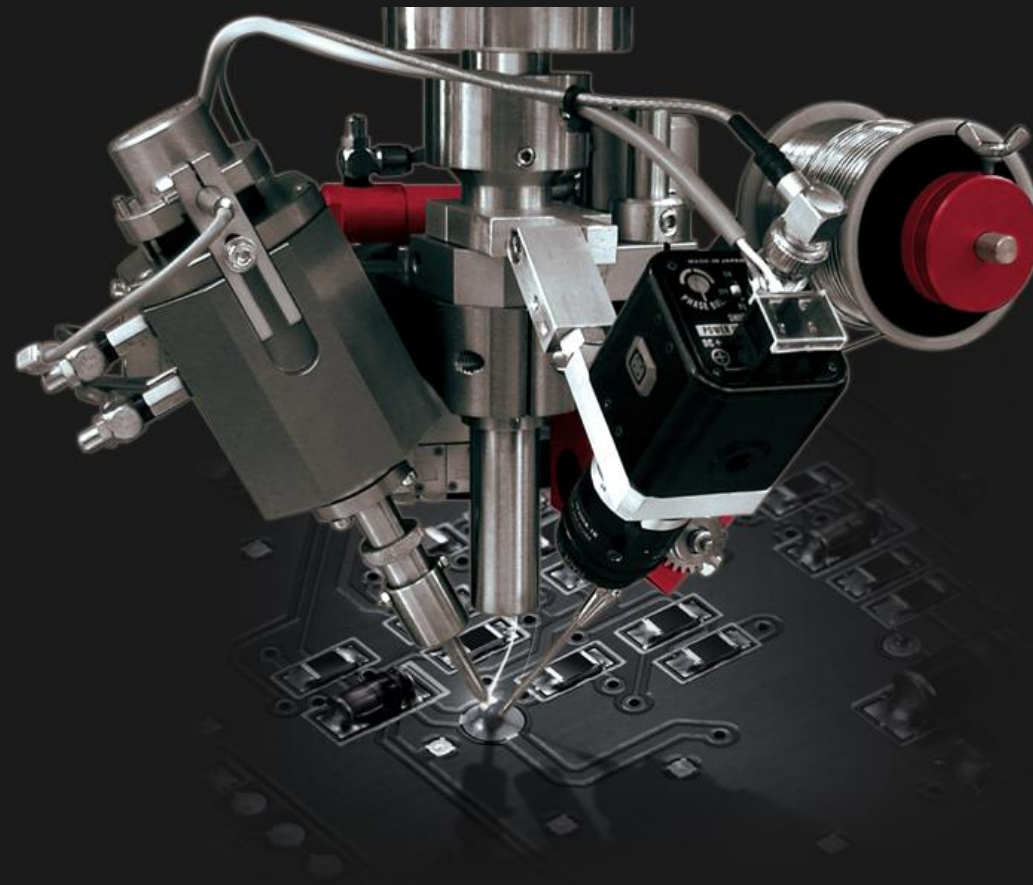


Design Guide for Using the Iron Soldering Head



mta

ROBOTICS FOR SOLDERING · DISPENSING





mta

ROBOTICS FOR SOLDERING · DISPENSING

Preface - caveats

- These guidelines provided in this document **are greatly simplified**.
- As a general rule, it is better to design the joints as large as possible, with as much space between other joints, and as large of a keep-out area as possible.
- Even if you follow everything in this guide, there can still be unaccounted variables that prevent us from being able to solder the joint.
- We are soldering experts – not board design experts! The boards must be functional and safe (*not overheat, etc.*)
- **If possible, it is always best to include us in your design review process.**



Why does robotic soldering have special requirements

Soldering robots are different from hand soldering in the following ways:

Hand Soldering

- Operators use their eyes to locate the joint, solder wire, and solder tip.
- Operators constantly watch the solder flow while making constant adjustments in real time.
- Operators can move heat around the joint by moving the soldering iron.
- Operators can push the solder around the joint by manipulating the solder and iron.

Robotic Soldering

- Robots always move to the same position, so the joint must be precisely located with a fixture *(or expensive vision systems)*.
- Robots cannot adapt to each joint in real time.
- Robots can only apply heat where the iron makes contact.
- Robots can only apply the solder wire in one place, so they rely on heat migration, the capillary effect, etc. to move the solder around the joint.

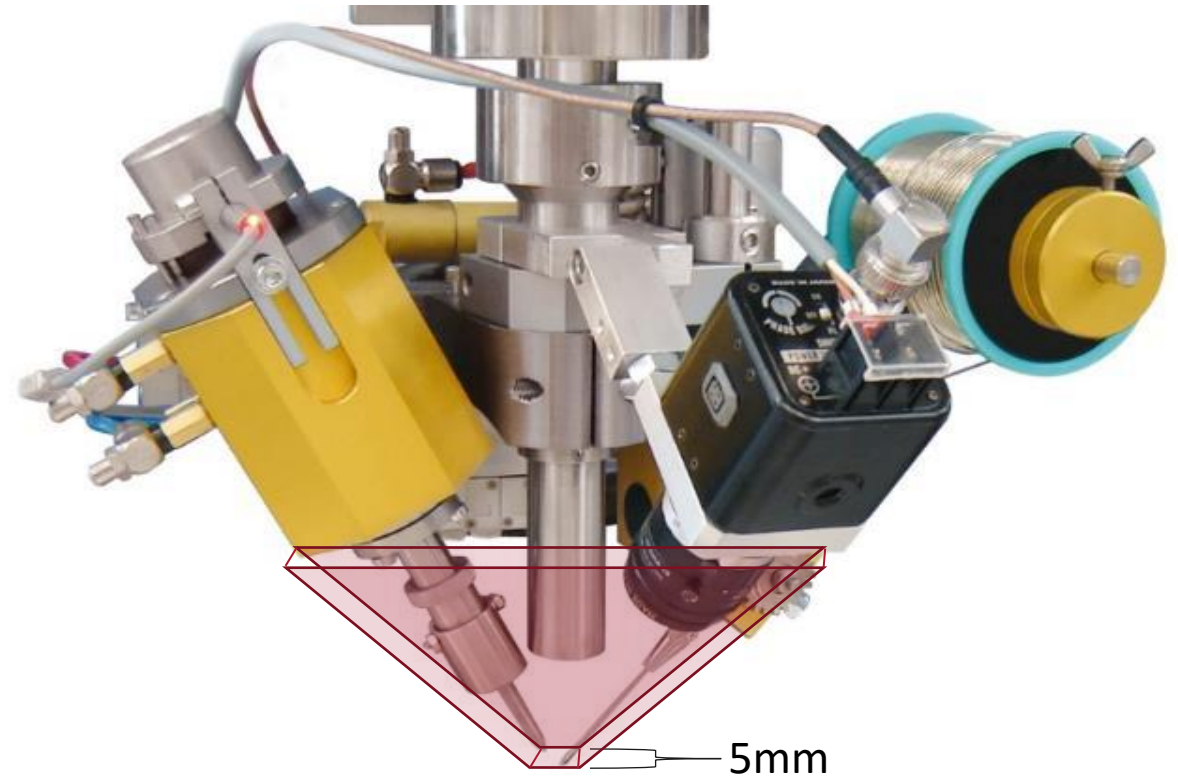


mta

ROBOTICS FOR SOLDERING · DISPENSING

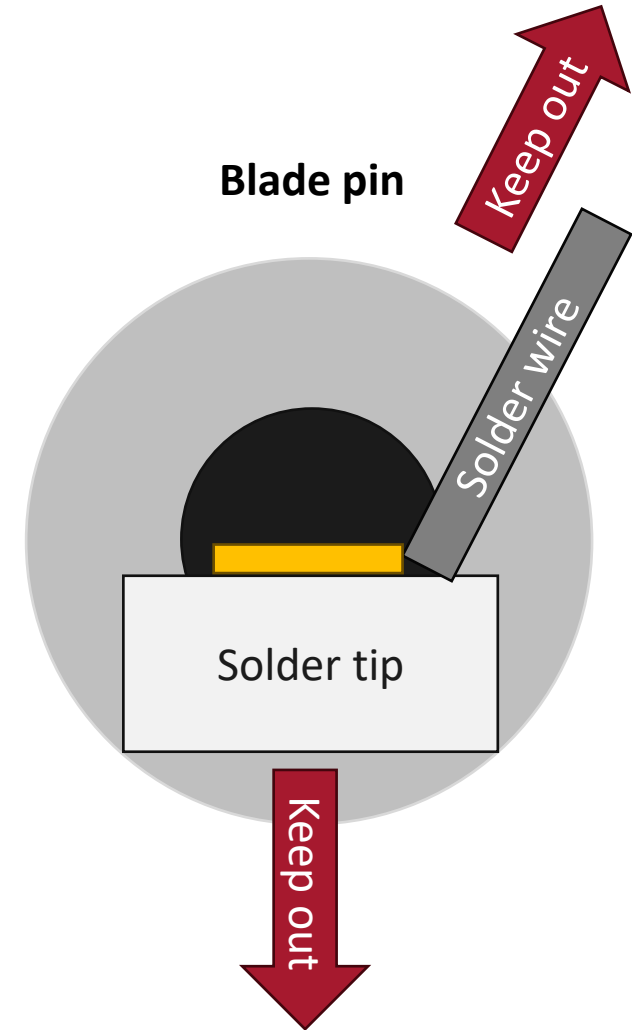
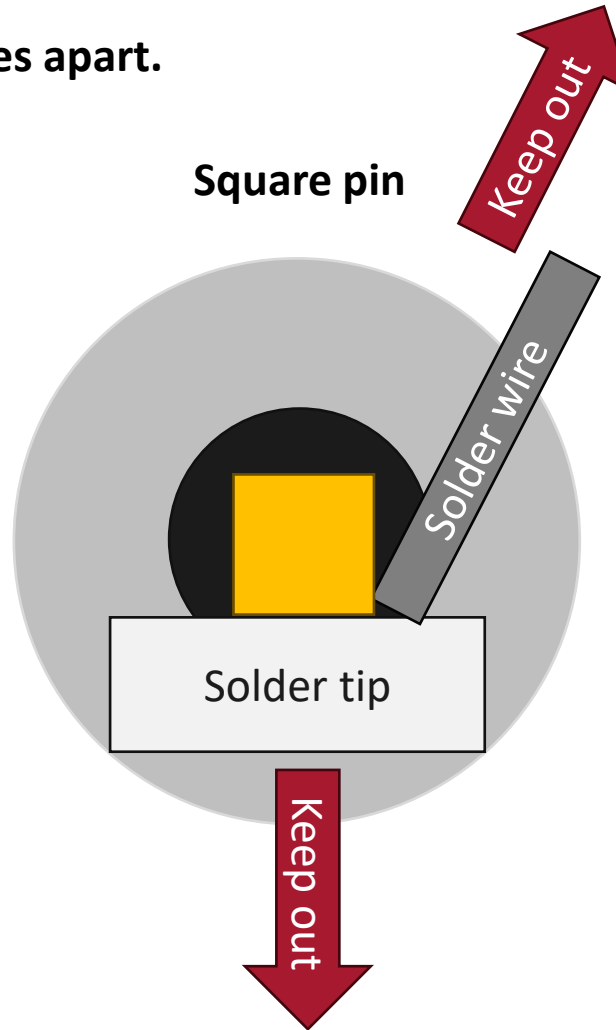
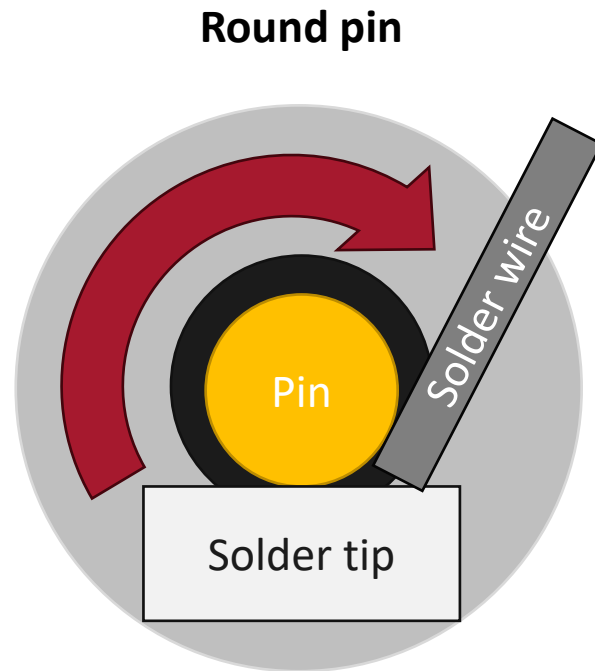
Keep-out area for the iron head

- The head can be rotated to avoid obstructions, but you can see in this photo that the wire feeder and soldering tip approach from opposite sides.
- The keep-out area applies to large components, side walls, or other tall obstructions. Additional solder joints and short components can be in the keep-out area.
- A **grossly oversimplified** rule of thumb is to allow a V-shaped opening 5mm wide to extend 45° from the base of the pad.
- We **highly recommend** using a solid model of the head in conjunction with your part to determine the safe keep-out area. We can provide this upon request.



Angle of approach

The tip and wire feed are about 170 degrees apart.



We can rotate the head to any angle on round pins, so the keep-out area is less critical.

The keep-out area should extend perpendicular to a flat surface on square and blade-type pins.



mta

ROBOTICS FOR SOLDERING · DISPENSING

Smallest standard tips for soldering irons

Liste Lötspitzen	
List of soldering tips	
Liste pannes de brasage	

mta offers three tips with small contact patches. If product design allows, it is always better to use a larger tip. Larger tips last longer, have a higher thermal mass, and this list of advantages goes on.

Nevertheless, this guideline is intended to represent the smallest allowable pad size for the soldering iron.

Note: If you must have a smaller pad size, we also have laser soldering available.

MTA 630	L = 35 mm		
	LT = 12 mm		
	a = 20.8 °		
	E = 1.5 mm		
	W = 0.8 mm		
	TH = 0.5 mm		

MTA 641	L = 35 mm		
	LT = 12 mm		
	a = 20.8 °		
	E = 1.5 mm		
	W = 0.8 mm		
	TH = 0.5 mm		

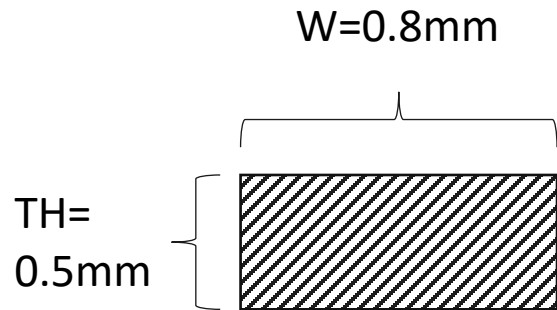


mta

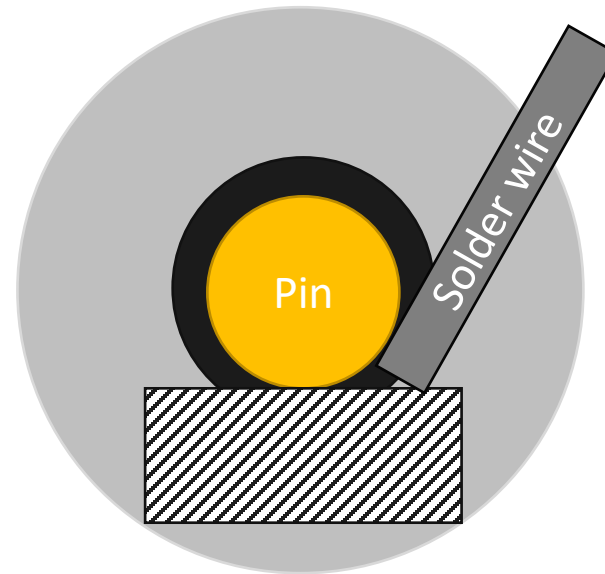
ROBOTICS FOR SOLDERING · DISPENSING

Tip's contact patch on the pad (*trough hole*)

Tip contact patch



Through-hole joint



This shaded area demonstrates the tip's contact patch on the pad. It is not intended to provide a dimensional reference for pad size.

Our soldering robot is repeatable to 0.001"; however, the pad design needs to be large enough to account for all the stacked tolerances within the part and the part fixture to allow the tip to accurately and repeatedly land on the pad.

Tolerance considerations are explained more on slides 15-17.



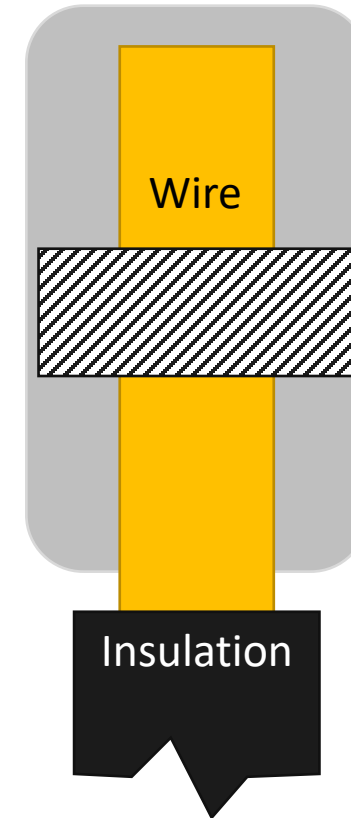
mta

ROBOTICS FOR SOLDERING · DISPENSING

Tip's contact patch on the pad (*surface mount*)

- For surface mount (wire on pad), the pad should be at least 2x the thickness of the wire (*if possible, wider is better*).
- The insulation should not touch the pad.
- We have tips with notches that can help center the wire, but **you cannot rely on this to position the wire**. The tip is always hot, so the solder will always be molten while the tip is touching it. If the wire is misaligned before soldering, it will spring back before the solder solidifies.

Surface mount

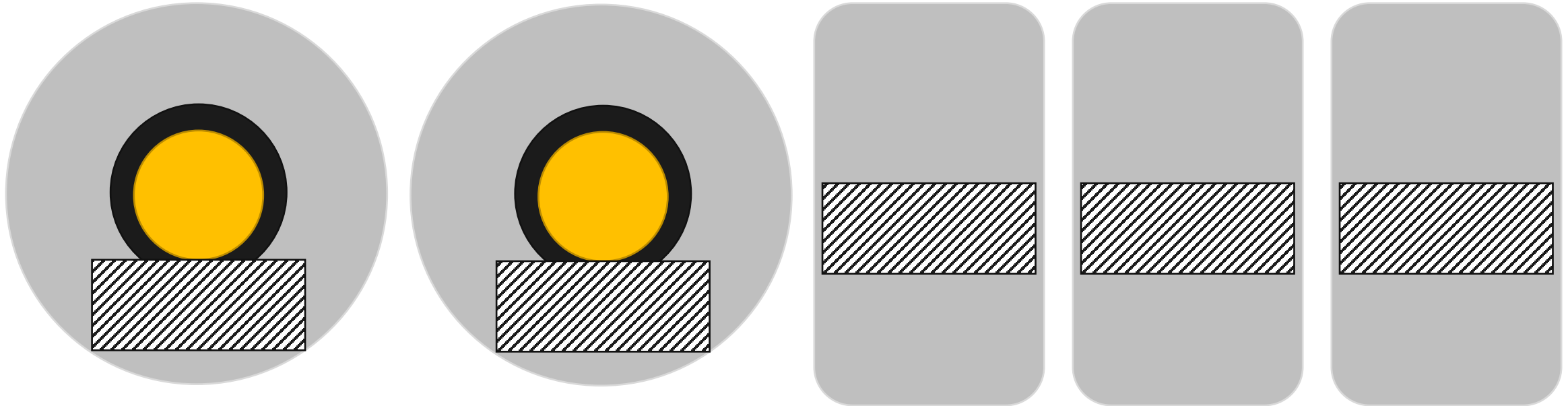




mta

ROBOTICS FOR SOLDERING · DISPENSING

Smallest recommended distance between joints

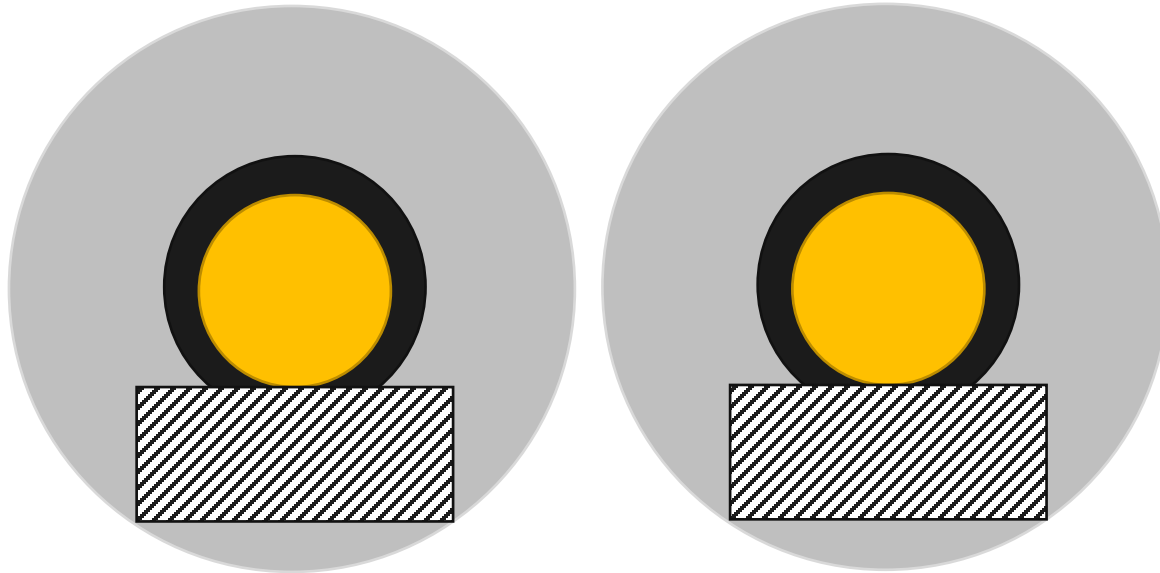


Unlike a selective mini-wave, the solder should not leave the pad, so as long as the solder pads are not touching, the solder should not bridge. If bridging occurs, it is most likely because too much solder is applied and exceeds the pad's capacity, so tight process control is critical. mta offers very tight control over the wire-feed amount.

Pragmatically speaking, this should be tested. Some solder materials work better than others, so if you are required to use a specific solder, testing with that solder is critical.

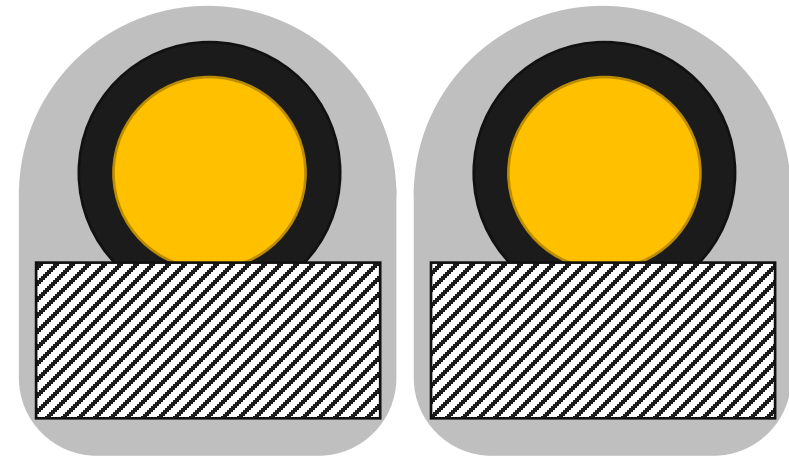
Modified solder pad shapes

Inefficient use of pad space



Round solder pads are an inefficient use of space. In the above illustration, the round pad is larger, but the contact patch is smaller. Also, the pitch between pins must be wider.

Solder pads optimized for tight spaces



The rectangular and D-shaped pads are smaller, but the contact patch is larger, and the pin pitch is closer.

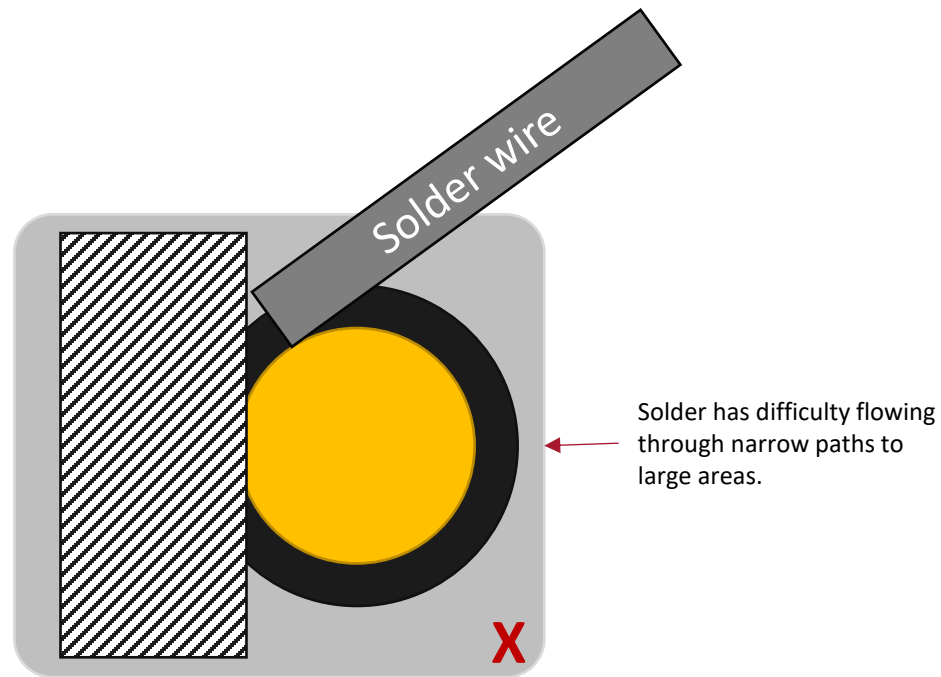
Be sure to keep the large side of the pad on the side with the most access for the tip (*towards the keep-out space*).



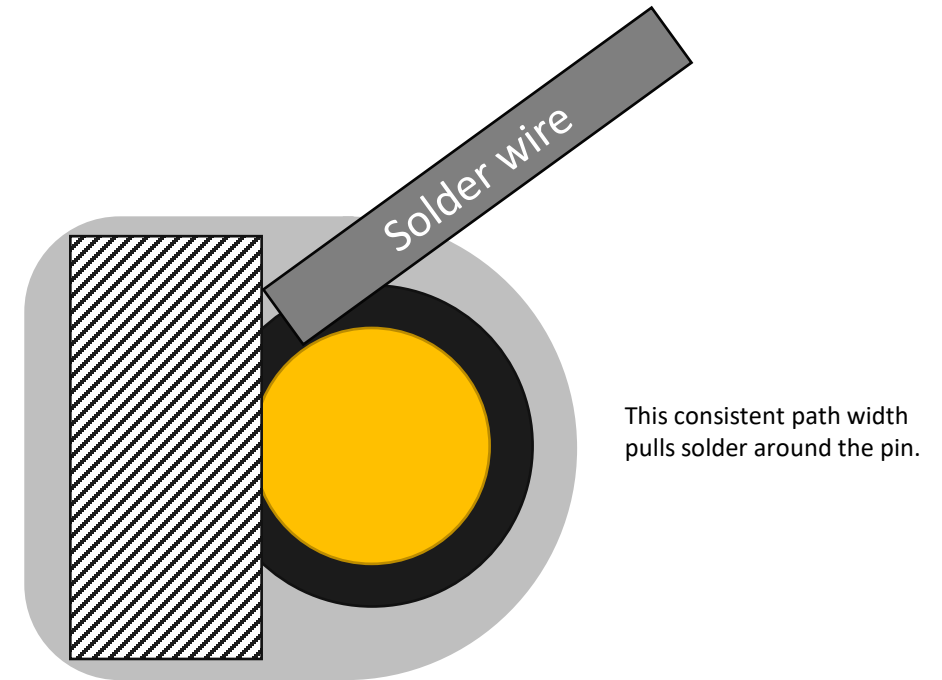
mta

ROBOTICS FOR SOLDERING · DISPENSING

Modified solder pad shapes (continued)



We sometimes see offset rectangular pads. While they use less space than round pads, solder has difficulty covering the area marked by the red X.



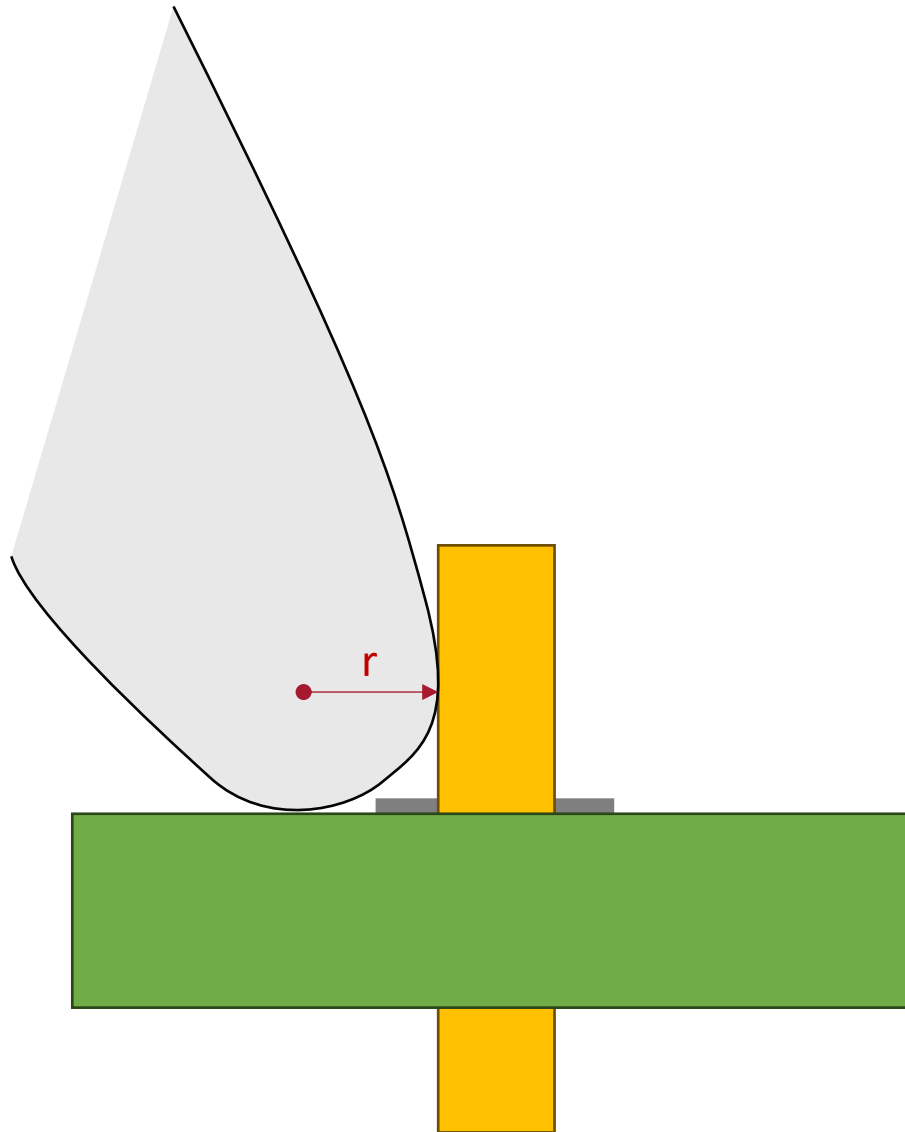
D-shaped pads do not have large open spaces opposite the wire feeder, so they facilitate better pad coverage.



mta

ROBOTICS FOR SOLDERING · DISPENSING

Pads can be too small



As explained earlier, larger solder pads are better whenever possible.

If small pads are unavoidable, the question becomes, “How small is too small?”

Solder tips are always slightly rounded to prevent abrasion on the part and excessive wear on the tip.

If the solder pad is smaller than this radius, the tip will not contact the pad at all.

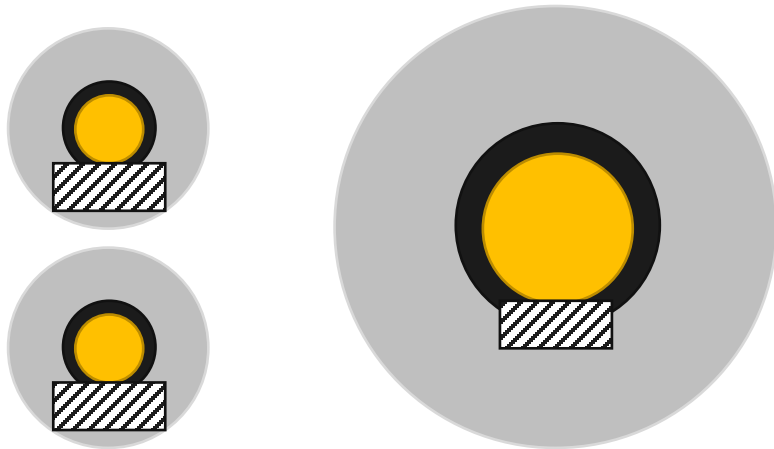


mta

ROBOTICS FOR SOLDERING · DISPENSING

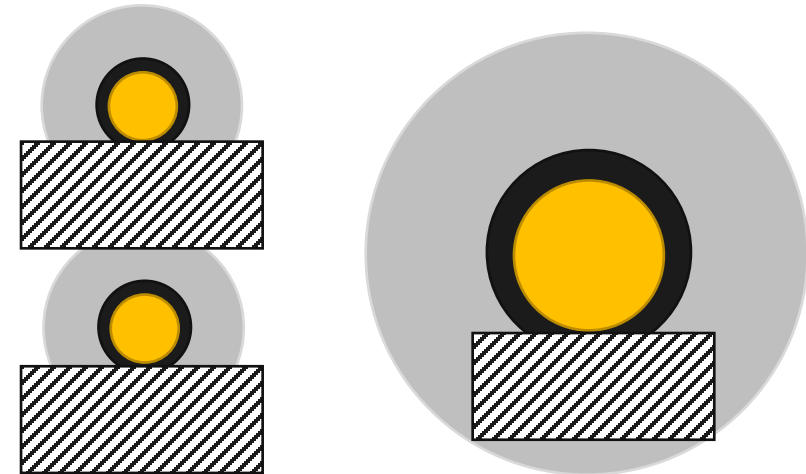
Avoid dissimilar-sized pads

Scenario 1: Use a tip sized for the small joint to solder both joints.



In this case, the large joint will take a very long time (if ever) to reach reflow temperature.

Scenario 2: Use a tip sized for the larger joint to solder both joints.



In this case, the large tip will can touch multiple pads and cause the solder to bridge the joints.

Medium-sized tips are an option, but not ideal. **A preferred solution is on the next slide.**

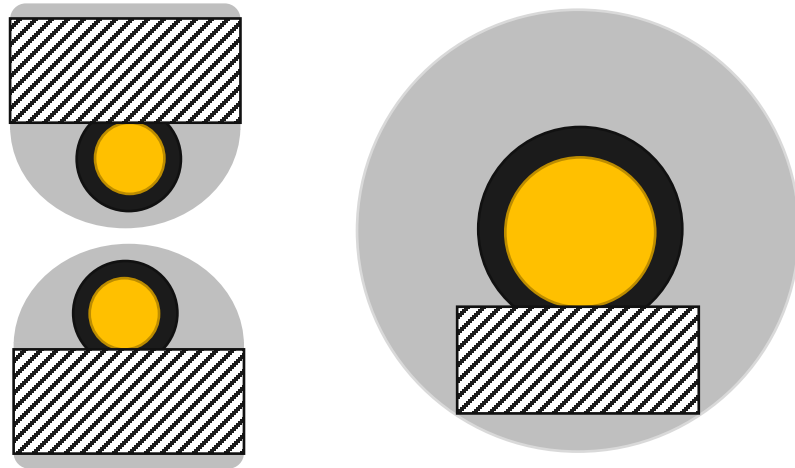


mta

ROBOTICS FOR SOLDERING · DISPENSING

Avoid dissimilar-sized pads (continued)

Preferred scenario: Create larger pads around the smaller joints to accommodate the larger solder tip.



By using D-shaped pads, the pitch between joints will be closer together than with large round pads.

Lastly, if there is no way to make the joints similar in size, the least preferred option is to run the boards in two batches and change the solder tip between batches.

While two batches will take longer, it is often faster than waiting for small tips to heat large joints, and larger tips will not bridge small pads.



mta

ROBOTICS FOR SOLDERING · DISPENSING

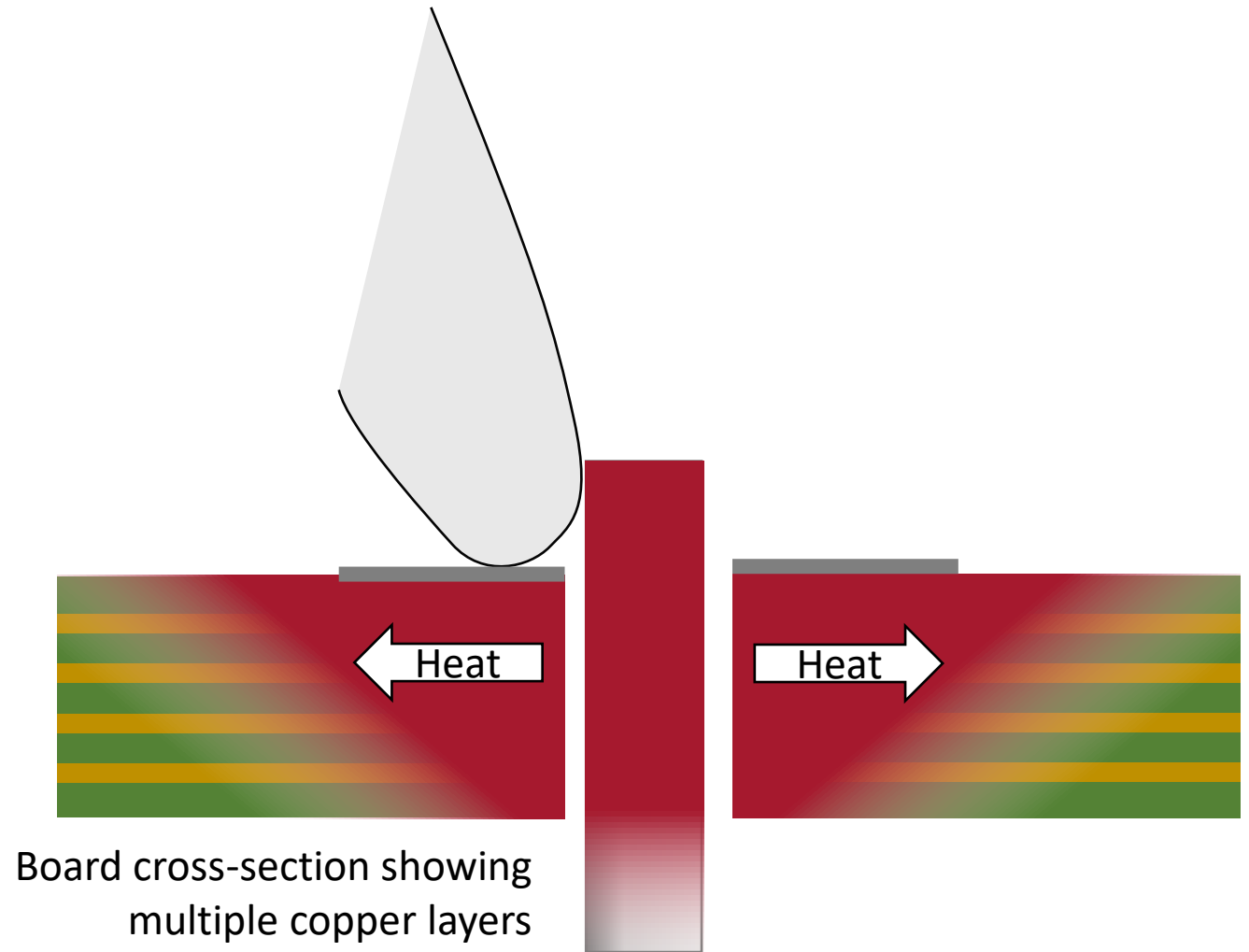
Thermal properties of the PCB

The heat needs to move through the PCB vertically to get a good fillet on the bottom side. However, copper planes in the PCB pull heat away from the solder joint laterally.

Multi-layer boards, and heavy copper planes can pull heat away from the joint faster than the tip can put heat into the joint, preventing a good bottom-side fillet.*

Heavy copper layers are common on the ground plane but also appear on boards that actuate devices directly from the board. These devices require more amperage than board components.

*It is a common misconception that more power will heat the board faster. However, it is the contact patch that limits how fast heat moves into the board.

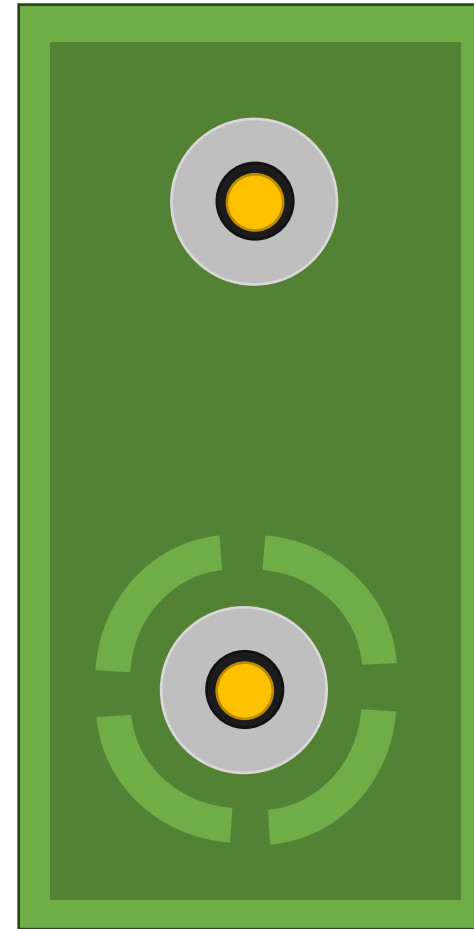


Thermal properties

Heavy copper planes can be mitigated with thermal reliefs around the joint.

There are many versions of thermal reliefs, this illustration shows the wagon wheel design.

Notice, we are soldering experts – not board design experts. The spokes of thermal reliefs must be able to handle the current expected to pass through them. Work with your board designer to reach the optimal balance.



Joint without thermal relief

Joint with a “Wagon wheel” thermal relief



mta

ROBOTICS FOR SOLDERING · DISPENSING

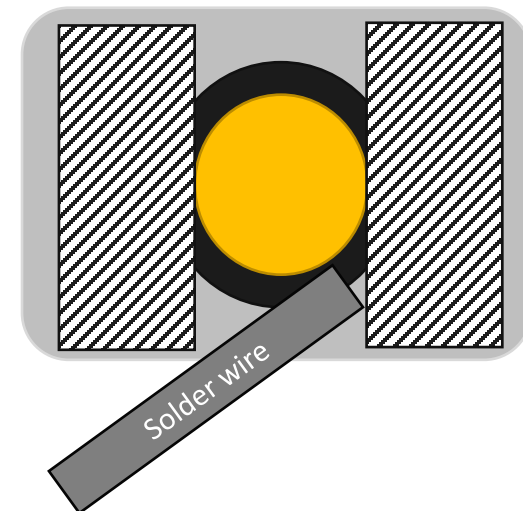
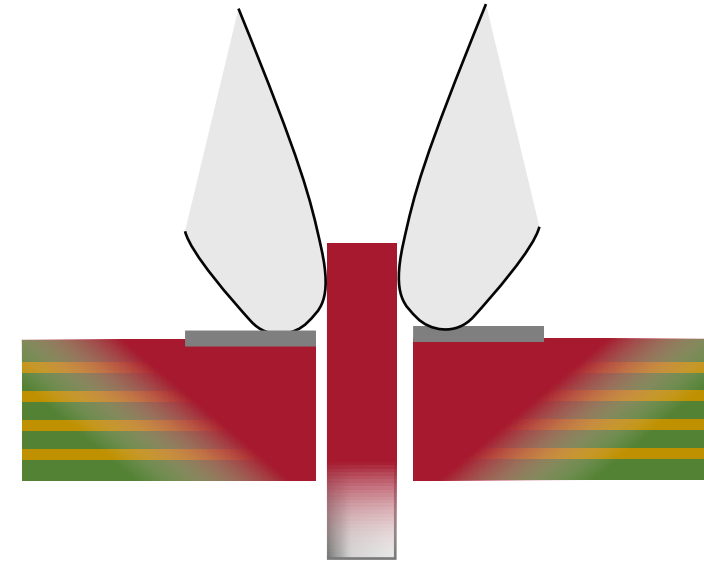
Double-Iron Head

In cases where thermal mass is extremely high, the heat can migrate away from the joint faster than a tip can apply heat.

One option is to use two irons at the same time. Again, the objective is to push heat into the joint faster than the board can pull it away. Two irons double the power, but more importantly, they double the contact patch for faster heat transfer.

High-mass joints should be tested early, so the solder pad can be optimized for two tips, and the budget is adjusted to include a second iron.

Note, we might also be able to use induction, but this comes with a different set of concerns we must address first.



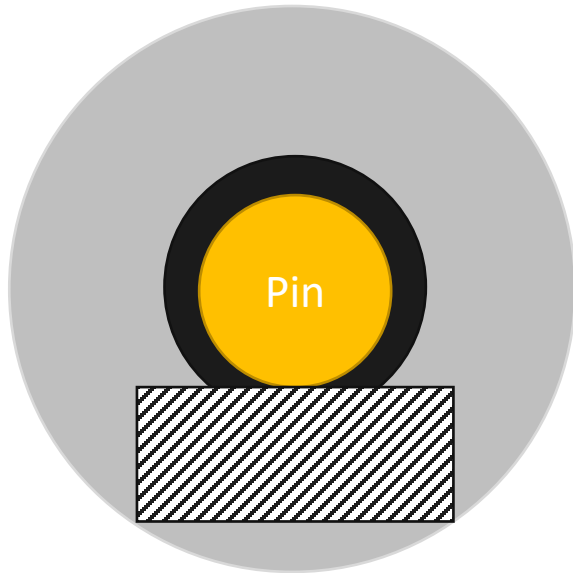


mta

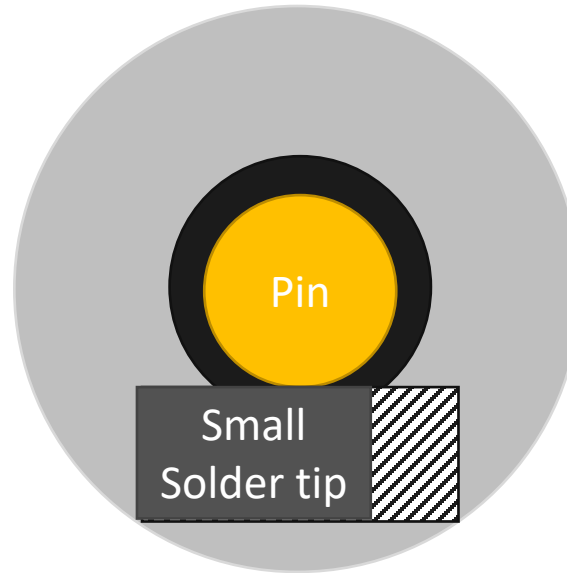
ROBOTICS FOR SOLDERING · DISPENSING

Why are tolerances so important to board design?

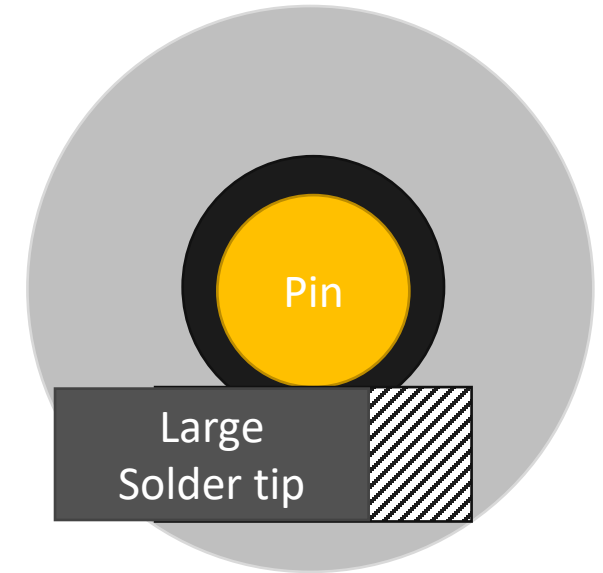
Consistency is paramount to automation. Tighter tolerances allow us to maximize the tip size, decrease process time, and improve joint quality.



Tolerances prevent us from ideally sizing a tip to cover the entire contact patch.



An undersized tip will fit on the solder pad at maximum tolerance. However, this comes at the expense of slower heat transfer (*longer process time*).



A tip sized to cover the entire contact patch will overhang the pad at maximum tolerance. This can damage adjacent components and bridge adjacent pads. Also, the portion overhanging the pad does not transfer heat into the joint.



Consider the tolerance stackup of the part AND the fixture

- With any application, you need to be sure that the pad can be located accurately and repeatedly under the tip's contact patch.

- There are at least 6 levels in the tolerance stack up between the robot and the solder joints.
 - The robot has a repeatability of 0.01mm.
 - The fixture's locating pins allow 0.005" so the operator can swap fixtures without binding.
 - The fixtures are machined within a tolerance.
 - There is a gap between the fixture and part, so the parts can be loaded and unloaded quickly.
 - If the board is in a plastic housing, the housing has tolerances.
 - Various surfaces of the board have different tolerances relative to the joints.
 - The fixture might be made of several parts, so assembly can create a variance from fixture to fixture.

- In cases where it is impossible to locate the pad accurately, we can use visual realignment to acquire the solder joints.
 - The best practices are to add proper fiducial markings (*not reference from pads or components*), and put the marking on each part (*not the panel*).
 - The vision system will locate the fiducial and set a new zero-point for each part and reference all the joints relative to the fiducial. (*The solder joints need to be accurately located relative to the fiducial.*)
 - **Visual realignment systems are an expensive, they restrict the work area, and increase the process time, so they should **not** be the first design approach. Precautions should be taken to avoid the need for visual realignment if possible.**



The part will need to be fixtured

Keep in mind that someone must design a fixture that can hold this part accurately.

- Panel breakaway tabs have huge tolerances, so avoid placing them where the board must interface with the housing or fixture.
- Instead, provide features to locate the board with smooth edges or internal features that have tight tolerances.
- If possible, design the product so the fixture can locate the PCB – not the plastic housing the board is sitting in.
- If soldering wires to the board:
 - Through-hole joints are better than surface mount (*wire-on-pad*) because they capture the wire ends, so they are easier to manage.
 - If you must have surface-mount wires, locate them close to the edge, as it is difficult (*or impossible*) to extend clamps to the center of the board.
 - You might be able to incorporate wire holders/clips in the housing design, which will simplify the fixture.

An added advantage to parts that are easy to fixture is that they require less time to load and unload from the fixture. Faster throughput is always appreciated.



Double-sided boards

Try to keep all the solder joints on one side. If you must place components on both sides, there are three primary options to solder them.

- **Sometimes we can solder a through hole from either side, but we must test this.**
 - Does the solder move through the board sufficiently to create a quality joint?
 - Does the component block access to the joint under it?
 - If we solder wires from the insulated side, does the insulation melt?
 - There are more considerations, but the point is, do not assume we can solder the joint from either side. We must test this using real parts, so prototypes will be required.

- **Flip the entire fixture:**
 - This adds a manual step (*flipping*) and a second soldering process.
 - The fixture will need to hold the part securely upside down.
 - The fixture design needs to allow access from both sides.

- **Flip the part:**
 - Flipping the part is a last resort option.
 - This will require another set of fixtures.
 - This will require substantially more process time to unload and reload the part a second time.
 - This will require a second soldering process.

- As previously mentioned, these guidelines are intended to improve the odds we can solder the joints, but there is no guarantee that following these guidelines will result in good solder joints. There are too many other variables to consider: materials, layers, thermal mass, solder alloy, etc. it will work.
- Testing is the only way to really know if we can solder the joints.
- We are happy to evaluate your drawings and test your prototypes, so please keep us involved in the design process.