

Emulation Solutions

An Embedded Systems Solutions Company
www.adapters.com

PolyPod

The PolyPod is the most versatile QFP adapter line in the industry. The adapters can be used as either an emulator interface, as a logic analyzer interface, or even as a surface mountable IC socket.

The user surface mounts the adapter base on the target board **in place** of the IC to be tested or emulated. The adapter base is ready to accept the actual IC if so desired, depending on the application.

■ Option 1.

Attach the emulator, utilizing the emulator interface cover. The emulator cable plugs directly into the cover through an optional receptacle. The user can install the actual IC if the emulator requires it.

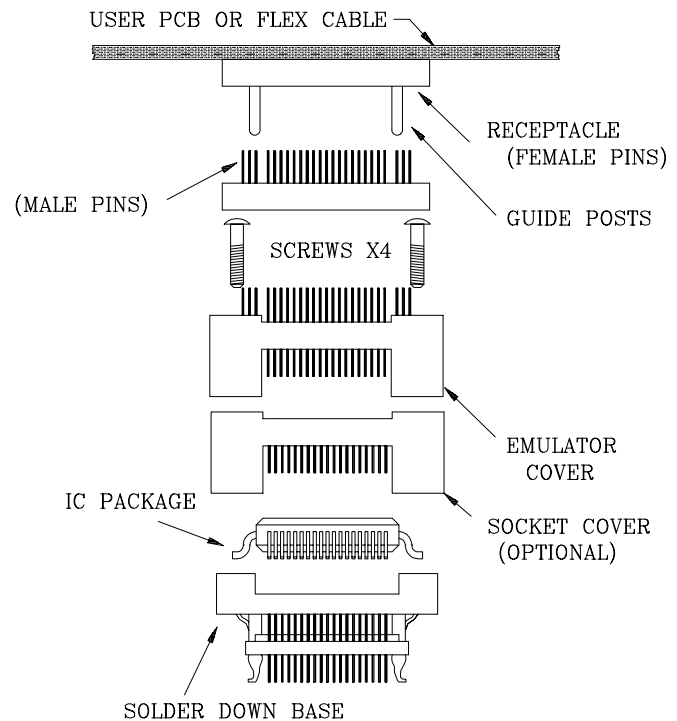
■ Option 2.

Install the actual IC into the base and connect a logic analyzer through the emulator interface cover and an interface board assembly. (The interface board assembly is available from Emulation Solutions.)

■ Option 3.

With the actual IC installed into the adapter base place the IC socket cover over the adapter and you are ready to ship the finished product.

Create a flexible debug environment



The flexibility of this adapter line allows other options, dictated by the requirements of the application.

Part number example: BASE-NQ-208Q5-SMT

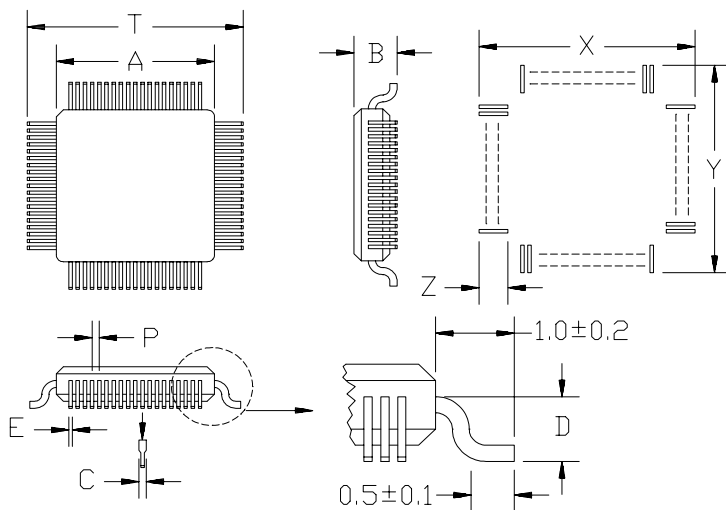
BASE-NQ	208	Q	5	SMT
Product code (socket Version)	Pin count (208)	Package Code (PQFP)	Pitch (0.5 mm)	Function code (Surface mount)

IC Dimensions							PCB pad layout (*see note below)		
Pin count	Pitch(P)	Body Size(A)	Tip to Tip (T)**	C & E max.	D min.	B max.	X min.	Y min.	Z min.
	(mm)	(mm)	(mm)	(mm)	(mm)	(mm)	(mm)	(mm)	(mm)
64	1.00	14.0 x 20.0	17.6 x 23.6	0.35	0.65	3.0	18.6	24.6	2.0
64	0.80	14.0 x 14.0	17.6 x 17.6	0.35	0.65	3.0	18.6	18.6	2.25
64	0.65	12.0 x 12.0	14.8 x 14.8	0.35	0.65	3.0	15.6	15.6	1.8
80	0.80	14.0 x 20.0	17.6 x 23.6	0.35	0.65	3.0	18.6	24.6	2.25
80	0.65	14.0 x 14.0	17.6 x 17.6	0.35	0.65	3.0	17.6	17.6	2.3
80	0.50	12.0 x 12.0	14.0 x 14.0	0.25	0.65	3.0	15.0	15.0	2.0
100	0.50	14.0 x 14.0	16.0 x 16.0	0.25	0.65	3.0	17.0	17.0	2.0
100	0.65	14.0 x 20.0	17.6 x 23.6	0.35	0.65	3.0	18.6	24.6	2.0
120	0.40	14.0 x 14.0	16.0 x 16.0	0.20	0.65	3.0	17.0	17.0	2.0
128	0.50	18.0 x 18.0	20.0 x 20.0	0.25	0.65	3.0	21.0	21.0	2.0
128	0.50	14.0 x 20.0	17.6 x 23.6	0.25	0.65	3.0	17.0	23.0	2.0
144	0.50	20.0 x 20.0	22.0 x 22.0	0.25	0.65	3.0	23.0	23.0	2.0
144/160**	0.65	28.0 x 28.0	31.2 x 31.2	0.35	0.65	3.0	32.2	32.2	2.0
160	0.65	28.0 x 28.0	31.2 x 31.2	0.35	0.65	3.0	32.2	32.2	2.0
168	0.65	28.0 x 28.0	31.2 x 31.2	0.35	0.65	3.0	32.2	32.2	2.0
176	0.50	24.0 x 24.0	26.0 x 26.0	0.25	0.65	3.0	27.0	27.0	2.0
240	0.50	32.0 x 32.0	34.6 x 34.6	0.25	0.65	3.0	35.6	35.6	2.0
256	0.40	28.0 x 28.0	30.6 x 30.6	0.20	0.65	3.0	31.6	31.6	2.0

**depopulated 160.

***NOTE:** The pad layout recommendations are for the adapters only. If users intend to solder an actual package on the target, they must consider the physical footprint of that package. Some packages have much larger footprints, where footprint is defined as “Tip to Tip dimension minus the package size”, in mathematical terms:

FP = T - A as illustrated on the IC package diagram below.



The American (JEDEC) and Japanese (EIAJ) standardization committees defined the following footprints:

- 2.0 mm**
- 2.6 mm**
- 3.2 mm**
- 3.9 mm**

In other words, a package with a 3.9 mm footprint and a body size of 28 mm will have a “tip to tip” dimension (T) of 31.9 mm. The recommended X/Y dimension for that package would be a minimum of 33.9 mm.

The recommended X/Y dimensions in our table for that size is only 31.0 mm.