DESIGN CENTER

```
******************************************************************
```
CONTENTS
```
******************************************************************
```

****

Part 1. New Features

****

DESIGN CENTER

* Mentor integration for HP and Sun systems
* Expanded libraries now with over 9000 devices, including over 1,700
  Japanese devices
* New PLogic digital-only circuit simulator
* PSpice availability for analog-only and mixed-signal circuit
  simulations

SCHEMATICS (Microsoft Windows and Sun OpenWindows platforms)

* Extended support for the SCICARDS external board layout package and
  EDIF 2 0 0 netlist format
* Stand-alone Schematics evaluation package
* Global and local configuration of symbol and package library files

* Support for the PLSyn Programmable Logic Synthesis package
* Improved pan and zoom functions
* Attribute editing on a global or multiple-part basis
* Integrated stimulus editing within the Schematics environment (Windows only)
* New symbols for external input, output, and bi-directional ports to identify peripheral circuit connections, and facilitate persistent hazard identification and tracking, which is a new feature of Probe (Windows only)
* Menu naming conventions consistent with Probe and PSpice

PSPICE/PLOGIC

* Simulation status display showing the three fastest transmission lines
* New .EXTERNAL command which provides a means of identifying and distinguishing those nets representing the outermost (or peripheral) connections to the circuit
* The .STMLIB command which makes stimulus library files available to PSpice and PLogic
* The .STIMULUS command which allows you to define the source devices (V, I, or STIM). The .STIMULUS definition encompasses only the Transient specification portion of what is allowed in the V, I, or STIM device syntax
* Improved noise immunity modeling for digital devices

CONTROL SHELL

* New Control Shell user interface for Extended DOS platforms
* Integrated access to PSpice, PLogic, Probe, Parts, and StmEd
* Improved printing functions
* Interactive analysis specification through dialogs and prompts

PROBE

* Append function for combining trace data from multiple data files
* Marching waveforms on the OpenWindows platform
* Improved pan and zoom functions
* Timing Violations and Error Tracking (Windows only)

STIMULUS EDITOR

* New native Windows version

PARTS

* New Voltage Reference Template
* Improvements to the MOSFET template

POLARIS

* Support for TangoPro board layout package

```
*****************************************************************

Part 2.  General

*****************************************************************

2.1.  File Name Changes
The file "pspice.iff" used in previous versions has been replaced by the
file "msim.iff" in this release.

The Windows PSpice executable file name has been changed from
"wpspice.exe" to "pspice.exe."

2.2.  Environment Variable Name Change
The environment variable PSPICELIB used in previous versions has been
replaced by the environment variable SIMLIBPATH in this release.

2.3.  Version Compatibility -- Schematics
Warning:  Schematic and/or Symbol Library files which are created and/or
modified using the 6.0 version of Schematics may not be backward
compatible with previous versions of Schematics.  That is, if you have a
5.3 schematic that you modify using the 6.0 version of Schematics, you
may not be able to read that schematic back into 5.3 or 5.4 Schematics.
Similarly, if you create new symbols using the 6.0 Symbol Editor, these
symbols may not be compatible with previous versions of Schematics.

2.4.  Cautions for PC and Windows Network License Users
If you try to invoke PSpice/PLogic, but a license is not available, the
simulator may delete or inadvertently overwrite any previous output
(.out) and data (.dat) files you may have had for that circuit.

Please be aware of this and copy any required output and/or data files
to other names before subsequently simulating.

2.5.  Command Line Option: -i
PSpice, Probe, the Stimulus Editor (Windows version) and Schematics
support a -i command line option which allows you to specify a
configuration file other than "msim.ini."

To use this option, you can change the command line associated with the
program's icon to be:

   <program name>  -i <new_file_name>.ini

If no path is specified for the new ".ini" file, it is assumed to be in
your Windows directory (on the PC) or in your home directory (on the
Sun).

From Schematics, an initialization file can be configured through the
Options/Editor Configuration/App Settings dialog.  The initialization
file configured here will only be used by programs invoked from within
```

Schematics. (The file specified by the Options/Editor Configuration/App
Settings dialog is written to the "inifile" specification within the
[Schematics] section of the "msim.ini" file.)

2.6. Using Extended DOS Versions Under Microsoft Windows
The Probe "Mid-Analysis Snoop" feature will not work if PSpice (PLogic)
is being run from within a DOS "Window" (under Microsoft Windows). The
symptom is an error message,

   "DOS/16M error: [32] DPMI host error (possibly insufficient memory)"

To avoid this problem, run PSpice (or PLogic) directly from the Control
Shell or the DOS command line, not under Windows.

****

Part 3. Installation Considerations

****

3.1. Extended DOS Versions
The environment variable PSPICELIB used in previous releases has been
replaced by the environment variable SIMLIBPATH in this release. This
variable can be set from the DOS command line, in your "autoexec.bat"
file, or from within the Control Shell using the Analysis/Analysis
Options dialog.

3.2. Network Versions
The following information applies to the use of NetHASP on Novell
networks consisting of several interconnected networks.

3.2.1. Definitions

Internetwork: a series of two or more networks that are linked via
internal or external routers.

Local network: a single cabling scheme identified by a unique network
number to which one or more stations are attached.

By default, when HASPSERV is loaded, it advertises its name to the
entire internetwork and serves all of its networks and stations.
However, you can instruct HASPSERV to serve only a specific local
network.

3.2.2. Specifying Local Networks with HASPSERV.NLM (i.e., on file
servers)

To instruct HASPSERV.NLM to serve a local network, use the following
command:

        LOAD HASPSERV LOCALNET

If the file server is a "bridge," then HASPSERV will serve only those
stations which are part of the networks connected directly to this

"bridge."

To specify the network numbers of those networks with which HASPSERV
should advertise its name:

        LOAD HASPSERV NET m [NET n ...]

where "m" and "n" are network numbers on the internetwork.

Note: Although HASPSERV advertises its name with specific networks, it
will serve all of the stations in the entire internetwork. To instruct
HASPSERV to only serve the specified networks, use the LOCALNET switch:

        HASPSERV NET 111 NET 222 LOCALNET

SET NETHASPPROTOCOL=LOCALNET ensures that the station will access only
those HASPSERVs which advertise their names with the station's network
number. The station will access a HASPSERV.NLM which was loaded with
the "NET n" switch, where "n" is the local network number of the
station.

3.2.3. Specifying Local Networks with HASPSERV.EXE (i.e., on local
network PCs)

To instruct HASPSERV.EXE to serve a local network:

        HASPSERV -LOCALNET

SET NETHASPPROTOCOL=LOCALNET ensures that the station will access only
the HASPSERV.EXE which advertises its name with that station's local
network.

****

Part 4. PSpice and PLogic

****

4.1. New PLogic Digital Simulator
PLogic is available for the Windows and OpenWindows schematic capture
environments, and for the Extended DOS with Control Shell environment.

PLogic is a digital-only circuit simulator that analyzes timing behavior
during a transient analysis. Detailed analysis is conducted subject to
the timing constraints (e.g., setup/hold times and min/max propagation
delay ambiguities) specified for individual devices.

PLogic operates identically to PSpice. In the Design Center environment
with Schematics, a schematic is prepared for simulation from which
Schematics generates a circuit file set. In circuit file entry
environments, the circuit file is created directly through the Control
Shell or a text editor. The circuit file set (or circuit file)
containing the circuit netlist and analysis commands is read by PLogic

for simulation. PLogic calculates the states of the nodes connected to digital devices. Any timing violations or hazards are reported by PLogic in the form of messages written to the simulation output file and to the Probe data file.

With the Design Center for Windows, the results are formulated into meaningful graphical traces in Probe which can be marked for display directly from your schematic. Probe's messaging feature (Windows only) allows you to automatically display digital signals and buses along with detailed message text associated with a given timing violation or hazard; trace segments with diagnostics are visually tagged.

PLogic analyses can be classified as standard and simple multi-run. For more information on running a particular analysis as well as specifying analysis parameters and output, refer to the Circuit Analysis User's Guide provided with your system.

4.2. Using FILE Specification in PWL Forms
When using the FILE keyword within a PWL (Piece-Wise Linear) specification, be aware that anything following the file name is ignored. For example, if you are using the FILE form with a REPEAT...ENDREPEAT loop, the ENDREPEAT must appear on a continuation line if it follows the file name.

The following examples illustrate correct and incorrect forms:

Correct:
     I3   5,6   PWL   REPEAT FOR 5 FILE DATA1.TAB
     + ENDREPEAT

Incorrect:
     I3   5,6   PWL   REPEAT FOR 5 FILE DATA1.TAB ENDREPEAT

Correct:

     I4   7 8   PWL   TIME_SCALE_FACTOR=0.1
      +   REPEAT FOREVER
      +   REPEAT FOR 5 (1,0) (2,1) (3,0) ENDREPEAT
      +   REPEAT FOR 5 FILE DATA1.TAB
      +   ENDREPEAT
      +   ENDREPEAT

Incorrect:

     I4   7 8   PWL   TIME_SCALE_FACTOR=0.1
      +   REPEAT FOREVER
      +   REPEAT FOR 5 (1,0) (2,1) (3,0) ENDREPEAT
      +   REPEAT FOR 5 FILE DATA1.TAB ENDREPEAT
      +   ENDREPEAT

NOTE: The REPEAT ... ENDREPEAT specifications may be nested to any depth.

4.3. Noise Immunity Behavior Modeling for Digital Devices
PSpice A/D and PLogic now model the noise immunity behavior in digital
devices. By setting the Pulse Width Rejection Threshold (PPWRT)
parameter in a digital device's I/O Model, the simulator will
appropriately generate a glitch when the pulse width is greater than or
equal to TPWRT and the pulse width is less than the propagation delay.

Refer to the Circuit Analysis User's Guide for additional information.

4.4. Digital Library Updates
The Digital component libraries, DIG_1.LIB, DIG_2.LIB, DIG_3.LIB,
DIG_4.LIB and DIG_5.LIB, have been modified to support the new "Embedded
Constraint" form of PINDLY device syntax (in those models that use both
PINDLY and CONSTRAINT devices). These changes do not affect the logic
operation of the models, but they support the new Probe "Error Tracking"
functionality (offered with Windows-based packages at this time).

The new syntax, as of release 6.0, represents the preferred modeling
style for devices of this type; the old syntax (i.e., "separate"
PINDLY/CONSTRAINT devices) will still work (logically), but will
generally not provide correct results with respect to "Error Tracking"
behavior.

4.5. Intel Pentium(TM) IBIS Models
A library of Intel's Pentium (TM) 82430 (Mercury) PCIset IBIS models is
now available. These highly accurate driver/receiver buffer models
include effects such as switching times, nonlinear transfer
characteristics, and package parasitics. These allow you to perform
accurate signal integrity simulation of I/O interconnects on printed
circuit boards. The models are located in the "buffer.lib" file.

****

Part 5. Probe

****

5.1. Probe Accelerator Keys (Windows and OpenWindows only)
The following accelerator keys have been added to Probe:

     <CTRL X>   EDIT/CUT
     <CTRL C>   EDIT/COPY
     <CTRL V>   EDIT/PASTE

5.2. Marching Waveforms (Windows and OpenWindows)
Marching Waveforms was supported for Microsoft Windows in release 5.4,
and remains unchanged for 6.0. Support for OpenWindows is now included
in 6.0 as well.

5.3. Timing Violations and Error Tracking (Windows only)
When there are problems with your design, such as setup/hold violations,
pulse-width violations, or worst-case timing hazards, PSpice and PLogic
will log messages to the simulation output file and/or Probe data file.
Using Probe (Microsoft Windows only), messages can be selected causing

associated waveforms and detailed message text to be automatically
displayed.

In the digital domain, the types of problems identified by the simulator
are generally classified as either timing violations or timing hazards.
Timing violations include SETUP, HOLD, and minimum pulse WIDTH
violations of component specifications. The occurrence of such a
violation may result in a change in the state behavior of the design,
and potentially the answer. However, the effects of many of these
errors are short-lived and don't influence the final circuit results.

PSpice and PLogic now identify any digital net that may carry a possibly
invalid state value due to a timing hazard; Probe identifies digital
traces of these nets by highlighting them with a different color. The
color difference exists for the duration of the potential logic
difference due to the timing hazard. Changes in a trace's color may or
may not coincide with actual state changes, depending on the nature of
the timing violation. In Probe, when a digital trace is highlighted in
this manner, double-click on the actual highlighted portion of the trace
to see a Simulation Message pop-up dialog; clicking on "OK" will then
bring up a new window containing both the message text and a plot of the
signals involved in the hazard.

Timing hazards are most easily identified by simulating a design in
worst-case timing mode, usually close to its critical timing limits.
Under such conditions, the simulator will report things such as
AMBIGUITY CONVERGENCE hazards. Again, these may or may not pose a
problem to the operation of the design. However, there are identifiable
cases that cause major problems.

The types of error conditions reported by the simulator are summarized
in the "Simulation Condition Messages" section of the "PSpice Analyses"
chapter in the Circuit Analysis User's Guide.

****

Part 6. Schematics

****

6.1. User Interface Changes (Windows and OpenWindows)

6.1.1. Menu Bar Changes
To maintain consistency with the other Design Center programs, some
minor changes have been made to the commands in the Menu Bar:

```
      Old Command              New Command
      -------------------------------------------
      Zoom                     View
      Zoom/Normal              View/Fit(*)
      Configure                Options
      Edit/Search              Edit/Find
```

Analysis/Run PSpice   Analysis/Simulate(**)

(*)The View/Fit command replaces the Zoom/Normal command, although the
functionality is somewhat different. Refer to the Schematic Capture
User's Guide for details.

(**)The simulator (PSpice or PLogic) is selected through the
Options/Editor Configuration/App Settings dialog.

Opening a schematic (either via File/Open or Navigate/Push) now does an
automatic View/Fit to adjust the zoom factor so that all wires, parts,
etc. on the schematic are visible.

The View/Area command now deselects the viewed area after zooming.

6.1.2. Setup Block and View Functionality
The Edit/Setup Block command was removed and functionality combined with
the Edit/Views command.

Thus, for 6.0, to change the schematic associated with a block, the
Edit/Views command should be used. The value of the DEFAULT view is the
default schematic name. The command can also be used to add, delete, or
display all views and schematics associated with a block.

If a hierarchical symbol is chosen (as opposed to a block), Edit/Views
will display the list of views but not allow editing. Views for
hierarchical symbols can only be changed in the Symbol Editor.

6.1.3. Edit/Model Changes
When editing a model definition, <ENTER> should now be used instead of
<CTRL-ENTER> to go to the next line.

6.2. PLogic Digital Simulator Integration
The Analysis/Setup command will bring up a different dialog depending on
whether PSpice or PLogic is configured as the simulator. For PSpice
(analog and mixed A/D), the Analysis Setup dialog provides a new Digital
Setup button. For PLogic, the Analysis/Setup command displays only the
digital analysis options dialog.

6.3. The VIEWPOINT Symbol
If you are using PLogic, the VIEWPOINT symbol will not show any
information.

6.4. Creating Symbols/Netlisting--Illegal Characters
Do not use the "%" character in names that will appear in the template
for PSpice. Also do not use "%" in the value of an attribute which must
be substituted in the template by the netlister (e.g., DEVTOL=@DEVTOL
where DEVTOL=10%). This will cause the netlister to emit erroneous "No
such pin" errors.

6.5. App Settings Dialog for Application Settings
The Options/Editor Configuration dialog has been changed to include
another button, labeled "App Settings...". This button displays a
dialog which allows the user to specify the simulator (PSpice or PLogic)

and the command lines used to invoke the simulator, Probe, and StmEd.
(The SUN version of Schematics will not have the StmEd command line
field).

6.6. Windows Stimulus Editor Integration (Windows only)
The Analysis/Library and Includes dialog now includes Stimulus Library
files.

Using the symbols VSTIM, ISTIM, and DIGSTIM (from the "source.slb"
symbol library) on your schematic allows you to define voltage, current
and digital stimulus using the new Windows Stimulus Editor. Refer to
the Schematic Capture User's Guide, and the Circuit Analysis User's
Guide for complete details.

If you edit a stimulus using the Stimulus Editor, be sure that any
changes are saved (using the File/Save command in the Stimulus Editor)
prior to doing your simulation if you want these changes included in the
simulation.

6.7. Reconfiguring Symbol and Package Libraries
Prior to 6.0, if the list of libraries was changed via Options/Editor
Configuration, then all libraries in the list would be re-read. For 6.0
this has been sped up by recognizing which libraries were previously
loaded and only reading new libraries. This change means that you
should never replace symbol and package libraries manually (using the
DOS copy command, for example) while Schematics is running.

6.8. Symbols
Local symbol library support has been added. You can now associate
symbol libraries with a particular schematic. These symbol libraries
will be loaded when that particular schematic is loaded. This is done
by selecting the Options/Editor Configuration/Library settings and
selecting the new Add Local button. Local libraries are always searched
prior to global libraries (those loaded for all schematics). For a
hierarchical schematic, only the local libraries associated with the
topmost schematic are used.

6.8.1. New "misc.slb" Symbol - "555alt"
The 555alt symbol has been added to "misc.slb." This symbol is intended
to provide a more functional pin configuration than the other 555
symbols (i.e. 555B, 555C, and 555D). The 555alt symbol references the
555C model, which is currently available for use with the Mixed A/D and
Analog-only products from MicroSim.

6.8.2. Breakout Symbol Name Change - "DBreak3"
The name of D3Break symbol has been changed to DBreak3.

6.8.3. Analog Devices with Visible Pin Numbers
A symbol library file, "analog_p.slb," is shipped for those who wish to
have visible pin numbers on the analog devices R, L, C, R_VAR, and
C_VAR. The "analog_p.slb" file must be configured by adding it to the
list of configured symbol library files before or in place of the file
"analog.slb."

6.8.4. The "dig_prim.slb" Symbol Library File
The symbol library file "dig_prim.slb" is provided primarily for users
of PLSyn Programmable Logic Synthesis. The IMPL attribute for each of
these symbols is set to PLSYN, and the timing delay is set to zero. If
you are not using PLSyn but want to use these symbols, Use the Edit
Attributes command to remove the PLSYN assignment from the IMPL
attribute. (In other words, set the IMPL= value to a blank.)

6.8.5. Obsolete Symbols
The "dig_old.slb" symbol library file is included for compatibility with
earlier releases. The symbols in this file are considered obsolete and
have been replaced by new symbols in other library files. These symbols
are provided for compatibility, only, and should not be used for new
schematics.

6.9. Hierarchical Symbols and Blocks
Hierarchical symbols and blocks now use a new default reference
designator (REFDES) to avoid conflicts with packaging.

The default REFDES for hierarchical blocks is now HB?.

The default REFDES for hierarchical symbols is now HS?.

Pin numbers are now P1, P2, etc. instead of 1, 2, etc.

6.10. Pin Name Translation Changes
Within pin names, the characters "<," ">," and "=" are now replaced by
"GT," LT," and "EQ," respectively. (In 5.4 and prior, these characters
were replaced by "G," "L," and "E" respectively.)

6.11. The "msim.ini" File

Note: You cannot use tabs within the "msim.ini" file.

6.11.1. New [Schematics Colors] Entries

```
    ENTRY      DESCRIPTION
    -----------------------------------------------------------------

    plsyn      specifies the color of PLSyn parts and blocks
               (IMPL=PLSYN or blocks with a DSL view)

    stimulus   specifies the color of StmEd stimulus parts
               (those with a STIMULUS attribute)
```

6.11.2. New [MicroSim] Entries

```
    ENTRY        DESCRIPTION                       DEFAULT
    -----------------------------------------------------------------

    plsyncmd     Command to invoke PLSyn.          plsyn

    stmedcmd     Command to invoke StmEd.          stmed
```

```
     polariscmd  Command to invoke Polaris.        polaris.exe /FT /RC

     plogiccmd   Command to invoke PLogic.         plogic

     backup      Previously in [Schematics] section
```

6.11.3. New [Schematics] Entries

```
     ENTRY               DESCRIPTION                           DEFAULT
     ----------------------------------------------------------------

     viewfitpercent      See description of View Fit below

     lastfile1-4         Most-recently used Schematic files

     lastslb1-4          Most-recently used Symbol Editor
                         files

     plsyn               ON adds Tools/PLSyn command to           OFF
                         menu, and shows PLSyn Setup button
                         on the Analysis Setup dialog.


     polaris             ON adds Tools/Polaris to menu.           OFF

     simulator           PSPICE or PLOGIC.  Set using Options/
                         Editor Configuration/App Settings...

     inifile             Name of initialization file to use when invoking
                         other Design Center Apps (PSpice, PLogic, StmEd,
                         Probe).  If this is left blank, the default
                         "msim.ini" file is used.  Can be changed through
                         the Options/Editor Configuration/App Settings.

                         A file specified through one of the program-
                         specific commands in the [MicroSim] section
                         (such as pspicecmd, probecmd, stmedcmd, etc.)
                         will take precedence over the file specified
                         with the inifile command setting.

     usescicardsp2       Flag to govern use of P2 record when      ON
                         netlisting in SCICARDS format.

     pcbhierpathpsep     Path separator character for PCB         " - "
                         netlists;  the hierarchical path is added
                         to named nets of subschematics to
                         identify them uniquely.  The character
                         "." had been used previously.
```

6.11.4. "Marker.slb"
"Marker.slb" should not be included in the list of libraries. If it is,
you will get a warning when you start Schematics:

"marker libraries should not be in configured list."

This is just a warning; you can continue to use Schematics.

6.12. Using European Parts

There are symbol and simulation model libraries available for European
manufacturers' components. If you want to use these parts, you need to
add "europe.lib" to the list of globally available libraries AHEAD of
"nom.lib" using the Analysis/Library and Include Files menu command.
You also need to reconfigure the symbol and package libraries using
Options/Editor Configuration/Library Settings to add "ebipolar,"
"ediode," and "epwrbjt" AHEAD of the other libraries. Remember to check
the "plb" setting as well as "slb."

6.13. Font-Related Issues

6.13.1. Using the RASTERFONT=ON/OFF Switch

If the line RASTERFONT=ON is included in your "msim.ini" file, it causes
the Helvetica font to be used instead of the modern font for display in
Schematics. This will cause rotated text to appear vertically versus
being rotated on its side. If your symbols and schematics use rotated
text, setting this item to ON may cause problems with positioning text.

For this reason, we suggest using the RASTERFONT=OFF switch setting.

6.13.2. Using Adobe Type Manager with Schematics
If you have Adobe Type Manager installed and are using Schematics, you
may encounter problems with how rotated text is displayed. We recommend
that you edit the "ATM.ini" file and comment out the line

     Modern=Helvetica

in the [ALIASES] section. If this is not done, rotated text may be
positioned differently on the display versus on hard copies. We also do
not recommend using the FONTTYPE=ADOBE setting in your "msim.ini" file
at this time as it has the same problem.

****

Part 7. Control Shell for Extended DOS Versions

****
7.1. New Features
The 6.0 Control Shell provides the following features:

* A menu-driven environment for setting up your netlists, circuit
   files, and the analysis parameters

* 32-bit implementation

* Integrated Help, with entries for the commands supported

by PSpice and PLogic

*   Command entry through function keys, pull-down menus, or buttons
    for the most frequently used functions

*   Availability of multiple editing and browse windows

*   Context-sensitive help for menus and dialogs

*   Syntax checker for circuit files

*   Printing functions, including multiple-copy output and page
    customization options

*   File size is no longer limited to 32KB (All of the available free
    RAM and disk space may be used.)

The above features are described in detail in the Circuit Analysis
User's Guide and the Circuit Analysis Reference Manual.

7.2. Features Not Supported in the 6.0 Control Shell

1. The External Editor command (e.g. for calling OrCAD's SDT) is not
currently supported. Use the File/DOS Shell menu command to "call"
other DOS-based programs.

2. The use of the ";" character for user comments on a command line is
not supported. Comments must appear on a separate line which begins
with the "*" character.

3. Command pairs such as .ac/.noise and .tran/.four will appear
together when set up or altered using the Control Shell Analysis Setup
dialogs. These command pairs cannot be separated by comment lines.

4. As with previous versions of the Control Shell, you cannot have more
than one circuit description in a circuit file.

5. Global parameters (.param) cannot be edited through dialogs. You
must manually define .param parameters in the edit window.

7.3. Known Anomalies
The following problems and/or restrictions are inherent in the software
tools used to create the Control Shell, and are currently beyond our
control.

1. Positioning of the text within the edit window is spurious when some
of the dialogs are invoked, used, or closed.

2. Cursor positioning on a line in the edit window drifts to the right
when the mouse is used. The farther to the right on the line that the
mouse pointer is clicked, the more the cursor position drifts.

3. If a line in the circuit file contains a space character in the
first column, spurious "garbage" characters may be inserted when new

characters are entered in column one. These characters may be manually
deleted by the user.

4. During a Replace operation, the target text is not highlighted.
Instead, a solid block character (similar to a cursor) is placed in
front of the selected text.

5. Various anomalies have been observed when using the Control Shell in
a DOS window under Microsoft Windows. For best performance under
Windows, use the ".pif" file provided with the Control Shell. If you
encounter problems, try running the Control Shell directly from DOS.

****

Part 8. Stimulus Editor

****
8.1. Stimulus Editor for Windows
The Stimulus Editor (StmEd) program has been rewritten as a native
Windows-based program for the Microsoft Windows platform. The 6.0
version is tightly integrated with Schematics through new voltage,
current, and digital stimulus symbols.

The following sections provide information on digital stimulus editing
which is not included in the Circuit Analysis User's Guide:

8.1.1. Creating a STIM
When creating a STIM, a dialog appears which allows you to define
the following parameters of the STIM:

1. Bus Width - the number of individual signals which comprise the bus.
STIMs that are one bit wide are refered to as "signals" while those that
are more than one bit wide are refered to as "buses."

2. Time Step - allows the STIM to be rescaled in time by changing a
single value. This is analogous to the TIME_SCALE_FACTOR in PWLs. All
transtions added to the STIM when a Time Step has been defined will
automatically be recorded internally in terms of the Time Step.

3. The format for the bus - "Binary," "Octal," "Hex," and "Other."
This defines how the values for the bus appear in the plot and any
applicable dialogs. For "Binary," "Octal," and "Hex," the user selects
the Bus Width and StmEd determines the proper Format Array. (See
Section 5.1.2.1 of the Circuit Analysis Reference Manual for more
information on Format Arrays.)

When "Other" is selected, the user specifies the Format Array and StmEd
determines the Bus Width. Additionally, selecting "Other" causes the
bus to display in decimal.

8.1.2. Adding a STIM
After a STIM has been created (or whenever it is being edited) the Add
option on the Edit menu is enabled and given a submenu. The selections
on the submenu vary depending on whether the STIM is a signal or a bus.

The following selections are available for a signal:

1. To 0/1 - add a single transition to either zero or one. If the value is 0 or it is X or Z and the following value is 0, then the new transition is to one. Otherwise the new transition is to 0. Repeatedly clicking while moving the mouse along the X-axis will create a digital pulse.

2. To 0R1F - adds a single transition, cycling through the states 0, R, 1 and F. The value for the new transition is determined from the values of the states on either side. Repeatedly clicking while moving the mouse along the X-axis will create a digital pulse with rise/fall times.

3. To X - inserts a transition setting the STIM to the unknown state (X).

4. To Z - inserts a transition setting the STIM to the high impedance state (Z).

The following selections are available for a bus:

1. To Value - add a transition to a specified value. When this option is selected, a dialog pops up so the user may set the value of the transition that will be added. Additional transitions will have the same value until the user right-clicks the mouse. The initial default is zero and StmEd remembers the last value used in a given session of editing a STIM.

2. Incr by - add an INCR BY x command. When this option is selected, a dialog pops up so the user may set the amount of the increment.

Additional transitions will use the same amount until the user right-clicks the mouse. The initial default is one and StmEd remembers the last value used for INCR/DECR in a given session of editing a STIM.

3. Decr by - add an DECR BY x command. When this option is selected, a dialog pops up so the user may set the amount of the decrement.

Additional transitions will use the same amount until the user right-clicks the mouse. The initial default is one and StmEd remembers the last value used for INCR/DECR in a given session of editing a STIM.

4. To X - inserts a transition setting the bus to the unknown state (X).

5. To Z - inserts a transition setting the bus to the high impedance state (Z).

Pressing Alt-A repeats the most recent Add command for a STIM. For example, to make an eight bit bus that varies between the two values 00 and FF, add all the transitions to 00 and then right-click. Next, type Alt-A and set the new value to FF and add the other transitions.

If Alt-A is pressed before any Add command is selected from the menu,
after going into edit mode on a given STIM, it will act as a "To 0/1"
for a signal, and as a "To Value" for a bus.

8.2.   New Stimulus Editor Circuit File Commands

8.2.1. Stimulus Library Files (.STIMLIB)
General Form: .STMLIB <file name[.stl]>

Examples
     .STMLIB mylib.stl
     .STMLIB volts.stl
     .STMLIB dgpulse

Stimulus library files created by StmEd are made available to PSpice and
PLogic via the .STMLIB command. The <file name> specification
identifies a file containing .STIMULUS statements. The file extension
is not defaulted to ".stl." If you specify a file name, you must
include its extension.

8.2.2. Stimulus Definition(.STIMULUS)

General Form:
     .STIMULUS <stimulus name> <type> <type-specific parameters>*

Examples:
     .STIMULUS InputPulse PULSE (-1mv 1mv 2ns 2ns 50ns 100ns
     .STIMULUS DigitalPulse STIM (1,1)
     + 0S   1
     + 10NS   0
     + 20NS   1
     .STIMULUS 50KHZSIN SIN (0 5 50KHX 0 0 0)

.STIMULUS statements generally appear within stimulus libraries created
by StmEd. These define <stimulus name>s which are referred to by the
source devices (V or I), or by the digital STIM device. The .STIMULUS
definition encompasses only the Transient specification portion of what
is allowed in the V, I, or STIM device syntax.

****

Part 9. Polaris Signal Integrity

****

Polaris now supports Version 2.0 of the TangoPro PCB layout package with
the ASCII Database Translation option.

****

Part 10. Intel IBIS Model Support

```
****************************************************************
```

A library of Intel's Pentium (TM) 82430 (Mercury) PCIset IBIS models is
now available. These highly accurate driver/receiver buffer models
include effects such as switching times, nonlinear transfer
characteristics, and package parasitics. These allow you to perform
accurate signal integrity simulation of I/O interconnects on printed
circuit boards. The models are located in "buffer.lib."

```
****************************************************************
```

Part 11. New Libraries and Devices

```
****************************************************************
```

This section lists the new libraries and devices added to the 6.0
release which are not documented in the documentation set.

11.1. New Transmission Line Models
Four new twisted pair transmission line subcircuits have been added to
"tline.lib." The names are as follows:

TP19AWG   TP22AWG   TP24AWG   and TP26AWG

These models use a .SUBCKT named ABMlossyTline which implements a
general lossy transmission line with R, L, C, and G, as functions of
frequency.

The symbol library "tline.slb" has also been updated to reflect these
changes. A base part, TWSTPAIR, has also been added. The coax symbols
now automatically choose between the .MODEL and .SUBCKT implementations,
depending on whether the user specified an operating frequency with the
FRQ parameter.

11.2. New Comlinear Model
One new model from Comlinear Corporation has been added to the
"comlinr.lib" library since publication of the manuals. It is a CLC532
2:1 analog multiplexer.

11.3. Library Name Changes - "nom.lib"
The file "nom.lib" now references different files depending on the
simulator that was installed. If, for example, you are using the PLogic
digital simulator, the file "nom.lib" will reference the files in the
Digital Library. If you have the PSpice mixed-signal simulator, the
file "nom.lib" will reference files in both the Analog and the Digital
library. In the latter case, this is done through a reference to the
file "nom_dig.lib" which in turn references the Digital Library files.

```
****************************************************************
```

Part 12. Parts Program

****

12.1. New Voltage Reference Template
A new voltage reference template has been added for 6.0, with 5 screens.
A subckt-based model will be generated. It is added as menu item 9 in
the Parts program main menu. The voltage reference template is
described in the Circuit Analysis Reference Manual.

12.2. Miscellaneous Changes
The power MOSFET template in the Rev. Drain Current screen has been
changed and the extraction algorithm was modified to yield a better fit.

Minor changes to the BJT template for some model parameter ranges were
made.

12.3. Mouse Support
The use of the mouse is not supported for commands in Device_curve
submenus (such as Exit, Add, Change, or Delete).

12.4. Preservation of the MDT file in case of error
Under the following error conditions, the original MDT file will not be
over-written (as would happen with release 5.4 and previous releases):

i)   A corrupted MDT file is read - the MDT file is corrupted.

ii)   A non-compatible MDT file for BJT and MOSFET is read - the
      existing MDT file was created using Parts 5.3 or before.

iii) A wrong device type MDT file is read - the existing MDT file was
      created with a different device type (e.g., user trying to run
      Parts for a diode but using a BJT MDT file).

An error message is displayed and Parts will return to the Main menu.

****

Part 13. MODTOMDT Utility

****

MODTOMDT is a utility program to convert PSpice model files or library
files containing PSpice models to MDT files in Parts format. The MDT
files generated will be compatible with other MDT files generated by
Parts.

Currently, the following device types are supported:

    DIODE   JFET
    BJT   MOSFET (LEVEL 3)

The following lists features and usage of this utility:

1. Allow users to load a model file into Parts and see the performance
   curves of the model.

2. Allow users to perform a "what-if" analysis when they change the
   model parameters in Parts.

3. Allow our users to modify/update the model by entering screen data
   into the Parts program and re-fit some model parameters

Usage: modtomdt [options] <infile>

An input file must be provided. This could be a single model file, or
it could be a .LIB file that contains many .MODEL statements.

Command line options:
    -h   provide a summary of usage
    -n   force all answers to program queries to 'No'
    -y   force all answers to program queries to 'Yes'

Each .MODEL statement in a file will result in an individual MDT file.
If the output MDT file already exists, the user is prompted to confirm
overwriting the existing file.

An example:

A user wants to use a device, "A," for PSpice simulation, but cannot
find the model in the library. However, a similar device, "B," is in
the library.

Here is the procedure for creating the A.MDT file using Parts and the B
device:

1. Copy the model for device B from the appropriate library file to a
   file <infile>. (The model begins with a .MODEL statement.)

2. Run MODTOMDT <infile> to create B.MDT.

3. Rename B.MDT to A.MDT.

4. Run Parts, and load A.MDT.

5. Modify/Add data using the Parts program, then generate the file
   A.MOD.

****

Part 14. Filter Designer

****

Filter Designer is no longer supported on the Macintosh Platform.