Foundry Design Kit for ADS
User Guide

Revision History

Version	Date	Description
1.0	2007/09/11	Initial release

Table of Contents

Chapter 1. ADS Design Kit Installation

Please note that this user guide can be adapted in every released UMC FDK document and is written herein as an example for the 90 nm design rules.

1-1. Software Requirements

1. Agilent ADS **(2006A)**

1-2. Installation Procedure

1. Start ADS and click on the **"DesignKit"** pull-down menu to install design kits in your ***Advanced Design System (Main)*** window (Figure 1-1).

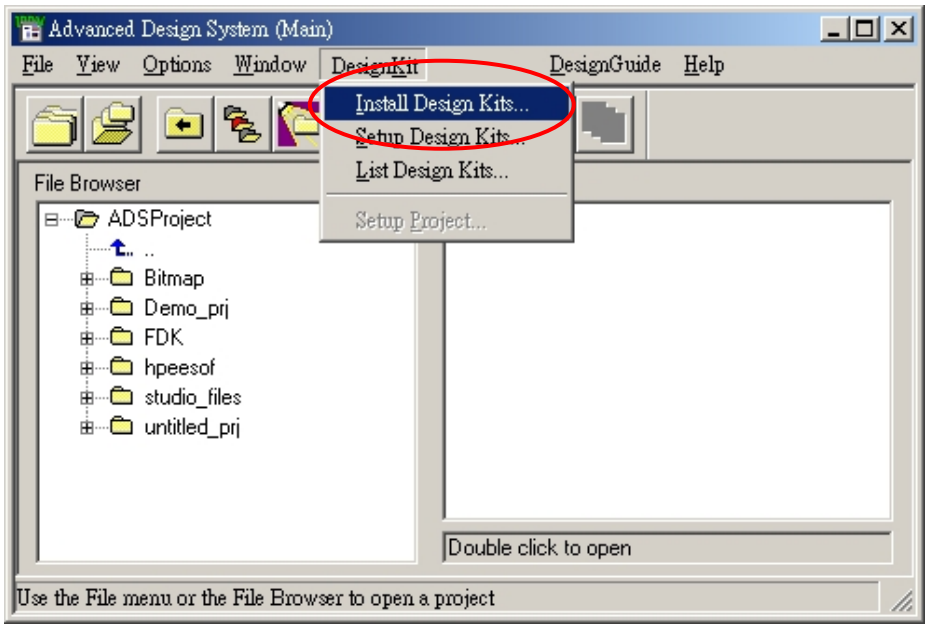

Figure 1-1. Design kit pull-down menu

2. Follow the procedure of the **"Install ADS Design kit"** dialog boxes to install design kit. The design kit compressed file is "UMC_90_1P7M1T1F_CMOS_FDKA_B01_PC.zip" (Figure 1-2).

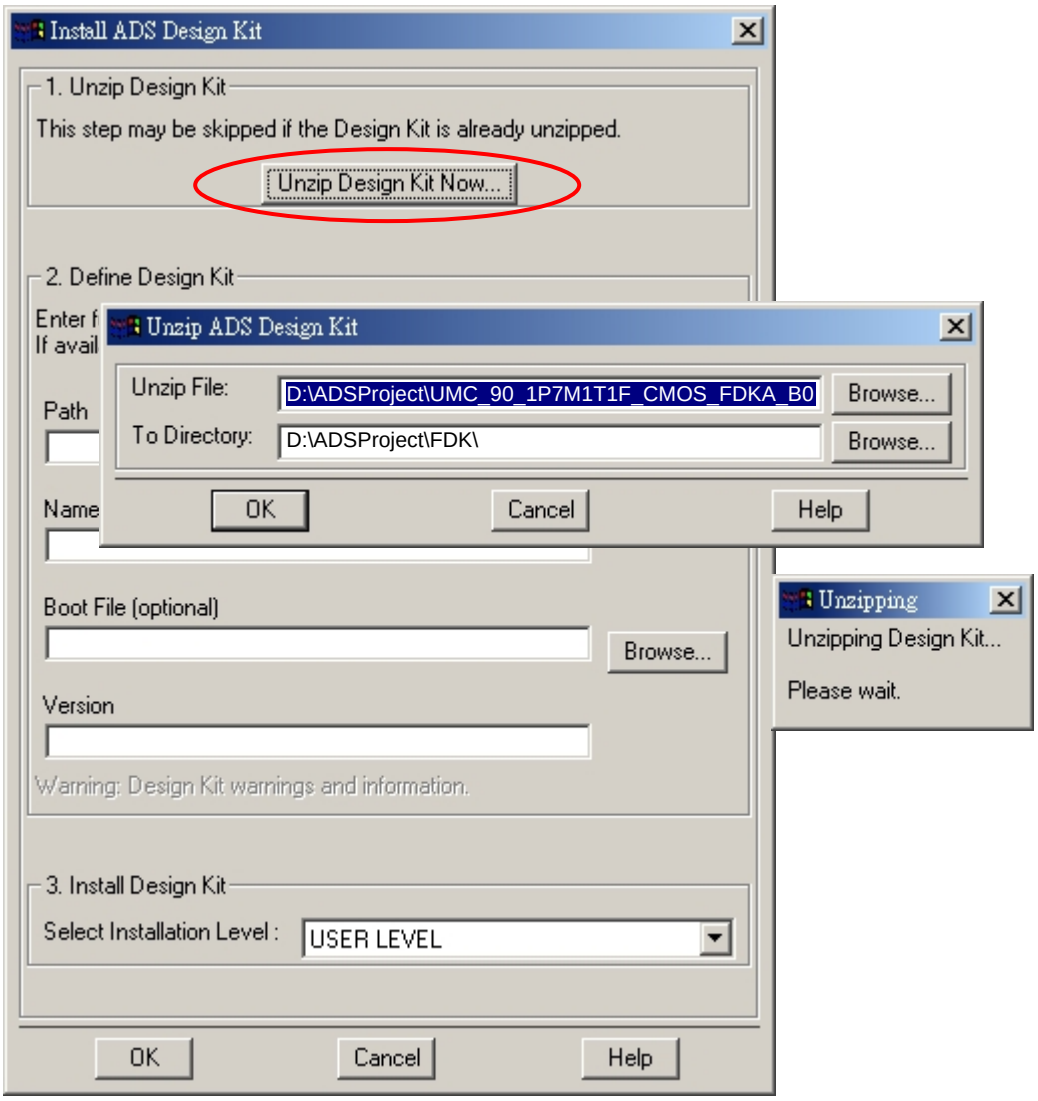

Figure 1-2. Unzip design kit compressed file

3. The new installed Design Kit will be included in your ADS environment now (Figure 1-3).

Note: Please refer to the document of “Agilent Design Kit Installation and Setup” for detailed information.

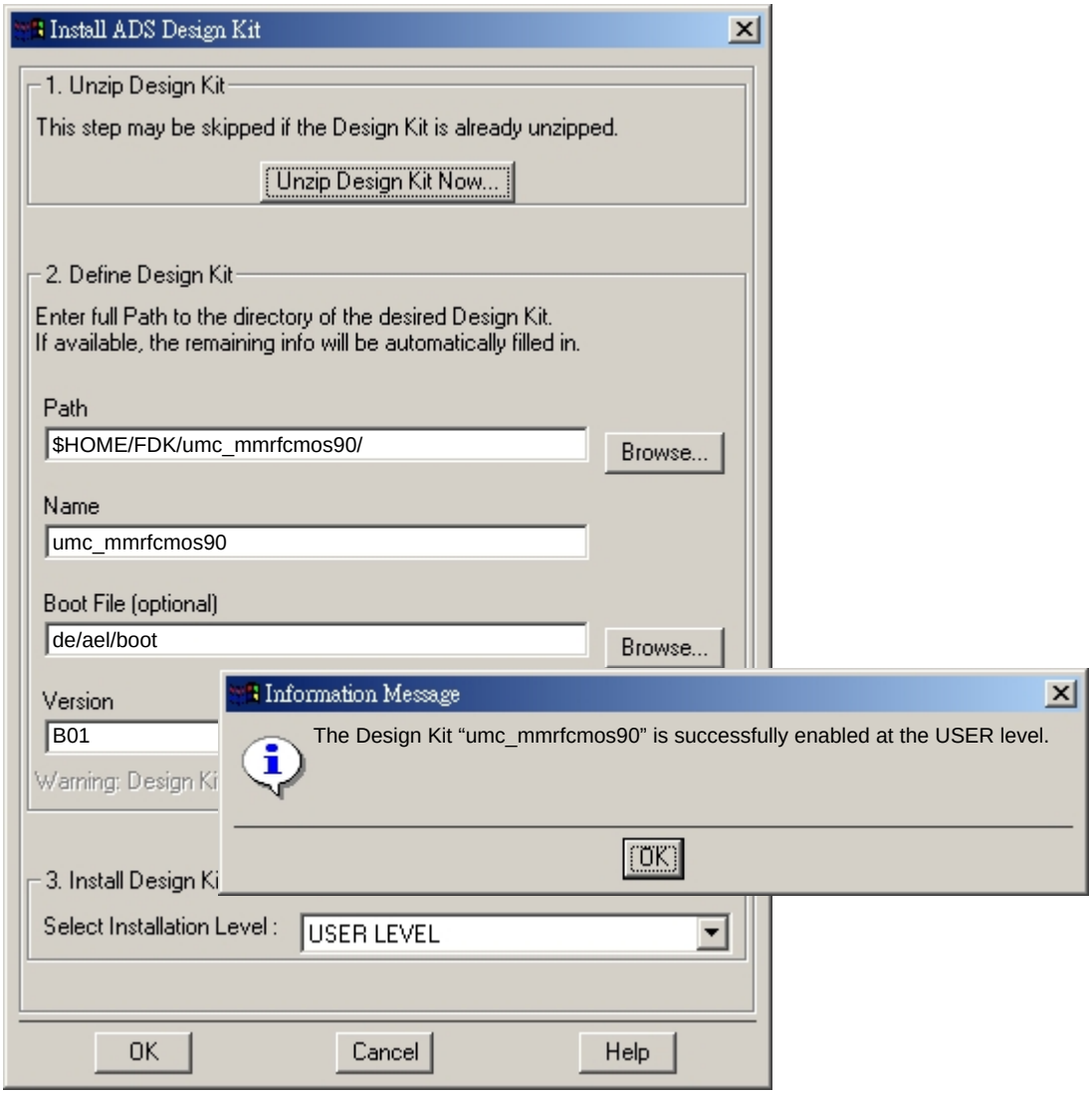

Figure 1-3. Design kit installed successfully

Chapter 2. Schematic Entry

At this stage, you should have the created New Project in your working directory for running ADS with UMC FDK.

2-1. Creating a New Schematic

1. Start Agilent ADS.
2. To use the components in Design Kit, users should create a new project, e.g., **"Demo_prj"** (Figure 2-1).

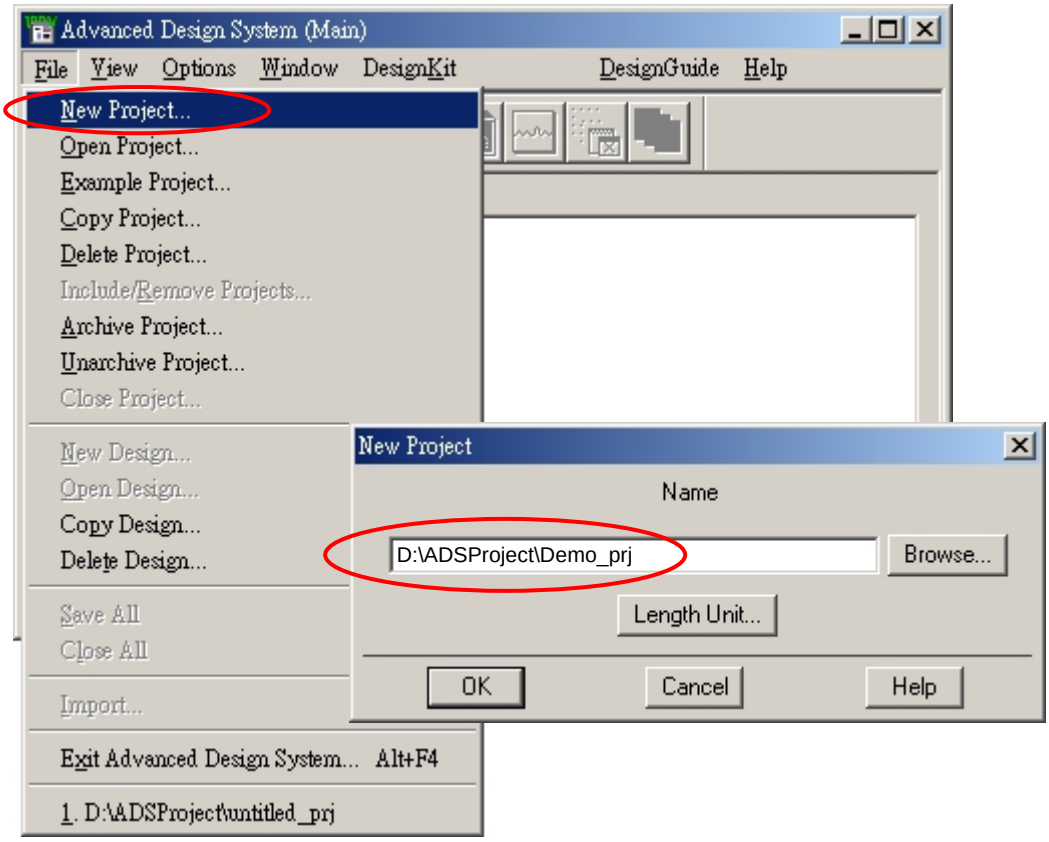

Figure 2-1. Create a new project

3. The ***ADS Schematic*** window will appear automatically after a new project is created (Figure 2-2).

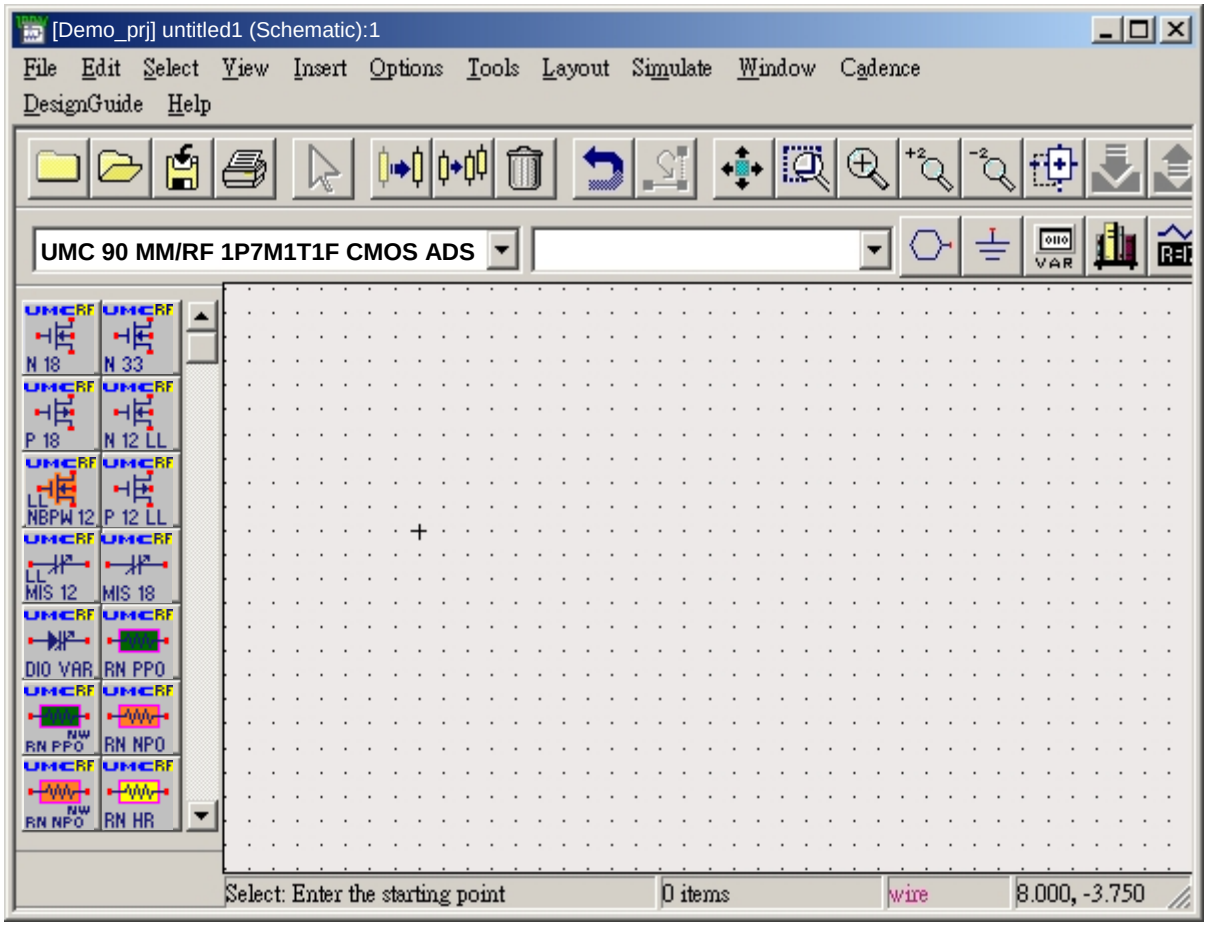

Figure 2-2. Schematic window of ADS

4. Choose the device in the palette window at the left side of the ***ADS Schematic*** window and put it in the Schematic (Figure 2-3).

   Note: For the detailed component list, please refer to the related release note.

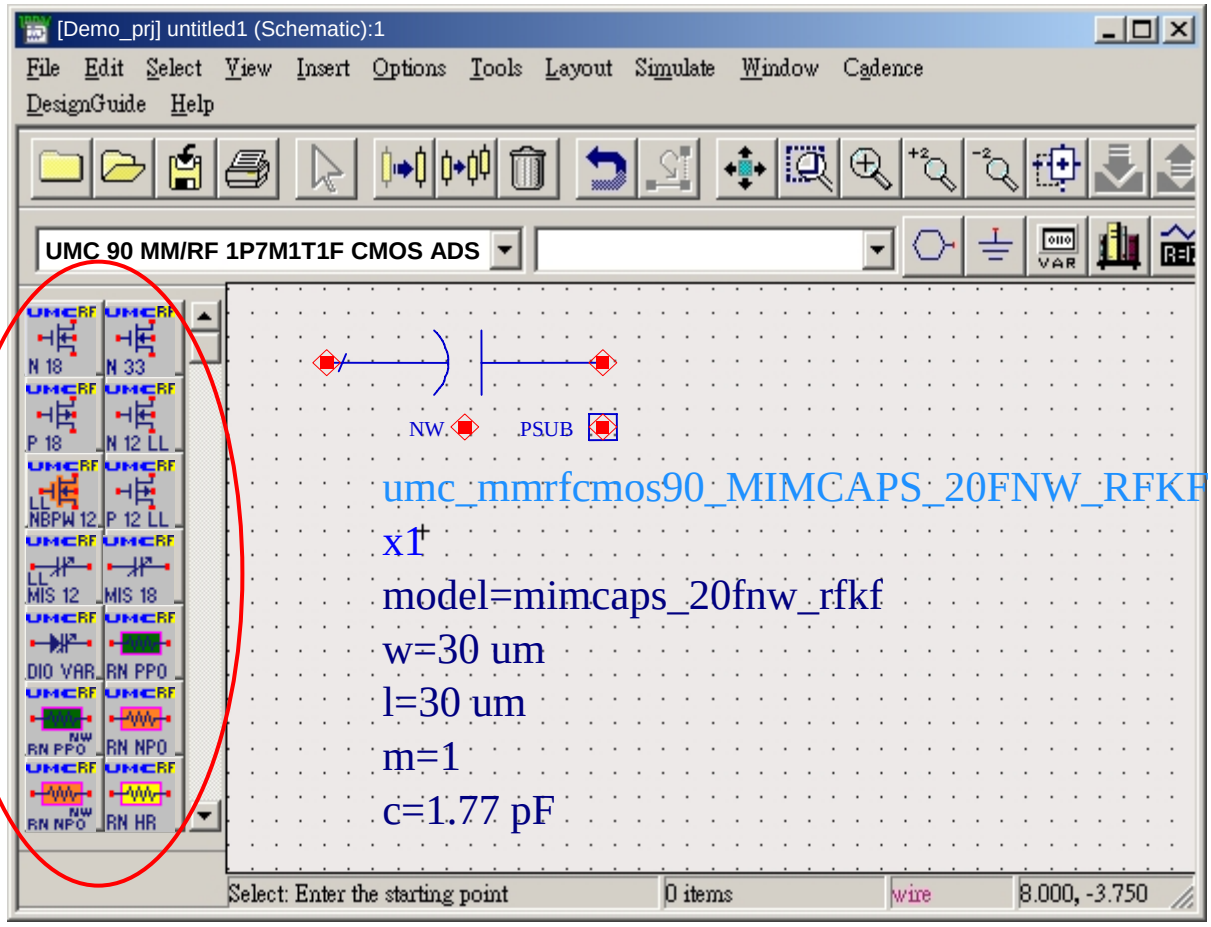

Figure 2-3. The new-added device in the ADS Schematic window

5. Select the device by clicking on **Edit → Component → Edit Component Parameters** to modify its parameter values (Figure 2-4).

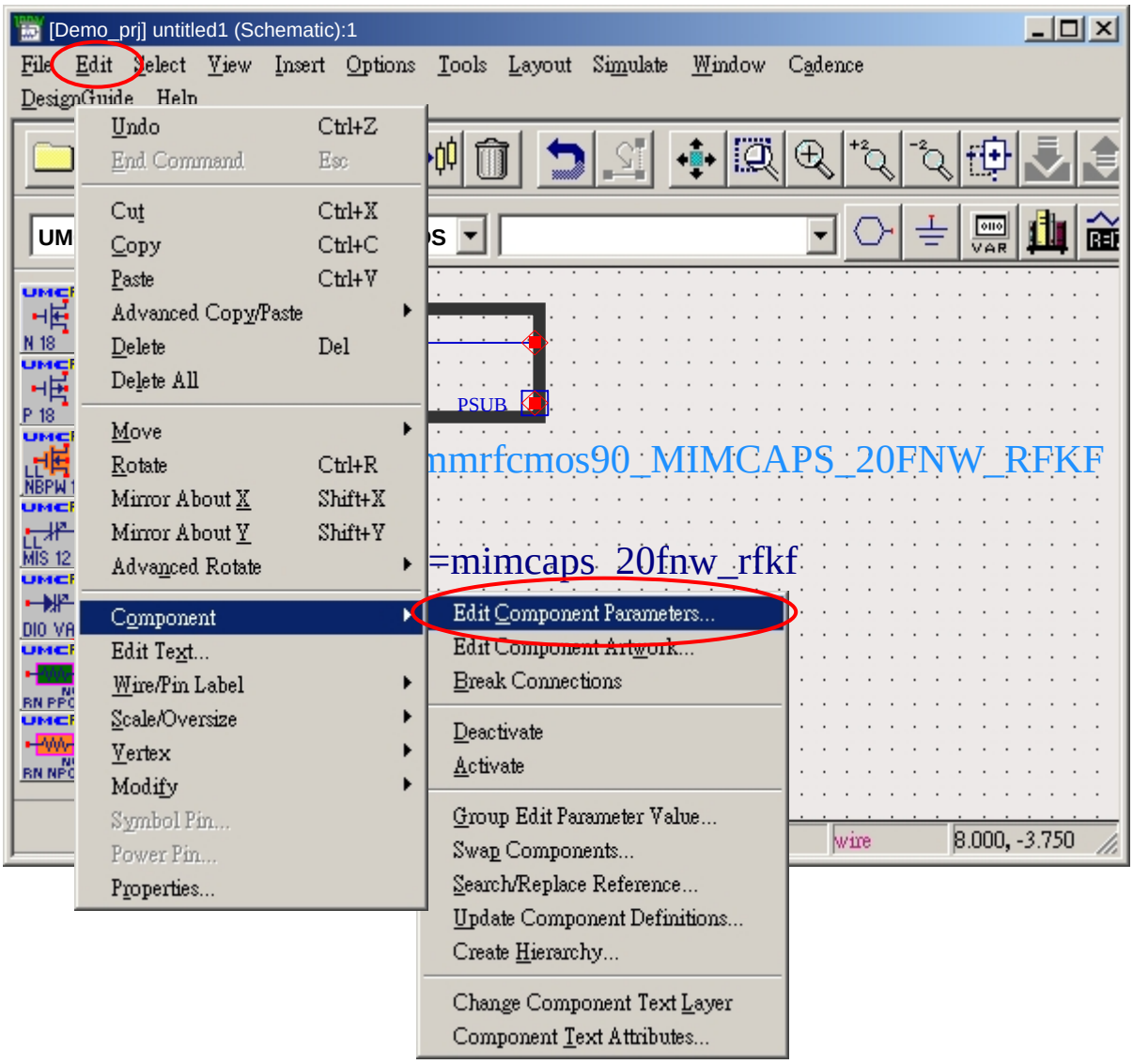

Figure 2-4. The new-added device in the ADS Schematic window

6. Modify the parameter values as you desired (Figure 2-5). If the value is out of model guaranteed range, the error message will appear (Figure 2-6) and the value will be fixed to the right one.
   Note: Please refer to the Agilent User Guide for detailed information.

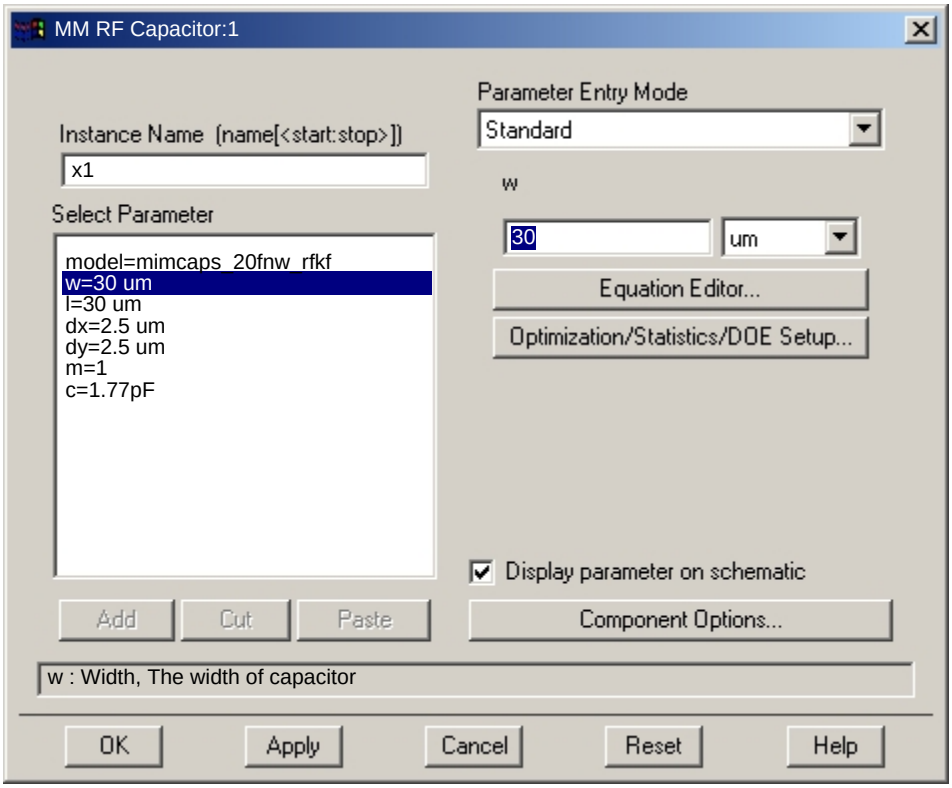

Figure 2-5. The new-added device in the ADS Schematic window

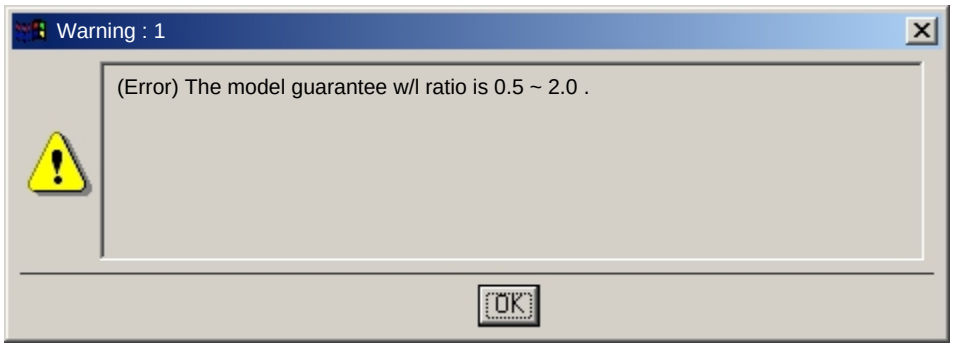

Figure 2-6. The error message window

2-2. Running Simulation

After the schematic and the related sources are completed, model files have to be set up before running simulation.

1. Key in **"NetlistInclude"** in **Component History** and put it in the ***ADS Schematic*** window (Figure 2-7).

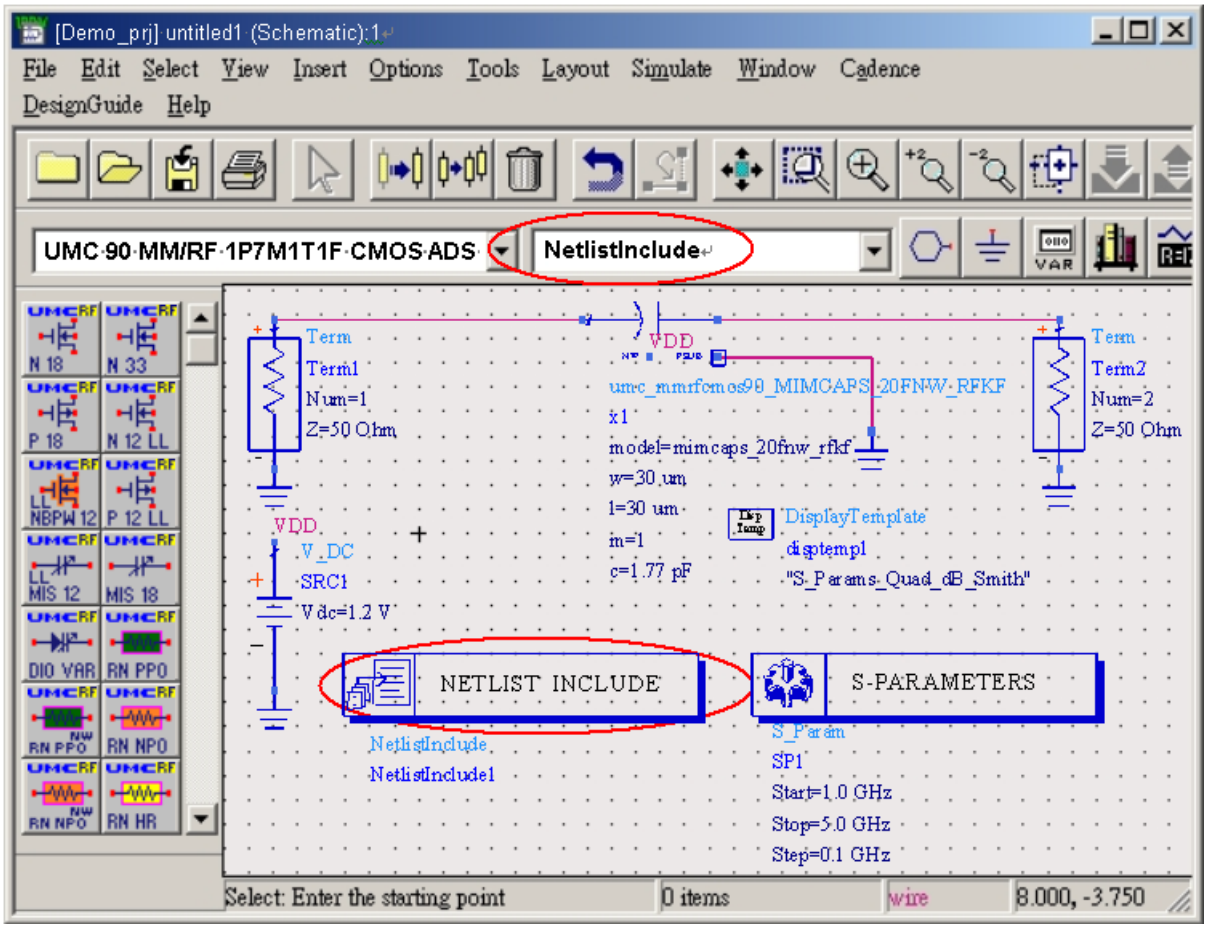

Figure 2-7. Add NetlistInclude

2. Edit the parameter of **"NetlistInclude"** component by following the steps shown in Figure 2-4.
3. Click on **Browse** to include the required model card for the simulation and assign the section name (Figure 2-8).

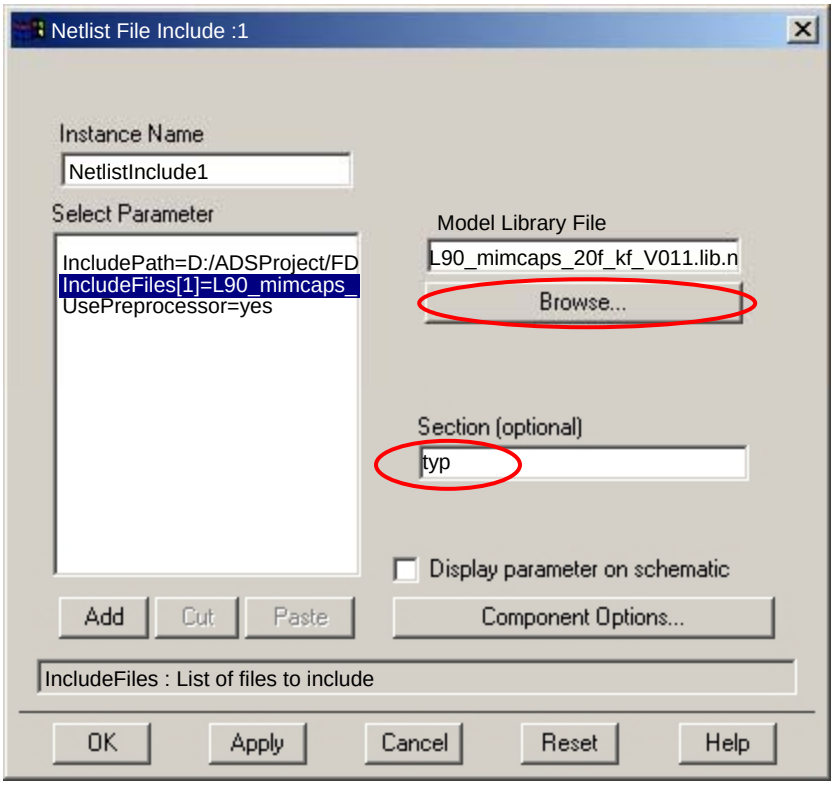

Figure 2-8. Include model cards

4. Click on **Simulate → Simulate** to start the simulation (Figure 2-9) and check the result.

   Note: Please refer to the document of “Agilent Circuit Simulation” for detailed information.

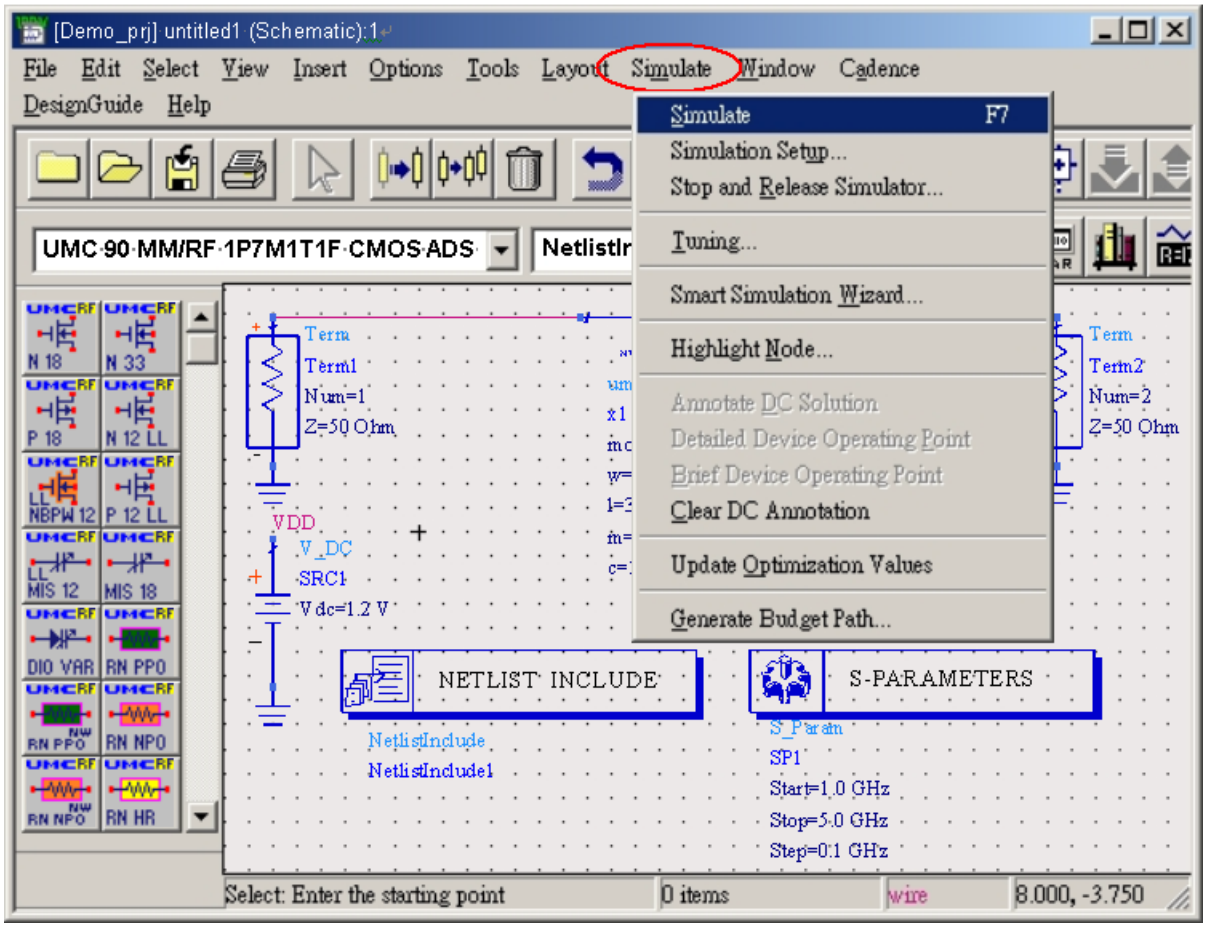

Figure 2-9. Start the simulation

2-3. Calibre Netlist Export

As schematic is completed, a netlist file is required for LVS.

1. In the ***ADS Schematic*** window, click on **Tools → Netlist Export → Create ADS Front End netlist…** (Figure 2-10) and the **Create ADS Front End Netlists** window will appear (Figure 2-11).

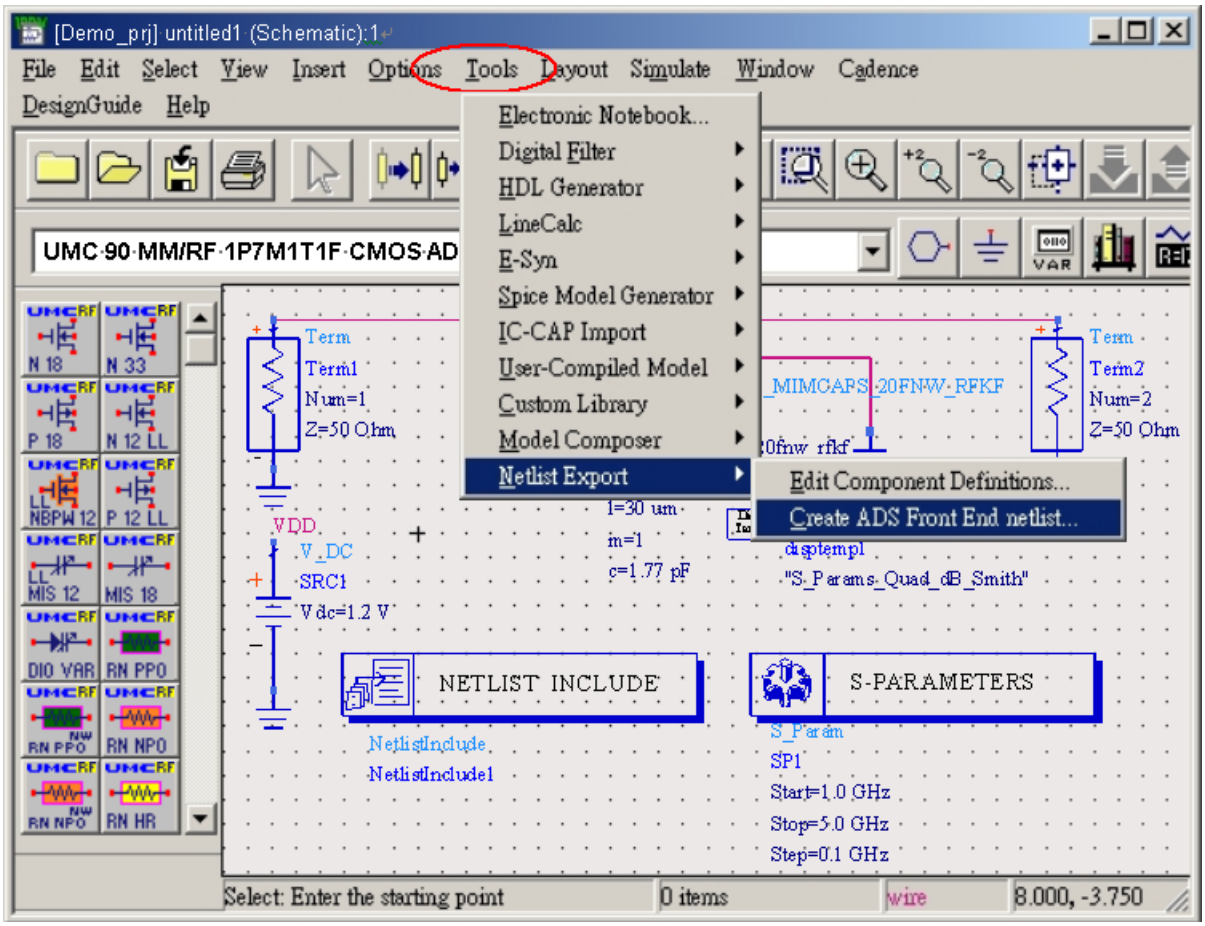

Figure 2-10. Calibre netlist export

2. Change the **Tool** option to **“calibre”** and assign the **Netlist file**.
3. Click **OK** to generate the Calibre netlist file.

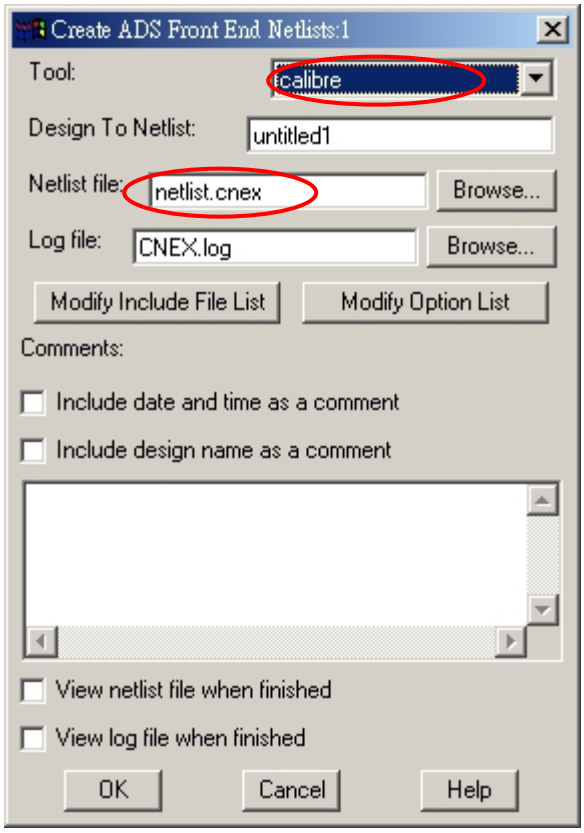

Figure 2-11. Create ADS Front End Netlists widow