Troubleshooting ORCAD File Losses and Failures

Troubleshooting ORCAD File Losses and Failures

File names

If you are to have any success in coping with various file losses and program crashes, it is a great help to keep every project in its own folder, named with the project name. It also is a very good idea to keep the names with <u>no</u> blank spaces, like **MyProject** or **My_Project** and <u>not</u> **My Project**. The reason is that words following a blank space can become dropped, or the blank spaces can become deleted.[1] During execution, a call inside a program for a file named **My Project•ext** cannot find it, possibly because it is now named **MyProject•ext**, but referred to in the call as **My Project•ext**. Naturally, failure to find the file causes a failure to run. However, there are some other causes of failure to find a file, examined next.

When Orcad closes the file automatically before you get started

A simple warning that often can be ignored tells you that not all components of the project can be found. Often this warning disappears if you simply try opening the file a second time.

A more serious problem that sometimes occurs when you open a •OPJ file with CAPTURE is that CAPTURE tells you that it is closing and that you will have to restart the program. This sequence of events simply repeats, and the file can never be opened. The cause of this behavior can be a corrupted PROBE-related file, and the problem can be corrected simply by deleting the files related to storage of various PROBE simulations. That is, delete all the project files ending in extensions •dat and •out. Then try again to open the file.

When Orcad cannot find the file

LOST •dsn FILE – MISSING SCHEMATICS!

Fairly frequently, you open a project and there is no schematic! An example is shown in Figure 1. To fix this problem, we have to attach the real •DSN file, which is clearly present in the file folder for the project when we look with the Windows File Explorer as seen in Figure 2. The •DSN file listed in Figure 1 is only a dummy, with no schematics.
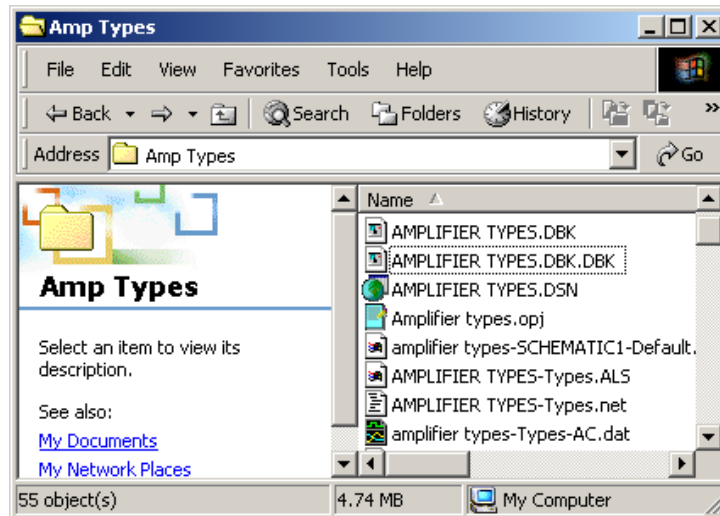
---

[1] Such filename changes are particularly likely to happen if you zip or unzip the project files, or if you copy the files from one directory to another one.
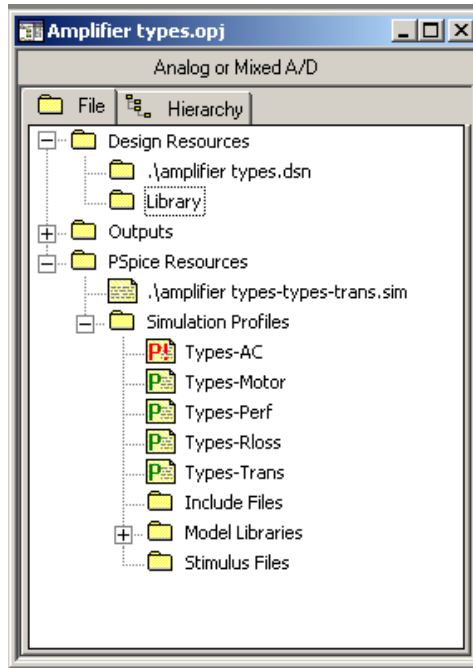
FIGURE 1:
    The • OPJ listing shows no schematic! The •DSN file listed is just a placeholder, and contains no schematics. Or, the •DSN file isn't even there.
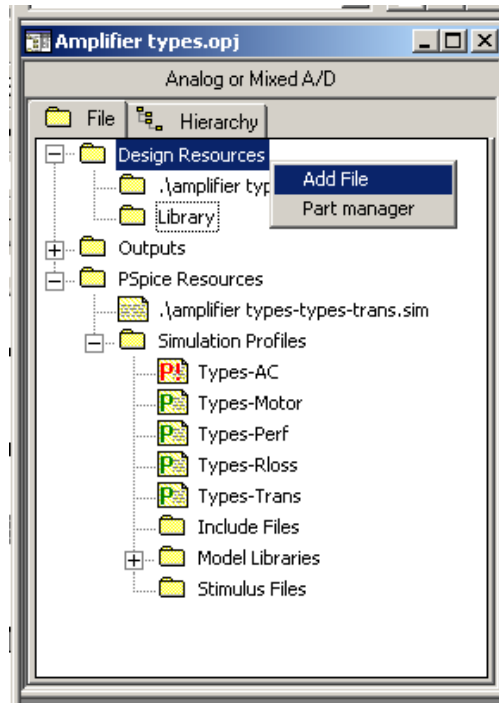

FIGURE 2:
    The •DSN file is in the project folder

If the dummy •DSN file is present as shown in Figure 1, we have to remove it by highlighting it in this menu and hitting the DELETE key. Then, to incorporate the real •DSN file we can proceed two ways:

- With the mouse drag the •DSN file from the Window Explorer over to the ORCAD menu and drop it in the DESIGN RESOURCES folder
- Formally add the •DSN file using the Orcad ADD FILE utility.

To follow the last method, highlight the DESIGN RESOURCES folder and right-click to obtain the ADD FILE menu, as shown in Figure 3.

FIGURE 3:
    Obtaining the ADD FILE menu
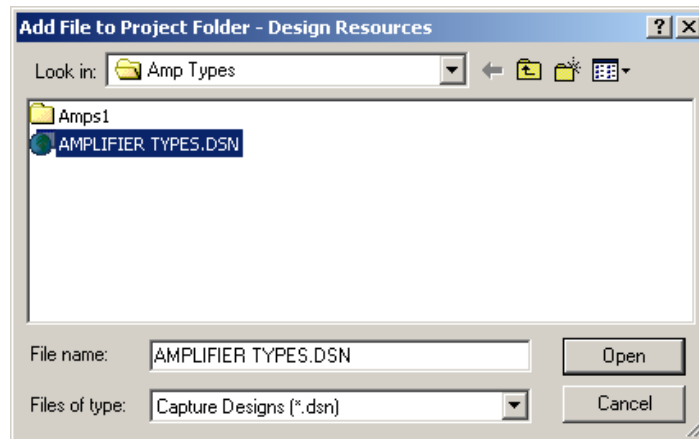
FIGURE 4:
    The ADD FILE menu. Select the •DSN file.

As shown in Figure 4, select the • DSN file and click OPEN.

LOST •sim – MISSING SIMULATION PROFILE!

It is extremely common for Orcad to lose a Simulation Profile[2]. When you look in the Window's File Explorer, the simulation profile is there, but Orcad refuses to find it until you try to recreate it, when it tells you that your Simulation Profile already exists! Go figure! You then have at least three choices:

• Create the Simulation Profile again under a new name
• Copy the schematic into a new project and reintroduce all the Simulation Profiles

---

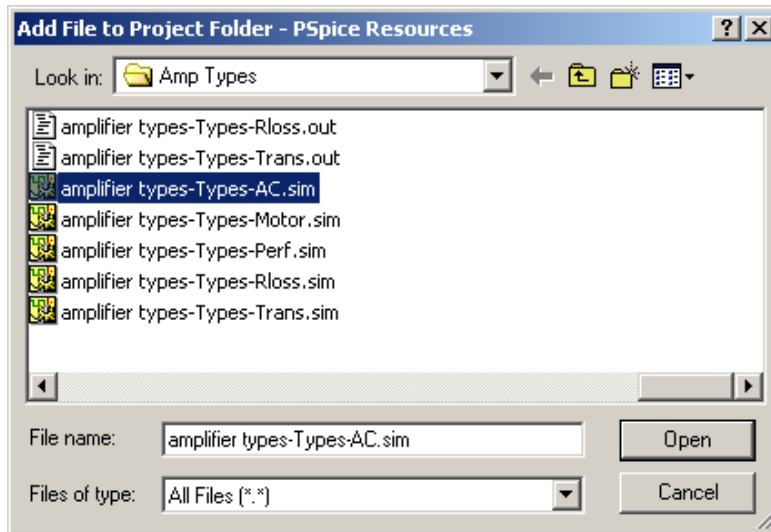[2] One cause is the system crashing – the auto-recovery loses or rewrites files

• Edit the ●OPJ file using NOTEPAD

It may be that the first choice is most expedient, but in case you want to edit the ●OPJ file, here's how.

In the ●OPJ listing in the Orcad window, select PSPICE RESOURCES and right click to get the ADD FILE menu, see Figure 5.
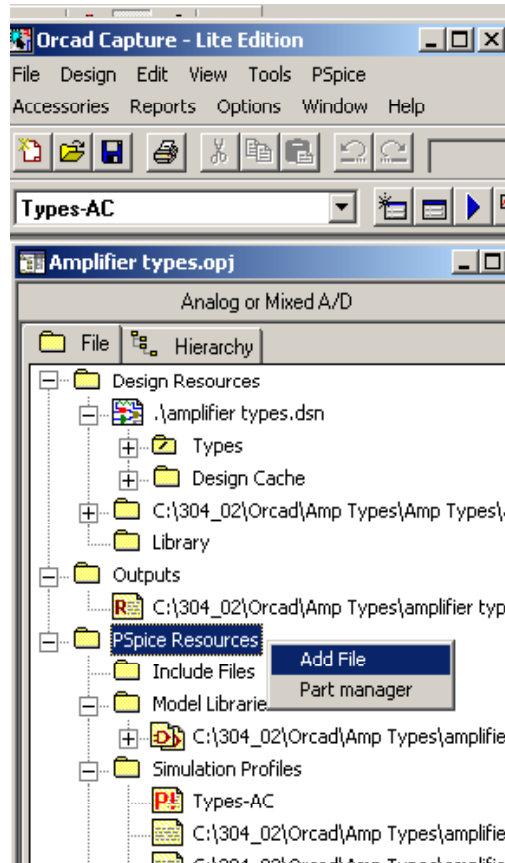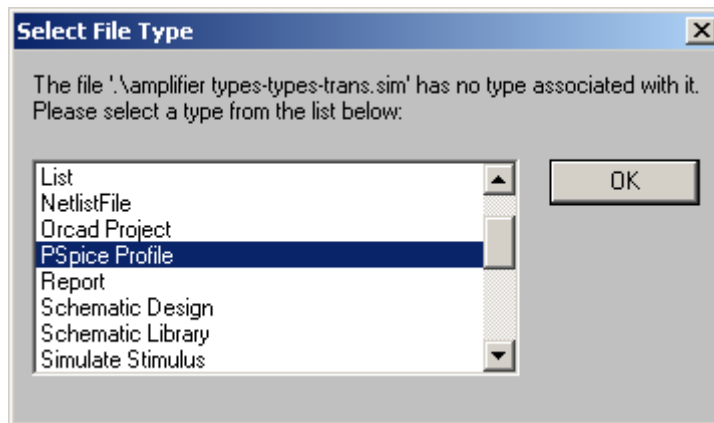

FIGURE 5:
    Obtaining the ADD FILE menu

FIGURE 6:
Arranging to add the •SIM files to •OPJ

With the ADD FILE menu, select the ALL FILES option so you can see the •SIM files. Highlight these files and add them, as shown in Figure 6.

FIGURE 7:
Identifying the file type as PSPICE PROFILE

As shown in Figure 7, identify them as type PSPICE PROFILE[3]. Once they are added, drag them down to the SIMULATION PROFILES block of the PSPICE RESOURCES as shown in Figure 5.

Now close CAPTURE. Next, open the •OPJ file using NOTEPAD. Find the section under PSPICE RESOURCES/ (FOLDER "SIMULATION PROFILES" as shown below.

Name of Project   Name of Schematic

(FOLDER "PSPICE RESOURCES"
  (FOLDER "SIMULATION PROFILES"   Name of Profile
    (FILE. "C:\304_02\ORCAD\AMP TYPES\AMPLIFIER TYPES-TYPES-AC.SIM"
    (TYPE "PSPICE PROFILE")))

Replace the (File <pathname>) with the following[4]

---

[3] If this option is not listed, don't worry: pick another entry. You can change it later in NOTEPAD.
[4] If the TYPE is incorrectly listed, change the TYPE to (Type "PSPICE Profile"))

                                       10/27/2002

```
(FILE.PSPICE. "C:\304_02\ORCAD\AMP TYPES\amplifier types-types-ac.sim"
    (DISPLAYNAME " TYPES-AC")
    (TYPE "PSPICE PROFILE")))
```

That is, FILE becomes FILE•PSPICE•[5] and a DISPLAYNAME line has to be added. After doing this for each of the missing simulation profiles, save the •OPJ file, and reopen the project using CAPTURE. (If you didn't close CAPTURE earlier, do it now, resave the • OPJ file in NOTEPAD, and reopen CAPTURE. That way, CAPTURE gets the modified •OPJ file.) The edited •OPJ file with one more profile added is shown below.

```
(FOLDER "PSPICE RESOURCES"
  (FOLDER "SIMULATION PROFILES"
   (FILE.PSPICE. "C:\304_02\ORCAD\AMP TYPES\amplifier types-types-trans.sim"
    (DISPLAYNAME " TYPES-TRANS")
    (TYPE "PSPICE PROFILE"))
   (FILE.PSPICE. ".\ TYPES-AC.SIM"
    (DISPLAYNAME "TYPES-AC")
    (TYPE "PSPICE PROFILE")))
```

The missing Simulation Profiles now will reappear.

A minor variation on this theme is that one of the profiles may appear further down the •OPJ listing under the heading ACTIVE PROFILE. Once found, it is changed in the same way, as shown below

```
(ACTIVEPROFILE ".\amplifier types-types-ac.sim")
    (FILE.PSPICE. ".\amplifier types-types-trans.sim"
    (DISPLAYNAME "TYPES-TRANS")
    (TYPE "PSPICE PROFILE")))
```

MISSING •olb FILE

This file often is missing from an e-mailed or archived project, and causes a warning menu to show up that not all the project files can be found. Then a second attempt usually opens the project. Once the project opens, assuming you have the *Parts.lib* library (*Parts* happens to be the project name in this example), you can make the *Parts.olb* library as follows.

Highlight a part in the *Parts* schematic. Then using EDIT/PSPICE MODEL, select the menu FILE/CAPTURE PARTS. The CREATE PARTS FOR LIBRARY menu appears, as shown in Figure 8. Fill in the location of the *Parts.lib* library using the BROWSE button. Then hit OK.

---

[5] Don't forget the final period in FILE•PSPICE• file.
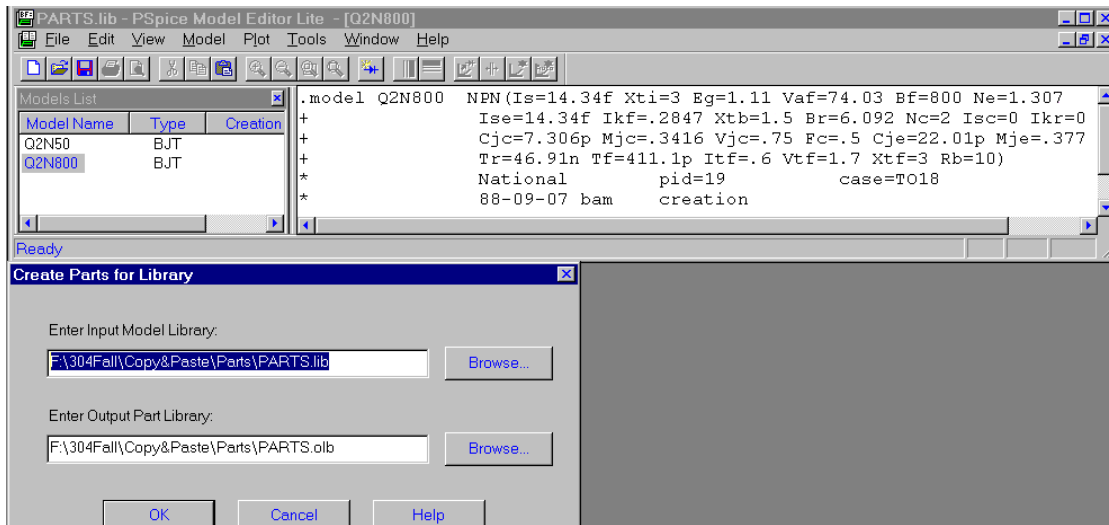
FIGURE 8
Creation of the *Parts•olb* model library using the PSPICE model editor with FILE/CAPTURE PARTS. The *Parts.lib* library is filled in using the BROWSE button.