

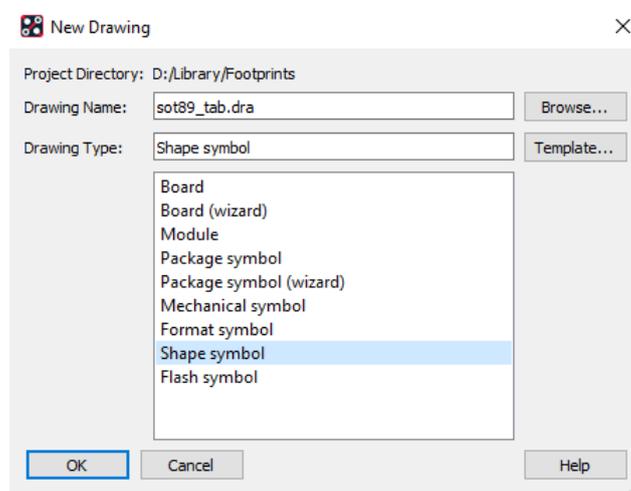


Introduction

The Cadence® OrCAD® and Allegro® Padstack Editor allows for the creation of the following pad shapes: - circle, square, oblong, rectangle, rounded rectangle, chamfered rectangle, octagon, donut and n-sided polygons, however there are times when something more complex is required, in this situation a shape needs to be created and used. This application note will show you how to create and use a shape for use as a SOT89 tab pin.

How to create an alternative shaped pad or pin

Launch PCB Editor, and create a new shape symbol: File > New... will open the New Drawing Dialogue, then change the 'Drawing Type' to Shape Symbol, enter a 'Drawing Name' and click OK.



Once in shape symbol editor draw your shape using Etch\Top to create the shape of the tab pin. The most important part of this step is to ensure that the origin point 0,0 is at the location inside the shape where you want the pin connection point to be. You can use the command line to enter specific co-ordinates to create this shape so that 0,0 is at the correct location.

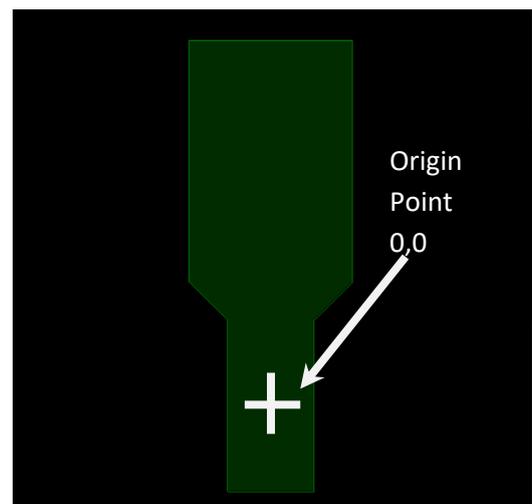
For the SOT89 tab pad enter the following at the command line after invoking the add shape command.

```
x -18 -28, ix 36, iy 72, ix 16 16, iy 102, ix -68, iy -102, ix 16 -16, iy -72.
```

An alternative method is to add this complete line at the command line and then the shape is drawn in one go: - x -18 -28; ix 36; iy 72; ix 16 16; iy 102; ix -68; iy -102; ix 16 -16; iy -72. By using the semi-colon means you can add a sequence of commands to draw a completed shape.

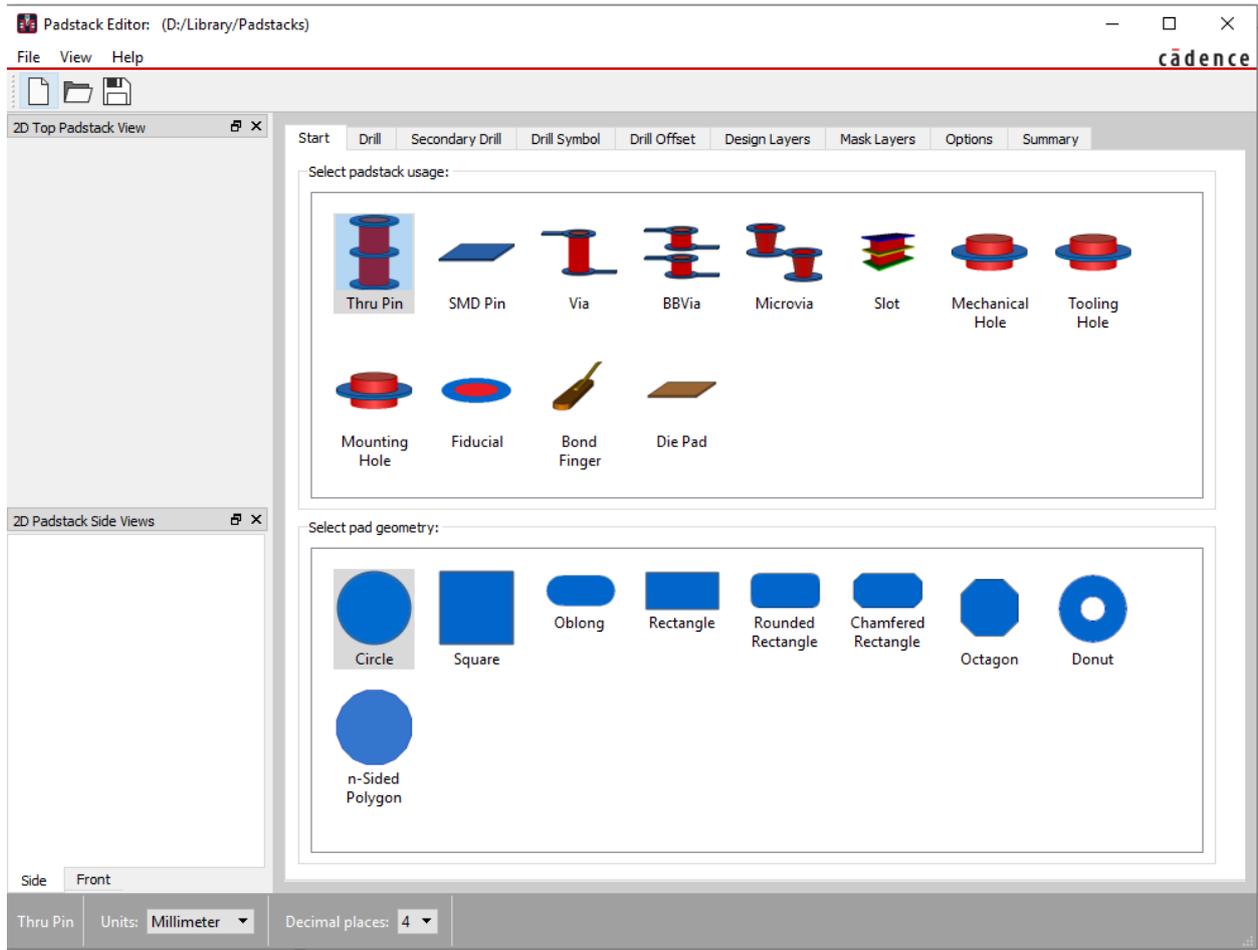
This will draw the shape shown above guaranteeing that 0,0 is at the centre of the pad shown.

Save the shape file as sot89_tab.ssm in the psmath location.

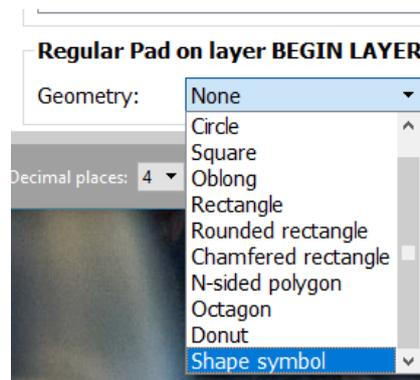


How to Create an alternative shaped pad/pin

Open Pad Designer (Start > All Programs > Cadence PCB Editor Utilities XX-XXXX > Padstack Editor XX.X)

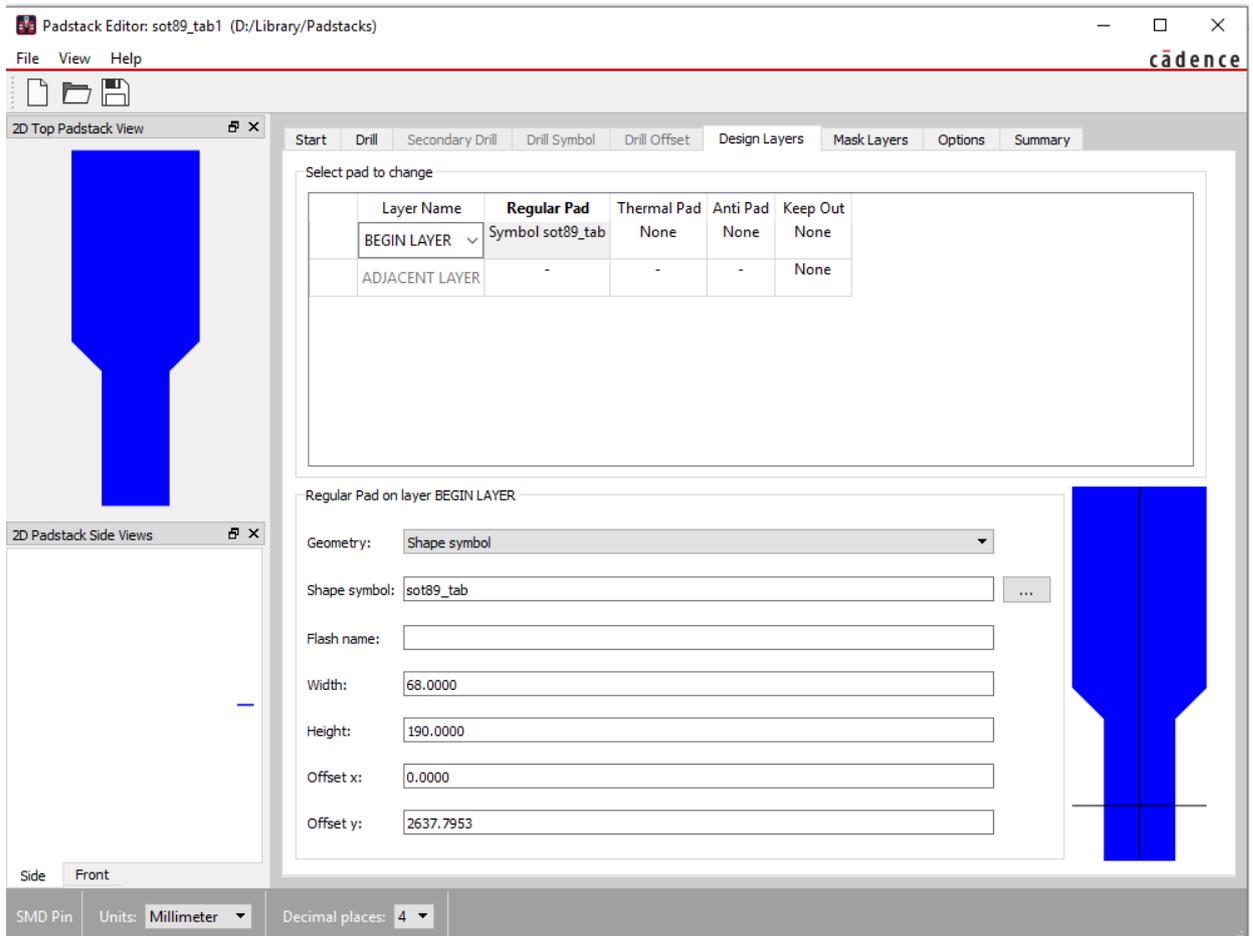
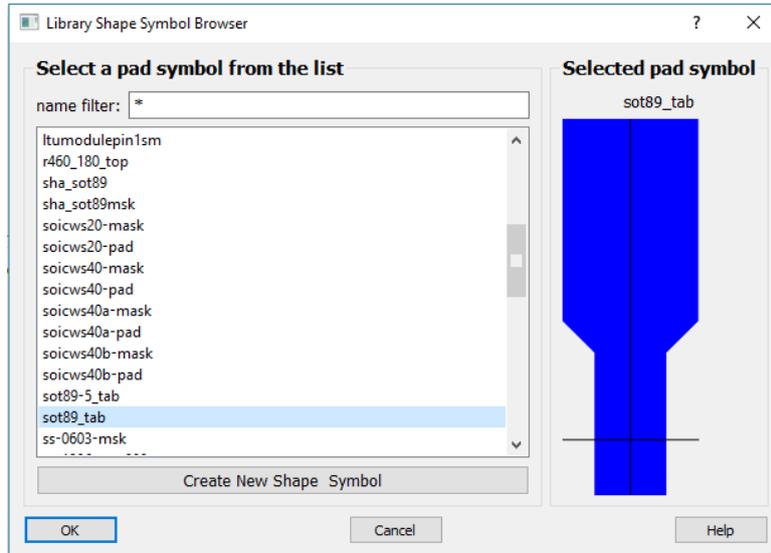


Select File > New, specify a name for the padstack ensuring the save directory is set to the padpath location. Set the Padstack usage to SMD Pin. Set the units and decimal places as required. On the Design Layers tab change the Geometry dropdown for Regular Pad on Layer BEGIN LAYER from None to Shape Symbol.



Select the browse button next to Shape symbol and choose the shape symbol you generated earlier. Another method if you haven't already created the shape symbol is to select the "Create New Shape Symbol" option which will launch PCB Editor and start a new shape symbol based on the name you specify. Once you have chosen the shape symbol and selected OK the shape name is applied to the Regular Pad for BEGIN LAYER.

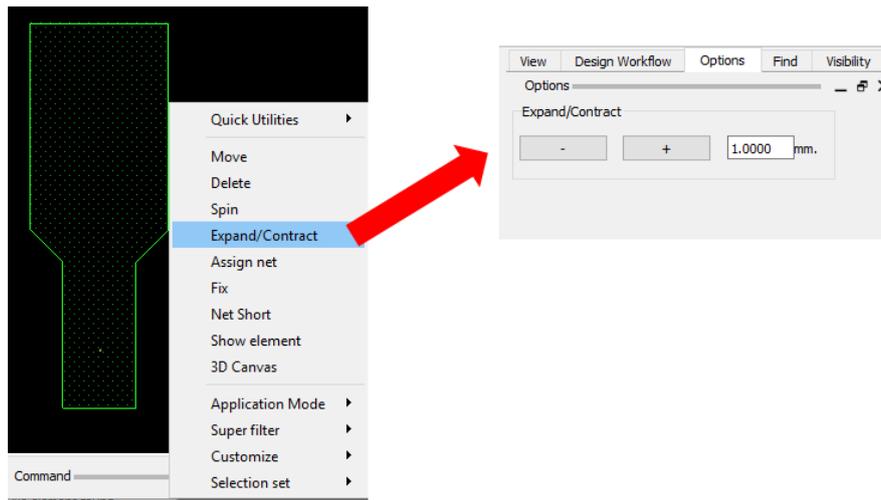
How to Create an alternative shaped pad/pin



]

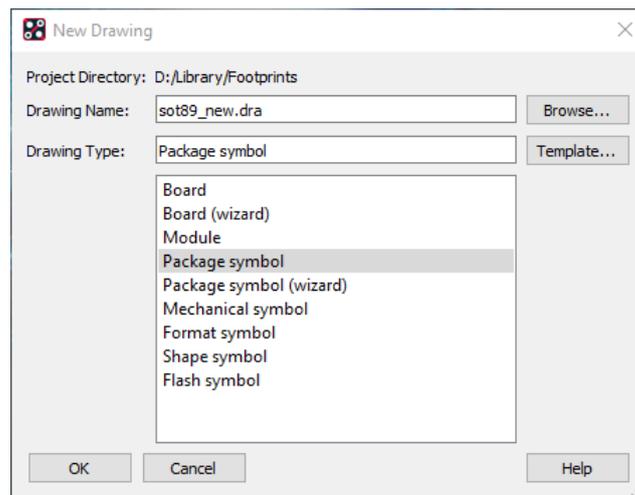
Repeat these steps for the soldermask and pastemask definitions via the Mask Layers tab. Remember if you under or oversize the pads for these layers you will need to generate different shape.ssm file as required. Once complete save the pad in the padpath location.

How to Create an alternative shaped pad/pin

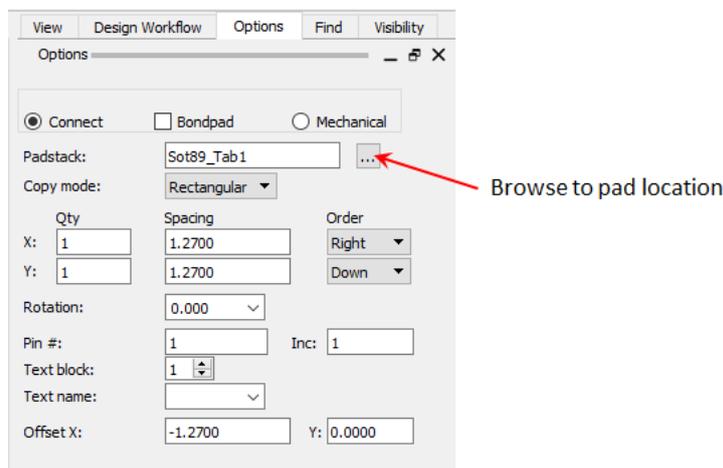


There is a useful option if you need to oversize/undersize the shape for the mask layers. Open the shape symbol then hover over the shape boundary until it highlights (ensure you are in General Edit App mode first) then right click > Expand/Contract and the set the values via the Options Pane. Then use File > Save As to save as a soldermask or pastemask version.

To use the pad when you create your Package Symbol (dra) file go to PCB Editor File > New > Package Symbol

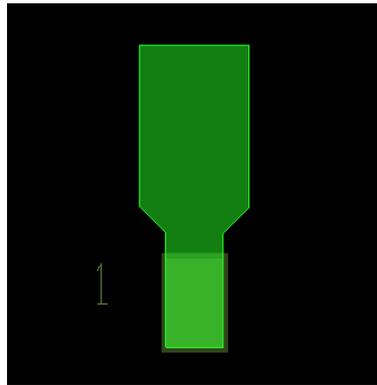


To add the pins use Layout > Pins. Browse to the pad name shown in the options menu



How to Create an alternative shaped pad/pin

Add the newly created pad as you would normally.



Creating padstacks for through hole pins using shapes

The same principle can be used for creating padstacks for through hole pins. In this case the same Shape is assigned to all of the padstack layers, Begin Layer, Default Inner and End Layer. If the Soldermask and Pastemask layers are going to be the same, the same shape can also be assigned to them. If different shapes are required for Soldermask and Pastemask layers, these can be designed as shape symbols in PCB Editor and then used by the Pad Designer. Use the Expand / Contract option for the shape as described above.

Note that, even though shape symbols in PCB Editor can only be defined on the Etch / Top and Etch Bottom layers, the Pad Designer and the Package Symbol Editor in PCB Editor will correctly process the shape to the required board design layer.

Pad	Geometry	Width	Height	X offset	Y offset	Shape name
Regular	Shape symbol	2.3876	1.7018	-0.7366	-0.5969	ss-pwrpak1212-2msk1
Thermal	None					
Anti	None					
Keep Out	None					

Pad	Geometry	Width	Height	X offset	Y offset	Shape name
Regular	Shape symbol	2.3876	1.7018	-0.7366	-0.5969	ss-pwrpak1212-2msk1
Thermal	None					
Anti	None					
Keep Out	None					

Pad	Geometry	Width	Height	X offset	Y offset	Shape name
Regular	Shape symbol	2.3876	1.7018	-0.7366	-0.5969	ss-pwrpak1212-2msk1
Thermal	None					
Anti	None					
Keep Out	None					

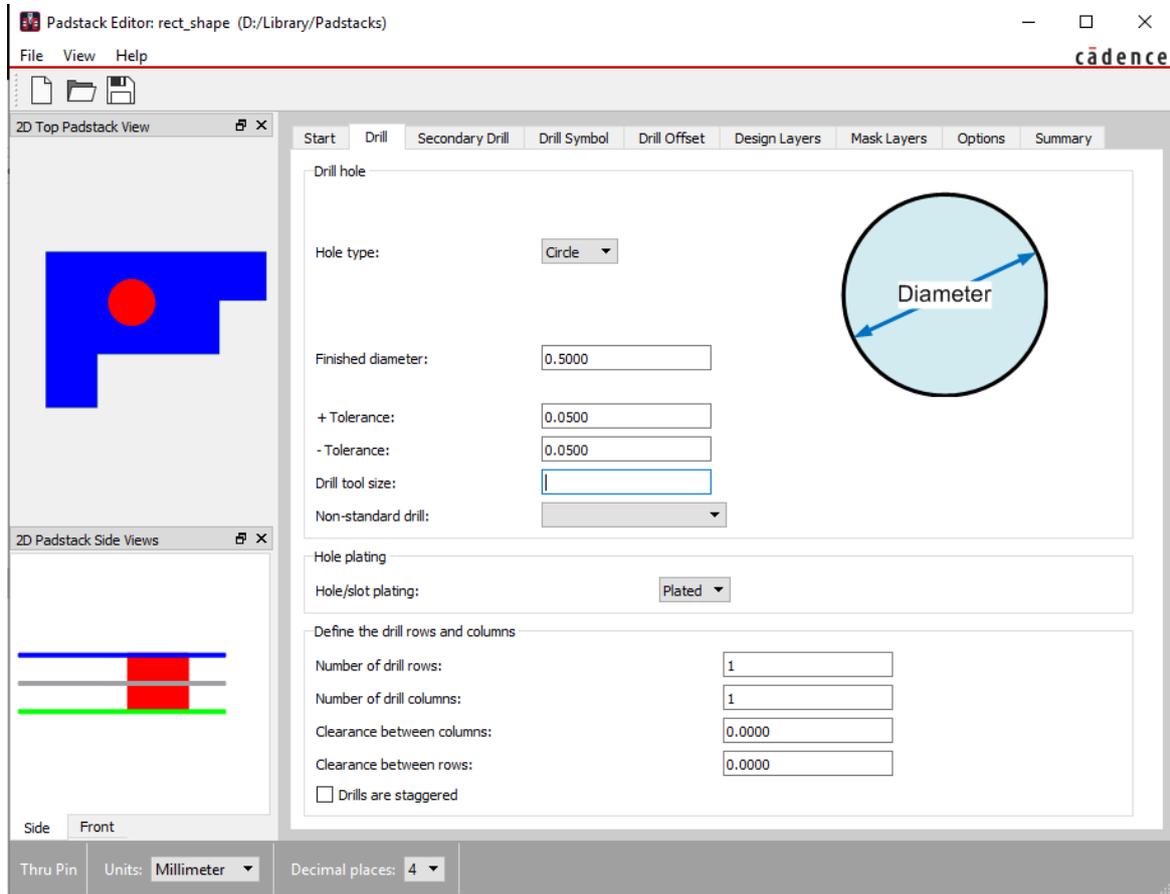
Pad	Geometry	Width	Height	X offset	Y offset	Shape name
Regular	None					
Thermal	None					
Anti	None					
Keep Out	None					

Layer	Geometry	Width	Height	X offset	Y offset	Shape name
SOLDERMASK_TOP	Shape symbol	2.3876	1.7018	-0.7366	-0.5969	ss-pwrpak1212-2msk1
SOLDERMASK_BOTTOM	Shape symbol	2.3876	1.7018	-0.7366	-0.5969	ss-pwrpak1212-2msk1
PASTEMASK_TOP	None					
PASTEMASK_BOTTOM	None					
FILMMASK_TOP	None					
FILMMASK_BOTTOM	None					

How to Create an alternative shaped pad/pin

In this case, the same shape has been assigned to all layers, the shape was added to the Begin Layer and then Copied to all the other layers using CTRL+C and CTRL+V including the Soldermask and Pastemask layers on the Mask Layers tab.

Define the drill as normal on the Drill tab:



Don't forget to define a Drill Symbol on the Drill Symbol tab or accept the warning.

Save the padstack definition to a "padpath" location so that PCB Editor can find it. Then File > New, Package Symbol and design a symbol using the new padstack.

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.