

# Using OrCAD<sup>®</sup> for Schematic Capture from MMICAD

This application note outlines the procedures for capturing a schematic from **OrCAD** and converting it to a **MMICAD** circuit file. A practical example will be given describing the schematic capture of an amplifier. It is recommended that the user first install the ORCAD-MMICAD interface according to Application Note No. 16, which includes creating the MMICAD design project.

### THE PART LIST:

The MMICAD component library is available in the MMICAD.LIB library supplied for OrCAD. Each component carries two identifications: (1) the **Part Name** which carries the name of the component, and (2) the **Reference Designator** which refers to a component label that can be annotated and edited. For example, a resistor has a part name **RES**, and reference designator **Res?**. When the Reference designator is annotated, the ? is replaced by a number relating to the order in which the part is placed in the schematic. The user may edit the Reference Designator to something meaningful such as R=10.

### SCHEMATIC CAPTURE EXAMPLE

In this example, a one stage amplifier is to be drawn in OrCAD. The amplifier consists of a single Field Effect Transistor (FET), with resistive feedback from the drain to the gate. The feedback path includes transmission line and a decoupling capacitor. The input and output matching circuits consist of a series transmission line with a decoupling capacitor. Identical biasing networks consisting of an inductor and capacitor to ground will be added for biasing the gate and drain of the FET. The schematic of the amplifier is shown in Figure 1. The OrCAD schematic file MMICAD.SCH is located in the \MMICAD directory on your hard drive. This schematic was created using the following procedure:

Within MMICAD, invoke **Schematic Capture** in the **Links** Menu. MMICAD then executes OrCAD. In ORCAD's **EDA Tools Menu**, click and execute **Schematic Design Tools**, then click and execute **Draft**. At this point, MMICAD.SCH file is opened. Using the **GET** command components can be selected from a scrolling menu by double clicking the mouse. Several recommendations for schematic entry are presented:

(a) Proper schematic capture requires the first and last

elements to be in shunt configuration. If the starting or ending elements are series elements, then simply add shunt **OPEN** components at the starting or ending points.

- (b) Place 0.5 (5 grid points) length wire between the two terminal parts, and 1.0 (10 grid points) length wire between two and three or more terminal parts. This facilitates adding custom Reference Designators.
- (c) Place Junctions at wire to element, wire to wire, and element to element intersections. This eliminates the possibility of floating nodes.
- (d) Do not overlap two wires, two elements, or a wire and element. This also eliminates the possibility of floating nodes.

After the schematic entry is complete, choose **Quit**, **Update File**, and **Abandon Edits**. Then start annotation by clicking **Annotate Schematic**. Execute **DRAFT** again, the amplifier schematic is shown in Figure 1.

Before converting the schematic into a MMICAD circuit file, three optional text editing steps can be performed. (The text added or edited will also appear in the MMICAD circuit file.)

**Editing Step #1:** Choose **Edit**. Point into any component and click the mouse. Now you can edit the **Part Value**, and **Reference**. For both, the **Location** and **Visibility** can be changed, and only for the **Reference** the **Name** may be changed. It is possible to make the **Part Value** invisible, while adding component labels to the **Reference**.

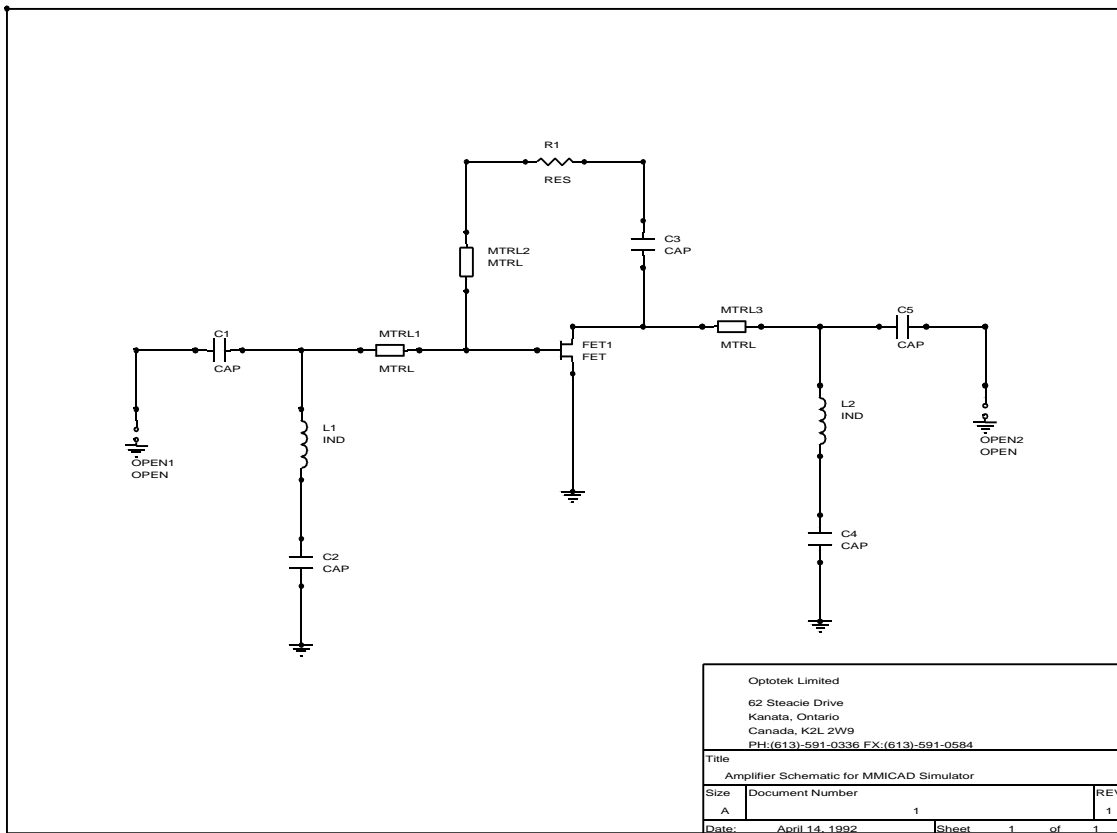
**Editing Step #2:** Choose **Edit**. Point into the Title block of the worksheet and click the mouse. A menu will appear from which the user may add a Title, Organization name and address, date, and revision number.

**Editing Step #3:** Choose **Place**, and select **TEXT**. The user may now add descriptive labels into the worksheet that will appear in the MMICAD circuit file. The procedure is to first add the

word MMICAD as text (in capital letters) preceded with | (the "pipe" symbol - above \ on the PC keyboard) symbol. The following text labels also have to be preceded with the broken vertical bar symbol and will appear after the CKT block in the MMICAD circuit file.

The result of the above editing into the schematic file of Figure 1 is shown in Figure 2. The capture of the schematic file proceeds by clicking **Create Netlist**. The user may now exit OrCAD, and MMICAD will automatically resume. To load the captured circuit file, change the current directory in MMICAD to \ORCAD\MMICAD and load the circuit file. This is shown in Figure 3.

OrCAD is a registered trademark of OrCAD Corporation.



**Figure 1** Amplifier Schematic



```

! *****
! ORCAD-MMICAD NETLIST CONVERTOR VERSION 1.0
! (c)Optotek Limited (1992)
!
! Amplifier Schematic for MMICAD Simulator
! Revised: April 14, 1992
! Revision: 1
! 1
! Optotek Limited
! 62 Steacie Drive
! Kanata, Ontario
! Canada, K2K 2A9
! PH:(613)-591-0336 FX:(613)-591-0584
! *****

! *****
! Edit With Defaults to obtain the Component Parameters
! *****

CKT
! STAGE #1 OF LNA
! INPUT CIRCUIT VARIABLES
! C=CIN
! W=WIN
! L=LIN
! L=LBIASIN
! C=CBIASIN
! FEEDBACK CIRCUIT VARIABLES
! R=RFB
! W=WFB
! L=LFB
! C=CFB
! OUTPUT CIRCUIT VARIABLES
! C=COUT
! W=WOUT
! L=LOUT
! L=LBIASOUT
! C=CBIASOUT
CAP 7 8 ! C=CIN
CAP 2 3 ! C=CFB
CAP 5 6 ! C=COUT
CAP 10 0 ! C=CBIASIN
CAP 9 0 ! C=CBIASOUT
FET 4 3 0 ! FET1
IND 8 10 ! L=LBIASIN
IND 5 9 ! L=LBIASOUT
MTRL 8 4 ! W=WIN,L=LIN
MTRL 1 4 ! W=WFB,L=LFB
MTRL 3 5 ! W=WOUT,L=LOUT
OPEN 6 ! OUT
OPEN 7 ! IN
RES 1 2 ! R=RFB
! Modify the DEF2P line below:
DEF2P 1 10 NETWORK

FREQ
SWEEP 1 10 1

OUT
NETWORK SPAR SPAR
NETWORK DB[S21] S21

LABEL
Amplifier Schematic for MMICAD Simulator

```

Figure 3 The Captured Circuit File

```

! *****
! ORCAD-MMICAD NETLIST CONVERTOR VERSION 1.0
! (c)Optotek Limited (1992)
!
! Amplifier Schematic for MMICAD Simulator
! Revised: April 14, 1992
! Revision: 1
! 1
! Optotek Limited
! 62 Steacie Drive
! Kanata, Ontario
! Canada, K2K 2A9
! PH:(613)-591-0336 FX:(613)-591-0584
! *****

GLOBAL

MSUB ER=12 H=625 T=2 RHO=1 TAND=0 @SUB0
SSUB ER=1 B=10 T=2 RHO=1 @SUB1

CKT
! STAGE #1 OF LNA
! INPUT CIRCUIT VARIABLES
! C=CIN
! W=WIN
! L=LIN
! L=LBIASIN
! C=CBIASIN
! FEEDBACK CIRCUIT VARIABLES
! R=RFB
! W=WFB
! L=LFB
! C=CFB
! OUTPUT CIRCUIT VARIABLES
! C=COUT
! W=WOUT
! L=LOUT
! L=LBIASOUT
! C=CBIASOUT
CAP 7 8 C=0 ! C=CIN
CAP 2 3 C=0 ! C=CFB
CAP 5 6 C=0 ! C=COUT
CAP 10 0 C=0 ! C=CBIASIN
CAP 9 0 C=0 ! C=CBIASOUT
FET 4 3 0 GM=25 CGS=0.3 RDS=500 CGD=0.01 CDS=0.1 RGS=5&
TAU=3 RG=1 RS=1 RD=1 LG=0.001 LS=0.001 LD=0.001 ! FET1
IND 8 10 L=0 ! L=LBIASIN
IND 5 9 L=0 ! L=LBIASOUT
MTRL 8 4 W=10 L=0 @SUB0 ! W=WIN,L=LIN
MTRL 1 4 W=10 L=0 @SUB0 ! W=WFB,L=LFB
MTRL 3 5 W=10 L=0 @SUB0 ! W=WOUT,L=LOUT
OPEN 6 ! OUT
OPEN 7 ! IN
RES 1 2 R=0 ! R=RFB
DEF2P 7 6 NETWORK

FREQ
SWEEP 1 10 1

OUT
NETWORK SPAR SPAR
NETWORK DB[S21] S21

LABEL
Amplifier Schematic for MMICAD Simulator

```

Figure 4 The Captured Circuit File After EDIT WITH DEFAULTS