CTP ENGINEERING WHITE PAPER

How to Design Magnetic Pogo Pin Connectors into a PCB Assembly

Mounting architecture, PCB footprint control, working stroke, soldering, load paths, contamination control, and validation.

ABSTRACT
Practical PCB integration guidance for magnetic pogo pin connectors, covering mounting, footprint control, working stroke, mechanical load paths, soldering, contamination control, and assembly validation.

Author	Steven Wan
Company	CTP / Shenzhen Yongtan Electronics
Original article	magneticpogo.com/magnetic-pogo-pin-pcb-mounting-guide/
Technical note	Model-specific parameters must be confirmed against the approved drawing and test conditions.

PCB -> CONNECTOR -> MAGNETIC MATING INTERFACE -> ENCLOSURE

The connector should be designed as part of the complete electrical and mechanical architecture, not as an isolated PCB component.

1. Engineering Summary

Integrating a magnetic pogo pin connector into a PCB assembly requires coordination between the connector footprint, pogo pin working stroke, device enclosure, mechanical load path, soldering process, cable or PCB termination, and final magnetic mating position. The mounting method should be selected from the complete device architecture rather than from connector size alone.

A connector can pass a bench test and still fail after installation if the PCB bends, the enclosure changes the working stroke, the solder joints carry repeated mating loads, or the final magnetic air gap differs from the prototype. PCB and mechanical engineers should therefore develop the connector footprint, functional datums, mechanical support, and validation plan together.

Key principle: The mechanical stop - not uncontrolled solder-joint height or magnetic force - should define the final seated position and pogo pin compression.

2. Choose the Mounting Architecture

Before selecting SMT or through-hole mounting, confirm what is included in the supplied connector assembly and which elements remain part of the customer's PCB, enclosure, and mating-side design.

Mounting method	Typical assembly process	Primary benefit	Primary engineering risk
SMT	Pick-and-place + reflow	Low profile and automated assembly	Solder joints may receive excessive mating or cable load
Through-hole	Wave, selective, or manual soldering	Terminal passes through the PCB	Hole design, solder fill, and support may be inadequate
Right-angle	SMT, through-hole, or mixed	Side-facing interface; reduced Z height	Board-edge and enclosure tolerance accumulation
Solder-cup / wire termination	Controlled manual wire soldering	Flexible cable or harness routing	Wire joint, strain relief, and contamination risk

Selection inputs:

- Connector orientation relative to the PCB and enclosure
- Available component height and board-to-enclosure distance
- Assembly process: reflow, wave, selective, or manual soldering
- Direction and magnitude of magnetic, spring, cable, and user-applied loads
- Production volume, automated placement requirements, and rework access
- Environmental sealing, potting, cable integration, and inspection strategy

3. Develop the PCB Footprint from the Controlled Drawing

The approved connector drawing should be the starting point for the PCB footprint. Avoid copying a footprint from a visually similar connector, because small changes in pitch, terminal width, locating features, housing overhang, or working height can make the pattern unsuitable.

Confirm that the drawing identifies:

- Terminal dimensions and pitch
- Housing outline and seating plane
- Locating pegs, tabs, and holes
- Recommended PCB pattern
- Pin-one or polarity reference
- Mating direction and working height
- Component tolerances and revision status

Any change to the recommended pattern should be reviewed against PCB fabrication capability, paste or soldering process, copper weight, thermal requirements, inspection access, and the selected assembly method.

4. Use Functional Datums and a Deliberate Mechanical Load Path

The connector should be located from datums that control the actual mating interface: board edge, enclosure bosses, connector housing surfaces, target-pad centerline, mounting holes, or mechanical stops. Using unrelated datums for the PCB, enclosure, and connector can create acceptable individual parts but a misaligned final assembly.

Design item	What to control
Functional datums	Connector face, pogo pin position, magnet position, enclosure opening, target pad, mechanical stop
Locating pegs	Positioning vs. retention function; plated or non-plated; electrical or non-electrical role
Load path	Transfer repeated mating and cable loads through housing/enclosure support where practical
PCB support	Board thickness, screw distance, cutouts, flexible regions, connector force, magnetic attraction
Working stroke	Minimum, nominal, and maximum compression across the complete tolerance stack

Preferred load path: Mating interface -> connector housing -> device enclosure or dedicated support, rather than mating interface -> terminals -> solder joints -> unsupported PCB.

5. Control Pogo Pin Working Stroke

Installed compression is affected by pogo pin free height, connector seating height, solder-joint height, PCB thickness and flatness, target-pad position, enclosure location, gasket or adhesive thickness, and the mechanical stop. Evaluate minimum, nominal, maximum, and uneven-compression conditions.

Condition	Possible risk	Required check
Minimum compression	Low contact force or intermittent operation	Minimum force and electrical stability
Nominal compression	Insufficient design margin	Resistance, force, and temperature

Condition	Possible risk	Required check
Maximum compression	Bottoming, high load, accelerated wear	Over-travel and structural loading
Uneven compression	Channel-to-channel force/resistance variation	PCB flatness and pin-by-pin comparison

6. Design the Mating Target and Offset Conditions Together

The mating side may use PCB copper pads, dedicated target contacts, through-hole targets, machined metal contacts, or insert-molded components. Target-pad dimensions, pitch, surface finish, position tolerance, flatness, mechanical support, wear area, and spacing from adjacent electrical functions should be defined with the connector.

Review the interface in all realistic positions: correct seating, lateral offset, angular offset, rotation, reversal where physically possible, and partial mating. Check whether any incorrect position could connect power to ground or signal, bridge two pogo pins to one pad, touch exposed metal, or energize before full seating.

7. Soldering, Reflow, and Contamination Control

Assembly process limits are part-specific. A connector housing, magnet adhesive, spring-loaded contact, and internal barrel should not be assumed to tolerate the same soldering profile as another product with similar geometry.

For SMT designs, review:

- Pick-and-place surface and nozzle access
- Placement accuracy and center of gravity
- Paste volume, stencil thickness, and component movement during reflow
- Supplier-approved reflow profile
- Post-reflow working height and spring return
- Housing deformation, connector tilt, magnet movement, wetting, and voiding

For through-hole or manual processes, review:

- Finished plated-hole size and annular ring
- Solder fill, terminal solderability, PCB thermal mass, and fixture access
- Flux exposure, cleaning method, and inspection access
- Application time and temperature near the spring-loaded barrel or plastic housing

Contamination risk: Solder, liquid flux, cleaning solvents, adhesive, conformal coating, potting compound, dust, or particles entering the pogo pin barrel can affect plunger movement and electrical performance.

Define coating and potting keep-out boundaries around moving contacts, electrical contact surfaces, target pads, mechanical stops, and magnet features. Inspection and rework procedures should be documented before production release.

8. Design the Complete Electrical Channel

Connector current capability cannot be evaluated independently from the PCB and cable. The complete path may include the pogo pin, contact interface, target pad, solder joint, PCB pad, via structure, copper trace or plane, and wire or cable termination.

- Review continuous and peak current, contact resistance, voltage drop, copper thickness, trace width, via quantity, thermal spreading, ambient temperature, and enclosure heat dissipation.
- When multiple pogo pins are paralleled, route channels symmetrically where practical and evaluate individual channel resistance and temperature because current may not divide equally.
- For signal and data contacts, review protocol, return path, differential routing, impedance, pin spacing, crosstalk, cable construction, and the connector transition. Pin count alone does not guarantee support for a specific data standard.

9. Review Magnets Near the PCB

Evaluate the magnetic structure together with nearby Hall sensors, magnetometers, relays, inductors, speakers, microphones, ferromagnetic screws or shields, and any area where metal particles may collect. Required keep-out distance depends on the actual magnet arrangement, field strength, shielding, component sensitivity, and enclosure.

10. Define the Assembly and Validation Sequence

1. Confirm the approved connector drawing and PCB footprint.
2. Confirm PCB fabrication and assembly capabilities.
3. Build initial PCB and connector samples.
4. Measure connector position and working height.
5. Install the PCB into a production-intent enclosure.
6. Confirm target-pad alignment and pogo pin compression.
7. Measure magnetic holding and release behavior.
8. Measure electrical resistance and voltage drop.
9. Review cable, housing, and PCB load paths.
10. Validate the selected soldering process, then freeze footprint, fixtures, and inspection criteria.

11. PCB Assembly Validation Matrix

Requirement	Possible evaluation	Example output
Connector position	Optical or dimensional measurement	X, Y, Z and angular position
Working stroke	Installed-height stack-up measurement	Minimum, nominal, maximum compression
Solder-joint quality	Approved workmanship inspection	Wetting, fill, alignment, condition
Plunger movement	Post-assembly compression and return check	Free return and working height
Contact resistance	Defined resistance measurement	Individual-channel and complete-path values
Voltage drop	Powered test under specified current	Voltage and stabilized temperature
Magnetic mating	Capture, holding, and separation-force measurement	Force / displacement behavior
Mechanical load	Mating, cable-pull, or device-level load evaluation	PCB deflection and joint condition
Assembly heat	Production-intent soldering trial	Housing, force, dimensions, electrical result
Contamination control	Post-process inspection and return test	No prohibited flux, coating, adhesive entry

12. Common PCB Integration Mistakes

Design mistake	Possible consequence	Better engineering approach
Copying a similar connector footprint	Terminal, peg, or housing mismatch	Use the controlled drawing for the selected part
Allowing solder joints to carry every mating load	Fatigue, PCB bending, connector movement	Create a housing/enclosure load path
Ignoring enclosure tolerance	Misalignment after final assembly	Use shared datums and a complete stack-up
Freezing PCB before the mating side	Target pads and pins may not align	Develop both sides together
Using magnetic force to define working stroke	Compression varies with air gap and housing	Use a controlled mechanical stop
Applying one reflow profile to every connector	Housing, spring, or magnet damage	Use approved process limits for the specific part
Allowing flux/coating into the barrel	Plunger sticking or resistance instability	Define orientation and keep-out controls
Ignoring PCB support	Board deflection changes compression	Add supports near the connector
Assuming through-hole is always	Poor hole/solder design can still fail	Evaluate terminal, hole, solder, and

Design mistake	Possible consequence	Better engineering approach
stronger		housing together
Assuming pin count defines high-speed capability	Channel may not meet protocol needs	Perform protocol-specific PCB/cable design

13. Release Checklist for the Connector PCB Package

- Controlled connector mechanical drawing and revision
- STEP model for interference review
- Recommended PCB footprint and schematic symbol
- Pin map and polarity identification
- Terminal and locating-hole requirements
- Mating-target drawing
- Working-stroke and spring-force requirements
- Magnetic-force requirements and mechanical stop definition
- Soldering-process limits and coating/potting keep-outs
- Inspection method and applicable electrical limits

A STEP model helps verify mechanical interference, but it does not replace a controlled drawing containing dimensions, tolerances, process limits, and functional requirements.

About CTP

CTP / Shenzhen Yongtan Electronics provides OEM / ODM custom interconnect solutions including magnetic connectors, magnetic cable assemblies, pogo pin connectors, and pogo pins for electronics applications. Engineering decisions should always be confirmed against the applicable product drawing, device architecture, and production-intent validation conditions.

Original engineering article: How to Design Magnetic Pogo Pin Connectors into a PCB Assembly

Website: https://magneticpogo.com/

Submission note: This white paper is adapted from CTP's engineering article for technical publication and industry education. It avoids model-specific ratings; applicable dimensions, electrical limits, and process limits must be verified for the selected connector.