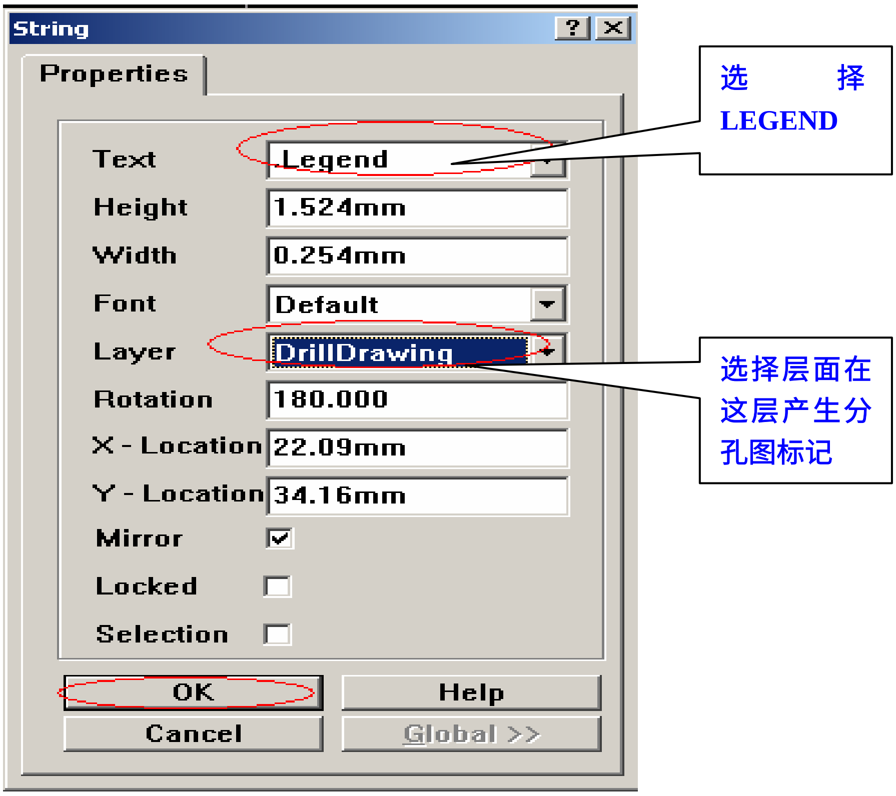
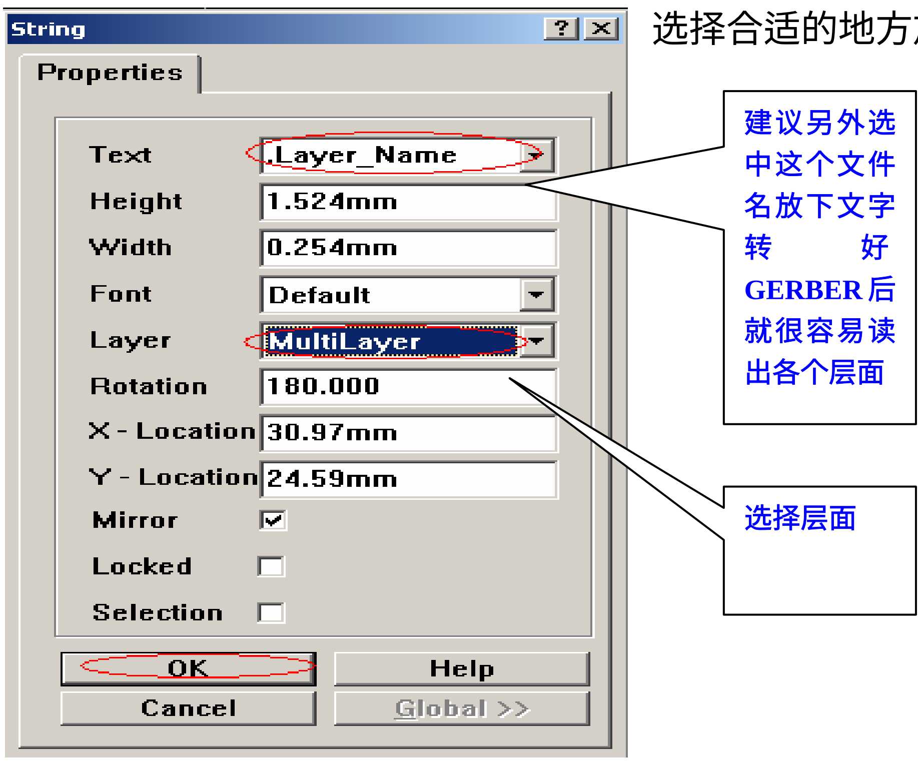
PROTEL99SE 转 GERBER 步骤

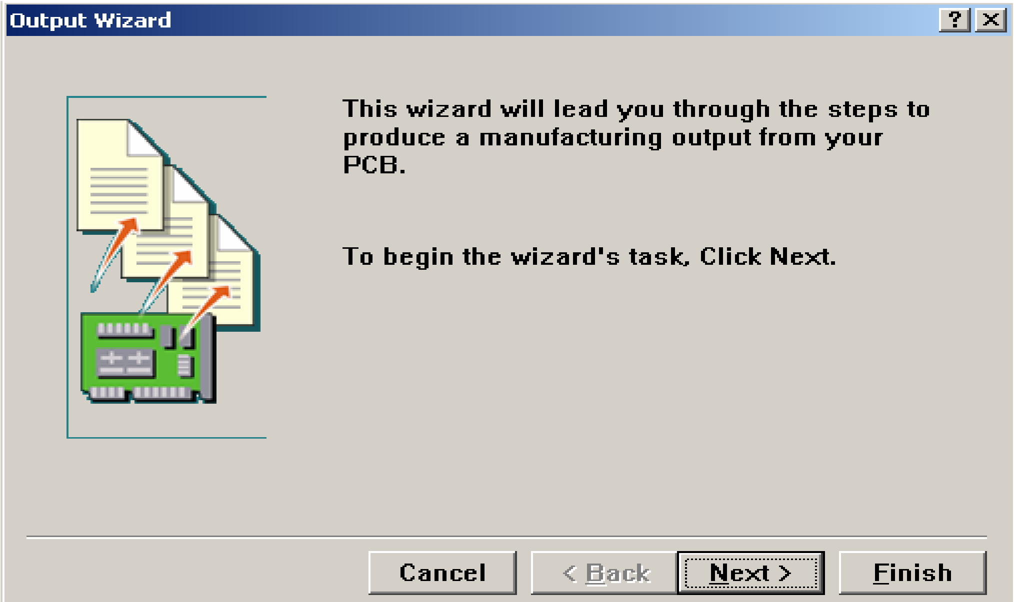
1 先打开PCB图档最好先检查一遍PCB图档是否有问题确认没问题后放孔径标记，点击工具栏上的 T 或者用快捷键PS这时会出现一个字符串STRING的字样再用鼠标点两下或按键盘TAB键出现如下对话框



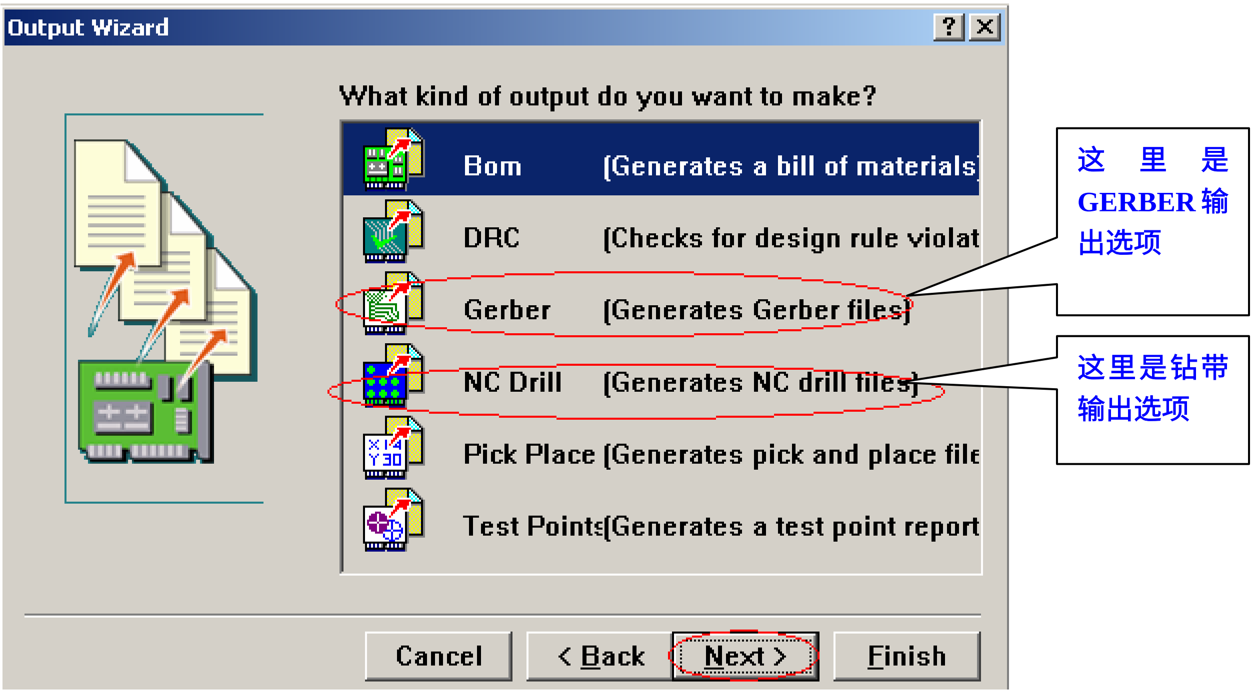
2 按OK ， 选择合适的地方放下文字。



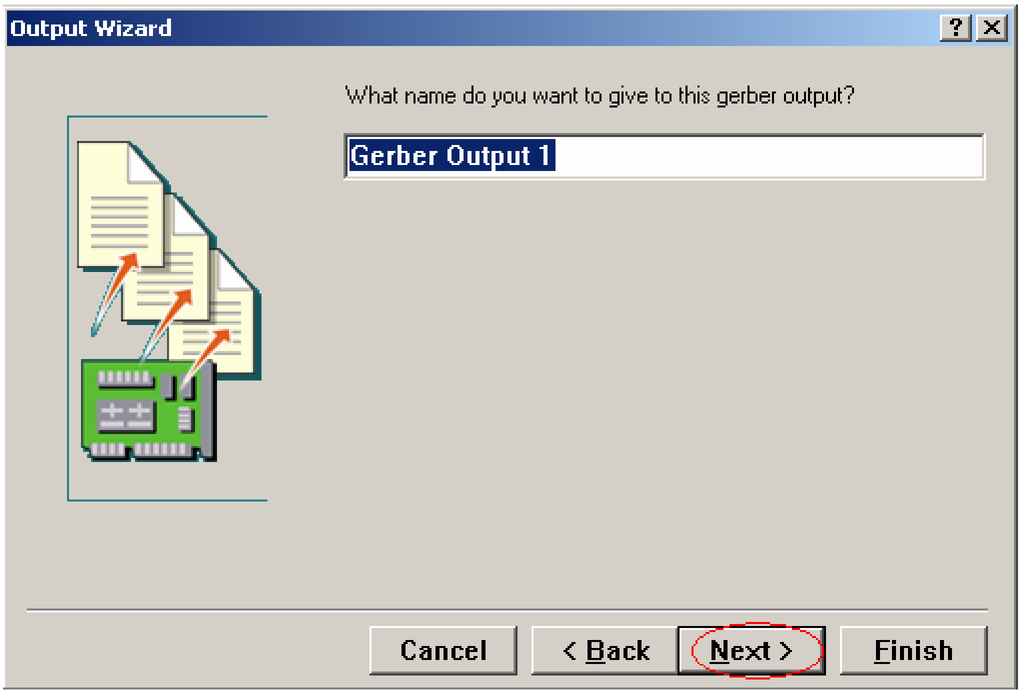
3 开始转GERBER在FILE——CAM MANAGER或快捷键FM出现OUTPUT WIZARD输出GERBER向导如下图

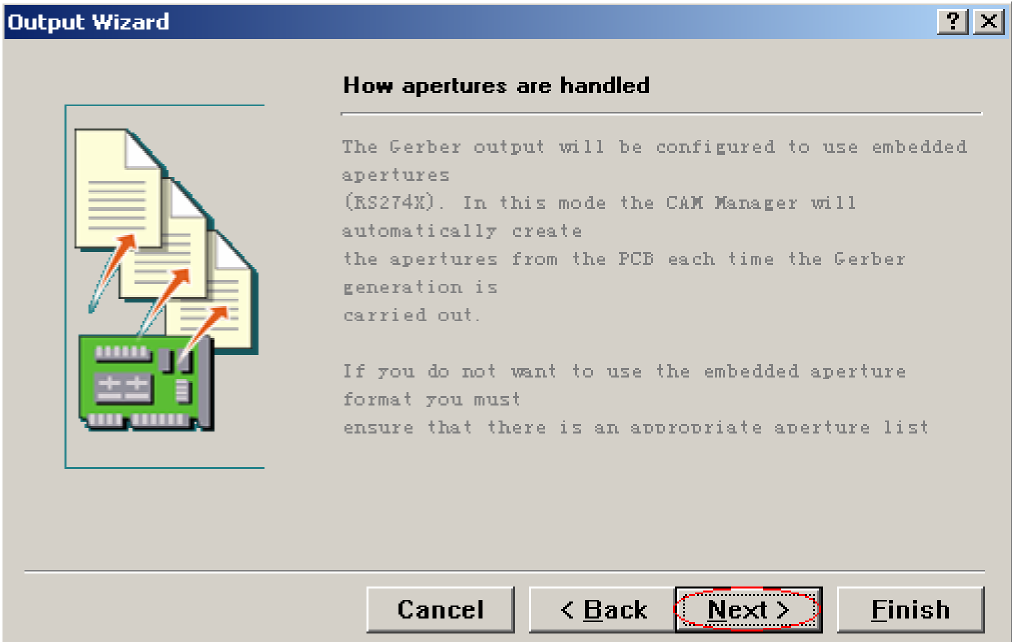


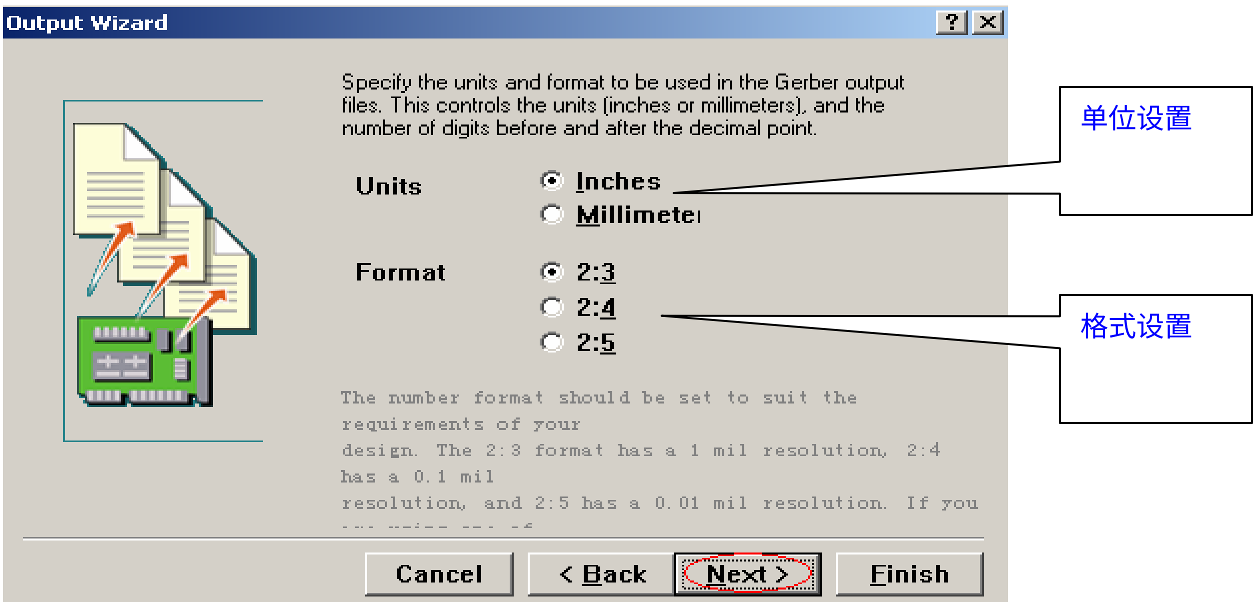
按NEXT进入下一步如图



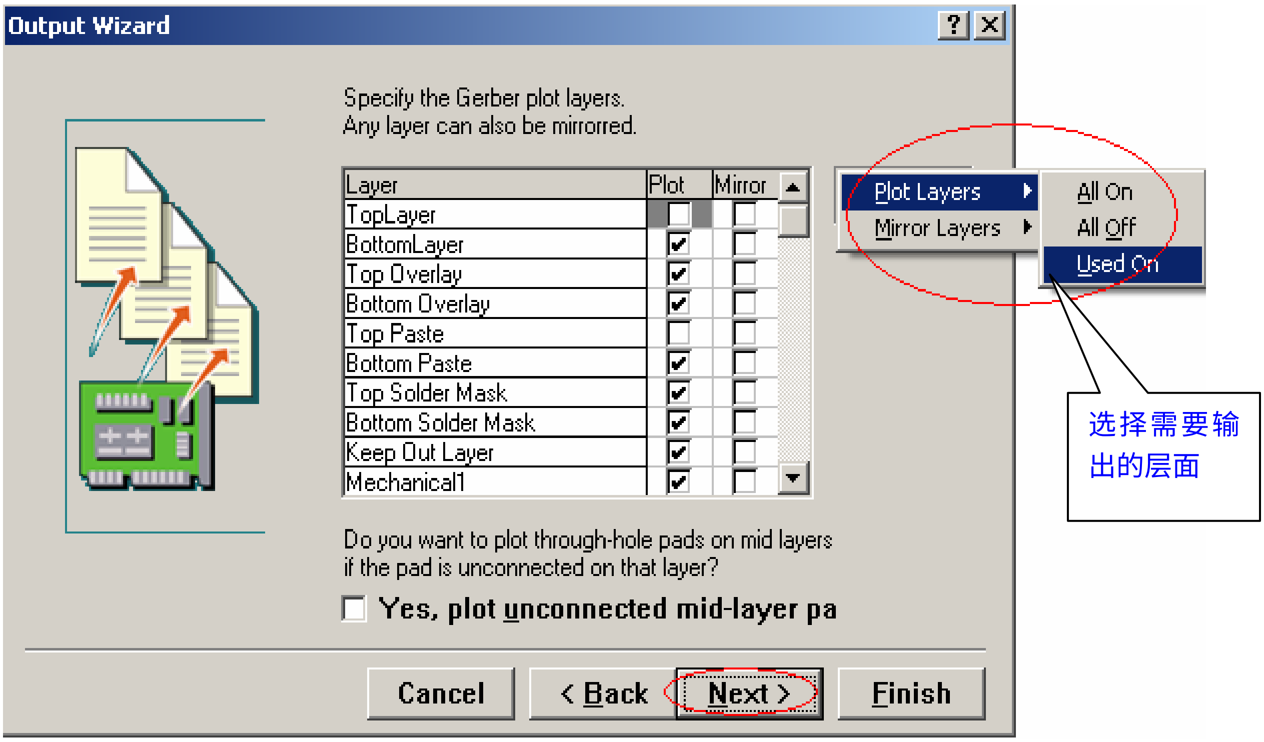
选择红色圈内的Gerber (Generates Gerber files)按NEXT进行下一步操作如下


Output Wizard
What name do you want to give to this gerber output?
Gerber Output 1
Cancel
< Back
Next >
Finish

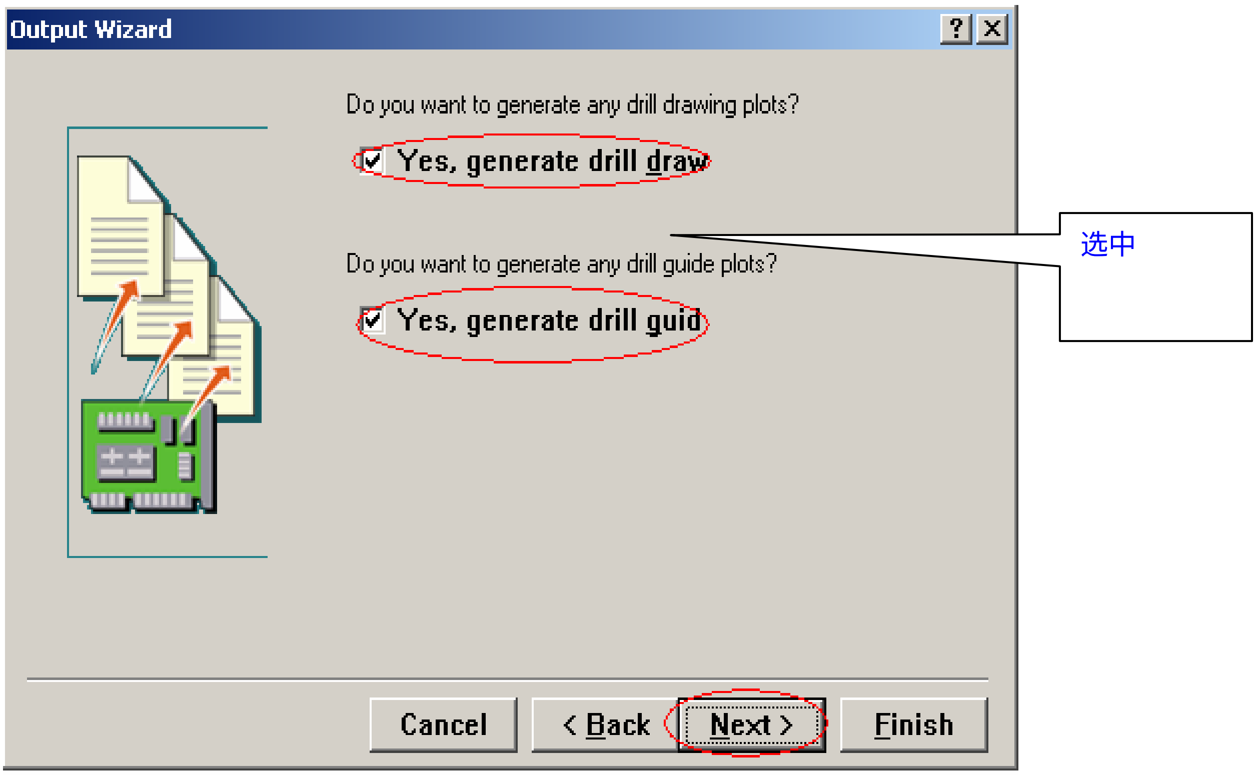

Output Wizard
How apertures are handled
The Gerber output will be configured to use embedded apertures
(RS274X). In this mode the CAM Manager will automatically create
the apertures from the PCB each time the Gerber generation is
carried out.
If you do not want to use the embedded aperture format you must
ensure that there is an appropriate aperture list
Cancel
< Back
Next >
Finish


Output Wizard
Specify the units and format to be used in the Gerber output files. This controls the units (inches or millimeters), and the number of digits before and after the decimal point.
Units
Inches
Millimeter
Format
2:3
2:4
2:5
The number format should be set to suit the requirements of your
design. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil
resolution, and 2:5 has a 0.01 mil resolution. If you
Cancel
< Back
Next >
Finish
单位设置
格式设置

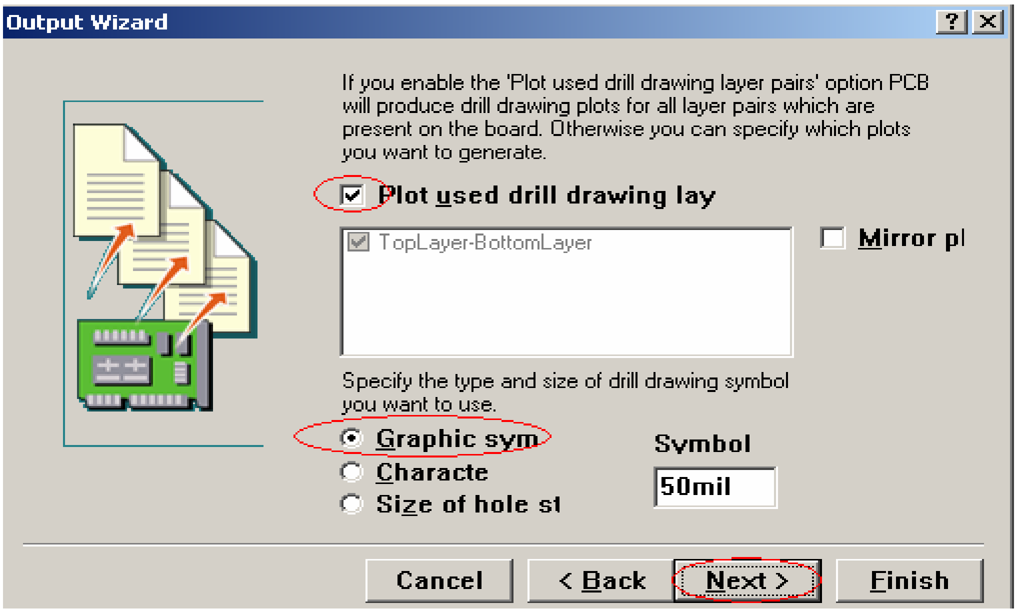
如上图全都按默认即可，直接按NEXT进入下一步

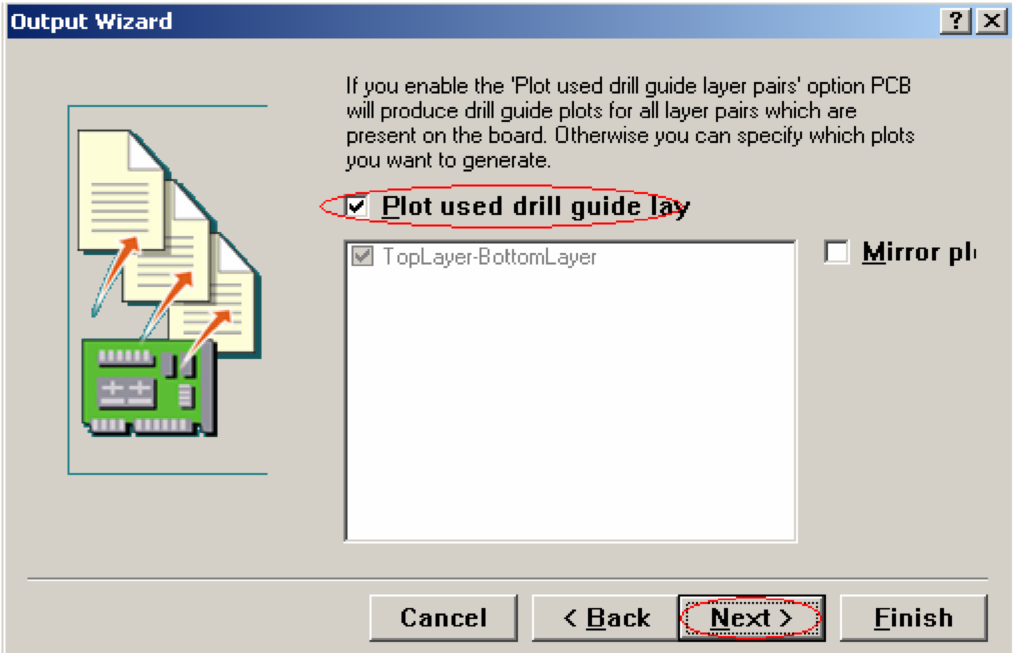


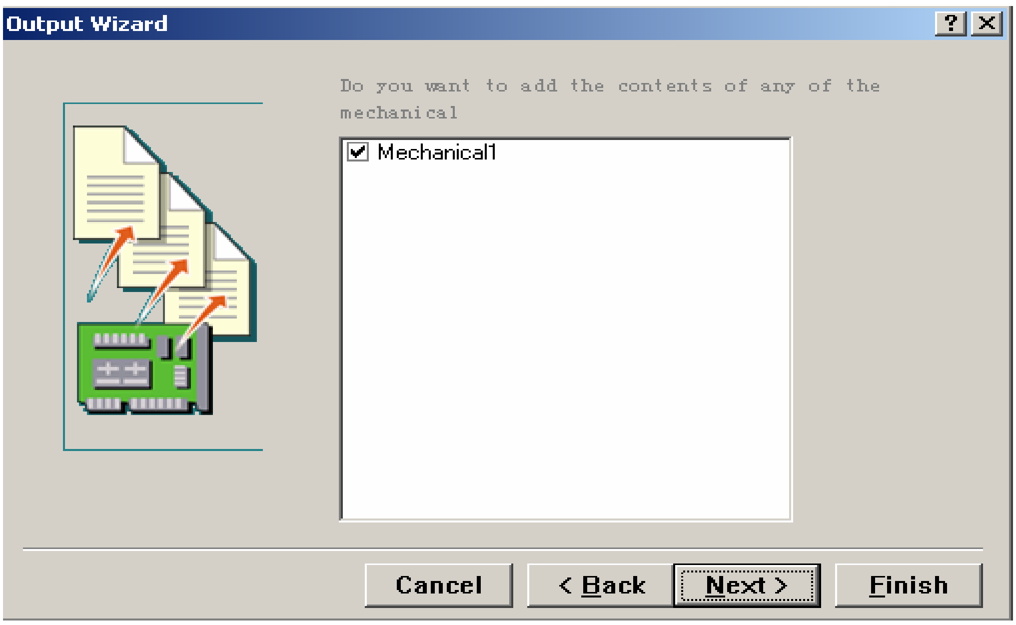
选择需要输出的层面，设置好后按NEXT

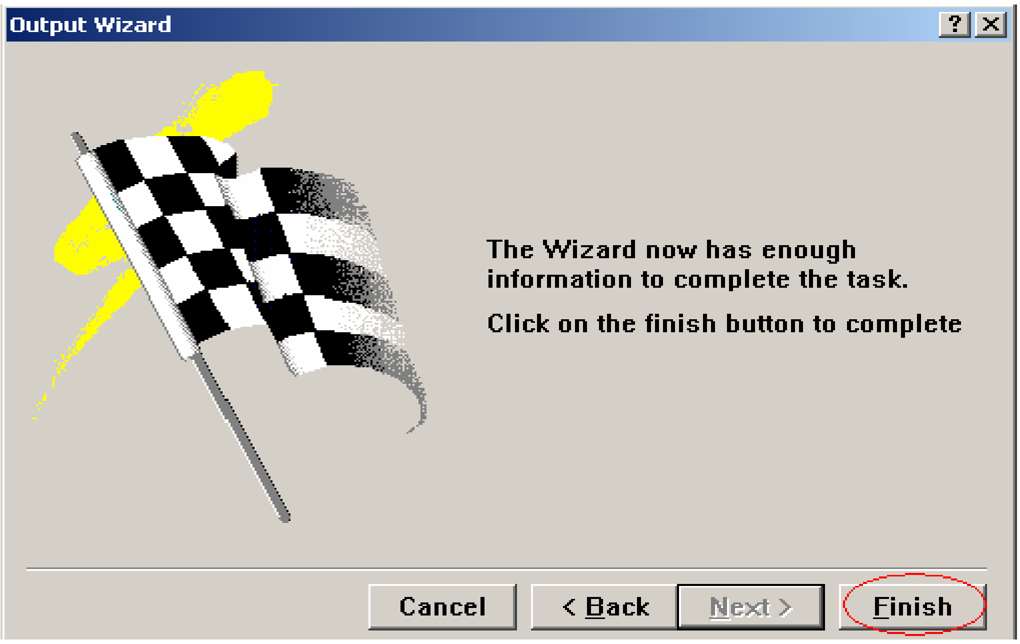


按NEXT后如下


Output Wizard
If you enable the 'Plot used drill drawing layer pairs' option PCB will produce drill drawing plots for all layer pairs which are present on the board. Otherwise you can specify which plots you want to generate.
Plot used drill drawing lay
TopLayer-BottomLayer
Mirror pl
Specify the type and size of drill drawing symbol you want to use.
Graphic sym
Characte
Size of hole st
Symbol
50mil
Cancel
< Back
Next >
Finish


Output Wizard
If you enable the 'Plot used drill guide layer pairs' option PCB will produce drill guide plots for all layer pairs which are present on the board. Otherwise you can specify which plots you want to generate.
Plot used drill guide lay
TopLayer-BottomLayer
Mirror pl
Cancel
< Back
Next >
Finish

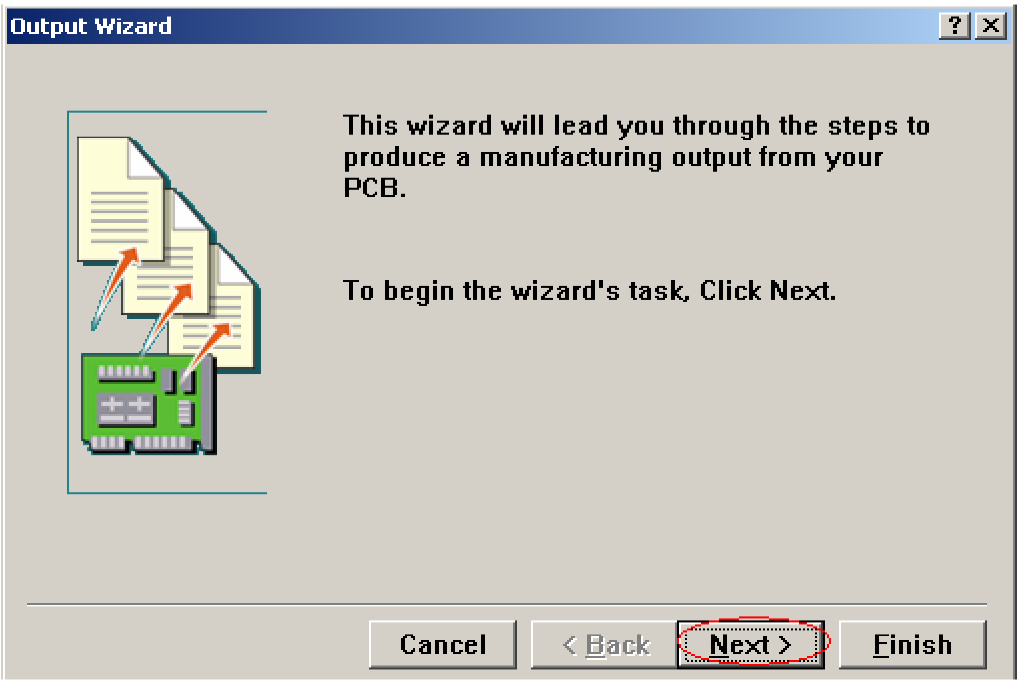

Output Wizard
Do you want to add the contents of any of the mechanical
Mechanical1
Cancel
< Back
Next >
Finish



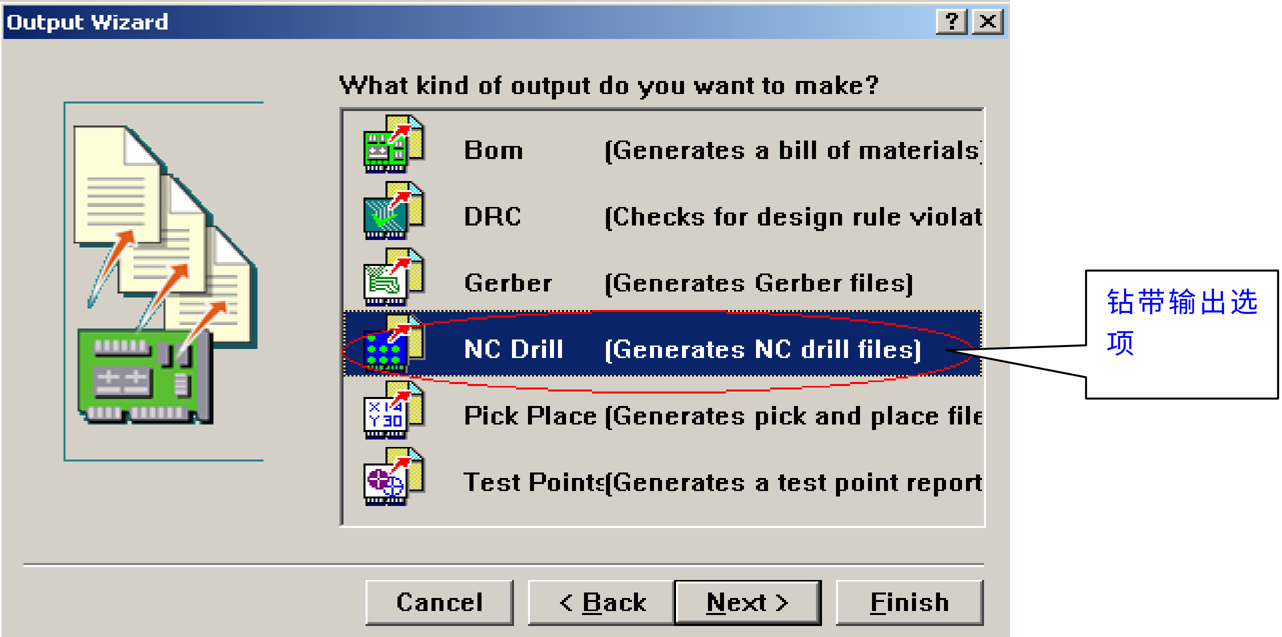
按FINISH完成

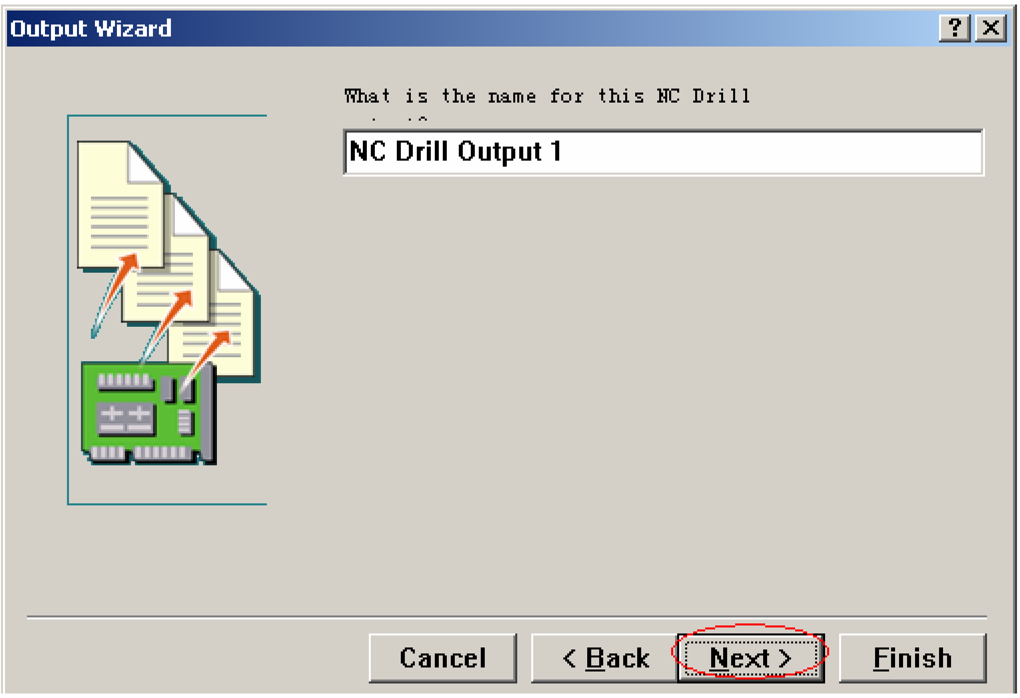
到了这里GERBER基本上转好了但是先不忙按F9导出GERBER

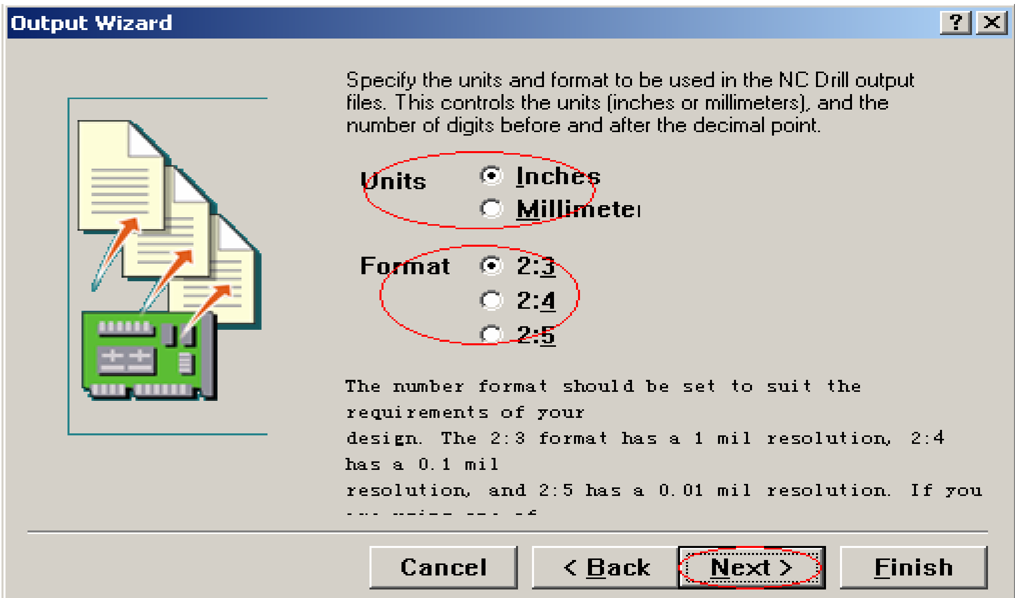
4 按TOOL——SCAM WIZARD就会出现如下图示按NC DRILL 转NC DRILL

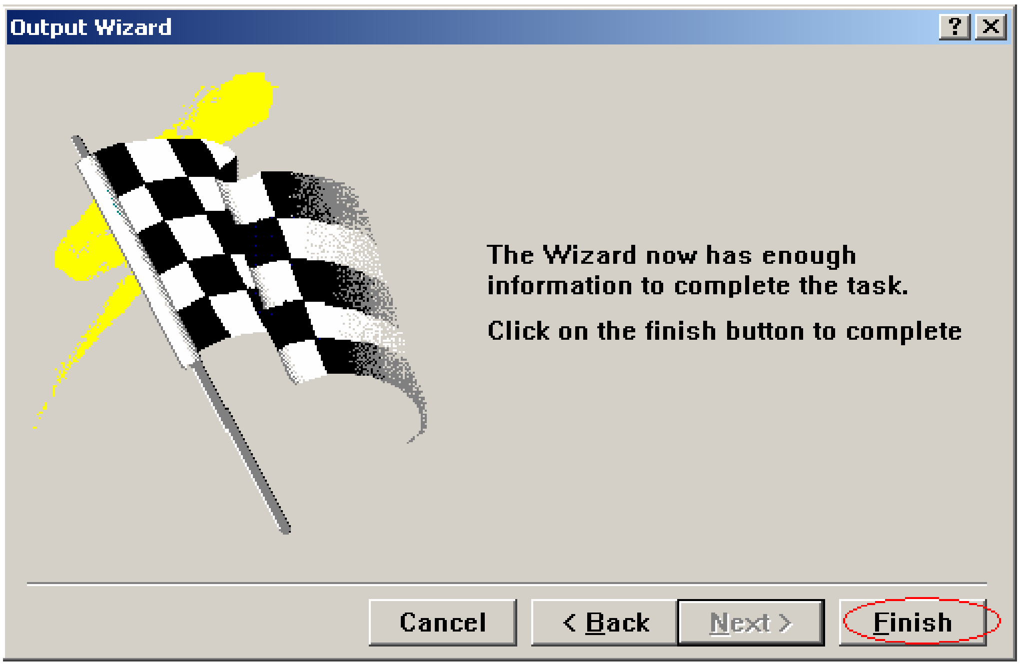


直接按NEXT进入下一步


Output Wizard
What kind of output do you want to make?
Bom [Generates a bill of materials]
DRC [Checks for design rule violat
Gerber [Generates Gerber files]
NC Drill [Generates NC drill files]
Pick Place [Generates pick and place file
Test Points [Generates a test point report
Cancel
< Back
Next >
Finish
钻带输出选项

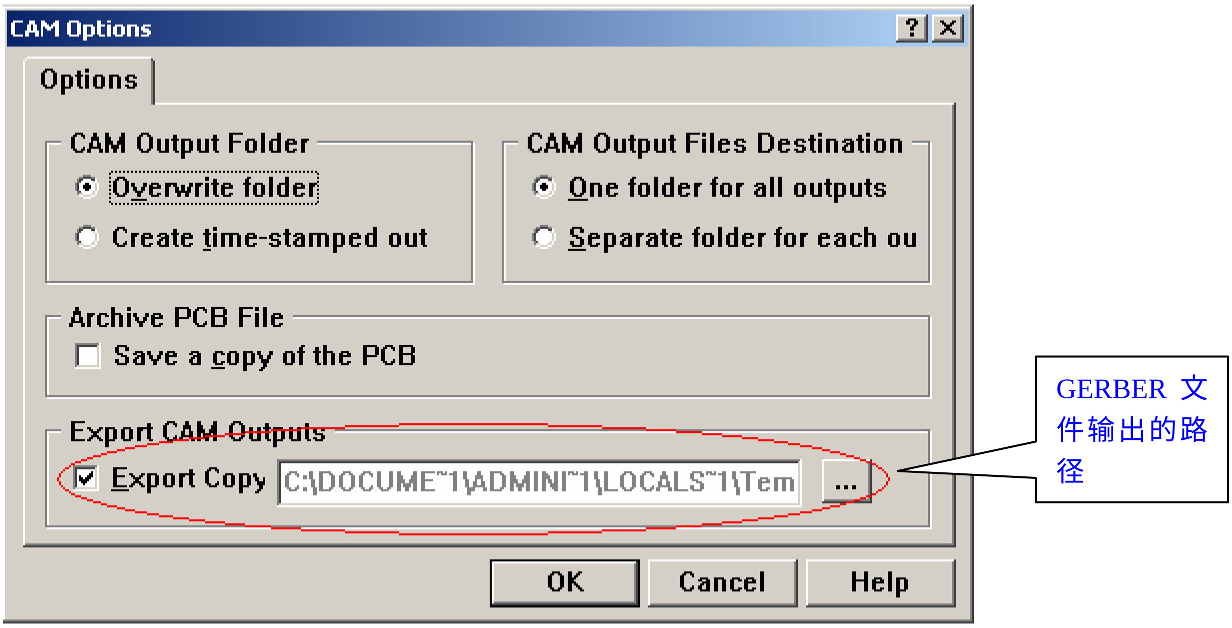

Output Wizard
What is the name for this NC Drill
NC Drill Output 1
Cancel
< Back
Next >
Finish


Output Wizard
Specify the units and format to be used in the NC Drill output files. This controls the units (inches or millimeters), and the number of digits before and after the decimal point.
Units
Inches
Millimeter
Format
2:3
2:4
2:5
The number format should be set to suit the requirements of your design. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0.01 mil resolution. If you
Cancel
< Back
Next >
Finish



按FINISH结束，到这里钻带层已经转好

5 点击Tools——Preferences如下



在这里你也可以不用设置路径，它默认在你的PCB图档文件里，然后在那里导出就可以了。

6 设置好你要输出GERBER文件的路径后按OK然后再用鼠标选中

☑ Gerber Output 1	Gerber
☑ NC Drill Output 1	NC Drill

按F9即可输出GERBER文件，GERBER已经转好了它会存放在你所设定的目录里.

这样GERBER已经转好，然后在CAM软件里进行处理