

Package Land Patterns for CAD Tools

Package land patterns are available on the external web at:

<http://focus.ti.com/quality/docs/gencontent.tsp?templateId=5909&navigationId=12626&contentId=5071>

Once there, click on

[By Package Type](#)

Then on

[SOIC](#)

Then on the table

Pkg	Pins	Footprint	Industry Standard Term	Description	Width (mm)	Length (mm)	Thickness (mm)	Pitch (mm)	Maximum Height (mm)	JEDEC
DW	16		SOIC	Plastic Small Outline	7.52	10.28	2.35	1.27	2.65	R-PDSO-G

Click on the box under Footprint next to the pin count and body size desired. This gives you:

Footprints for Pin 16 Pkg DW

Select a CAD format for this footprint*

Format	Size	Date
Cadence Allegro v13.6+	7.91 KB	04 Jan 2008
Mentor Boardstation v8.7+	1.04 KB	04 Jan 2008
Mentor PADS-PowerPCB v3.5+	1.79 KB	04 Jan 2008

* my.TI login will be required

Don't see your format? [Suggest additional formats](#)

Data provided by Soft Solutions and Data. See "Warranties and Disclaimers" under [Terms of Use](#)

Which gives three choices for formats that can be put into the board design software.

(Or you can just click on the link above—if the link survived the email and you can log into it—for the one he wants, download it, and send it to him.)

Regards,

Jeff Holloway

Analog Packaging
Texas Instruments Incorporated
12500 TI Boulevard, MS 8740
Dallas, Texas 75243
214-480-2391