Assembling SRD Products Onto Customer's PWBs

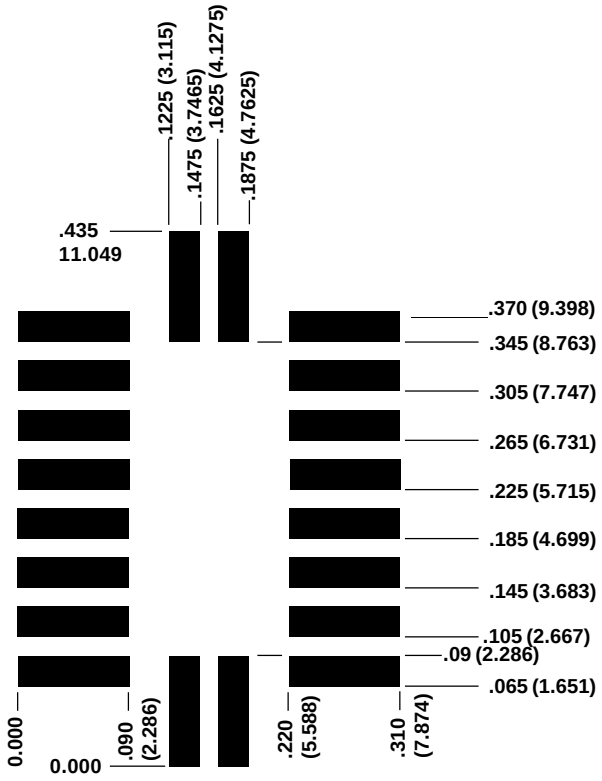
Assembling SRD Products Onto Customer's PWBs

- Considerations/Topics
  - PWB footprint (layout)
  - Solder mask use/layout
  - Type of solder
  - Temperature profile
  - Type of soldering equipment
  - Stencil
  - Flux
  - Cleanup
  - Standards

PWB Footprint/Layout

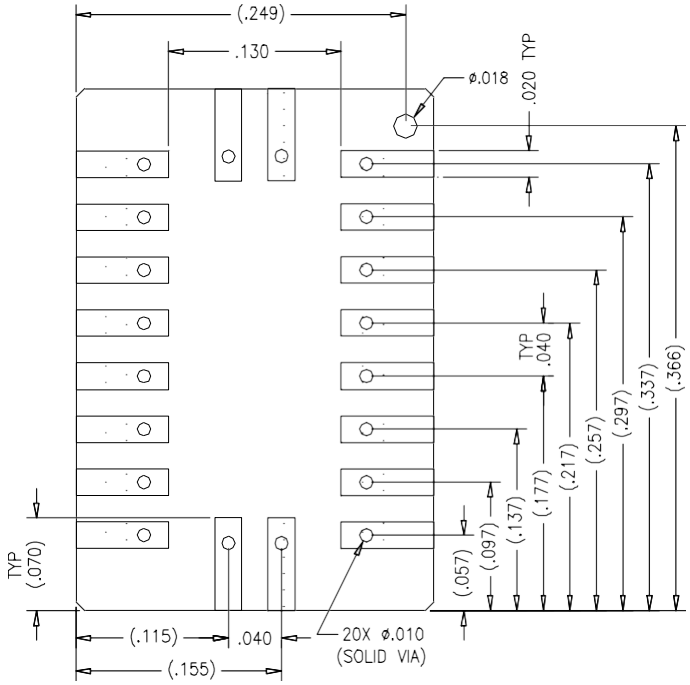
- Depending on which reference manual you use, or internal process preferences, the PWB footprint will be:
  - the same size as the part solder pad layout, or
  - slightly larger for each feature (typical).
- We chose to make the PWB footprint for each package slightly larger
  - SM20H: Pads are 0.005” wider and 0.020” longer than those on the package
  - SM20L: Pads are 0.005” wider and 0.025” longer than those on the package
    - Allows for some misalignment during part placement
    - Solder flow is visible - No castellations on these parts.

Footprint Comparison



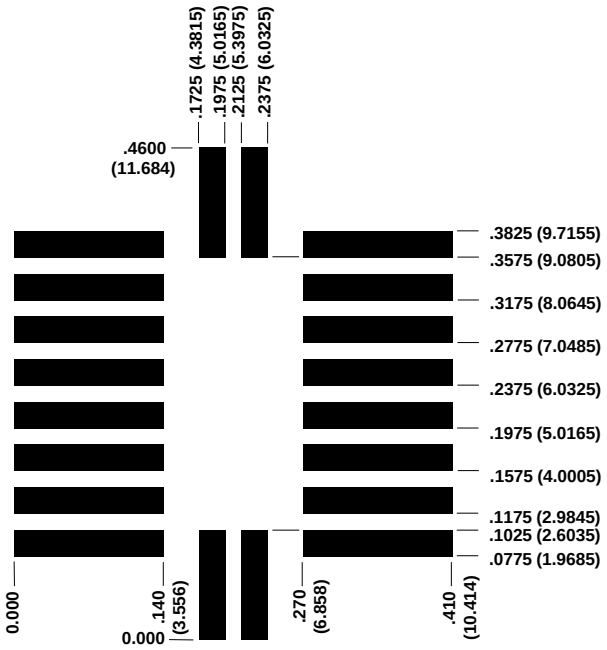
Dimensions in inches (mm).

SM20H



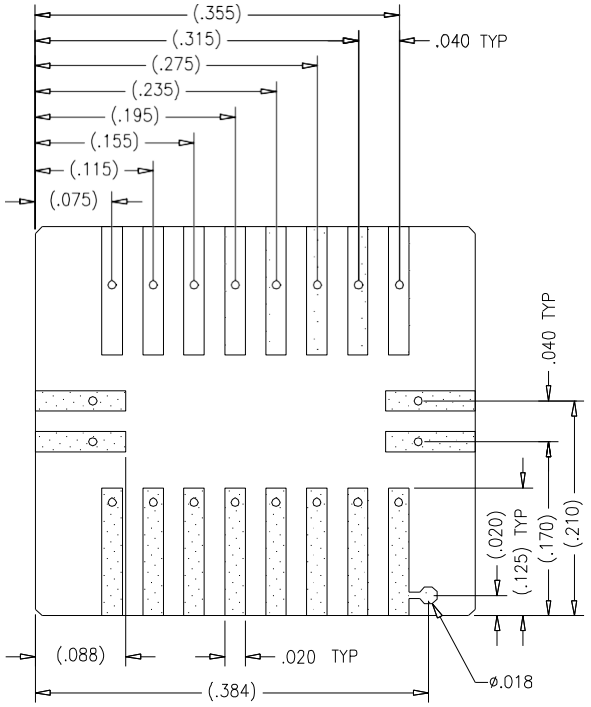
DETAIL D
(MP-400, BOTTOM VIEW)

Footprint Comparison



Dimensions in inches, (mm)

SM20L



DETAIL D
(MP-400)

Castellations

- These parts/packages do not have castellations
  - Castellations are expensive (NRE & recurring cost)
  - The contribute to distortions of the substrate which can complicate assembly (lower yield)
  - Customers want lower cost products
- Customers like castellations
  - They can see a solder fillet - visual inspection
- Castellations are allowable by industry specifications
  - IPC-A-610 allows & specifies a minimum of 25% solder flow up into the castellations, if they are present (Section 10.2.4, paragraph 6)
  - IPC refers questioners to Section 10.2.1, “Chip Component/ Bottom Only Terminations” for parts without castellations.
  - IPC plans a revision which will specifically address non- castellated parts sometime in the next three years

Solder Mask

- Depending on which reference manual you use, or internal process preferences, the Solder Mask footprint will be:
  - the same size as the PWB solder pad layout, or
  - slightly larger for each feature
  - slightly smaller for each feature
- We chose to make the solder mask openings the same size as the PWB pads

Solder

- Single or two-sided assembly?
  - Two sided typically requires a higher melting point solder for one side of the assembly
- Form
  - Typically a screenable paste consisting of very small solder particles suspended in flux
- Metal content
  - The amount of solder metal in pastes is specified as “% solids” or “% metal”
  - Metal content is a determiner of the final thickness of the soldered joint
  - “Thicker is better”

Type of Solder

- We use SN62 (Federal Spec. QQ-S-571F)

– Approximate composition = 62%Sn/36%Pb/2%Ag
– Liquidus (melting point) = 189°C
– Solidus (freezing point) = 176°C
– 80% metal content
– RMA flux

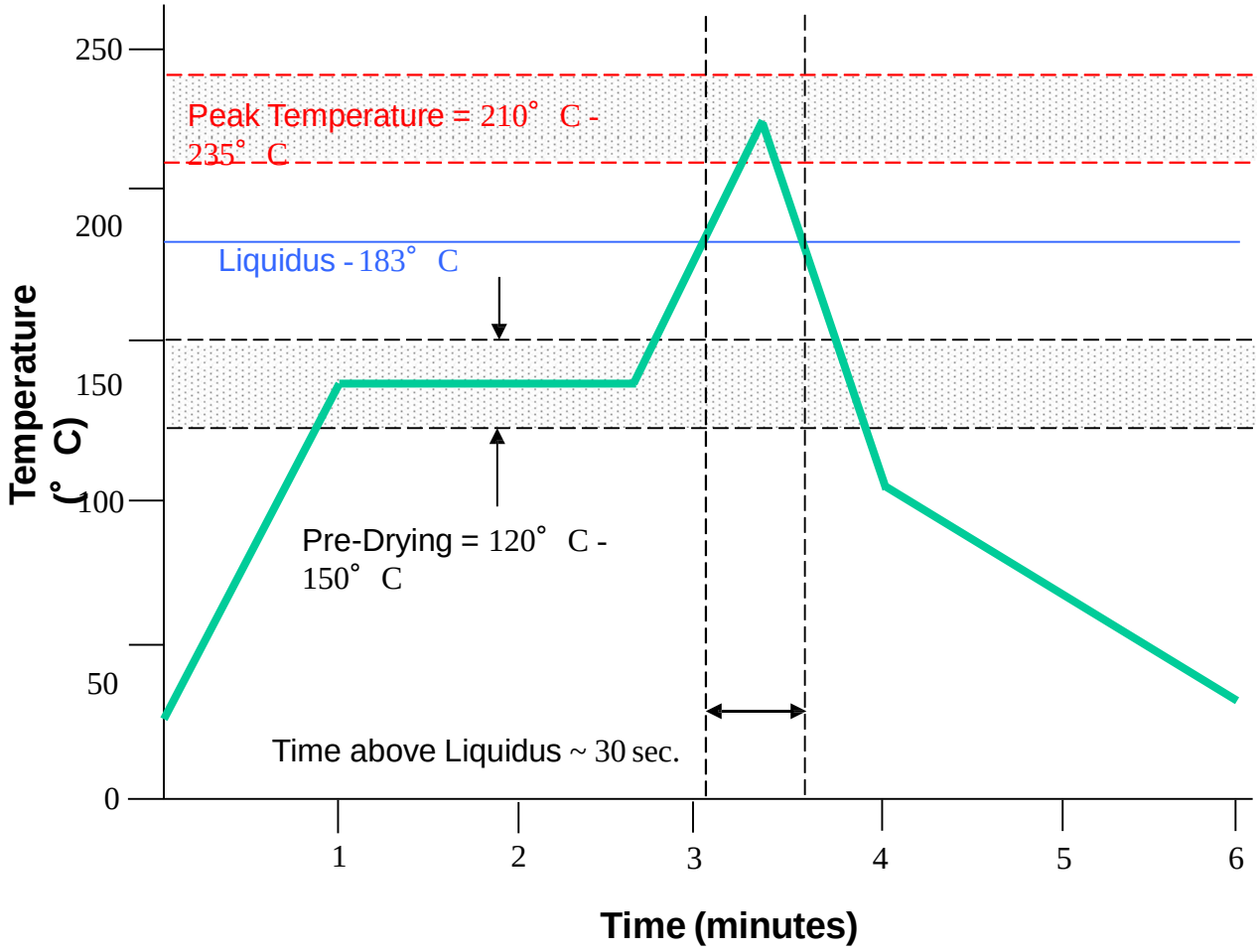
} A non-eutectic solder

- Other popular solders
  – SN63; eutectic at 183° C
  – SN60; non-eutectic; liquidus = 190° C; solidus = 182° C
- Choice of solder influences soldering profile

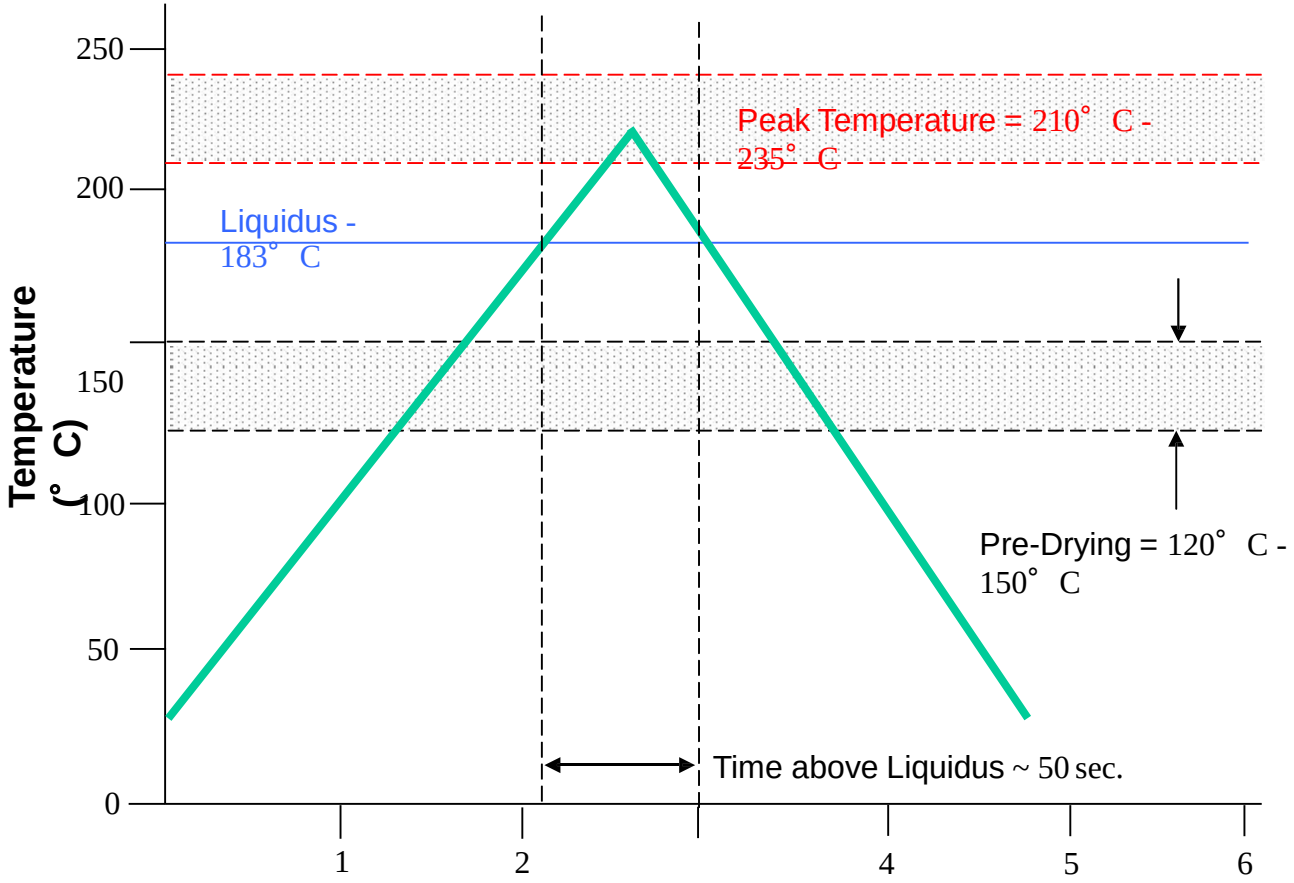
Soldering Profile

- Two Classic Profiles:
  - Ramp & hold
  - Triangular
- Exact profile depends on the equipment used
  - Never exactly duplicated
- Most popular types of equipment are:
  - Convection (U.S. typically)
  - IR (Europe & Asia)

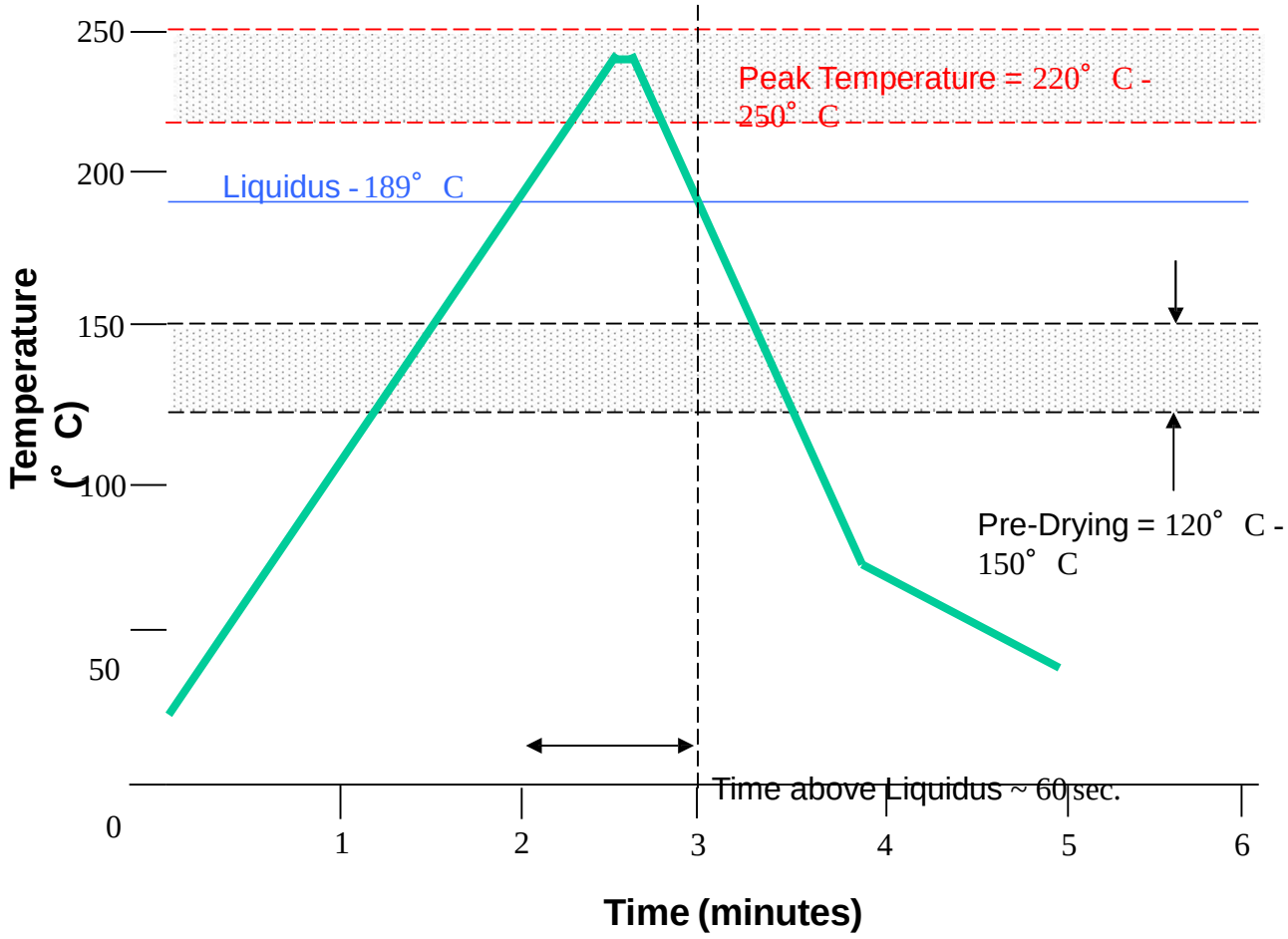
Ideal Reflow Soldering Profile for SN63 (Not SAC) Solder (Ramp & Hold)



Reflow Soldering Profile for SN63 (Not SAC) Solder
(Triangular)



Profile We Used fo SN62 (Not SAC) Solder



Reflow Soldering Equipment

- Vapor Phase - Environmental concerns
- Wave - Won’t typically be used with our products
- IR - Widely used in Europe and Asia
- Convection - Recommended

Reflow Soldering Equipment

- IR systems depend on radiated energy to heat joints to soldering temperatures
- Wavelength/spectra of the energy radiated varies with system brand and age
- Wavelength absorbed by each component varies
  - Metals typically absorb in the near-IR region
  - Organics (plastics) & oxides (ceramics) typically transparent in near-IR, but absorb middle- and far-IR to varying degrees
  - Thus, one can “burn up” one component while soldering another

Reflow Soldering Equipment

- IR systems depend on radiated energy to heat joints to soldering temperatures
- Wavelength/spectra of the energy radiated varies with system brand and age
- Wavelength absorbed by each component varies
  - Metals typically absorb in the near-IR region
  - Organics (plastics) & oxides (ceramics) typically transparent in near-IR, but absorb middle- and far-IR to varying degrees
  - Thus, one can “burn up” one component while soldering another

Reflow Soldering Equipment

- Convection systems avoid problems of IR systems while offering similar high throughput, reliable soldering
- We used a Heller, model 1700 to solder the TR products to the DR boards

Flux

- Many different flux types are available
  - QQ-S-571 designators are still used to identify fluxes (RMA, WSF-O, WSF-1, LR)
    - Water soluble and “No Clean” (LR) fluxes are increasingly popular
  - ANSI J-STD-004 reclassified fluxes according to activity
    - L0 = Some RMA, some LR
    - L1 = Most RMA
    - M0 = Some LR
    - 33 = Some water soluble
    - H1 = Most water soluble

Stencil/Screen

- Stencils are preferred by most assemblers
  - Better solder joint thickness control
- A Stencil is a thin sheet of metal with openings corresponding to where the solder joints are to be made
  - The openings in the stencil can be made the same size as the PWB solder pads, or slightly smaller or larger
- We made the openings in our stencil the same size as the PWB solder pads

Solder Joint Thickness

- Determined by “% Metal,” solder mask openings, Stencil openings, thickness of solder plating on the PWB and Stencil thickness

Wt% Metal	Vol% Metal
75	24.32
80	30.00
85	37.78
90	49.10
95	67.06

- In our case for the TR1000, (80% metal, Stencil = PWB pads, Solder Mask = PWB pads, PWB pads larger by 60% than part pads, and solder plating on PWB
  - Solder joints will be 0.002” to 0.003” thick

Cleaning

- “To Clean or Not To Clean, That Is the Question”
  - The move is on to “No-Clean” fluxes
- With any other type of flux, cleaning after assembly is very important
  - Long Term Reliability
- Height of the component above the PWB (as determined by solder joint height) and the spray pressure, fluid volume and solvent type (probably water) are keys to producing properly cleaned assemblies
- Poorly cleaned PWBs
  - Shorts/Signal Loss
  - Residue/Contamination

References

- Modern Solder Technology for Competitive Electronics Manufacturing - *Jennie S. Hwang, PhD; McGraw-Hill*
- Soldering Handbook for Printed Circuits and Surface Mounting - *Howard H. Manko; Van Nostrand Reinhold*
- Design Guidelines for Surface Mount Technology - *Vern Solberg; Tab Books*
- Surface Mount Technology for PC Board Design - *James K. Holloman. Jr.; Howard W. Sams & Company*
- Surface Mount Land Patterns (Configurations and Design Rules) - *ANSI/IPC-SM-782*
- Surface Mount & Mixed Technology PCB Design Guidelines - *David Boswell; Technical Reference Publications Limited*

Applicable Standards

- ANSI/IPC-A-610 - Acceptablity of of Electronic Assemblies
- QQ-S-571F - Federal Specification, Solder, Electronic (96 to 485°C)
- ANSI/J-STD-001 - Requirements for Soldered Electrical and Electronic Assemblies
- ANSI/J-STD-002 - Solderability Tests for Components, Leads, Terminations, Lugs, Terminals and Wires.
- ANSI/J-STD-003 - Solderability Tests for Printed Circuit Boards
- ANSI/J-STD-004 - Requirements for Soldering Fluxes
- ANSI/J-STD-005 - Requirements and Test Methods for Solder Paste
- IPC-TM-650 - Test Methods Manual
- IEC-1191-1 - Generic Standard, Requirements for Soldered Electrical and Electronic Assemblies Using Surface Mount and Related Assembly Technology
- ISO-9453-1990 - Soft Solder Alloys -- Chemical Compositions and Forms