

Electronic Design for Windows
EDWINXP



Library Component Creation

VISIONICS

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved.

No part of the Library Creation document can be reproduced in any form or by any means without the prior written permission of Visionics. Library Creation document is subjected to change without notice. Visionics will make changes in a manner that will not affect dependent systems.

Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics.

Visionics, EDWinXP, Docone, EDComX, SimWinXP and Mixed Mode Simulator and their respective logos are trademarks or registered trademarks of Visionics. Unauthorized duplication of this work may also be prohibited by local statute.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Visionics. The information contained herein is the proprietary and confidential information of Visionics or its licensors, and is supplied subject to, and may be used only by Visionics's customer in accordance with, a written agreement between Visionics and its customer. Except as may be explicitly set forth in such agreement, Visionics does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Visionics does not warrant that use of such information will not infringe any third party rights, nor does Visionics assume any liability for damages or costs of any kind that may result from use of such information.

Contents

LIBRARY COMPONENT CREATION	4
LIBRARY CREATION USING WIZARD	5
<i>Creation of Symbol</i>	<i>5</i>
<i>Creation of Package</i>	<i>10</i>
<i>Creation of Part</i>	<i>15</i>
<i>Creation of Padstack</i>	<i>18</i>
LIBRARY CREATION WITHOUT USING WIZARD	20
<i>Creation of Symbol</i>	<i>20</i>
<i>Creation of Package</i>	<i>23</i>
<i>Creation of Part</i>	<i>25</i>

Library Component Creation

Basic Terminologies

Grid: Graphical aid for placing the component

Snap: The minimum distance that the cursor can jump from one position to another position with an object tagged to it.

Angular Snap: The minimum angle that the cursor can jump with an object tagged to it

Entry Point: The point at which the electrical connection given or taken to/from the component pin.

Pad: A circular/ oval/ rectangular/ line shaped seat for component pin with necessary diameter and hole (optional) created with copper.

Units

1"=1000mils

1"=25.4mm

0.1"=2.54mm=100mils

From View choose **Units → inches**

Set Grid as 0.1" and Snap as 0.05"

Naming Conventions

Packages in EDWinXP are named in the format Package type No: of pins in the package/horizontal pitch between the pad in mils. Eg: **DIP14/300** where DIP is the package type, 14 is the no: of pins in the package and 300 is the horizontal pitch between the pads in mils.

To create a library part first you need to create the symbol and package of the part. Part constitutes of Symbol and package.

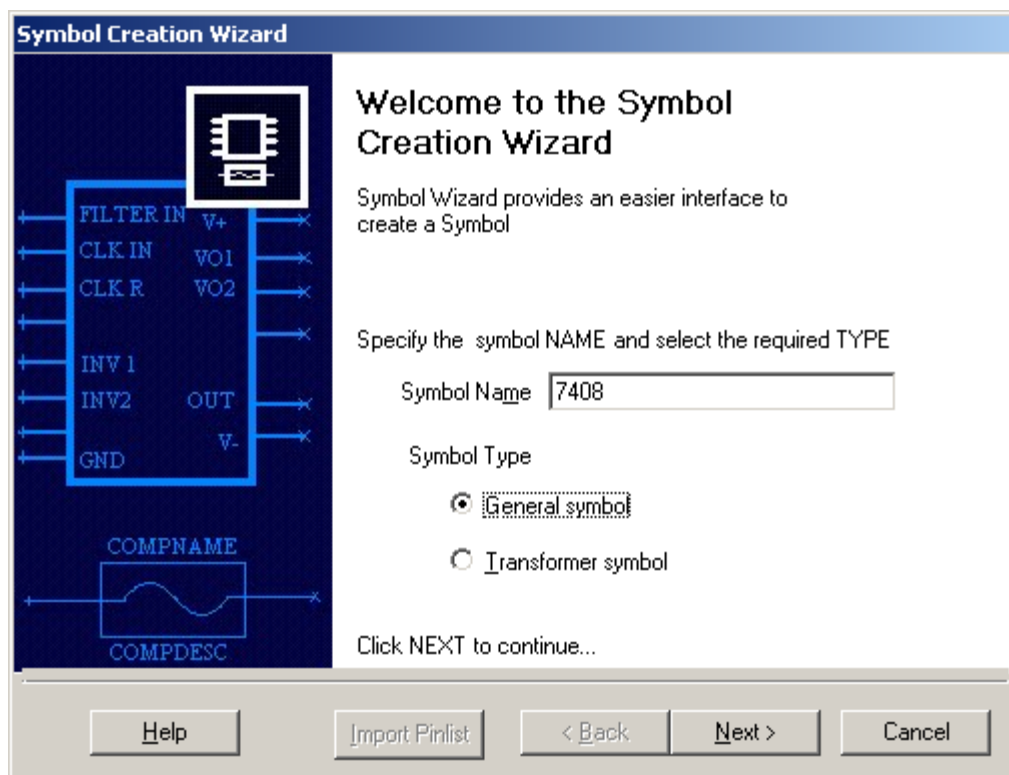
Library creation Using Wizard

Open EDWinXP Main from Start →Programs→EDWinXP →EDWinXP Main. Choose **Library** right click Library and choose **Library Editor**.

Creation of Symbol

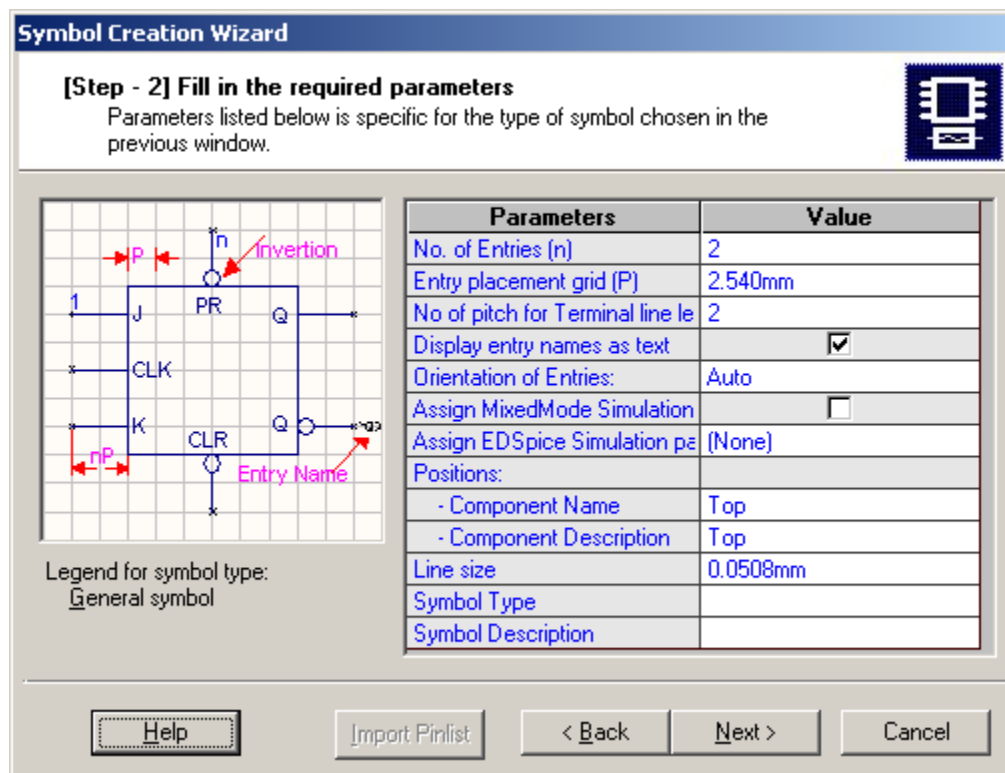
For creating symbols automatically, select Symbol tab / File/New Symbol using Wizard from Library Editor. This utility provides an easier interface for creating symbols. The Symbol creation wizard is as shown in figure below.

1. Specify the name and type of the symbol. For eg : Specify the symbol name as 7408 and symbol type as general symbol. Click on *Next* button to move to the parameter specification window.

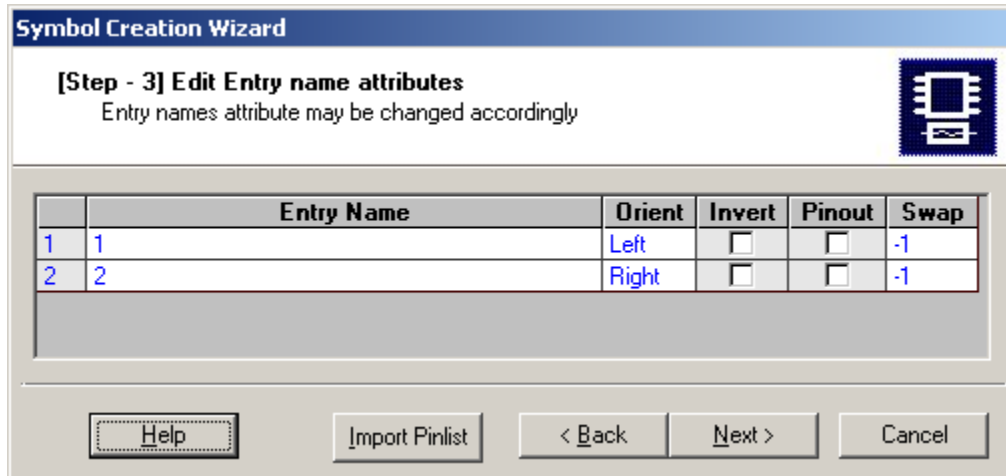


2. Specify the parameters, the description about the parameters that is to be specified while creating the symbol for 7408 is given below. Click on *Next* button, a new window appears which specifies the orientation of all entries, whether or not they are invertible, invisible etc.

No: of Entries (n) :3
 Entry placement grid (P) :.1000"
 No: of Pitch for terminal line length (nP) :2
 Orientation of entries : Auto
 Positions
 Component name : Top
 Component description : Top
 Line Size : .0020"
 Symbol Type : Double click on the field, Select **Digital**→**Gate** →**AND** from the drop down menu.
 Symbol Description : AND Gate
 Check display entry names as text check box.



- In the new window edit, the entry name attributes. You can also indicate the swap level and Sim Name. An import pinlist option is also available.



The Pin details format should be like Pin-Number, Pin-Name, Pin-Orientation (valid Pin-Orientations are *Left (L), Right (R),Bottom (B),Top (T)). Any invalid format will produce error in operation. Comments should start with a '*' character.

```

Pin List file
*-----
1,First Pin,Left
2,Second Pin,Left
3,Third Pin,Bottom
4,Fourth Pin,Bottom
5,Fifth Pin,Right
6,Sixth Pin,Right
7,Seventh Pin,Top
8,Eight Pin,Top

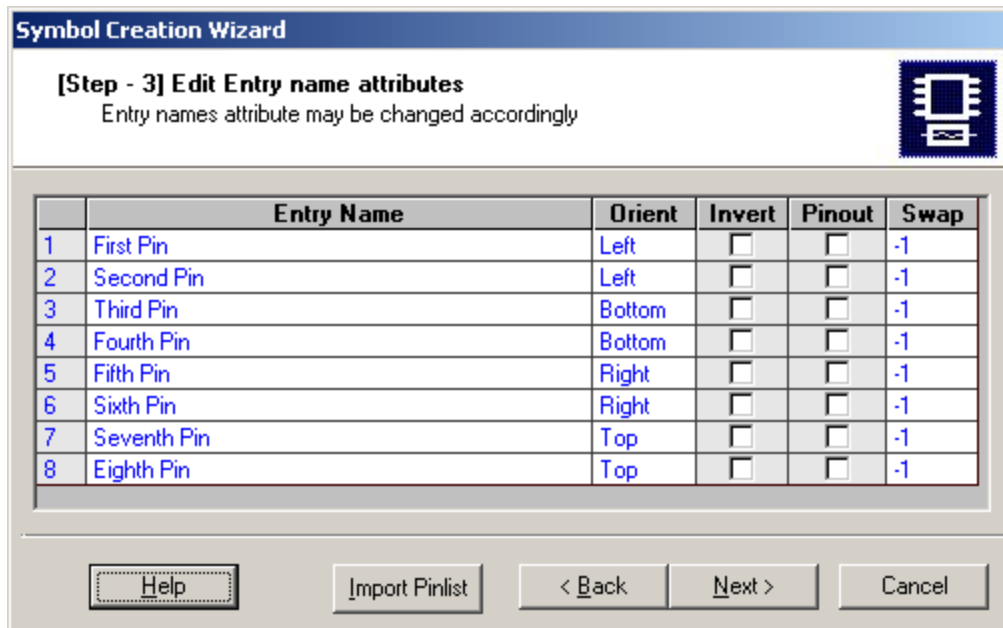
*nextpin details
*nextpin details

*End Pin List File

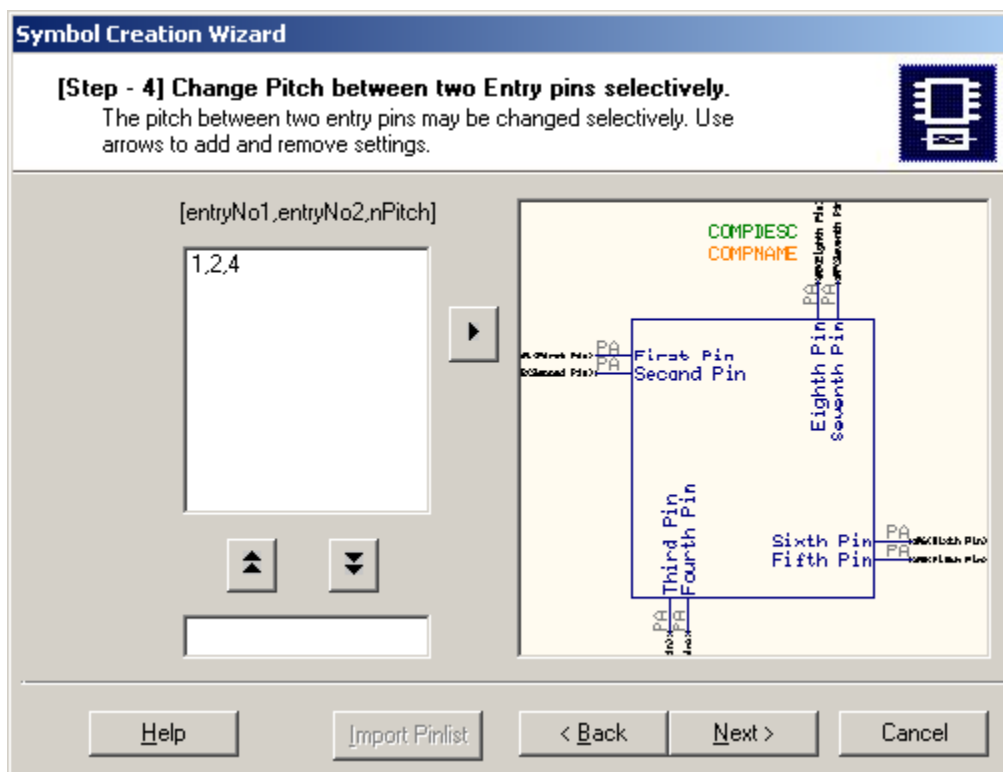
*Instructions
*-----
*1.Comments should be start with a
*'character
*2.Pin details format should be like
Pin-Number,Pin-Name,Pin-Orientation(val
id Pin-Orientations are *Left(L),
Right(R),Bottom(B),Top(T)). Any invalid
format will produce error in operation.

```

Window after importing the list is as shown below.

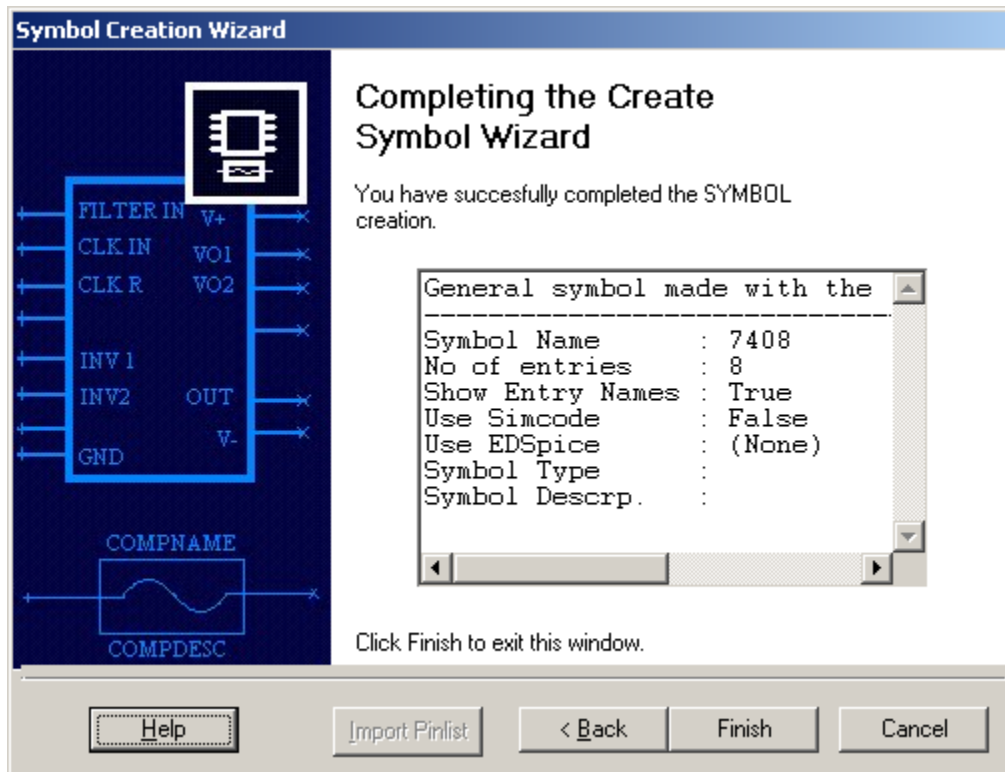


- Click on Next button moves to a new window where the pitch between two entry pins can be specified.



For example if the pitch between the pins has to be changed to 4 then give 1, 2, 4 in the field allowed for entering and click on the up arrow and then click on the arrow directed to right. Then the pitch between the pins is changed as applied.

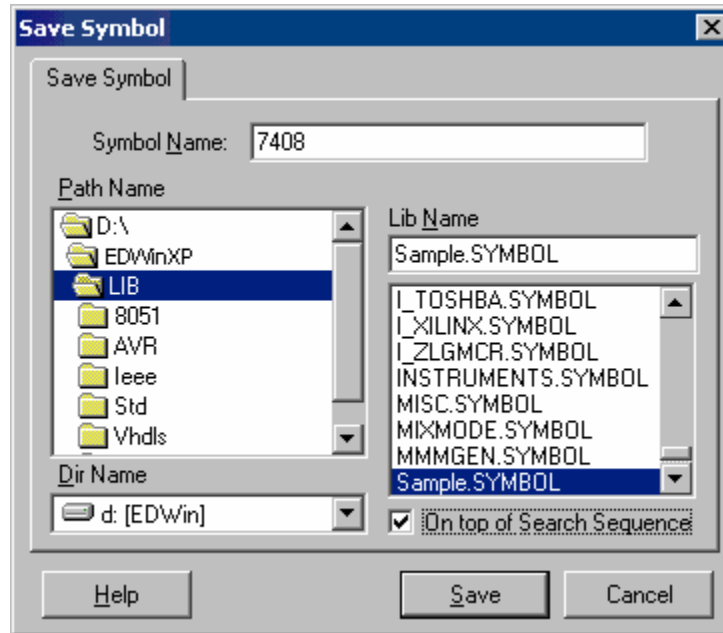
Click on Next button moves to a new window, which displays the summary of information about the symbol created using the wizard.



5. Click *Finish* to complete the process.

From the File Menu choose **Save Symbol As**. In the new Window Set the Symbol Name and the Lib Name.

Check On top of search sequence check box and Save.



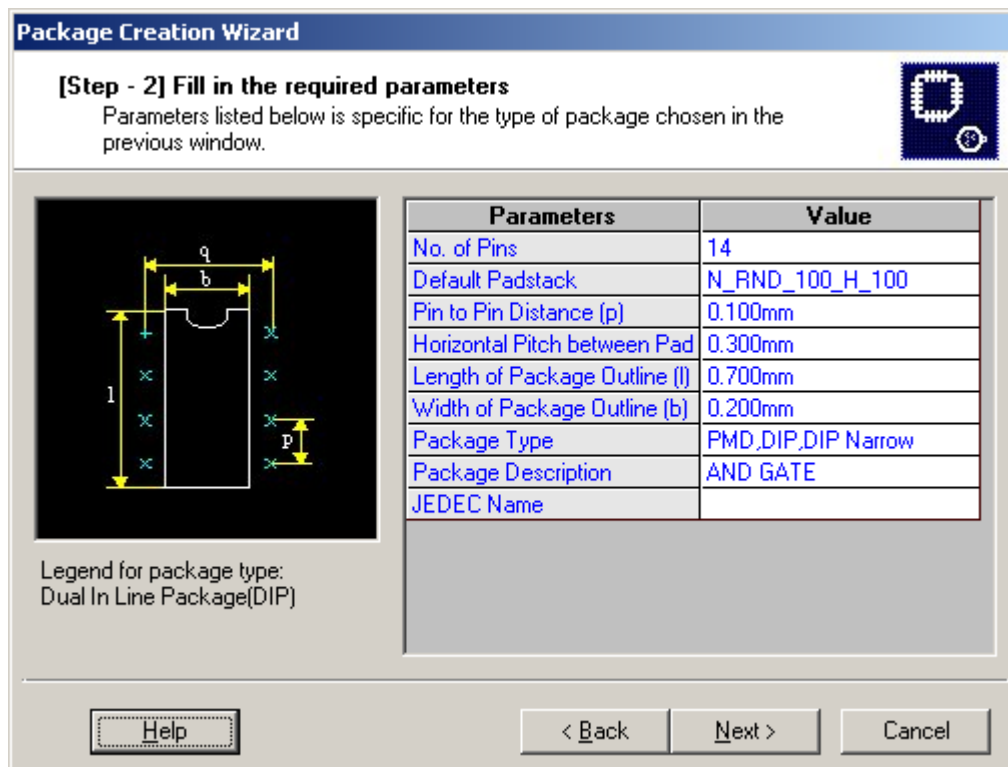
Creation of Package

In order to create package, click on **Package** tab.

1. Select **New Package using Wizard** from the file menu for creating the package for example IC7408.
2. Set the **Package Name as DIP 14/300**, 14 denote number of pins in this package and 300 denotes the width of the package. **Package Type as Dual In Line Package (DIP)**.



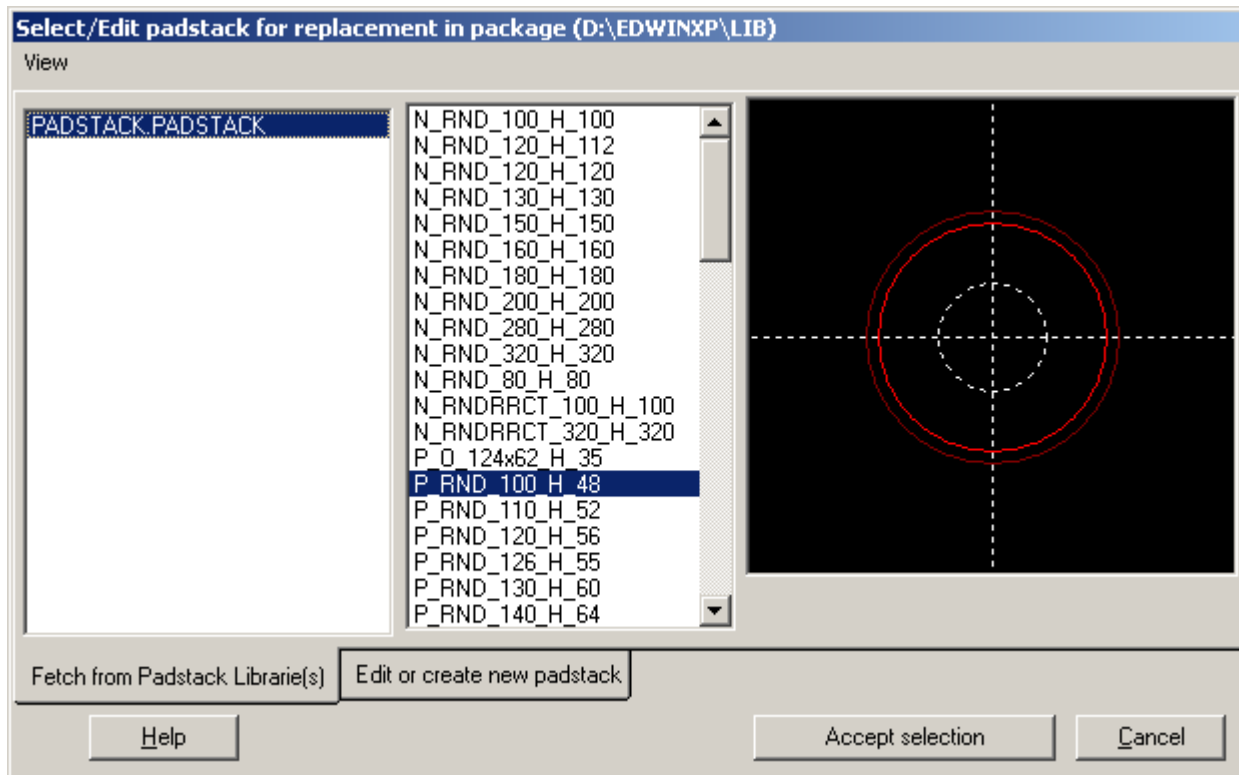
3. Click *Next* button to continue.



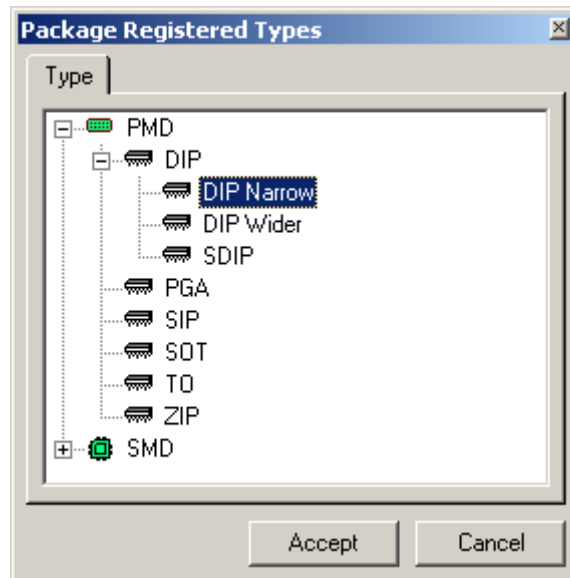
In the Window, which appears, enter the following values

No. of pins	: 14
Pin to pin Distance	: 1000"
Horizontal pitch between pads	: .3000"
Length of Package outline	: .7000"
Width Package outline	:. 2000"
Package Description	:AND GATE

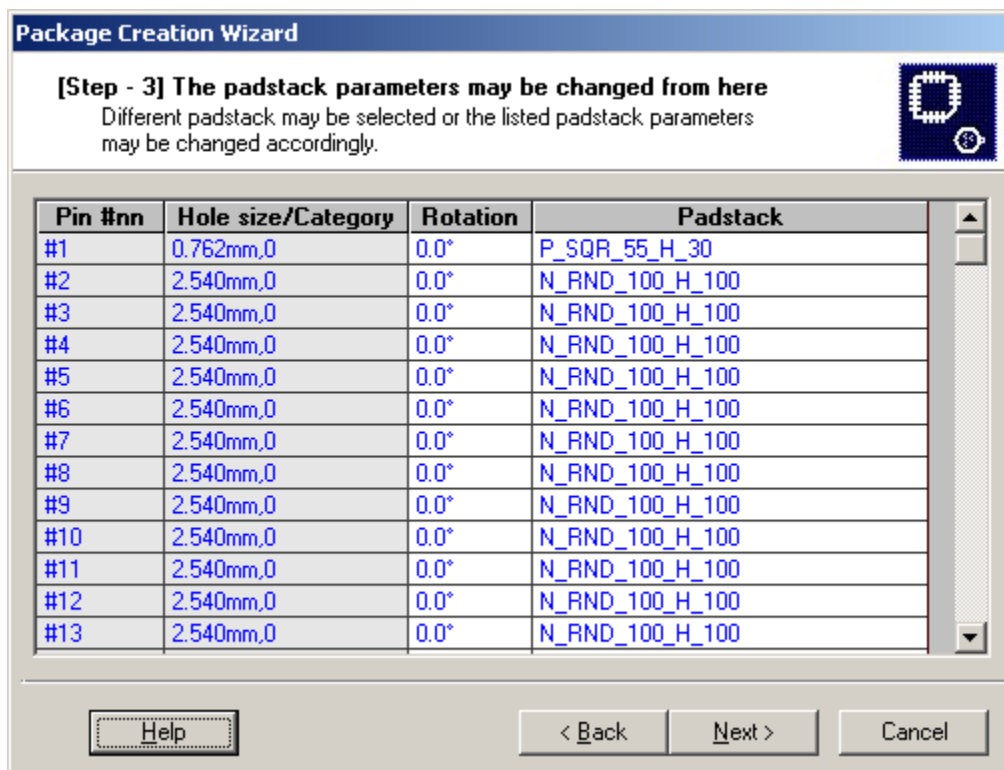
4. Padstack is selected by double clicking on the field **Default Padstack**, a window for selecting padstack is displayed as shown in figure below. From the menu select the padstack and click *Accept Selection*. Then automatically the padstack name is assigned to the required field.



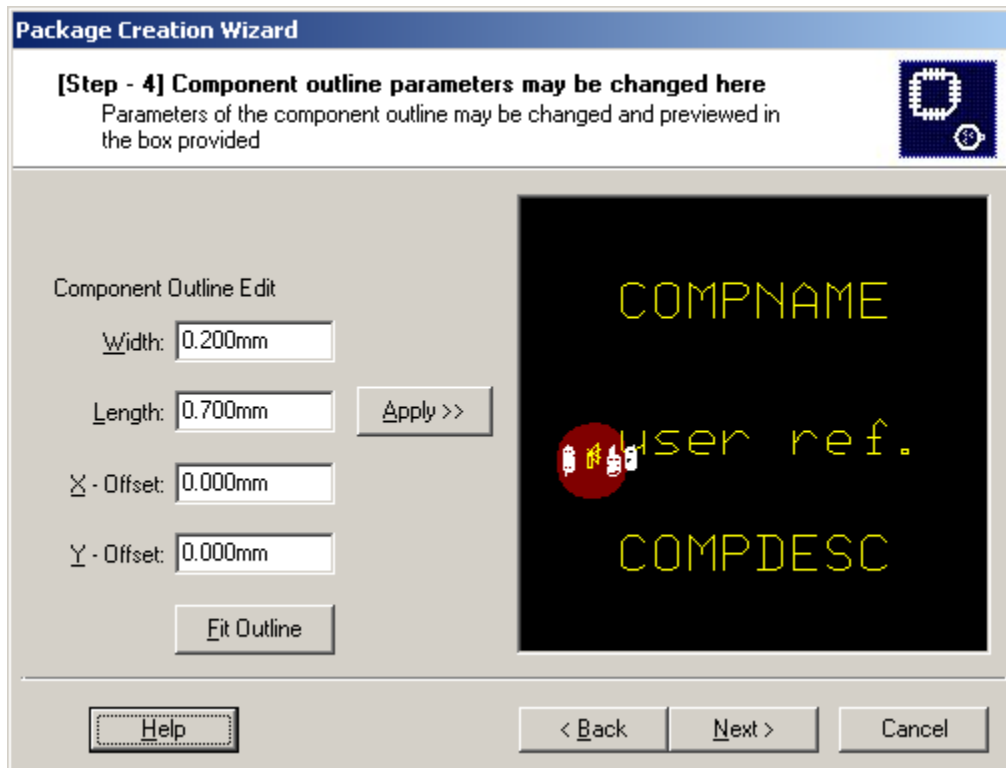
5. Package Type is given by double clicking on this field, a window for selecting the package type is displayed. From the drop down menu select **PMD** → **DIP** → **DIP Narrow**. Click *Accept*.



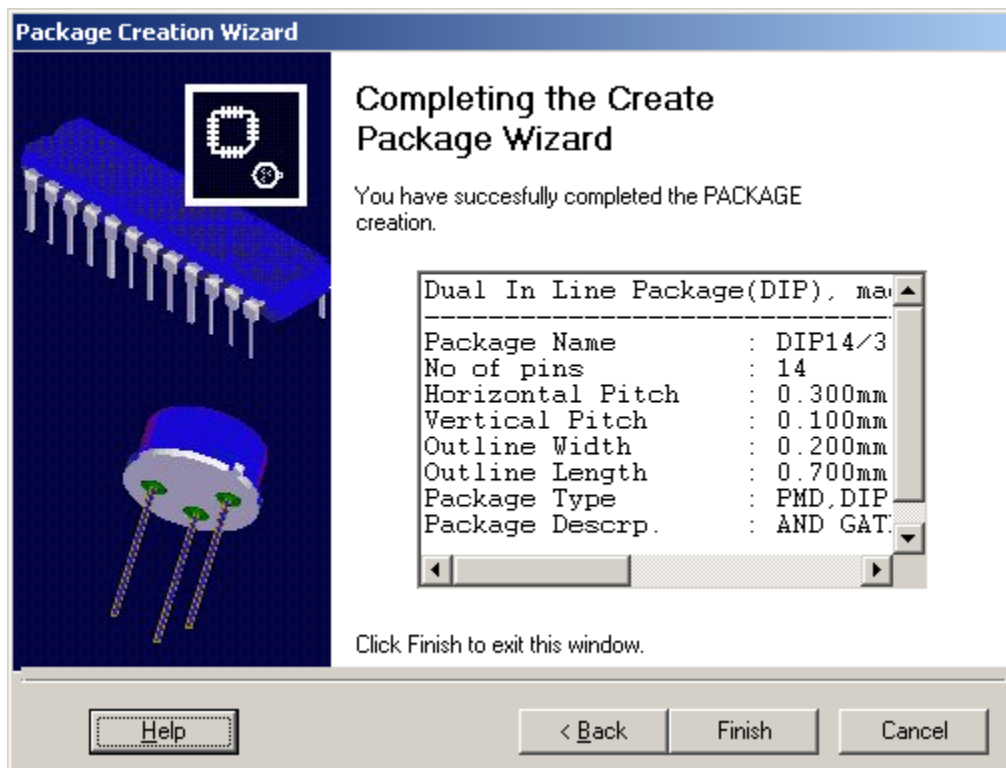
6. Click *Next*, Change the Pad Name of first pad (#1) to P_SQR_55_H_30.



7. Click *Next*.



8. Click *Next*.



9. Click *Finish*.

10. From the File Menu choose **Save Package As**.

11. Give Package name as DIP14/300 and Lib Name as Sample
12. Check **On top of search sequence** check box and Save.

Creation of Part

1. Click on **Part Edit** tab, enter the Part and Package details as

Part details

Name : 7408
Description : AND Gate

Select manufacturer, technology and type from the drop down menu by double clicking on the required field.

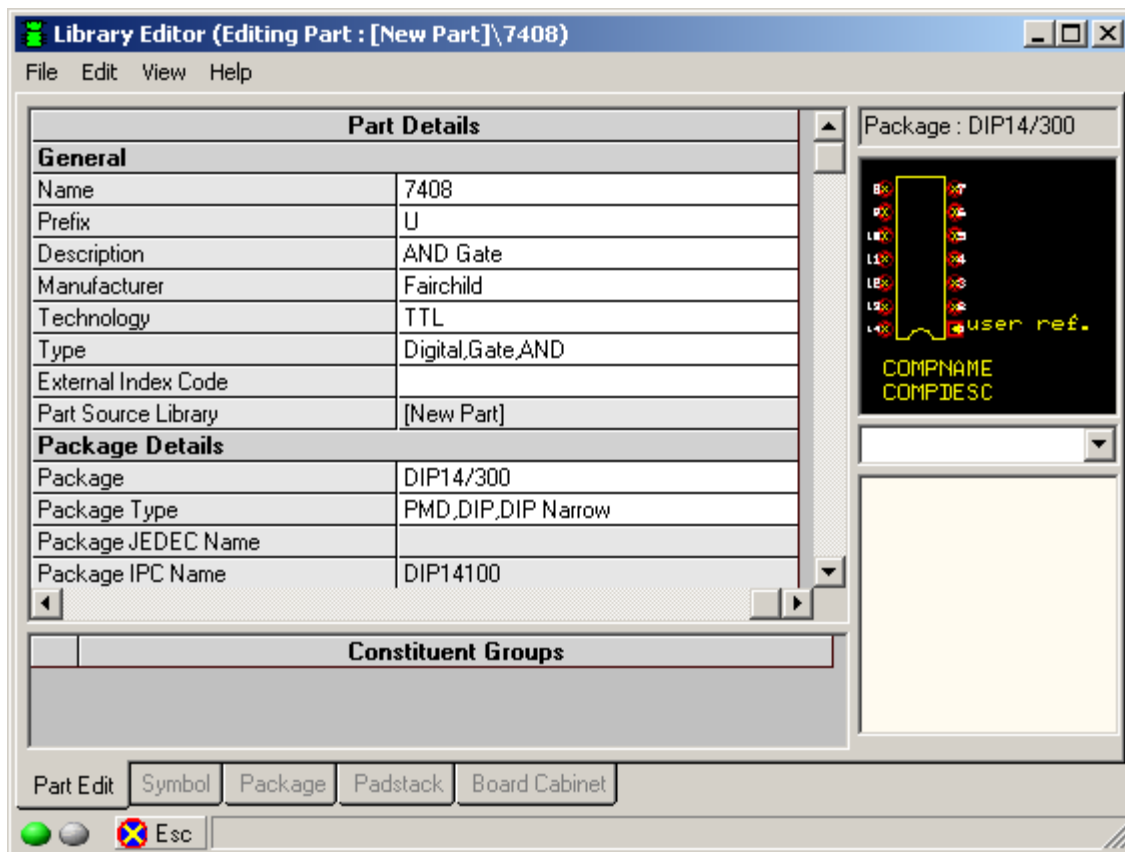
Manufacturer : Fairchild
Technology : TTL
Type : **Digital→Gate→ AND**

Package Details

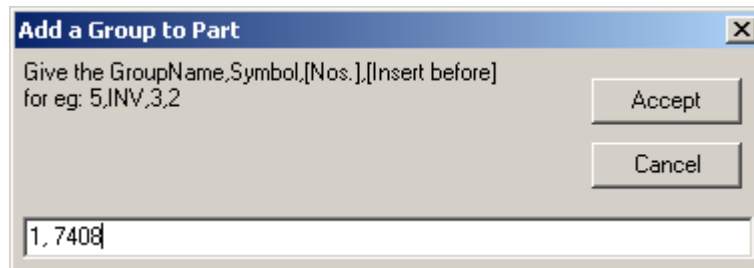
Package : DIP14/300

Click Yes for the question "Are you sure to change the package?"

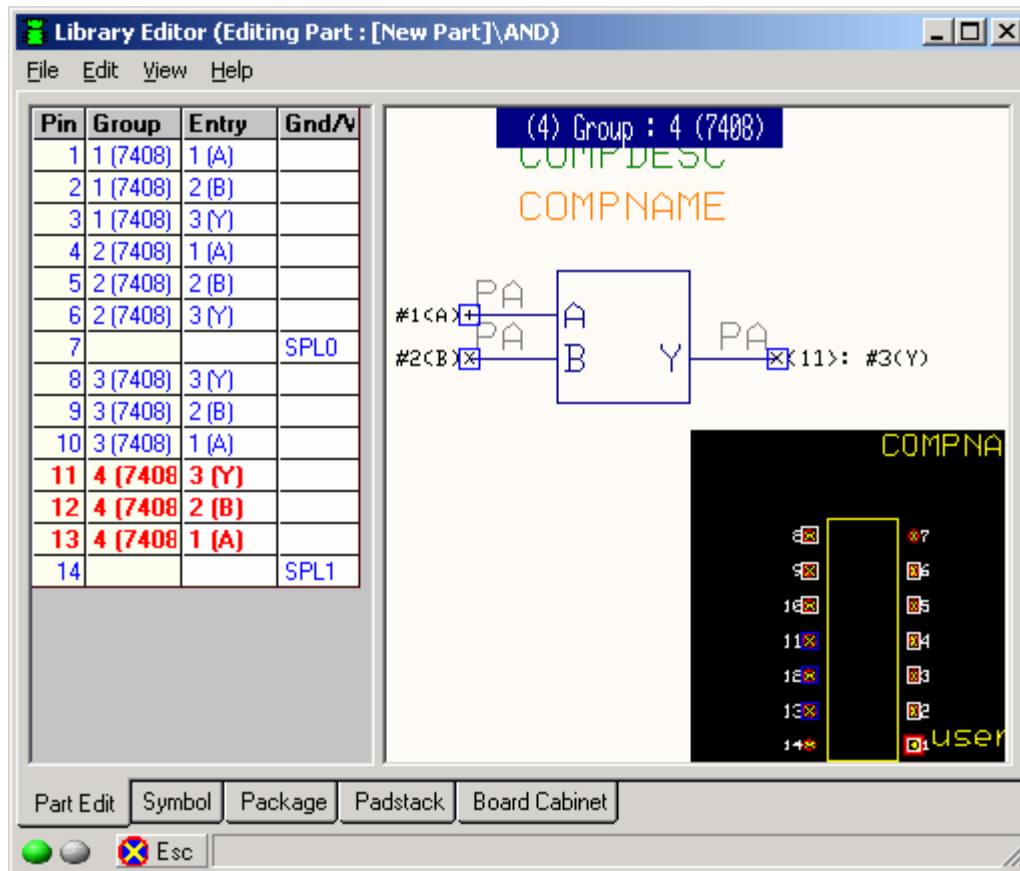
Select Package type by double clicking on the field and select **PMD →DIP →DIP Narrow.**



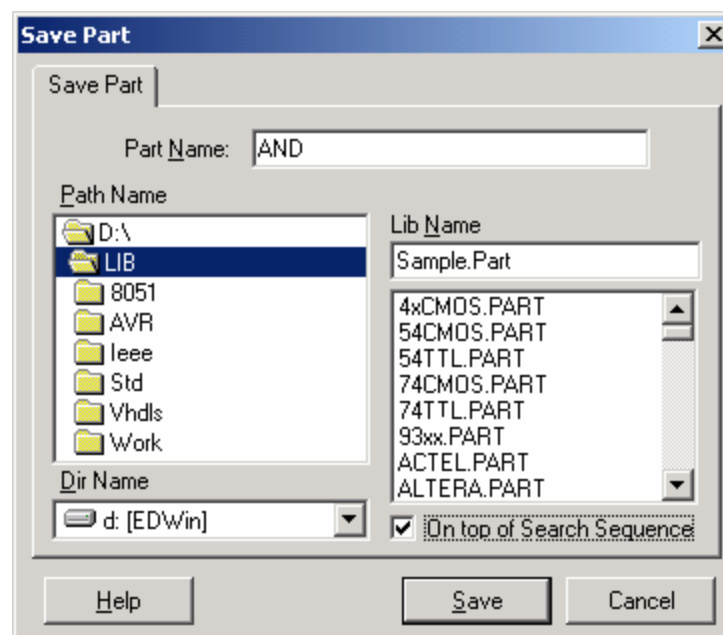
2. From the Edit Menu choose **Add Group**, In the new window, Give 1, 7408 (Symbol name given during creating symbol). Similarly add 3 more groups by repeating **Edit** →**Add Group**.



3. After adding four groups, Choose **Edit** →**Edit Pinout**. In order to view assigning each group to the package select **Edit** →**Show Package Window**.
4. Choose **Edit** →**First group** and assign first group to the package by clicking on the Entry field. Select inputs and output pins according to the package.
5. Select **Edit** →**Next Group** to assign the second group to the package. Name Pin no: 7 as SPL0 and Pin No: 14 as SPL1. Assign groups third and fourth similarly as done early.

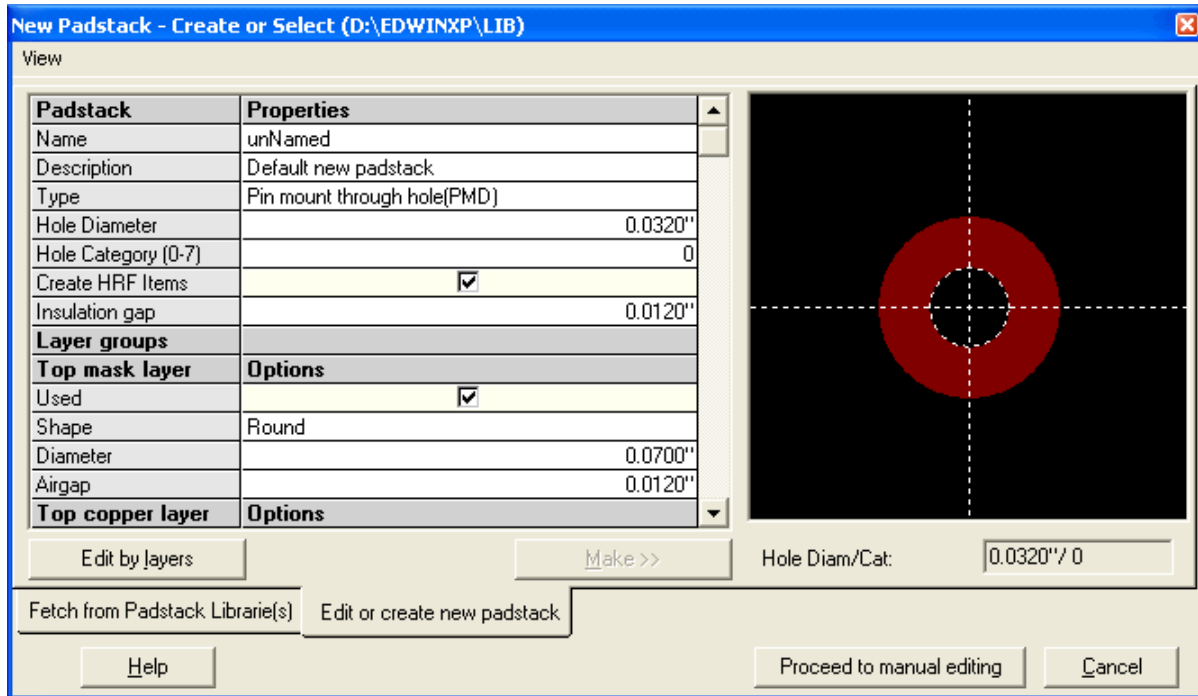


6. Select File → Save Part, give Part Name as AND and Lib Name as Sample.
7. Check On top of search Sequence check box and Click Save.



Creation of Padstack

1. Select **File | New Padstack using Wizard**. to pop up window as shown in figure below.



2. In the window, a miniature representation of the padstack is shown in the preview. Right click on the preview for options to view True size, Hrf items and Airgap view. From the **Fetch from Padstack Library(s)** tab, select the padstack "N_RND_100_H_100". Now on moving to the **Edit or create new padstack** tab, the padstack details of "N_RND_100_H_100" appears.
3. The minimum information required to create a New Padstack is given in the figure below. The user may edit the values.

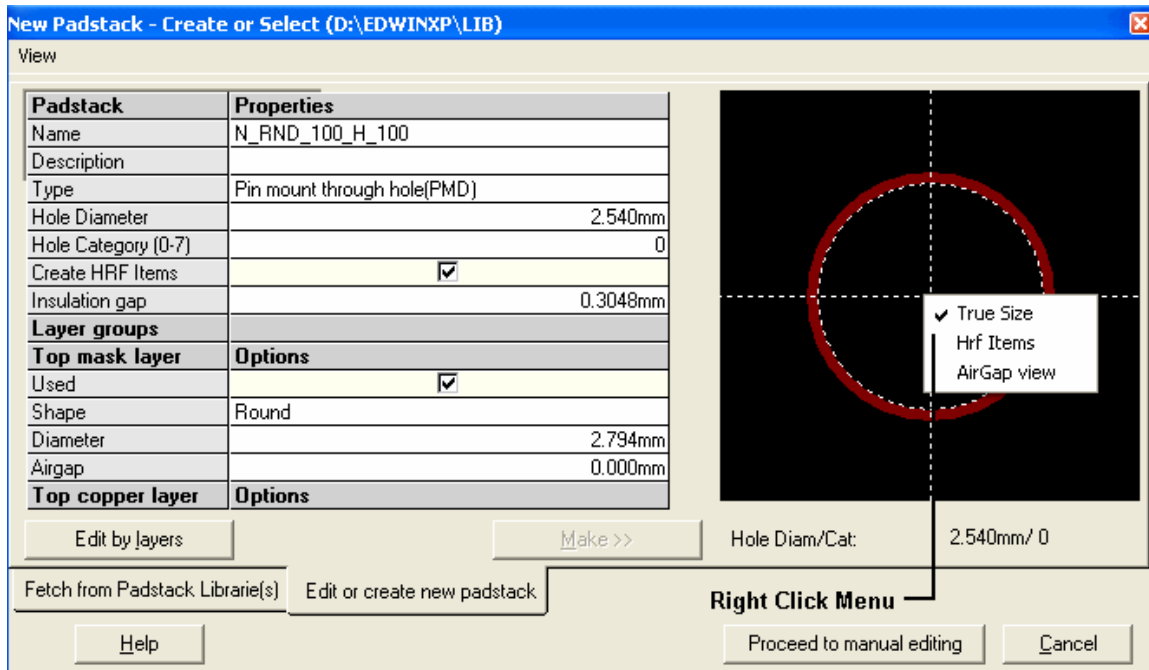
Name = N_RND_100_H_100 and press the <enter> key.

Create HRF items = select the checkbox

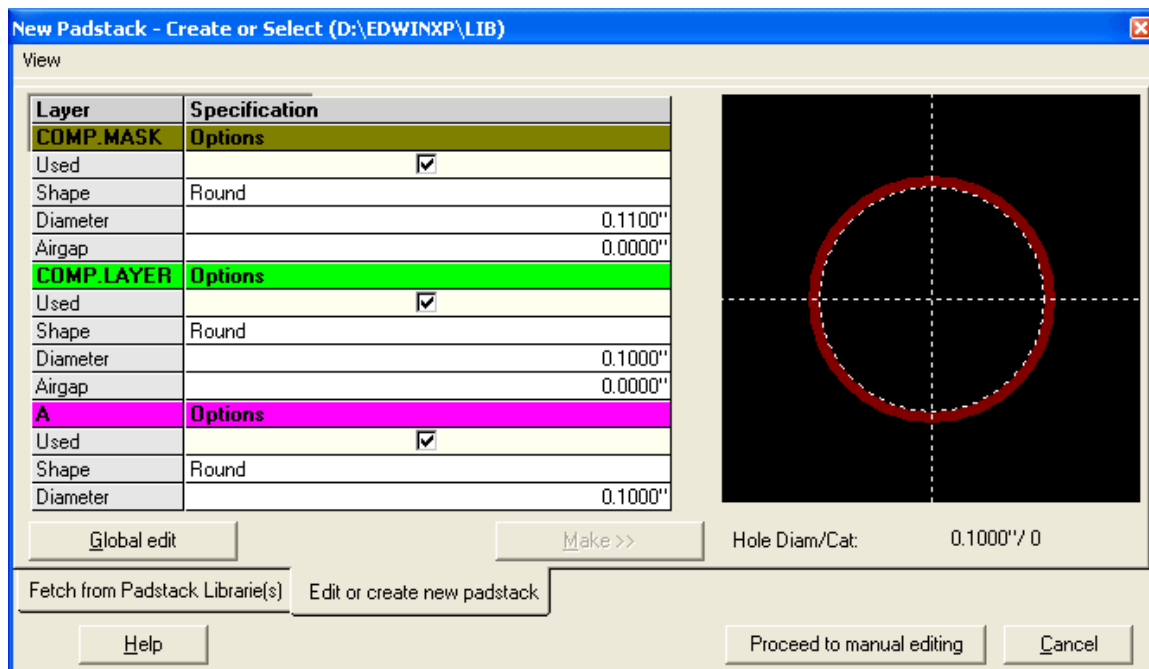
Airgap = 0.0120"


Hole Category = 0

Hole diameter = 0.1000"



4. On clicking the *Edit by layers* button, the pad details of different layers (Shape, diameter, Airgap and the option of selecting the layers to be used) appears.



 Global Editing of layers is also allowed.

5. Finally click the button *Make* to create the new padstack. Now the edited changes are visible in the Preview in the right pane. Click on *Proceed to manual editing* to effect the changes and exit.




This completes the Padstack creation using wizard. The edited padstack appears in the Padstack editor and further edition can be done using the tools available.

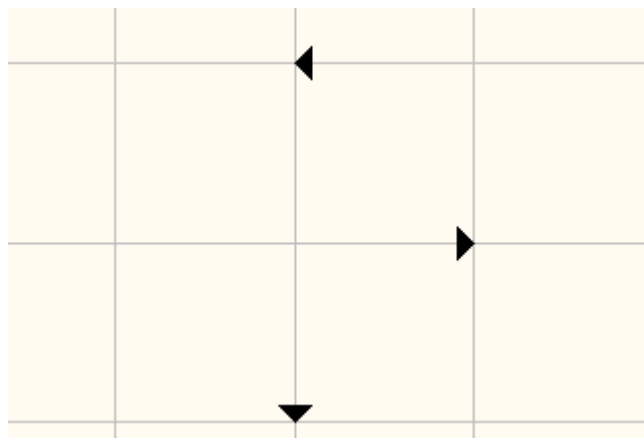
Library Creation without using Wizard



Open EDWinXP Main from Start →Programs→EDWinXP →EDWinXP Main

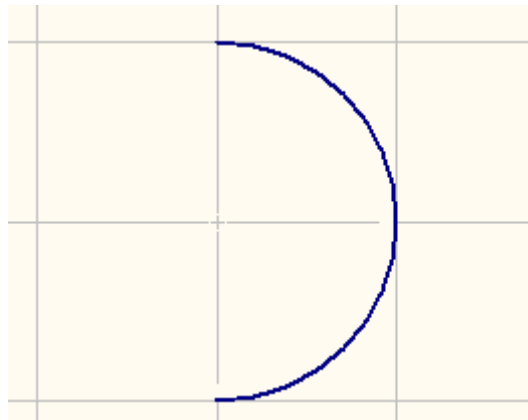
Choose **Library**, right click Library and choose **Library Editor**.




Creation of Symbol

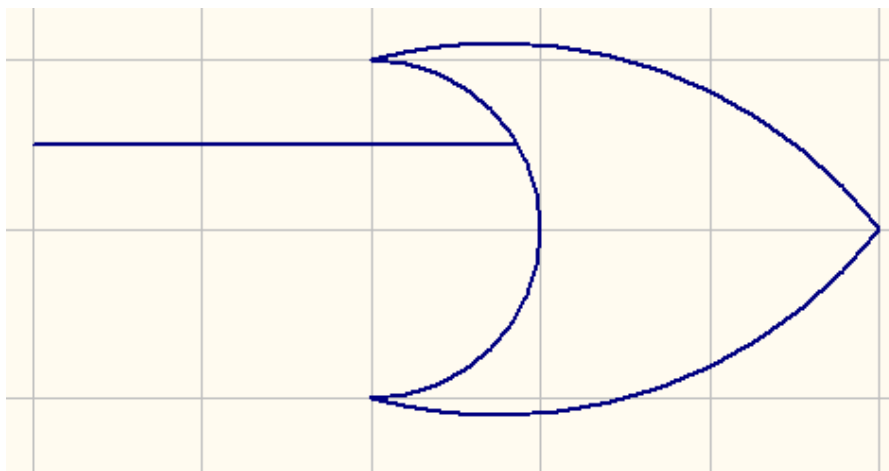
1. From **Library Editor** choose **Symbol** tab at the bottom of the page.
2. To create a symbol of OR gate, select the  **Set Contact Point** (function tool)→ select the first option tool  and place it on the grid. Similarly select the second one  and place it on the next grid as shown.






3. Select the  Create graphic item (function tool) →  Create arc (option tool). Click anywhere near the contact point to get an arc tagged to the cursor.

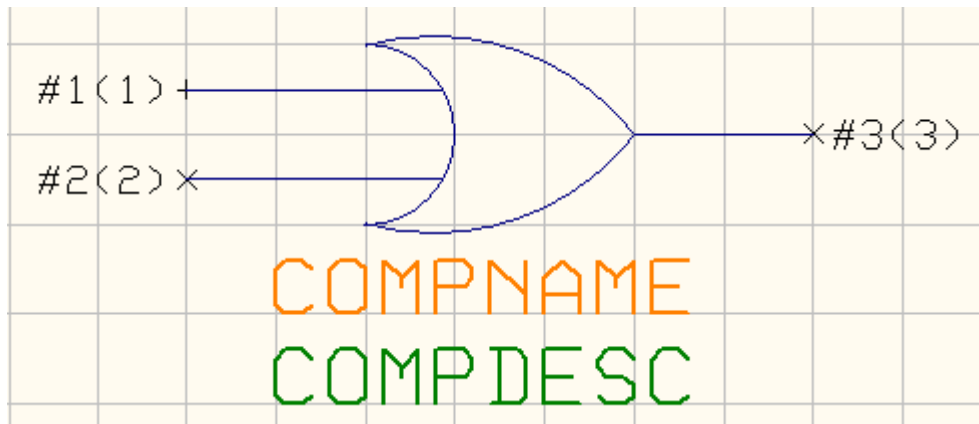


4. After adjusting the size of the arc, click to place it. Adjust the arc, if necessary, by selecting the  Stretch Item function tool and click the  Stretch arc radius tool or press F1 key, to change the radius of the arc.
5. Repeat the arc as needed, after placing the arc press ESC.
6. Select  Create line and click on the workspace where the line needed.



7. Repeat the same line using the tool  Repeat Graphic Item, select this tool and click on the workspace then a line is tagged with the cursor, place it where required.

8. Place the first/default entry (+) present in the workspace, exactly at the starting point of the first pin of the OR gate using the  Relocate item tool.
9. Select the  Create graphic item tool and click the  Create entry pin option tool or press F6 key, for creating the remaining entry points. Click on the starting point of the pins.

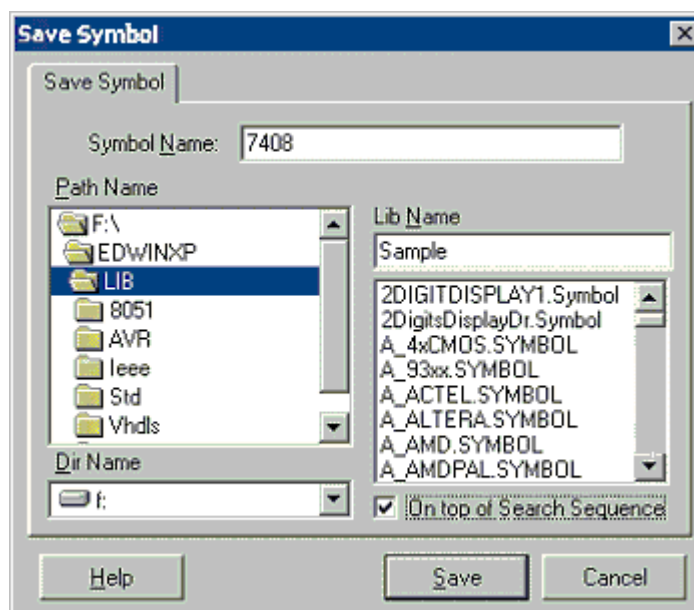


10. From the File Menu choose Save Symbol As

11. In the new Window.

Set the Symbol Name and the Lib Name

Check **On top of search sequence** check box and Save.




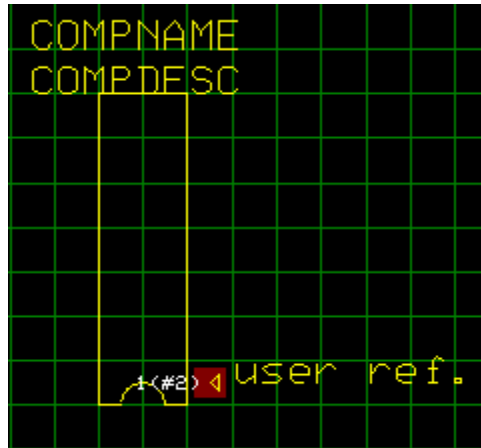
Creation of Package


From **Library Editor** choose **Package** tab at the bottom of the page.

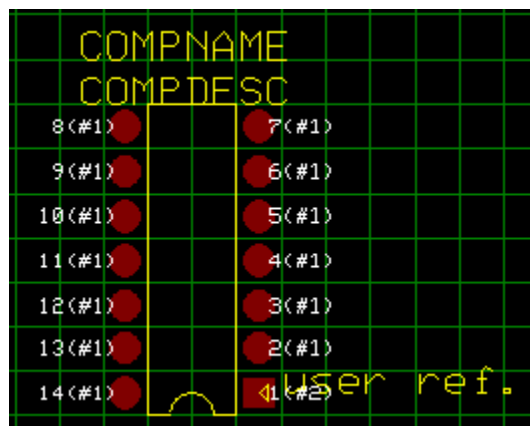
Select the Component Print, by default component layer will be selected.



To create the outline of the package, select **Create Graphic Item** (Function tool) → **Create rectangle** (Option tool).

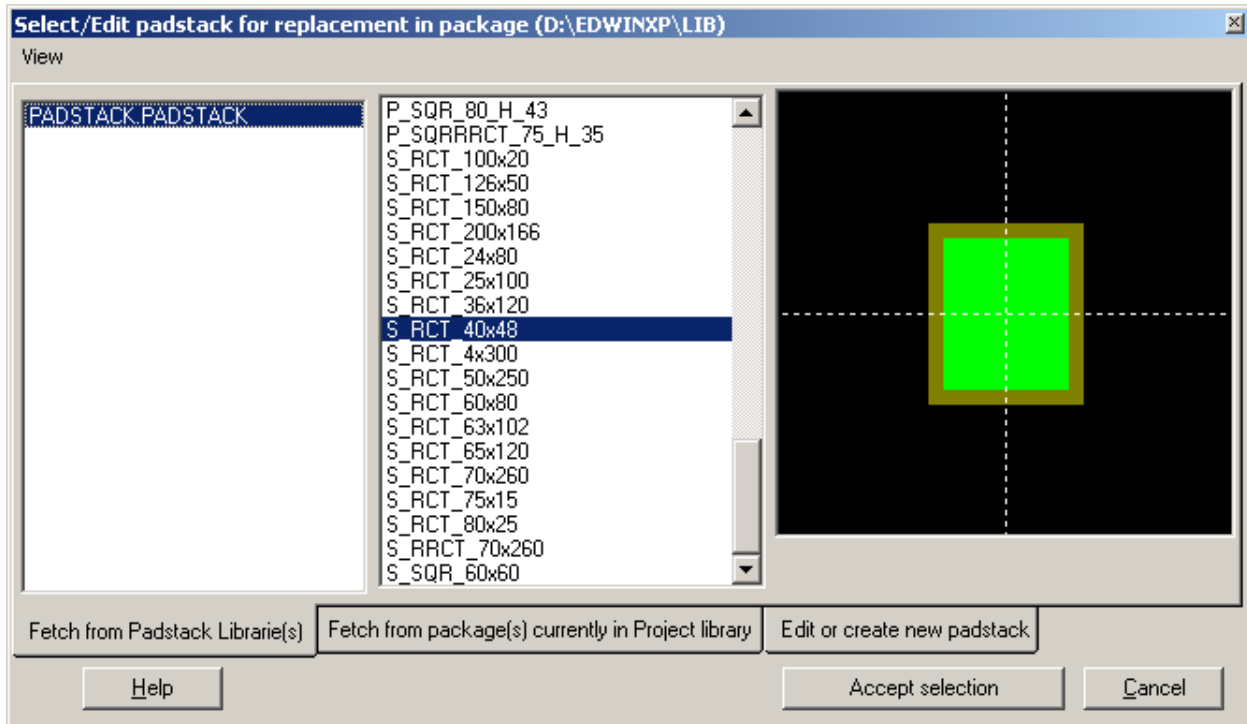
Place the first/ default pad (+) exactly at the place of the first pad using the  **Relocate item** tool.



For placing the remaining second pad select the  **Create graphic item** tool and click the **Create Pad** option tool.



The pad is placed at the required position and displayed as a small "x". To change the padstack, click  **Change Padstack** tool and select  **Select Padstack** option tool.



Select an SMD pad, Eg: Select S_RCT_40*48 SMD pad.

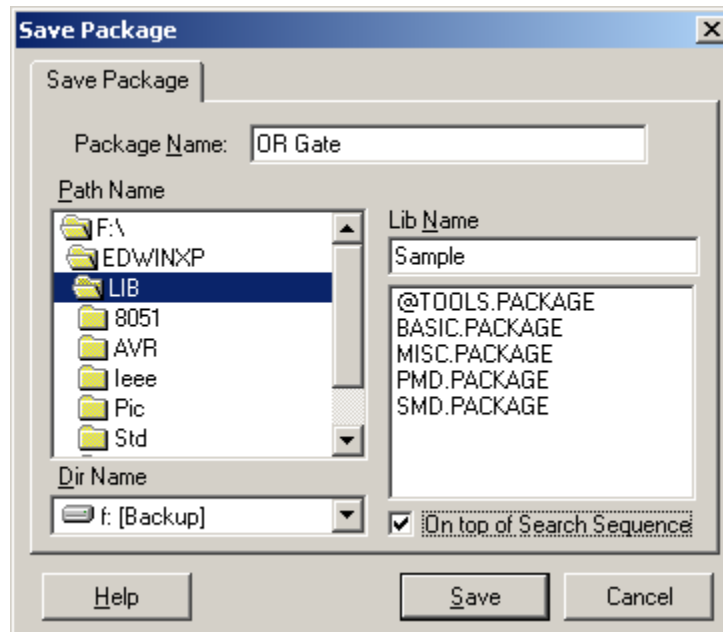
Click **Edit or create new padstack** tab → **Edit by layers** tab to edit the shape and size of the pad.

To reflect the changes made in the selected padstack click  button.

Click the *Accept Select* button to select this padstack. Click on the center of each pad to locate the pads made.

From the **File** Menu choose **Save Package As**.

Give Package name as OR Gate and Lib Name as Sample.



Check On top of search sequence check box and Save.

Creation of Part

Same as in the creation of part using wizard.