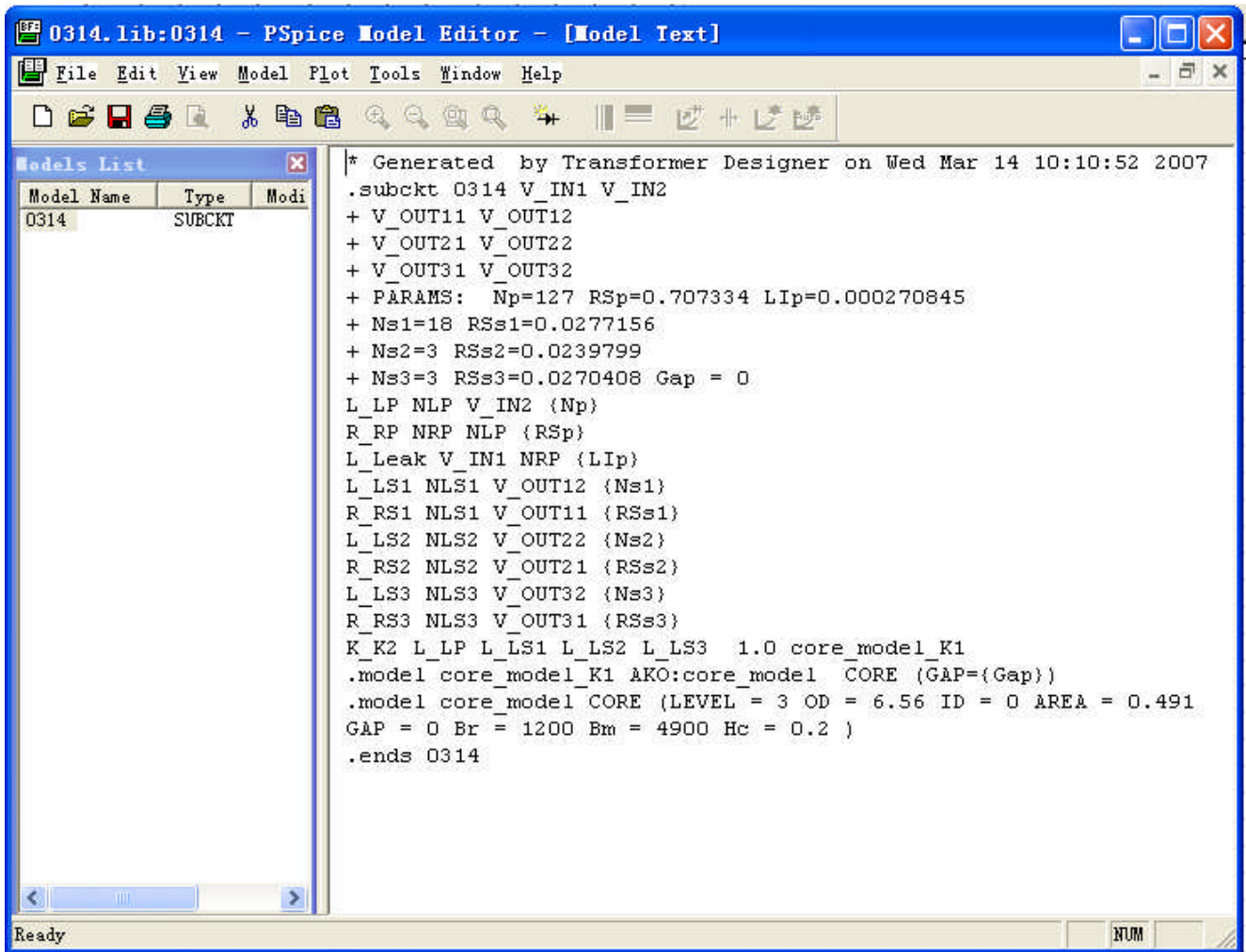


ORCAD PSPICE 导入问题

编辑好的LIB文件

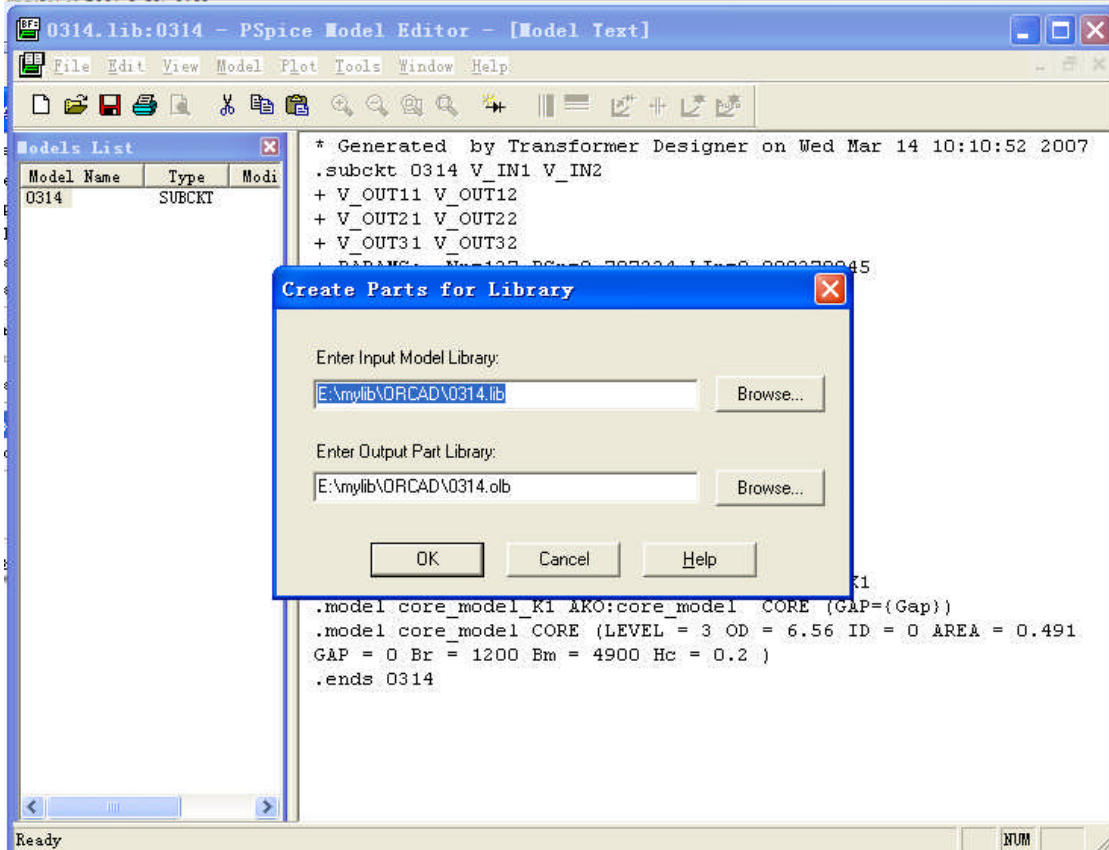
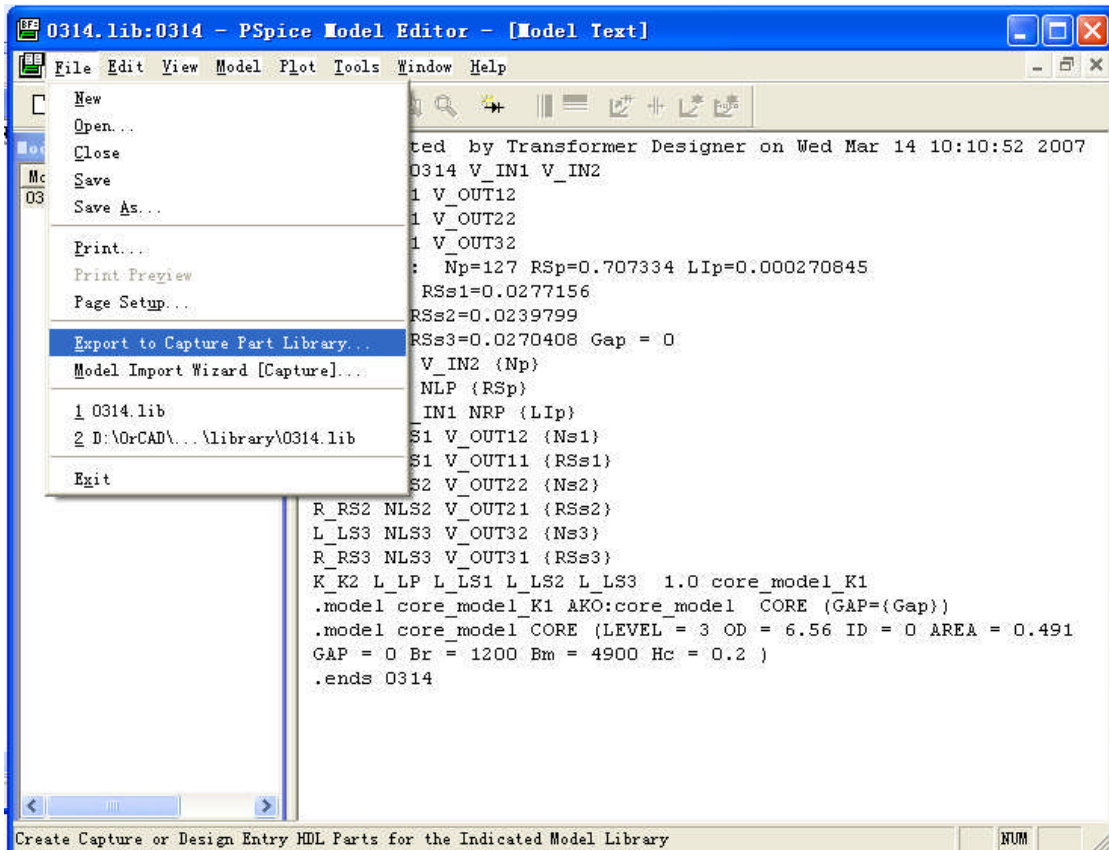
下面是一个已编辑好的变压器仿真模型,



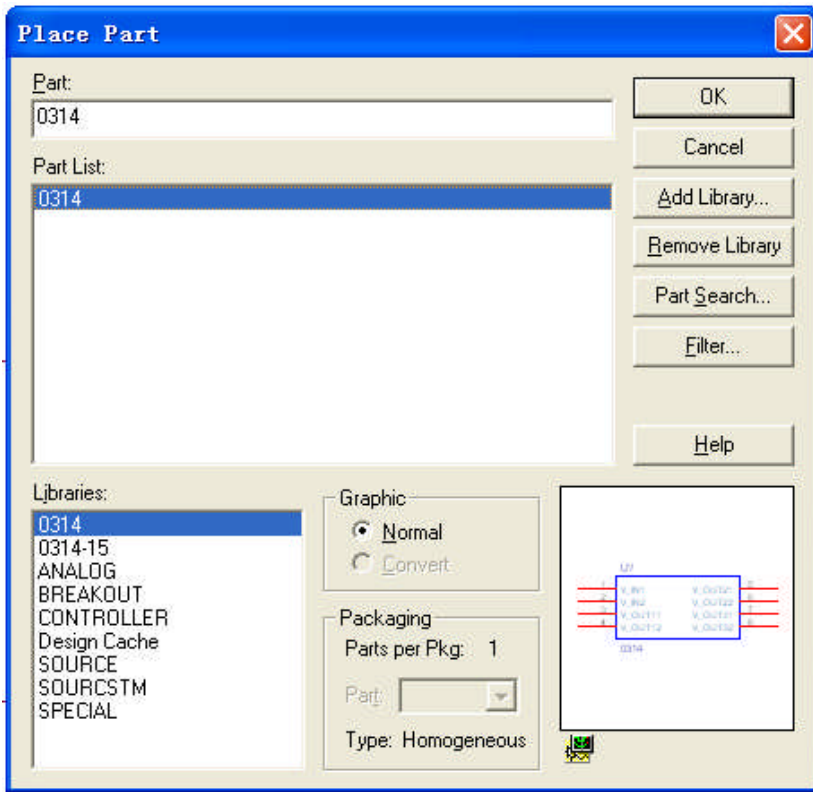
The screenshot shows the PSpice Model Editor window titled "0314.lib:0314 - PSpice Model Editor - [Model Text]". The window contains a menu bar (File, Edit, View, Model, Plot, Tools, Window, Help) and a toolbar. On the left, a "Models List" table shows the model "0314" of type "SUBCKT". The main text area contains the following code:

```
* Generated by Transformer Designer on Wed Mar 14 10:10:52 2007
.subckt 0314 V_IN1 V_IN2
+ V_OUT11 V_OUT12
+ V_OUT21 V_OUT22
+ V_OUT31 V_OUT32
+ PARAMS: Np=127 RSp=0.707334 LIp=0.000270845
+ Ns1=18 RSs1=0.0277156
+ Ns2=3 RSs2=0.0239799
+ Ns3=3 RSs3=0.0270408 Gap = 0
L_LP NLP V_IN2 {Np}
R_RP NRP NLP {RSp}
L_Leak V_IN1 NRP {LIp}
L_LS1 NLS1 V_OUT12 {Ns1}
R_RS1 NLS1 V_OUT11 {RSs1}
L_LS2 NLS2 V_OUT22 {Ns2}
R_RS2 NLS2 V_OUT21 {RSs2}
L_LS3 NLS3 V_OUT32 {Ns3}
R_RS3 NLS3 V_OUT31 {RSs3}
K_K2 L_LP L_LS1 L_LS2 L_LS3 1.0 core_model_K1
.model core_model_K1 AKO:core_model CORE (GAP={Gap})
.model core_model CORE (LEVEL = 3 OD = 6.56 ID = 0 AREA = 0.491
GAP = 0 Br = 1200 Bm = 4900 Hc = 0.2 )
.ends 0314
```

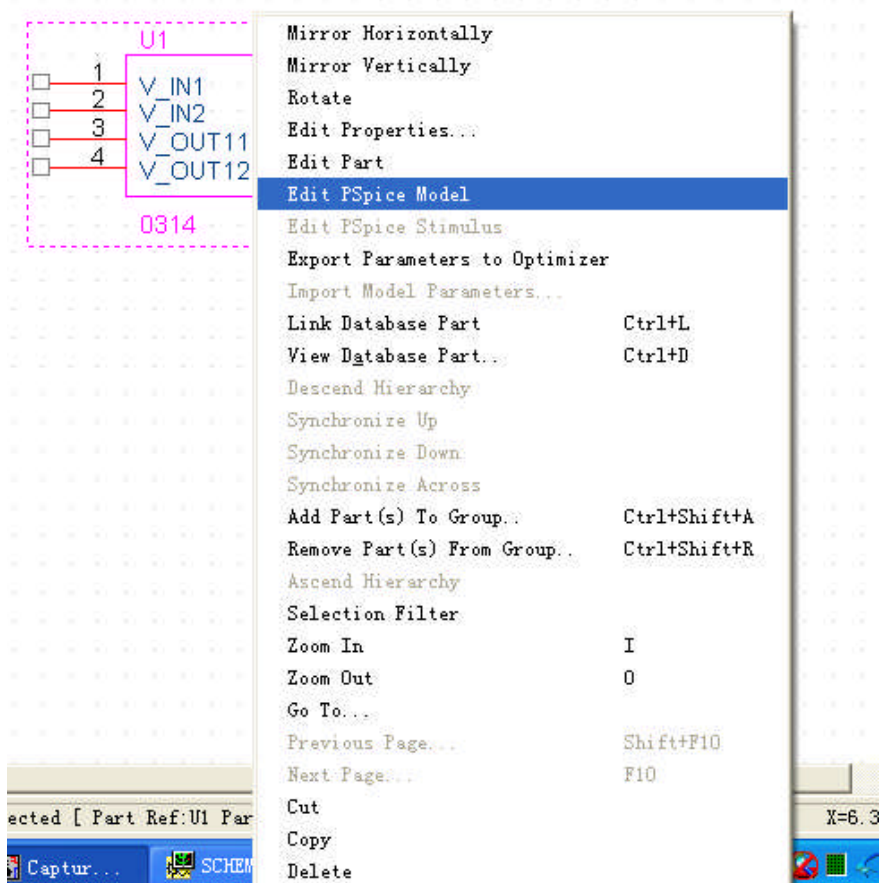
导出到可用的原理图符号:



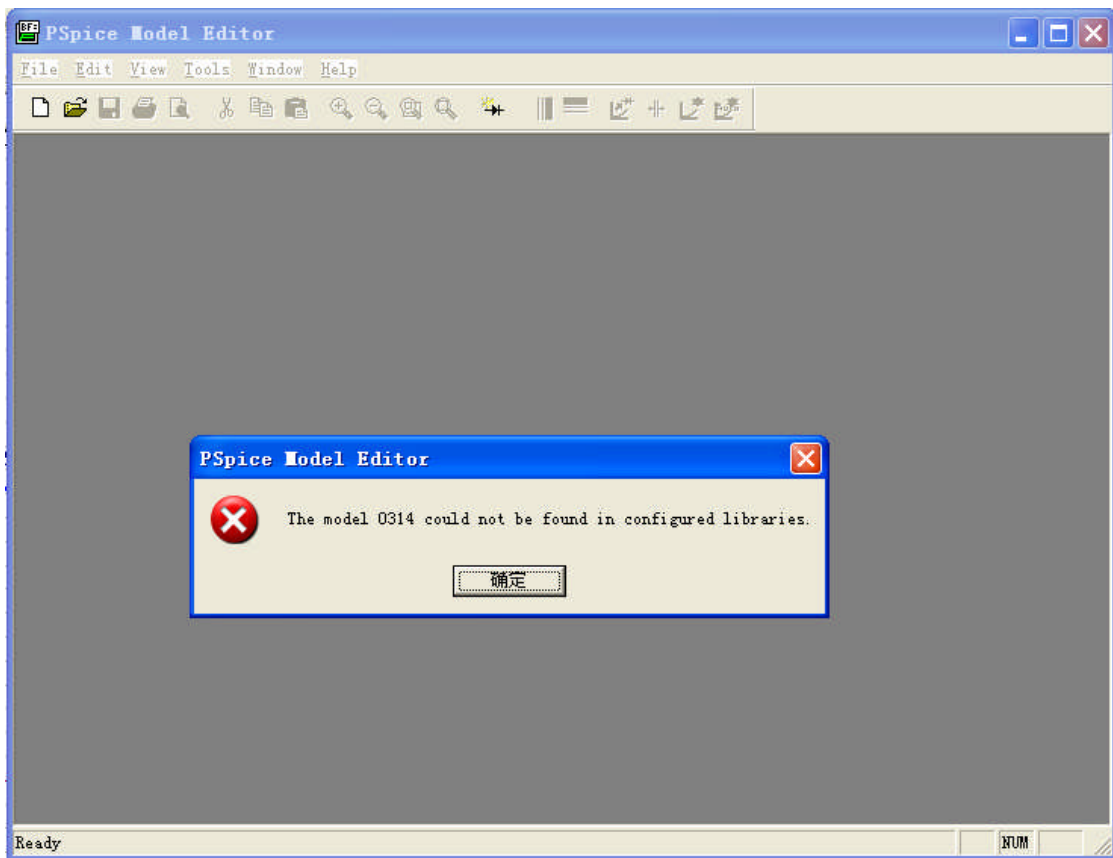
接下来使用刚做好的元件放置于原理图中:



仿真当然不行啦！左键单击选中这个元件，然后点击右键如下图所示：



Choose Edit PSpice Model
然后出现下面对话框：



看看上面这个错误的提示：

The model 0314 could not be found in configured libraries