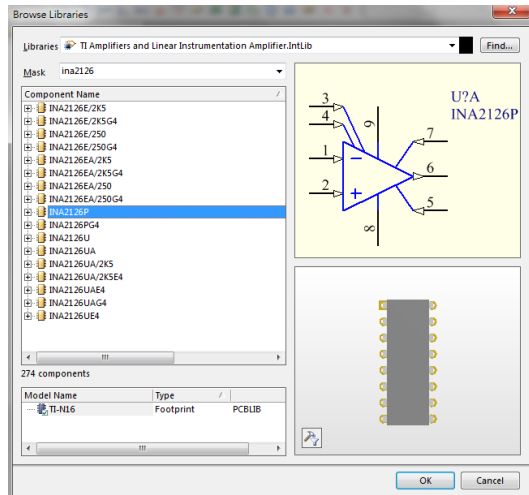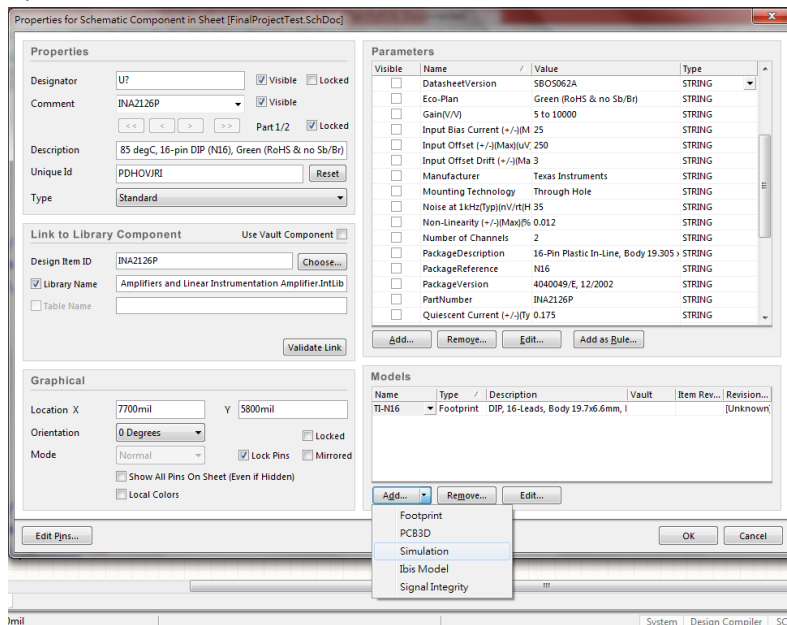# Tutorial

-Adding simulation model to INA2126

1) Pick the right part with footprint (DIP-16) as describe in the project.
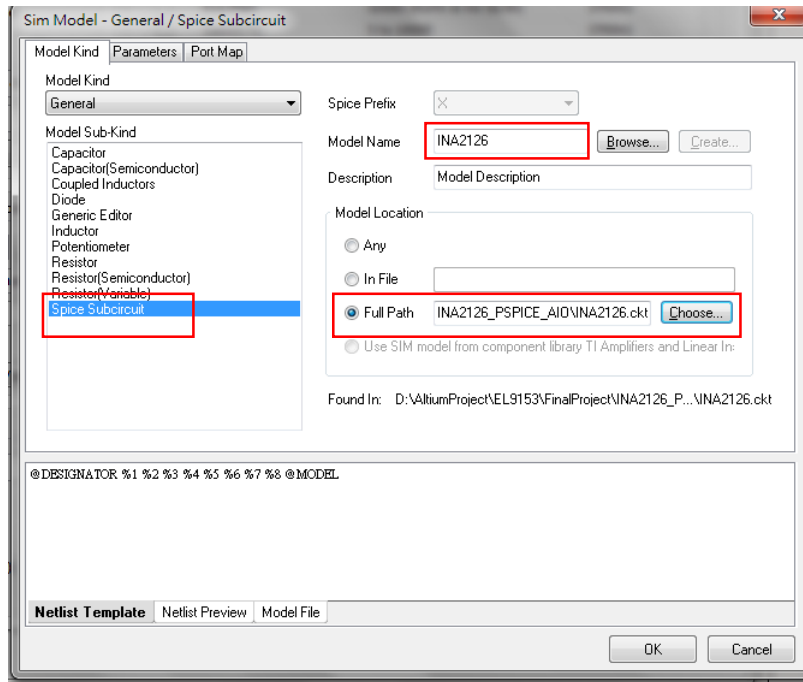

When you are placing this component, it will contains both "U?A" and "U?B" as there are 2 set of instrumentation amplifier in this chip. (For detail, please refer to the datasheet).

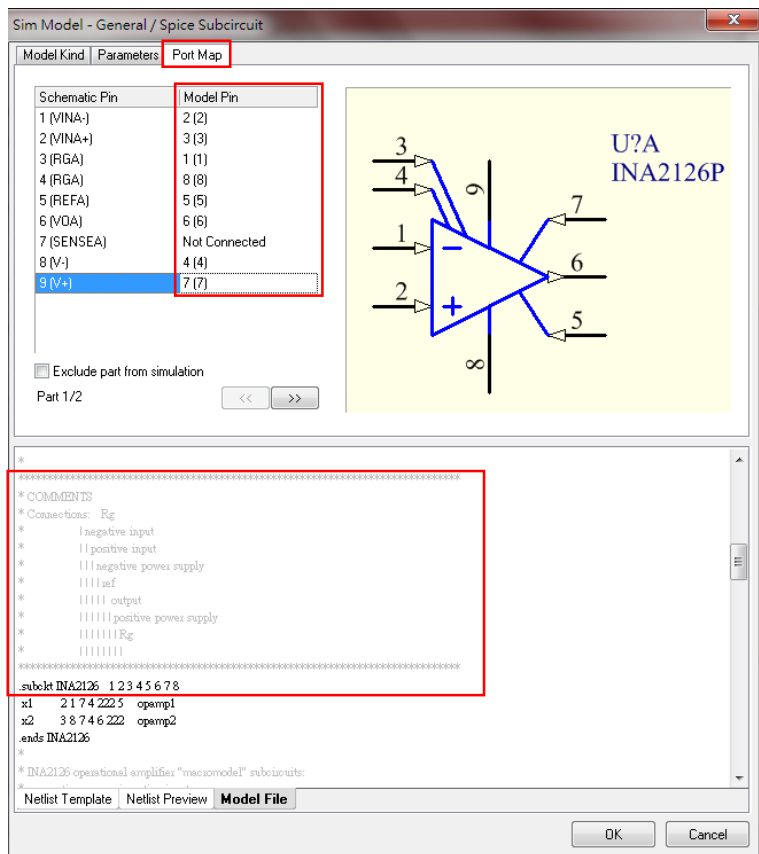2) Double click the "U?A" and add the simulation model
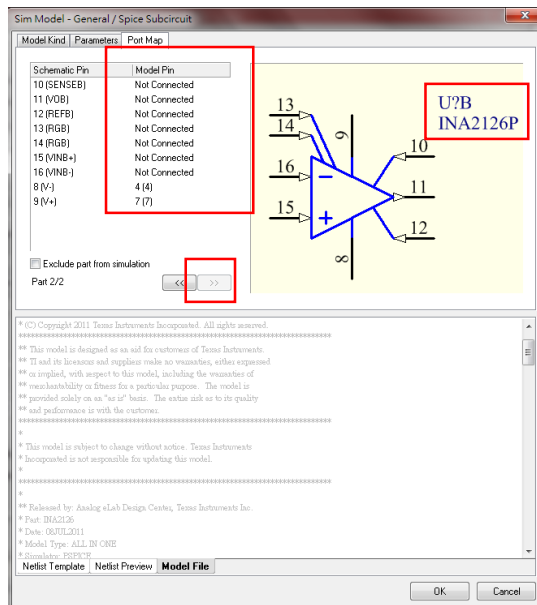
3) Select "Spice Subcircuit", manually type in "INA2126" in the <u>Model Name</u> section, and click the <u>Full Path</u> to link up your "INA2126.ckt" file

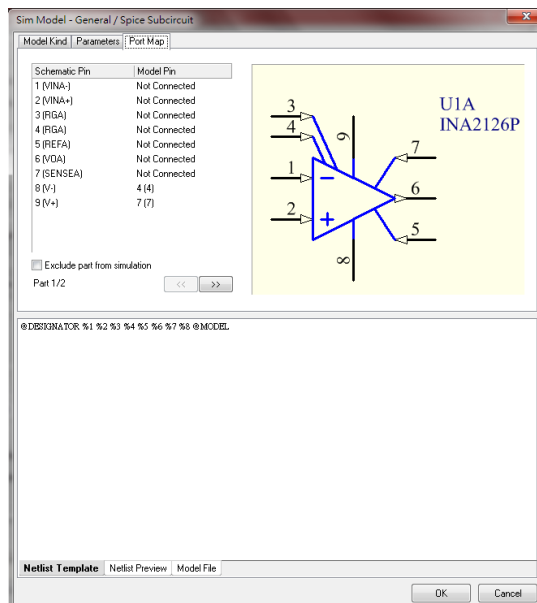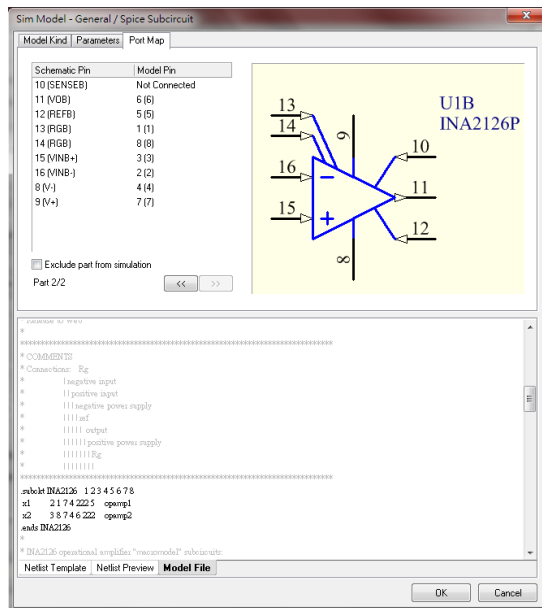4) Map the model to the schematic according to the model file

Sim Model - General / Spice Subcircuit

Model Kind | Parameters | **Port Map**

Schematic Pin	Model Pin
1 (VINA-)	2 (2)
2 (VINA+)	3 (3)
3 (RGA)	1 (1)
4 (RGA)	8 (8)
5 (REFA)	5 (5)
6 (VOA)	6 (6)
7 (SENSEA)	Not Connected
8 (V-)	4 (4)
9 (V+)	7 (7)

☐ Exclude part from simulation

Part 1/2   `<<`   `>>`

U?A
INA2126P

```
*
************************************************************
* COMMENTS
* Connections:  Rg
*           | negative input
*           | | positive input
*           | | | negative power supply
*           | | | | ref
*           | | | | | output
*           | | | | | | positive power supply
*           | | | | | | | Rg
*           | | | | | | | |
************************************************************
.subckt INA2126  1 2 3 4 5 6 7 8
x1     2 1 7 4 222 5   opamp1
x2     3 8 7 4 6 222   opamp2
.ends INA2126
*
* INA2126 operational amplifier "macromodel" subcircuits:
```

Netlist Template | Netlist Preview | **Model File**

OK   Cancel

5) Since we are only adding the simulation model to "U?A" right now, disconnect the portion for "U?B".

6) Click "OK" and you have successfully added the simulation model to half of the INA2126

7) Repeat step 2-6 but adding the simulation model to "U?B" now.
The Port Map becomes:

Congratulations!

You may now simulate the INA2126 in your design.