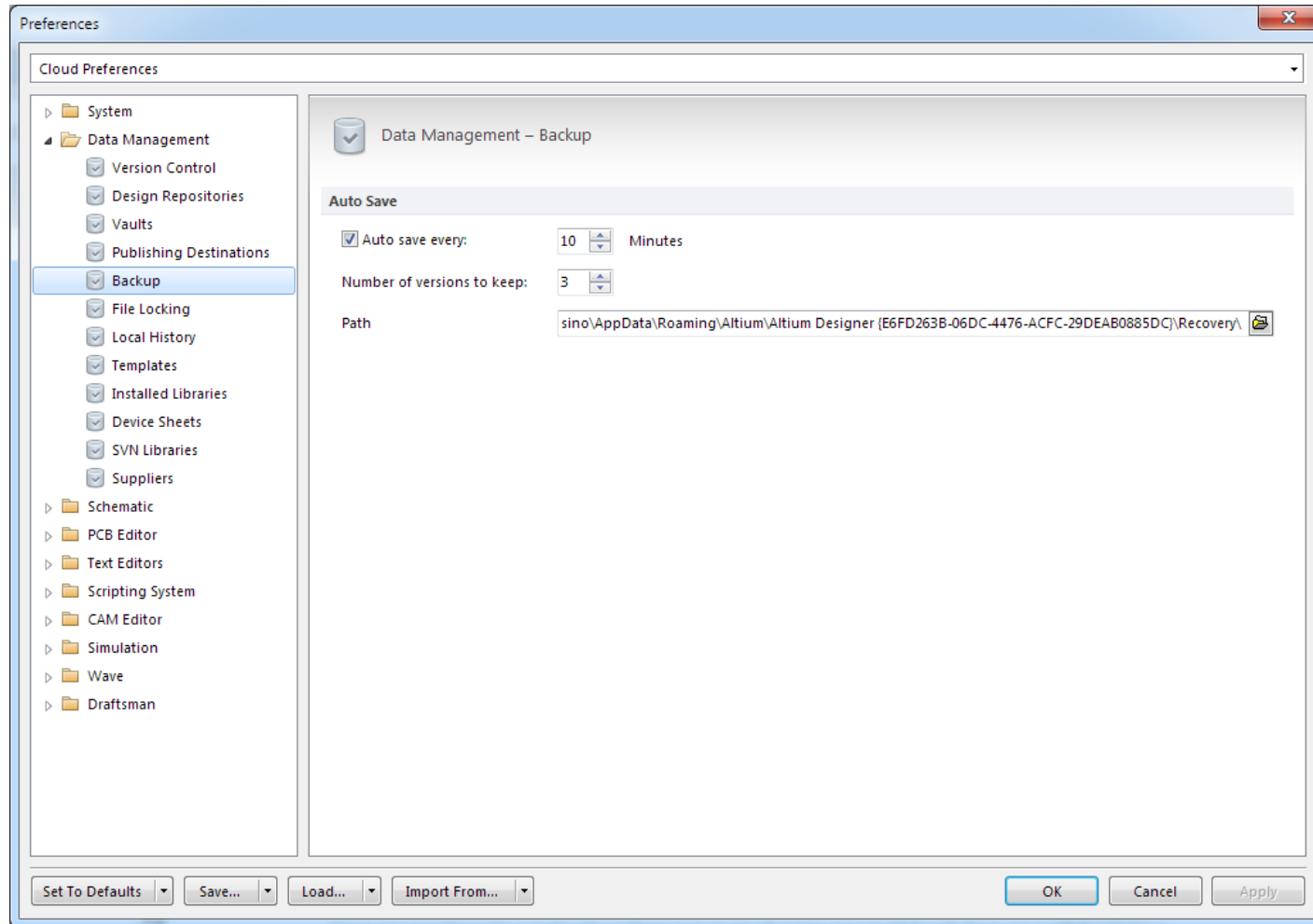
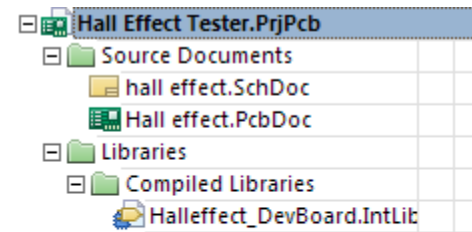# Example Project – Getting Started
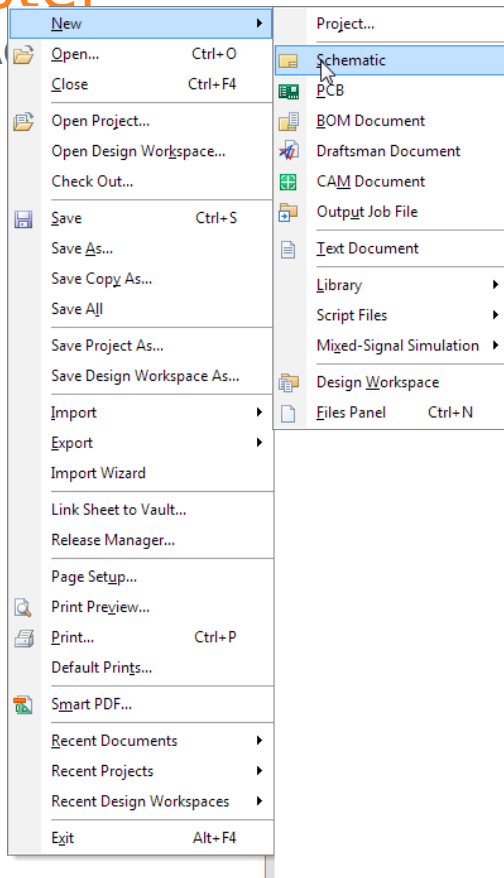
- Setup Altium to find libraries:
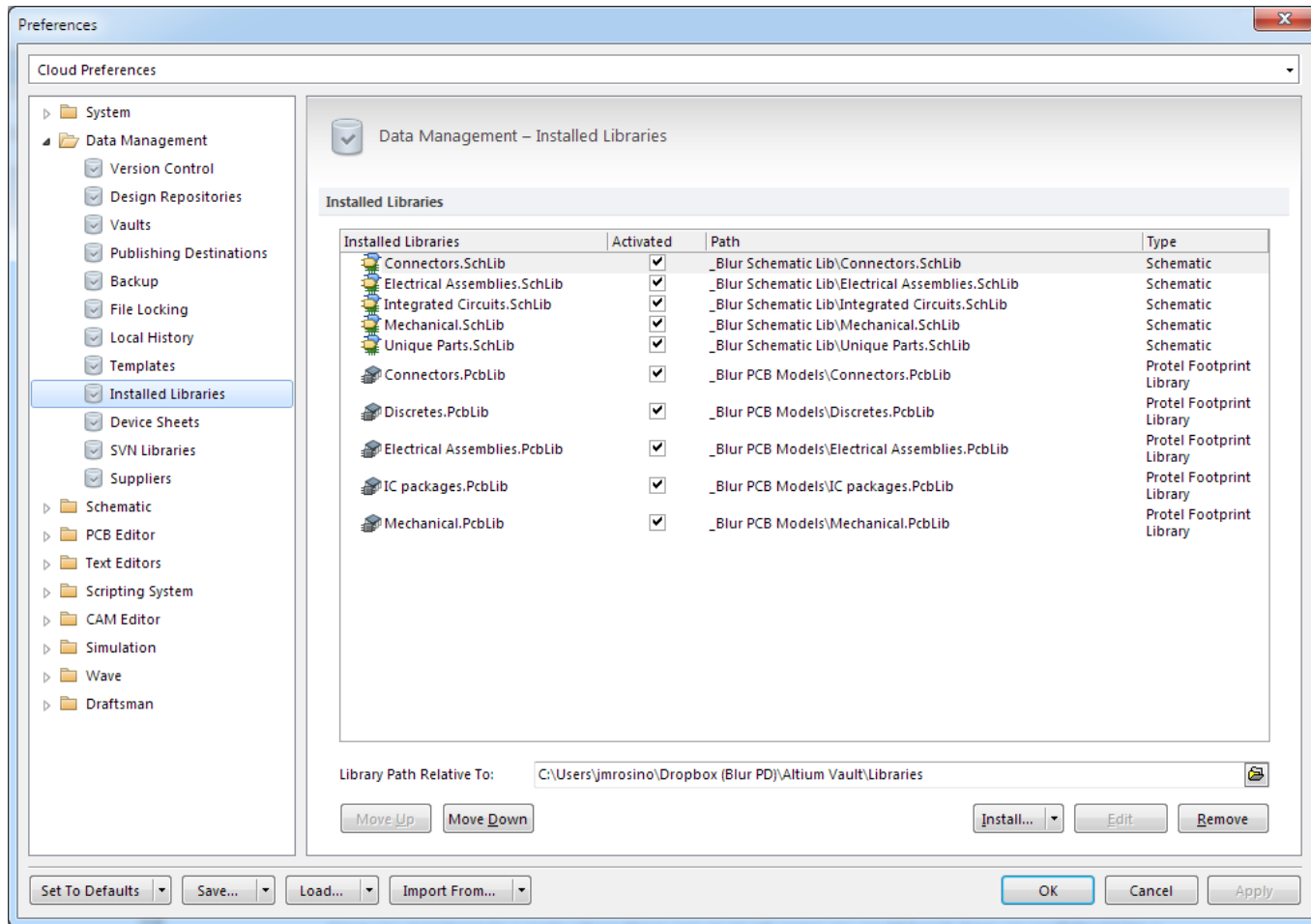
Example Project – Hall Effect Sensor Magnet Tester

- Start a new project:

Example Project – Hall Effect Sensor Magnet Tester

- Add source documents, projects, and Integrated Library (by add existing)

Example Project – Hall Effect Sensor Magnet Tester

- Design Requirements
    - Hall Effect Sensors/switch must be placed 1 inch apart along the Y axis edge and centered.
    - Board size - "C" shaped: 1.0"x1.5" with 0.5"x0.4" notch in center.
    - When magnet is in range, LED illuminates (active high).
    - Run on 3.3V rail.

Example Project – Hall Effect Sensor Magnet Tester
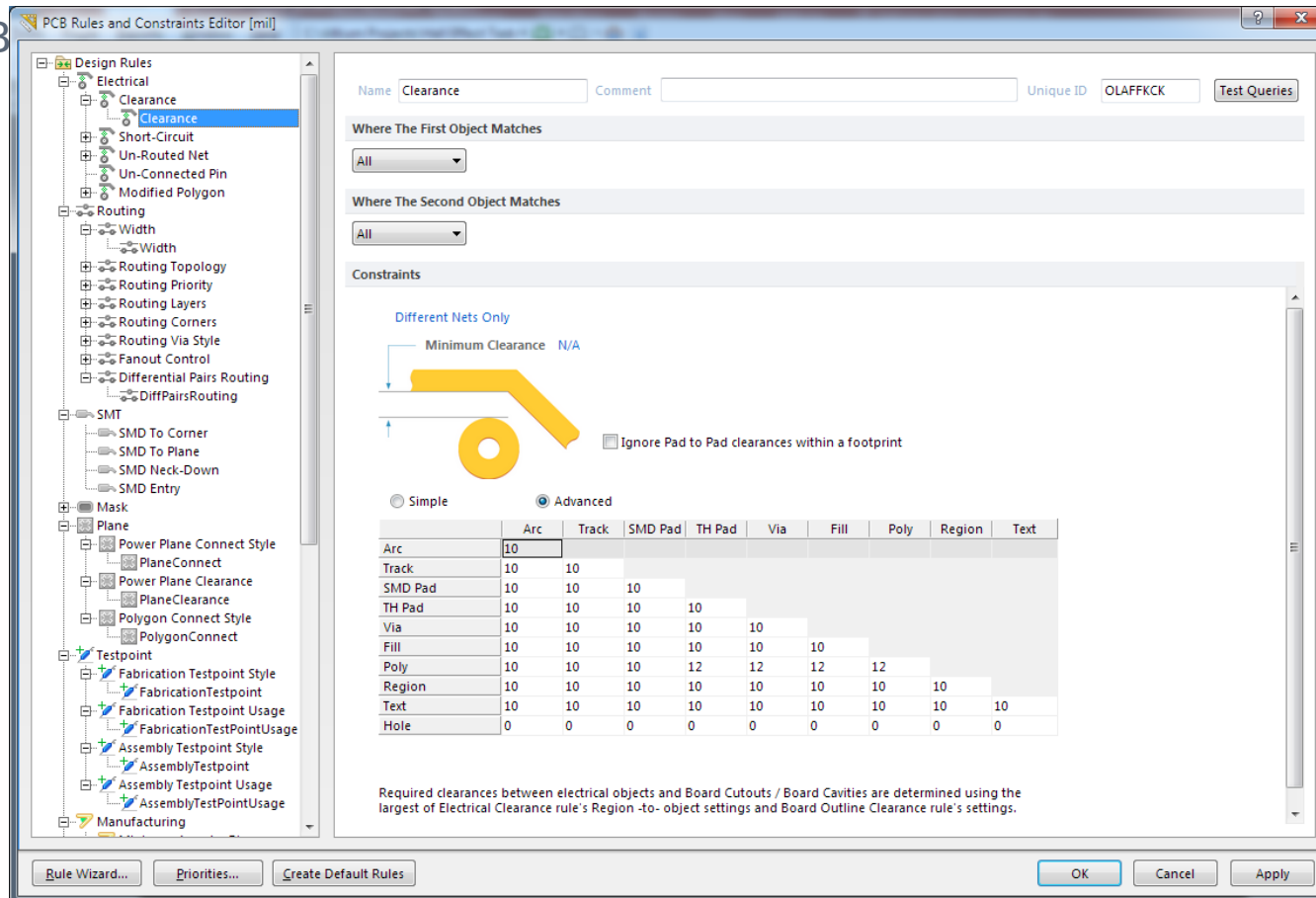
- Save Project and Start Capturing

BLUR
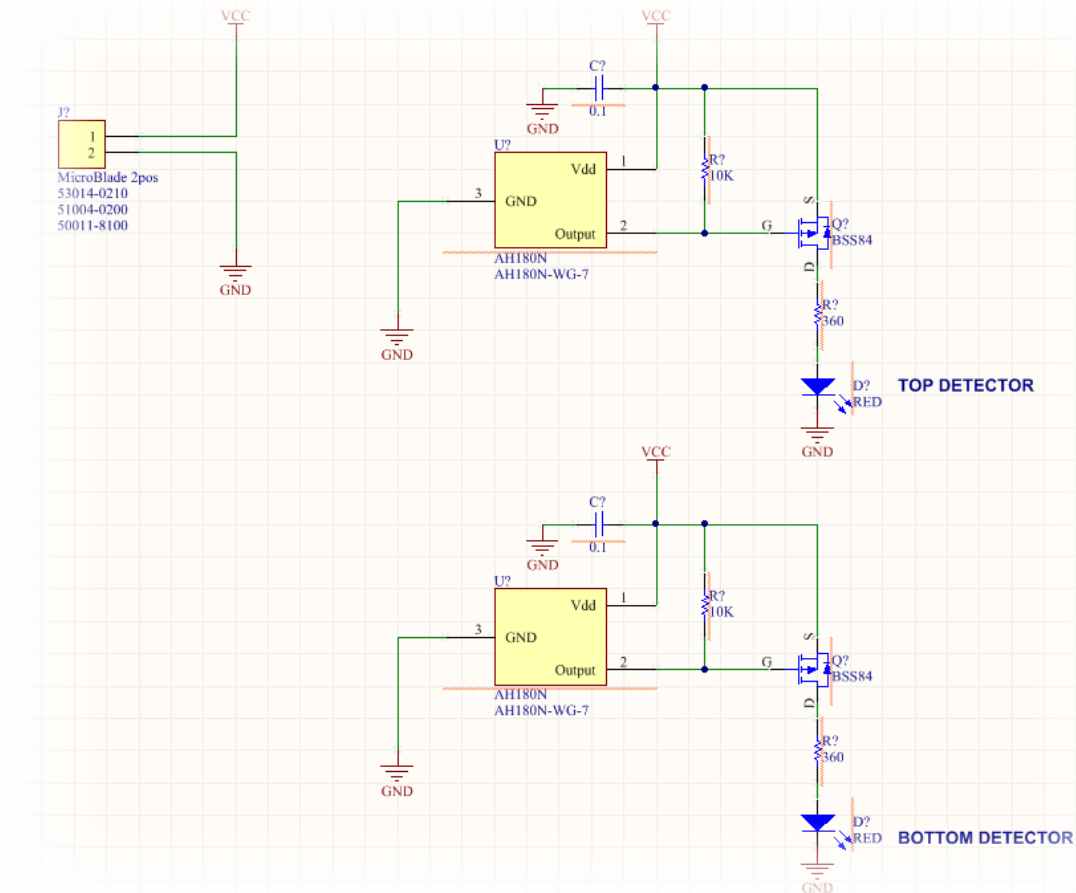
Example Project – Hall Effect Sensor Magnet Tester

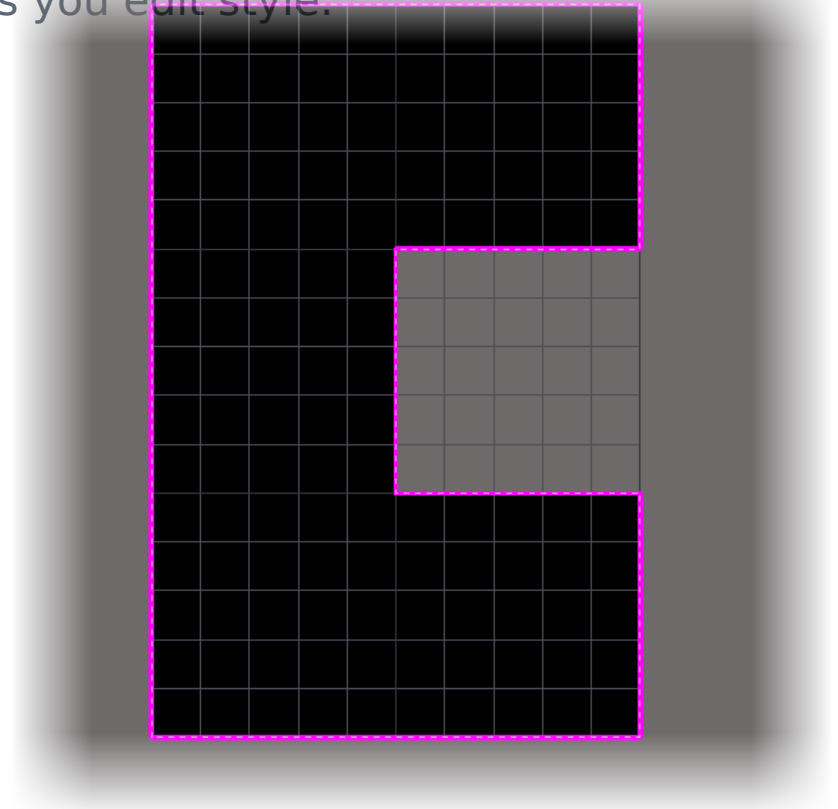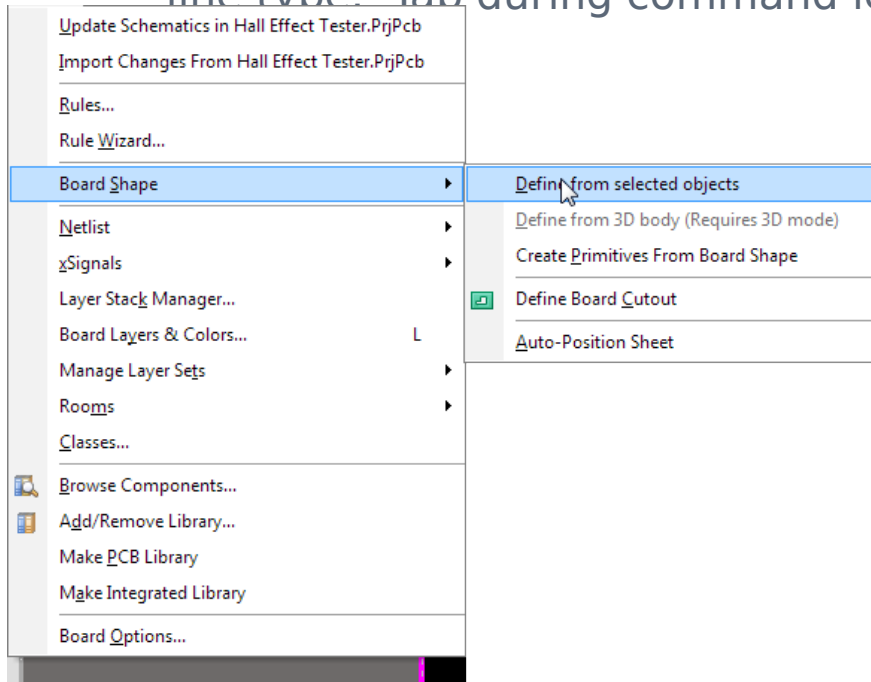- Annotate Schematics and Compile Design – Providing a unique designator to each component
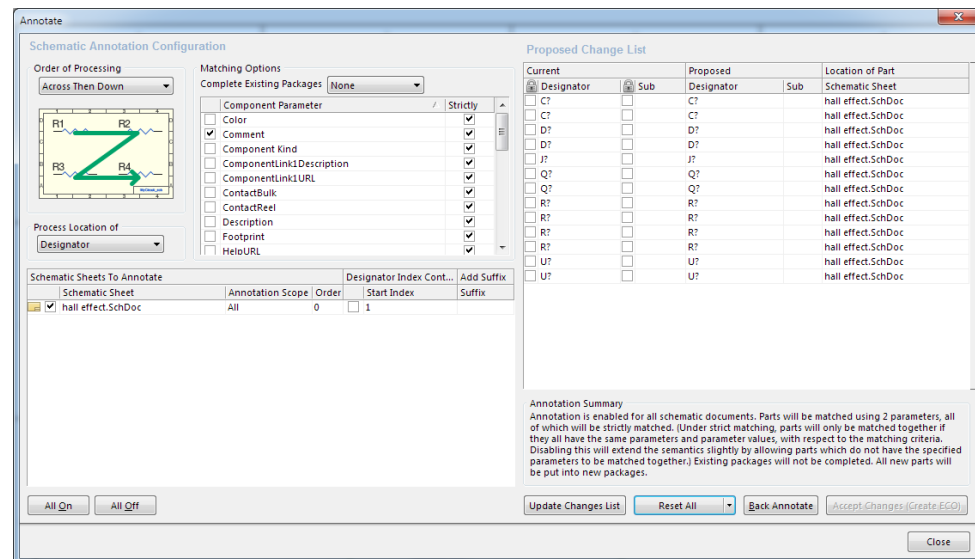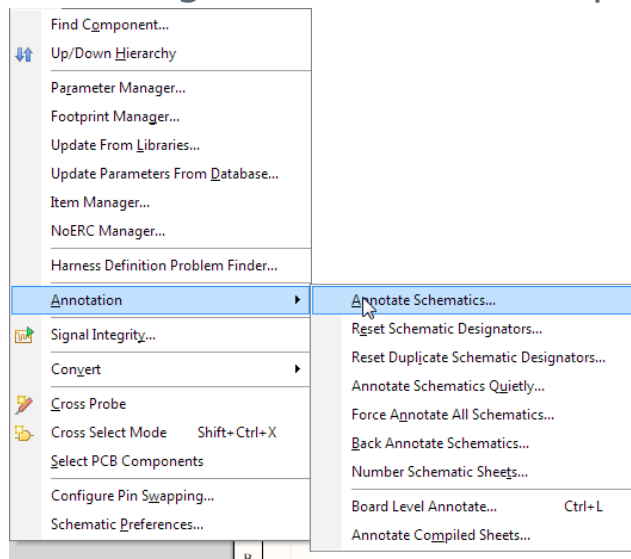
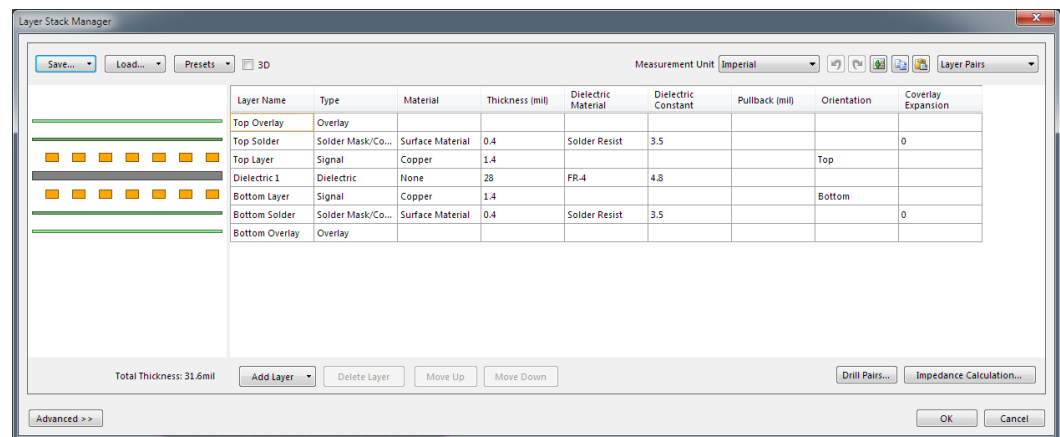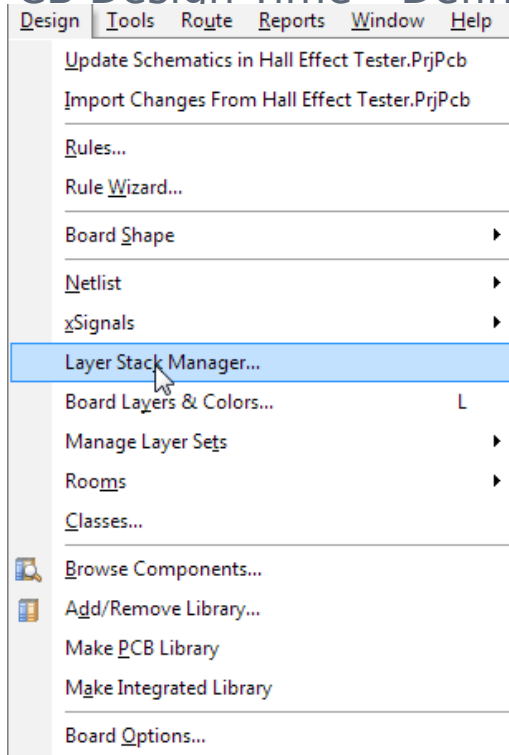## Example Project – Hall Effect Sensor Magnet Tester

- PCB

BLUR

Example Project – Hall Effect Sensor Magnet Tester

- PCB Design Time – Setup PCB outline
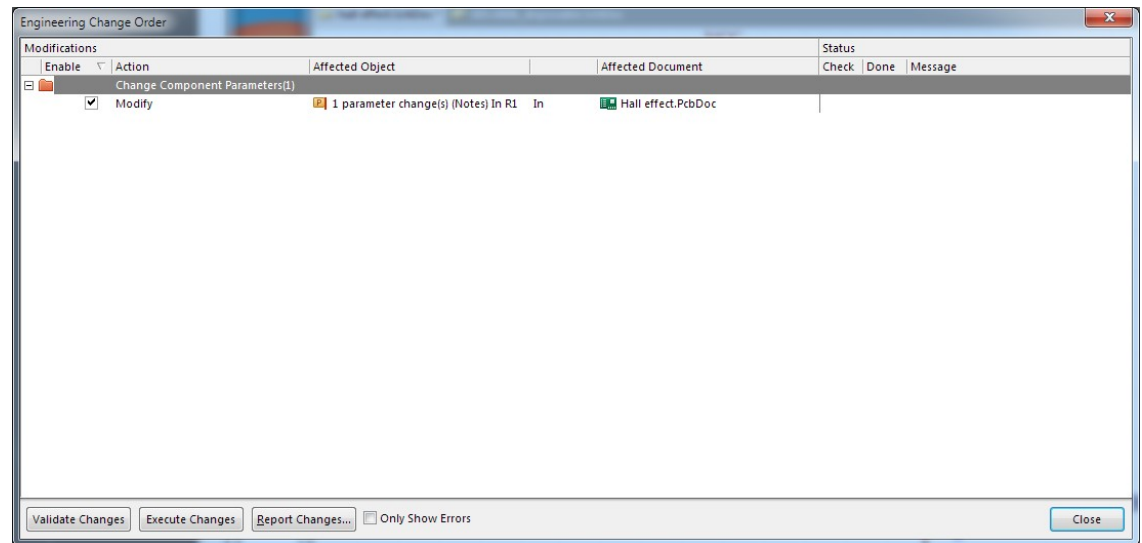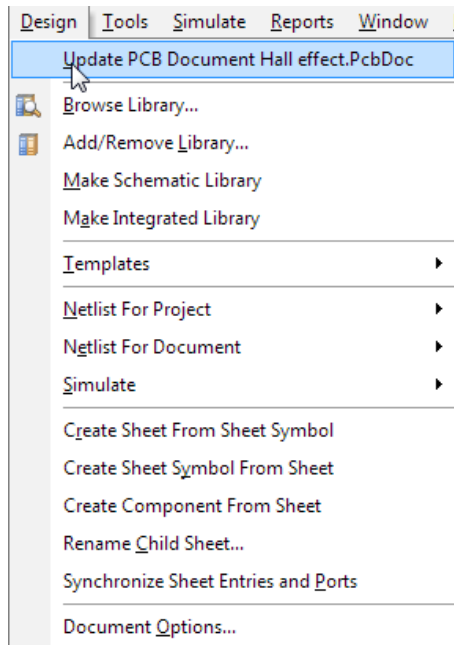 - Tips – Space bar changes end angle. Shift +Space Bar changes line type. Tab during command lets you edit style.

BLUR

Example Project – Hall Effect Sensor Magnet Tester
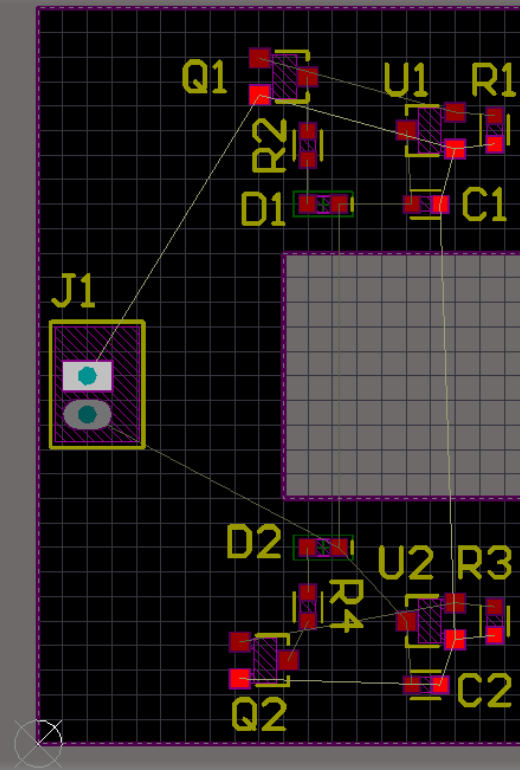
- PCB Design Time – Define Layer Stack

Example Project – Hall Effect Sensor Magnet Tester

- Update PCB Design – From the schematic page, click "Update PCB Document…"

Example Project – Hall Effect Sensor Magnet Tester

- PCB Design Time – Layout (Place Components)
  - Tips – Press "M" then "C" to move component. Make sure to select "Component jump to cursor".

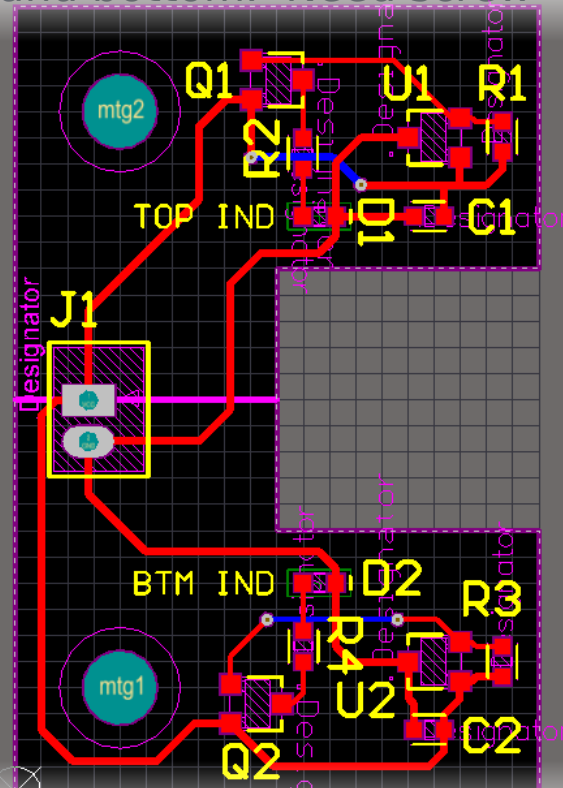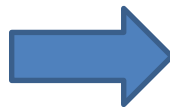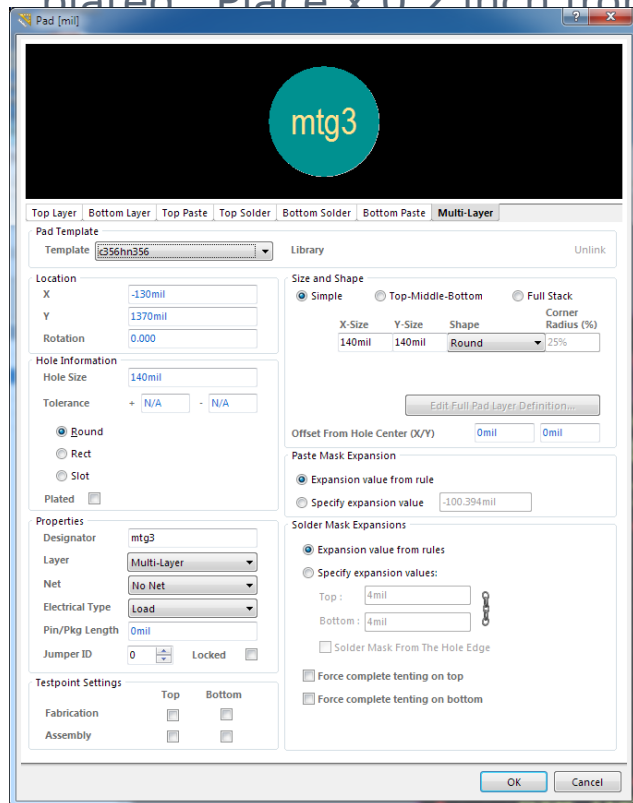# Example Project – Hall Effect Sensor Magnet Tester

- PCB Design Time – Route… not auto
  - Rules – No 90 deg bends. Hit center of pads. 15mil wide trace for power (3.3 and GND). All other nets can be 10mil.
  - Tips – Press TAB to change properties of a route. Use SHIFT + E to enable/disable hot snaps.
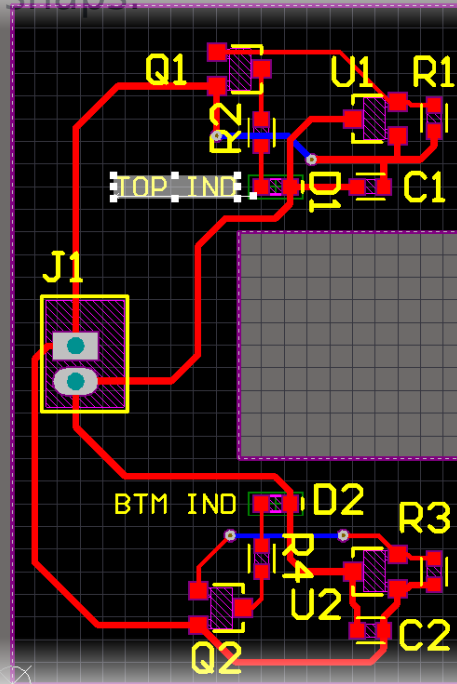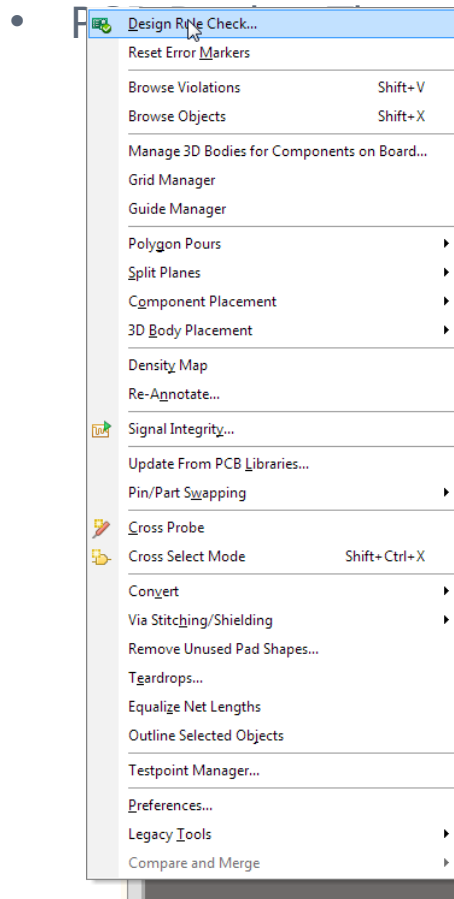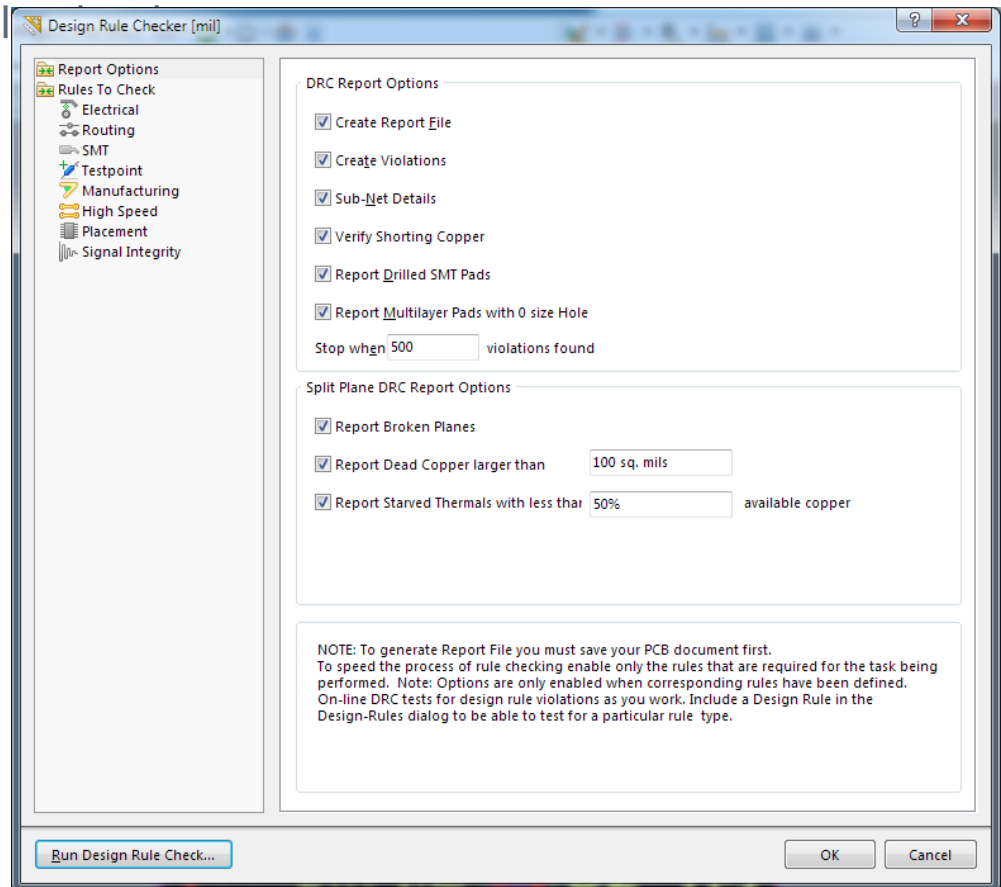
Example Project – Hall Effect Sensor Magnet Tester

- PCB Design Time – Add Mounting holes: #6 fastener -> 0.140"dia non plated. Place x 0.2 inch from left side top and bottom. Need screw

BLUR

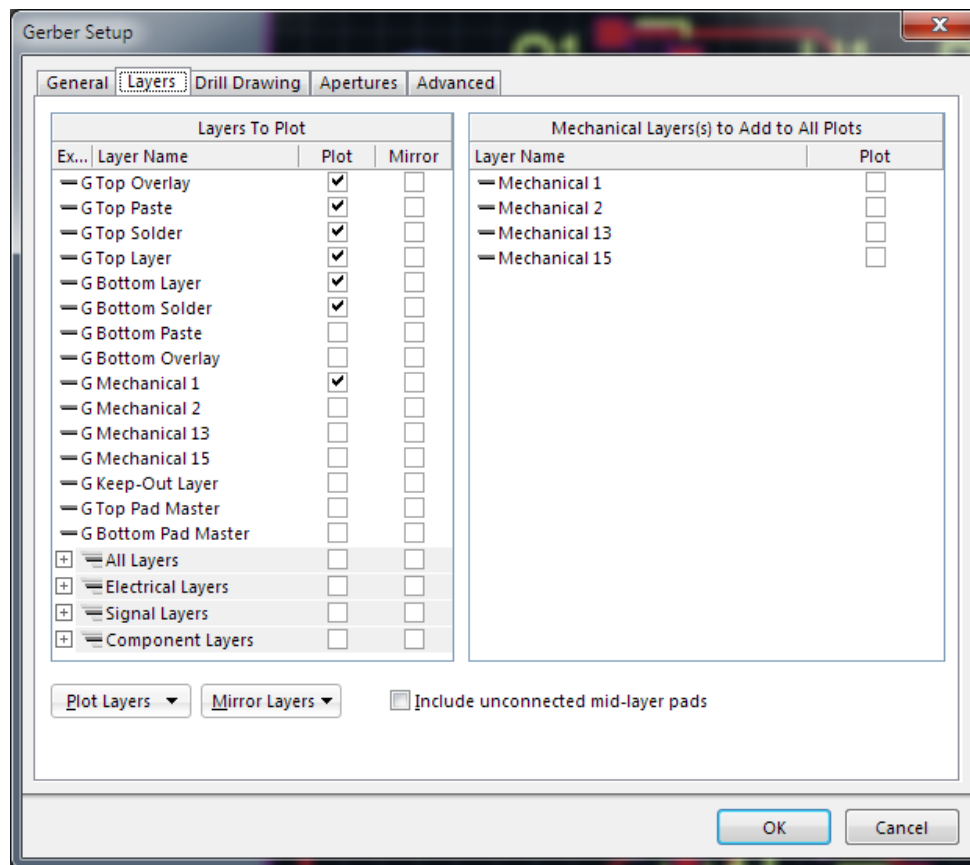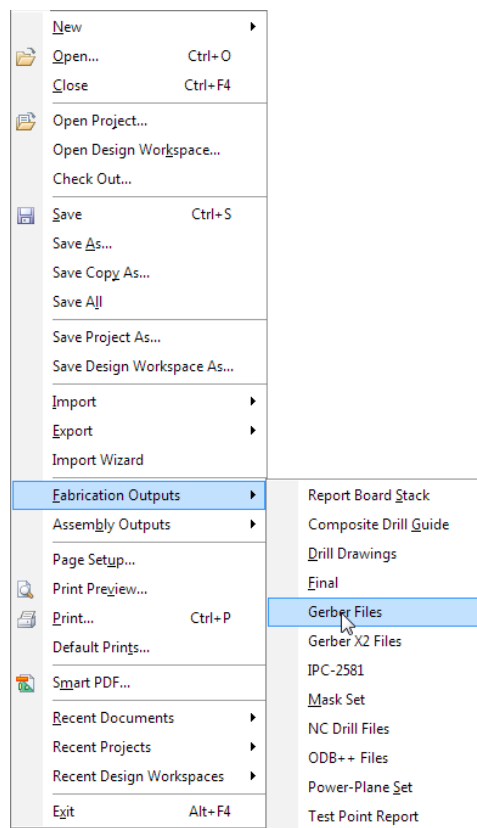# Example Project – Hall Effect Sensor Magnet Tester

- Design rule check

Example Project – Hall Effect Sensor Magnet Tester

- Fabrication Outputs (Using Advanced Circuits for PCB Fab).
  - Gerbers and drills

Competition Time

- Update design by adding two more hall effect sensor positions on the right side. Separate sensors by 0.5" and add in an EEPROM to identify the board. Need a separate connector to interface with the EEPROM.

- Objective – Design for cost efficiency.
  - Cost of PCB - $1.00/SQIN
  - VIAs - $0.25 ea
  - Spacing – 10/10 no adder; 8/8 add $0.12 per net.