cadence®

OrCAD X Capture Messages Reference Guide

Product Version 23.1
September 2023

Document Last Updated: October 2022

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

1

[ALGnnnn] Messages

The [ALGnnnn] messages are errors and warnings generated by the Capture netlister during netlisting and back annotation. Each of these messages may be caused by more than one type of problem. In general, most error and warning information may be found in the session log. You can also click in the text of the session log message and press F1 to view more details.

Error

A design or environment condition encountered that prevents the netlisting from proceeding. An error is always fatal and the netlisting process is aborted.

Warning

An undesirable design or environment condition encountered which you should be aware but one that allows netlisting to continue.

Warning[ALG0051] Pin <Pin Name> is renamed to <Pin Name2>after substituting illegal characters on < Part Reference>: <Schematic> , <Page> (<LocationX> , <LocationY)>

This message is flagged when the netlister detects:

- duplicate pin names per package.
  or
- a pin name with an illegal character, such as ' (a single quotation mark).

In addition to flagging the message, Capture also renames the pin.

You can use Library Correction utility to rename the pin names. This utility scans through all the parts in the library and identifies those components that have duplicate logical pin names. However, if a component has a duplicate Power pin, then that pin is not considered as duplicate by the Library Correction utility.

2

[DRCnnnn] Messages

The [DRCnnnn] messages are Design Rules Check errors and warnings. Check to see if you are following the procedures correctly to perform a specific action. You should also check to see if all the design rule check options you want specified are selected. Verify that buses connect to bussed pins, and wires connect to scalar pins. If you are creating a design to be translated back to SDT, check that you have followed the rules for creating an SDT compatible design.

Note: It is always recommended to run the Design Rule Check (DRC) before creating a netlist.

WARNING [DRC0041] Multiple Hierarchical Ports of same name exist within Hierarchical Block.

If a hierarchical block contains multiple hierarchical port of the same name, you may experience board level problems. It is advisable to maintain unique port names across a block.

3

[FMTnnnn] Messages

The [FMTnnnn] messages are Create Netlist format errors and warnings. Each of these messages may be caused by more than one type of problem, but end up with the same result. In general, you should check to make sure that all files you work with are not read-only or corrupted. For parts, pins, nets, and properties, check that you are using valid character names. If Capture is unable to create new objects or documents, or if Capture is unable to allocate memory for a task, check to see if you have sufficient system resources for Capture. You may need to close down applications that you are not using.

[FMT0002] Internal error -- unrecognized type in .INS file %02X, pos = %06lX

This is an internal error message. It appears when some unrecognized characters is encountered in the Netlist creation file (.INS).

[FMT0006] Internal error - Expecting ""("" in .INS file, ref-des %s

This is an internal error while generating the netlist creation file (.INS). There might be a missing closing parenthesis " (" in the .INS file.

[FMT0007] Internal error -- Expecting "")"" in .INS file, ref-des %s

This is an internal error while generating the netlist creation file (.INS). There might be a missing closing parenthesis " (" in the .INS file.

[FMT0010] Out of memory allocating Nets table

This error message appears if Capture is unable to allocate memory for a task. Check to see if you have sufficient system resources for Capture. You may need to close down applications that you are not using to solve this error.

[FMT0011] Out of memory allocating Parts table

This error message appears if Capture is unable to allocate memory for a task. Check to see if you have sufficient system resources for Capture. You may need to close down applications that you are not using to solve this error.

[FMT0012] Can't open first output file

This error message appears when Capture is unable to create or open the netlist file (.NET) specified in the Netlist File 1 text box.

To resolve this error make sure that the netlist file (.NET) path is correct in the Netlist File 1 text box and you have checked the View Output check box.

[FMT0013] Can't open second output file

This error message appears when Capture is unable to create or open the component/part list file (.CMP) specified in the Netlist File 2 text box.

To resolve this error make sure that the component/part list file (.CMP) path is correct in the Netlist File 2 text box and you have checked the View Output check box.

[FMT0014] Two output file names required

This error message appears if you don't specify the second filename in the Netlist File 2 text box in the Other tab. In addition to the netlist file, Capture also creates a part list file when you select the Vectron netlist format or component list file when you select the Calay90 netlist format from the Formatters list.

To resolve this error, specify an appropriate file in the Netlist File 2 text box.

[FMT0015] Can't create temporary file

This error message appears when Capture is unable to create a temporary file for the netlist.

[FMT0016] Internal error - can't open .INS file

This error message appears when Capture is unable to open the Netlist creation file (.INS). This may happen if the .INS file is corrupted.

[FMT0017] Internal error -- can't open .RES file

This error message appears when Capture is unable to open the Netlist creation file (.RES). This may happen if the .RES file is corrupted.

[FMT0018] Error loading .INS file

There is a set of characters regarded as illegal by all of the netlist formatters available on the Other tab of the Create Netlist dialog box. The use of these illegal characters in a design is the most common cause of the FMT0018 error. Additionally, each formatter has a particular set of characters considered illegal only by that specific formatter. For information on these format-specific sets of characters, refer to the Capture online help; each netlist format has a respective topic.

The following are examples of characters accepted by none of Capture's netlist formatters (quotation marks delimit text strings). Using these characters in a design causes the FMT0018 error.

1. Spaces in reference designator prefixes (for example " U", "U " or "J P")
2. Spaces in pin numbers (for example, "1 1" or "B 5")
3. Spaces in pin names where no pin number exists (for example "RE SET")
4. Spaces in hierarchical pin names on a hierarchical block that is not implemented (in this case, the hierarchical block is treated as a primitive part)

The FMT0018 error message is too ambiguous to be of any help in identifying and correcting errors. The NETLIST.DLL file located in your Capture installation directory detects netlisting errors. With the NETLIST.DLL file in place, the following error messages appear when you try to create a netlist with a formatter on the Other tab (given the corresponding conditions):

[NET0075] Reference designator contains spaces %s

Error [NET0075] indicates that your design contains spaces in reference designator prefixes. To solve this problem, first locate the part with the faulty reference designator in your design. Then double-click on the part and remove the leading space.

[NET0076] Pin Number 4 4 blank or contains spaces for part %s

Error [NET0076] indicates the presence of spaces in some or all of a design's pin numbers. This error is also caused by spaces in a pin name when there is no corresponding pin number. To remove this error, find the questionable part or parts in your design and remove the space or spaces by editing the part. You may also edit the part in the part library and then update the cache to bring the edited part into the design.

This error also occurs when spaces exist in pin names on a hierarchical block that is not implemented. To solve this problem, check the implementation path or implementation name of the specified hierarchical block. Make sure that you can descend through the block. If you can descend

through it, then spaces in the hierarchical pin names are inconsequential.

In rare cases, a corrupted net will cause the FMT0018 error as well. To find out if this is the case, choose Edit - Browse - Nets after highlighting the schematic page in the project manager. If a net on that page is corrupt, Capture does not display a window with a list of nets, but rather a message stating: Object not found. Use this process to isolate the page which contains the corrupt net (s). Then copy and paste portions of that page to another page, using the above test after each portion is copied. After you find the offending net, delete and then redraw it.

In addition, however, this error can occur if any part property value in the design includes a carriage return. If this is the case, you can isolate the problem as follows:

1. Select the design file in the project manager.
2. Choose the Browse Parts command.
3. In the resulting list, select all parts.
4. Choose the Edit Properties command. This causes the Edit Properties spreadsheet to appear.
5. Scan the property values for a carriage return, which appears as two vertical lines (||).
6. Remove any such characters from the property values.

Often, you can find carriage returns in property values that are long text strings.

After you remove the carriage returns, you can create your netlist normally.

[FMT0019] Warnings processing .RES file, netlist may not be usable

There were some warnings flagged while generating the netlist creation file (.RES). You may not be able to use the netlist. To solve this problem, check the .RES file and fix the warnings to proceed.

[FMT0020] Errors processing .RES file, netlist may not be usable

There were some errors while generating the netlist creation file (.RES). Capture was unable to generate the netlist creation file (.RES).

[FMT0021] No parts in the design

The Create Netlist tool could not find any parts in the design. Complete the design by including appropriate parts and create the netlist again.

[FMT0022] No nets in the design

This error occurs when the Create Netlist tool does not find any nets in the design. Complete the design by including appropriate wires, buses, parts, and symbols wherever required and create the netlist again.

[FMT0023]

This error message appears when there is a bus pin on a part or the same part number has different pin configurations.

[FMT0024] Ref-des not found. Possible Logical/Physical annotation conflict.

This error message indicates that there are illegal characters, such as dashes, in a reference designator prefix. To remove this error choose Edit - Browse - Parts after highlighting the design name in the project manager. Scan the list of reference designators for possible illegal characters. Double-click on the reference designator that contains such a character; this highlights the related part in your design window. Double-click on the highlighted part and remove the dash from the reference designator prefix in the Edit Part.

4

ORCAP Messages

ERROR[ORCAP-1130] - "Error(s) encountered while attempting to export properties. See the Session Log for more information."

This message occurs when Express encounters errors while attempting to export properties. Check the session log for details.

Legacy Message ID: EXP0003

ERROR[ORCAP-1184] - "No Pin-Name/Pin-Number headers found. Generate Part Failed."

During part generation, if the Pin-Name / Pin-Number headers are missing for APD BGA/Die-Text files then this warning appears. To successfully generate parts from APD BGA/Die-Text files, the pin-Name and pin-number headers need to be defined.

Legacy Message ID: GSM0016

ERROR[ORCAP-1255] - "Unable to create VHDL file '%s', file error (%d)"

This error message appears when you click the Remove Library button on the Place Part dialog box with a library selected in the Libraries list box.

You cannot remove Design Cache from your project.

Legacy Message ID: CAP0048

ERROR[ORCAP-1358] - "Part Occurrence for %s Not Found; Please check if there is any recursive reference."

This error message comes up when during incremental annotation Capture encounters parts (components) in your design, which are already annotated with part references that are outside the range specified in the grid. Make sure that the part reference range does not conflict with the parts already annotated in the design.

Legacy Message ID: ANN0010

ERROR[ORCAP-1359] - "An attempt was made to change %s to part %d, however it is a %d part per package device"

This error arises when Update Part References tried to change the Part Reference's package value to use the part's package as the suffix, but the part's package does not exist as a valid suffix to change to. This problem may exist because of an improperly configured combined property string in the Update Properties dialog box.

Legacy Message ID: ANN0003

ERROR[ORCAP-1360] - "Memory limit, unable to complete updating references"

A memory limit was encountered while processing. Currently this is only encountered while organizing the parts on the page for the annotation order. Free up some memory and rerun.

Legacy Message ID: ANN0002

WARNING[ORCAP-1374] - "Properties specified in combined property string not found on part, '%s'. Using default combine property string {Value} for this part."

This warning message comes up when the properties specified in the combined property string are not found in a part while performing annotation. Capture assumes default combined property string in such case. User should check the combined property string that was put while annotating.

Legacy Message ID: ANN0006

WARNING[ORCAP-1375] - "Warning limit exceeded. Please check combined property string for invalid specification. "

This warning message comes up when there are too many warning messages, and the number exceeds the limit. Currently the limit is 10. User needs to check the messages and rectify. One situation while this message comes up is if the user has specified some properties as part of combined property string, and they are not available in any of the parts in the design.

Legacy Message ID: ANN0007

ERROR[ORCAP-1376] - "Cannot perform annotation of heterogeneous part '%s(Value %s) at location (%s) on page %s', part has not been uniquely grouped (using a common User Property with differing Values) or the device designation has not been chosen"

This error arises when the tool encounters a second heterogeneous part with an identical part reference to the first part. Group the parts into two different packages. You can avoid this error by annotating all heterogeneous parts before running the tool.

Legacy Message ID: ANN0005

WARNING[ORCAP-1377] - "Component %s%s has different common pin connection for two instances and hence packaged separately."

When a common pin of two instantiated sections of a part has different net connections, Capture packages the two instances separately to eliminate design errors. This warning can be ignored if the connections are done intentionally.

Legacy Message ID: ANN0009

ERROR[ORCAP-1404] - "Part Count Overflow: %s."

This error arises when the tool is creating a report of unused parts and finds more than 4096 parts in a package.

Legacy Message ID: BOM0012

ERROR[ORCAP-1407] - "Part reference %s is out of range (%d to %d) %s(x=%d, y=%d)"

This error message comes up when total number of parts (components) on your schematic page exceeds the part reference range specified in the grid. Specify an End Value for the range, which is more than the total number of parts on the page.

Legacy Message ID: ANN0011

Q[ORCAP-1409] - "Allegro/Layout PCB Netlist has been already generated for this design; this operation may cause physical design to go out of sync, and result in problem(s) during Back-Annotation. Do you want to continue?"

This warning message comes up when annotation is attempted on a design after creating Layout/PCB Editor netlist. This message is to inform the user, in case the user is going to perform annotation by mistake after creating Layout/PCB Editor netlist, as annotate operation may result in the design to go out of sync with the physical board. This warning can be ignored if the operation is performed intentionally.

Legacy Message ID: ANN0008

ERROR[ORCAP-1411] - "The total number of components on %s exceeds the range supplied for it. Increase the End value of the range."

This error message comes up when during incremental annotation in the hierarchical blocks mode, Capture encounters a schematic page for which a part reference range has not been specified and the schematic page contains annotated parts with part references, which conflict with the part reference range specified for the hierarchical blocks referring the schematic page.

Legacy Message ID: ANN0012

ERROR[ORCAP-1412] - "(line %d) No valid update type parameter was found, either ""%s"" or ""%s"" must be specified as a parameter"

This error occurs when a swap file includes update properties information, and the swap file attempts an update without specifying either parts or nets. In the swap file, find the section attempting an unspecified properties update, and add the appropriate parameter (Parts or Nets).

Legacy Message ID: GAT0025

ERROR[ORCAP-1413] - "Memory limit occurred while allocating data space"

Gate and Pin Swap is unable to obtain sufficient memory to start. Free up some system memory, and then run Gate and Pin Swap again.

Legacy Message ID: GAT0008

ERROR[ORCAP-1414] - "Unable to open '%s'"

The swap file is not found, or there is a problem opening it.

Legacy Message ID: GAT0007

ERROR[ORCAP-1415] - "Possibly incorrect design loaded. Current design is %s, back annotations are for %s."

This error occurs when the current design name doesn't match the name Capture stores in memory. For example, this message is generated when the design is renamed (using the Save As command on the File menu), and then Capture attempts to swap gates or pins. Avoid renaming a design just before using Gate and Pin Swap. If the design has been renamed, close it, and reopen the design.

Legacy Message ID: GAT0027

ERROR[ORCAP-1416] - "Memory limit occurred while enlarging internal data space"

Gate and Pin Swap is unable to obtain sufficient memory to continue. Free up system memory, and then run Gate and Pin Swap again.

Legacy Message ID: GAT0009

ERROR[ORCAP-1417] - "(Line %s) Duplicate swap specification, reference (%s) has already been swapped or changed"

The part reference shown (partRef) is listed more than once. If a part reference is changed in either a GATESWAP or CHANGEREF specification, then another GATESWAP or CHANGEREF specification with the same part reference is not allowed in the same swap file.

For example, the following two swap specifications would not be allowed in the same swap file:

GATESWAP U1 U2 ; U1 appears in two specificationsCHANGEREF U1 U10 ; in the same swap file.

However, a device within a package can be swapped along with the package. For example, the following specification is legal:

GATESWAP U1A U1B ; Swap gates A and B on U1.CHANGEREF U1 U2 ; Swap U1 and U2.

[*Legacy Message ID: GAT0010*

ERROR[ORCAP-1418] - "(Line %s) Misplaced %s keyword, %s is only allowed as the first keyword in a swap file"

The VERSION keyword is found somewhere other than the first line. Gate and Pin Swap ignores the VERSION keyword anywhere else in the swap file.

Legacy Message ID: GAT0004

ERROR[ORCAP-1423] - "(Line %s) Expecting a reference designator: %d"

An incomplete swap specification is found. The swap specification (swapSpec) may be missing the first part reference. In the case that the swap specification is a gate swap or reference change, the second part reference may be missing.

Legacy Message ID: GAT0001

ERROR[ORCAP-1424] - "(Line %s) Unterminated string: %s"

A string begins with a quote but does not end with a quote, or swapSpec (including the quotation marks) is greater than 126 characters. An extremely long part reference or pin number that has been quoted by mistake can also cause the error.

Legacy Message ID: GAT0002

ERROR[ORCAP-1425] - "(line %s) Multi and single part per package swap is illegal: %s"

A GATESWAP or CHANGEREF specification attempts to change a multiple-part package with a single-part package. The following example swap specification causes this error:

CHANGEREF U1A U2

[*Legacy Message ID: GAT0019*

ERROR[ORCAP-1426] - "(Line %s) Expecting a pin identifier: %s"

An incomplete pin swap specification is found. Two pins are expected in a pin swap specification after the part reference; one or both of these pin identifiers were missing.

Legacy Message ID: GAT0003

I[ORCAP-1427] - "(Line %s) Ignoring extra tokens on the line: "

Extra tokens are found on a swap specification line. The swap specifications typically consist of a keyword followed by either two part references, or a part reference and two pins. The additional tokens are ignored.

Legacy Message ID: GAT0005

ERROR[ORCAP-1428] - "Swap file is empty"

There are no swap specification lines in the swap file.

Legacy Message ID: GAT0006

ERROR[ORCAP-1429] - "Multiple pins contain the same name, cannot swap '%s' on '%s' by pin name, swap by pin number instead"

A PINSWAP specification refers to a pin by name, but the device contains multiple pins with the same name (differing only by pin number). To specify the pin unambiguously, refer to it by pin number.

Legacy Message ID: GAT0020

ERROR[ORCAP-1430] - "Unable to perform pinswap, multiple pins contain pin number %s on %s, swap using pin name instead"

This error arises when the tool encounters two or more pins on a part with identical pin numbers. Capture responds by swapping the pin names instead. To avoid this problem, check that all pins on parts have unique pin numbers.

Legacy Message ID: GAT0023

ERROR[ORCAP-1434] - "Unable to change %s to %s because %s was not found"

A GATESWAP or CHANGEREF specification is not performed, typically because the part (partRef) is not found in the selected schematic pages.

Legacy Message ID: GAT0016

WARNING[ORCAP-1436] - "Unable to restore '%s' from '%s'."

Some portions of the design were not found. Please close the design and relocate the missing schematic(s). Saving will lose any occurrence properties associated with the missing schematics. See the session log for details. A typical cause for this error is that the referenced library or design files have been moved or are not present in the paths relative to this design.

Immediately preceding this error message in the session log is a list of referenced schematics that are missing. From this list you should be able to determine why the schematics are no longer available. The message indicates that a hierarchical part references a schematic that is no longer present or has moved.

Actions that might generate this error include relocating a schematic, moving a library or referenced design to a new directory, or losing a network connection where the library or referenced design can no longer be reached. This message might also mean that you have a design in an intermediate state and a referenced schematic may not have been created yet. If this is the case then ignore this message and continue working. You do not want to save the design if a portion of it is not available because occurrence information for that section of the design will not be saved, such as reference designators, IDs, or properties. Only if this information is not important should the design be saved in this state.

In general, you cannot move externally referenced designs once they have been placed and used in a design without losing all occurrence properties. Re-structuring a library—by moving a reuse design location, renaming directories, or adding a level of hierarchy in the directory structure, for example—requires careful management of parent reuse designs and the reuse designs contained within them. If you do re-structure a library you must first place the reuse schematic in the same path it was originally found before doing the edits to the parent schematic. Editing reuse designs that are already referenced or contained within reuse schematics must be done with care.

Legacy Message ID: CAP0052

ERROR[ORCAP-1457] - "A file must be selected in order to perform back annotation."

This error arises when a swap file has not been selected in the Gate and Pin Swap dialog box. Select a swap file before choosing OK in the dialog box.

Legacy Message ID: GAT0022

ERROR[ORCAP-1459] - "Unable to open library. Specify a different library (.OLB) for replacement part."

This error message appears when you try to paste a design into a target design and the design being pasted contains parts that are out of sync with the parts in the target design.

To solve this error, compare the two design caches. Do replace cache on matching parts to pull them from the same library.

Legacy Message ID: DSM0020

ERROR[ORCAP-1490] - "Macro Manager Unable to Load Macros Only 50 Macros may be loaded at one time."

The macro manager detected an attempt to load more than 50 macros. The macro manager can only have up to 50 configured macros at any given time. If you need a particular macro or macros configured, scan through the current list of configured macros, remove unessential macros, and then configure the necessary macros.

Legacy Message ID: MCR0009

ERROR[ORCAP-1491] - "Macro Player Error Detected in Macro"

The macro player detected an error in the selected macro while attempting to play the macro. For example, this error occurs when the macro player encounters an undefined macro in a macro file. Check that your macro file uses macro commands defined in About macros of the OrCAD Capture User's Guide.

Legacy Message ID: MCR0001

ERROR[ORCAP-1492] - "Macro Player Error Detected in Macro %s: %s"

The macro player detected an error in the selected macro while attempting to play the macro. For example, this error occurs when the macro player encounters an undefined macro in a macro file. Check that your macro file uses macro commands defined in About macros of the OrCAD Capture User's Guide.

Legacy Message ID: MCR0001

ERROR[ORCAP-1493] - "Macro Player Error Detected in Macro %s"

The macro player detected an error in the selected macro while attempting to play the macro. For example, this error occurs when the macro player encounters an undefined macro in a macro file. Check that your macro file uses macro commands defined in About macros of the OrCAD Capture User's Guide.

Legacy Message ID: MCR0001

WARNING[ORCAP-1500] - "Unable to find object."

Capture was unable to find the specified object. If you think the part should be in the design, schematic, or schematic page, check that you spelled the object's name correctly (including the case if you selected Match Case). Check that you selected the correct scope if applicable. If the design manager is active, check that you selected the appropriate schematics and schematic pages.

Legacy Message ID: CAP0028

WARNING[ORCAP-1531] - "Part Name renamed from %s to %s as it exceeds the 31 chars OLB storage limit."

This warning appears if the Part Name has more than 31 chars for the generated part. Since OLB has a storage limit of 31 characters for part names, so during part generation, part Name is automatically truncated till 31 chars and the part is generated.

Legacy Message ID: GSM0015

ERROR[ORCAP-1545] - "File not found: %s."

This error arises when one of the tools cannot find the file to be backed up. This is usually due to an invalid path or file name.

Legacy Message ID: EXT0007

ERROR[ORCAP-1546] - "Cannot close the backup (.BAK) file."

This error arises when a critical system error prevents the tool from creating a backup of the existing .PLD file.

Legacy Message ID: EXT0008

ERROR[ORCAP-1548] - "Unable to open source file while making backup file."

This error arises when one of the tools cannot locate the original file that it needs to be backed up. It is probably due to an invalid path or file name.

Legacy Message ID: EXT0001

ERROR[ORCAP-1549] - "Problems writing to destination file while making backup file."

This error arises when one of the tools cannot write to a backup file. This is probably due to insufficient space, empty floppy disk drive, or lack of write access.

Legacy Message ID: EXT0003

ERROR[ORCAP-1550] - "Unable to close source file while making backup file."

This error arises when one of the tools cannot close the file being backed up. This is caused by a critical system error.

Legacy Message ID: EXT0004

ERROR[ORCAP-1551] - "Unable to delete the source file while making backup file."

This error arises when one of the tools cannot delete an existing version of the backup file. This is probably due to lack of delete access (especially on a network drive).

Legacy Message ID: EXT0006

ERROR[ORCAP-1552] - "Unable to close destination file while making backup file."

This error arises when one of the tools cannot close the backup file. This is caused by a critical system error.

Legacy Message ID: EXT0005

ERROR[ORCAP-1553] - "Unable to delete the backup (.BAK) file (Read Only File)."

This error arises when the Read-only attribute is set on an existing .BAK file, or when some other problem prevents the tool from deleting the old .BAK file.

Legacy Message ID: EXT0009

ERROR[ORCAP-1554] - "Unable to create destination while making backup file."

This error arises when one of the tools cannot create a backup file. This is probably due to insufficient disk space, empty floppy disk drive, or lack of write access.

Legacy Message ID: EXT0002

ERROR[ORCAP-1564] - "(Line %s) Zero length property string - %s"

Update properties encountered a zero length property string in the property update file. For example, this error occurs if one of the entries in the first line of the file is "". Check that all property strings have at least one character in the property update file.

Legacy Message ID: UPD0005

ERROR[ORCAP-1565] - "Unable to open report file '%s'"

Update Properties was unable to open the specified report file. For example, this error occurs when the report file doesn't exist, or isn't in the indicated directory. check that the file exists in the indicated directory.

Legacy Message ID: UPD0001

ERROR[ORCAP-1566] - "Unable to open stuff file '%s'"

Update Properties was unable to open the specified property update file. For example, this error occurs when the property update file doesn't exist, or isn't in the specified directory. Check that the file exists in the specified directory.

Legacy Message ID: UPD0003

ERROR[ORCAP-1567] - "No matches made, check the 'Combined property string' for spelling errors and nonexistent property names"

Update Properties didn't find any matches between the property update file and the design. For example, this error is generated if a key match string in the first column is spelled incorrectly, or nonexistent in the design. Check that all key match strings are in the design and spelled correctly.

Legacy Message ID: UPD0012

ERROR[ORCAP-1568] - "A file must be selected in order to update properties"

Update Properties couldn't update any properties because no property update file was selected. You must select a property update file.

Legacy Message ID: UPD0011

ERROR[ORCAP-1569] - "(Line %s) Missing stuff string, the number of stuff strings does not match the number of properties to stuff - %s"

Update Properties encountered a missing property string in the property update file. For example, this error occurs if the property update file looks like:

"{Value}" "{PCB Footprint}"

"74LS00"

Check that all property match string lines have the same number of columns as the first line in the property update file.

Legacy Message ID: UPD0007

ERROR[ORCAP-1570] - "(Line %s) Identifier is not quoted - %s"

Update Properties detected that an identifier was not quoted. For example, this error occurs if the first entry of the first column reads {PCB Footprint} instead of "{PCB Footprint}". Make sure that all identifiers in the property update file are quoted.

Legacy Message ID: UPD0002

ERROR[ORCAP-1571] - "(Line %s) The stuff file does not specify any properties to update - %s"

Update Properties detected no properties to update in the property update file. For example, this error occurs when the property update file only has one column in the first line. Check that the first line of the property update file has at least two columns.

Legacy Message ID: UPD0006

WARNING[ORCAP-1574] - "(Line %s) Too many stuff strings specified, the extra stuff strings will be ignored - %s"

Update Properties detected an extra update property string in the property update file. For example, this warning occurs if the property update file looks like:

"{Value}" "Schematic" "Schematic Library""74LS32" "HALFADD" "FULLADD.OLB" "FULLADD.TXT"

Check that each property string has a corresponding property identifier in the first line of the property update file.

Legacy Message ID: UPD0008

ERROR[ORCAP-1575] - "Stuff file '%s' contains no stuff specification"

The specified file contained no properties to update. For example, this error occurs when the property update file contains only one line of information. Check that your property update file contains at least two lines of information.

Legacy Message ID: UPD0004

ERROR[ORCAP-1576] - "(Line %d) Duplicate key match string encountered - %s"

Update Properties detected a duplicate key match string in the property update file. For example, this error occurs if the property update file looks like:

"{Value}" "PCB Footprint"

"74LS08" "DIP14"

"74LS08" "DIP14"

Check that each key match string is used only once in the property update file.

Legacy Message ID: UPD0009

I[ORCAP-1581] - "Errors were detected while reading a page. Please open the page in the schematic editor and examine it to determine whether you wish to save the page in its current state."

This error is caused by an improperly named bus pin on a hierarchical block. Check the syntax to ensure that the naming reads as Pin Name[Left Bound..Right Bound]. The bounding values must be numeric and not alpha characters.

Legacy Message ID: CAP0049

ERROR[ORCAP-1582] - "Cannot swap pins %s and %s of %s because they are not located on the same device"

One or both pins listed are not on the part shown (partRef). The pins must be on the same part.

Legacy Message ID: GAT0011

ERROR[ORCAP-1583] - "Unable to change %s to %s because %s does not contain a device with a suffix of '%s'"

One of the part references (partRef) refers to a device (partRefSuffix) that does not exist in the package definition. The following example swap specification causes this error by attempting to swap devices A and E on a quad NAND gate which defines suffixes A through D only:

CHANGEREF U1A U1E ; Quad NAND gate

[*Legacy Message ID: GAT0021*

ERROR[ORCAP-1584] - "Unable to perform pinswap because pins %s and %s were not found on %s"

A PINSWAP specification is not performed, because the pins listed (pin1 and pin2) are not found in the part shown (partRef).

Legacy Message ID: GAT0018

ERROR[ORCAP-1585] - "Unable to perform ChangePin because either or both of pins %s and %s were not found"

Capture is unable to update the design file with ChangePin information because one or both the pins are not available in the complete package.

Note that if the pins are spread across different parts in a heterogeneous package, Capture 9.2.3 or later can update the design with the ChangePin information. However, in case of a homogeneous package Capture will give GAT0028 error if the pins are spread across different packages.

Legacy Message ID: GAT0028

ERROR[ORCAP-1586] - "Unable to swap pins %s and %s of %s because %s was not found"

A PINSWAP specification is not performed, because the part (partRef) is not found in the selected schematic pages.

Legacy Message ID: GAT0017

ERROR[ORCAP-1588] - "Pins %s and %s of %s have different types/shapes, the pins will not be swapped"

The two pins differ in type or shape. Only pins of identical type and shape can be swapped.

Legacy Message ID: GAT0014

Q[ORCAP-1589] - "Net has two or more aliases - possible short?"

Two or more power nets have been connected together in the design. The following are the conditions that may cause this warning to appear:

1.	A Power and GND net are shorted together.

2.	Any two nets are shorted.

3.	A net has two or more aliases assigned to it.

This warning is displayed only if the Check power ground short option is checked in the Physical

Rules tab of the Design Rules Check dialog box.

Legacy Message ID: DRC0037

WARNING[ORCAP-1590] - "Visible unconnected Power Pins are connected to global nets"

The visible power pin on the part instance is not connected to a wire or other power object and hence has a default connection to a global net that has the same name as the power pin. You may override this default connection either by connecting the visible power pin to a wire or adding a wire between the pin and the power object.

This warning appears if you enable the Report visible unconnected power pins design rules check.

Legacy Message ID: DRC0038

ERROR[ORCAP-1591] - "Pin has been placed on top or bottom of hierarchical block"

Design Rules Check detected a pin placed on top or bottom of a hierarchical block. This error occurs when checking for SDT compatibility. SDT hierarchical blocks cannot have pins on top or bottom. Move the pin to the left or right side, as appropriate.

Legacy Message ID: DRC0019

ERROR[ORCAP-1592] - "Bus has no name and therefore defines no signals."

Design Rules Check detected a bus without a name. The specified bus defines no signals, even though it may be connected to other nets. Use the Net Alias command on the Place menu, or the Net Alias tool on the schematic page editor tool palette, to create a name for the net.

Legacy Message ID: DRC0029

ERROR[ORCAP-1597] - "More than 1 titleblock exists on this page"

Design Rules Check detected a second titleblock on the page while checking for SDT compatibility. Make sure that all schematic pages have no more than one titleblock each.

Legacy Message ID: DRC0021

ERROR[ORCAP-1598] - "More than 8 user properties exist on this part instance"

Design Rules Check detected a part with more than eight user properties while checking for SDT compatibility. Make sure that all parts in the schematic are limited to eight or less user properties when creating a design for SDT.

Legacy Message ID: DRC0020

ERROR[ORCAP-1599] - "The package contains more than 16 parts"

Design Rules encountered a package containing more than 16 parts while checking for SDT compatibility. Designs created for SDT cannot have more than 16 parts. Replace the parts with a package containing 16 or fewer parts.

Legacy Message ID: DRC0023

WARNING[ORCAP-1600] - "Net has fewer than two connections"

Design Rules Check encountered a net with one or no connections. For example, this warning is generated when one end of a wire connects to a pin, but the other wire end is unconnected. Check that the net has at least two connections.

Legacy Message ID: DRC0006

WARNING[ORCAP-1603] - "Type of pin above does not match the pin type of corresponding port below"

Design Rules Check detected a pin or hierarchical port in a hierarchical block whose type does not match the corresponding hierarchical port type in the child schematic. For example, this warning is generated when a hierarchical port in a hierarchical block named "out" has type output, but the corresponding hierarchical port named "out" in the child schematic has type input. Check that all pin and hierarchical block types in parent schematics match the corresponding hierarchical port types in the child schematic.

Legacy Message ID: DRC0014

ERROR[ORCAP-1604] - "Same Pin Number connected to more than one net."

Design Rule Check detected multiple pins with the same pin number connected to different nets instead of a single net. In a package there can be multiple logical pins having the same number as they physically represent a single pin only.

Note: To catch DRC0031 errors in your design, select the Check SDT Compatibility check box in the Design Rule Check dialog box.

Legacy Message ID: DRC0031

ERROR[ORCAP-1605] - "Other parts in this package have different values or PCB footprints."

The part's reference (such as R or U1) has been assigned to a different package type, or the part uses a different PCB footprint value than the rest of the package. For example, this error occurs when a 74LS08 has been assigned a part reference U1B, but a 74LS32 part has been assigned a part reference U1A. This error is also generated when part U1A uses the PCB footprint DIP14, but U1B has a PCB footprint of DIP24. If the reference values are inconsistant, then change the part's reference to an appropriate value, or use Update Part References to first reset, and then update the references. If the PCB footprint caused this error, change this value to match the rest of the package.

This error also occurs when one part in a multi-part package is edited in place on the schematic. For example, this warning occurs when a part with part reference of U1D has been edited in place in the schematic, but U1A, U1B, and U1C have not. The edited part has lost its connection with the original library part. However, the other parts in the package have not lost this connection. Either use Update Part References to reset the edited part, or update the other parts to match the edited part.

Legacy Message ID: DRC0027

ERROR[ORCAP-1607] - "User properties exist on an object that is not a part instance"

Design Rules Check encountered a user property on an object that is not a part instance. For example, this error occurs if a user property exists on a net, and you are checking for SDT compatibility. Delete the user property from the object.

Legacy Message ID: DRC0016

WARNING[ORCAP-1608] - "Net has no driving source"

Design Rules check encountered a net with no driving source. For example, this warning is generated when both ends of a wire are connected to input pins. Check that the net has a driving source, such as one end being connected to an output type pin.

Legacy Message ID: DRC0007

WARNING[ORCAP-1610] - "Pin has no matching port in implementation below"

Design Rules Check detected a hierarchical port in a hierarchical block with no matching port in the child schematic. For example, this warning occurs when the hierarchical block contains five hierarchical ports, but only four ports exist in the child schematic. Check that all hierarchical ports in hierarchical blocks match up with one hierarchical port in the child schematic, and that these names are spelled exactly the same.

Legacy Message ID: DRC0012

WARNING[ORCAP-1611] - "Two nets in same schematic have the same name, but there is no off-page connector"

Two nets in the same schematic, but on different schematic pages share the same name without connection by an off-page connector. Add off-page connectors between the two pages to connect the nets, or rename one of the nets.

Legacy Message ID: DRC0008

WARNING[ORCAP-1612] - "Two connected wires/buses form a T yet there is no junction to show a connection is being made."

This warning message appears when Design Rule Check detects a connection between two wires/buses, which does not have a junction placed on them to represent the connection.

Legacy Message ID: DRC0035

WARNING[ORCAP-1613] - "No matching off-page connector"

Design Rules Check detected an off-page connector without a matching off-page connector in the same schematic. Check that you are not trying to use an off-page connector as a hierarchical port. Off-page connectors cannot traverse hierarchies.

Legacy Message ID: DRC0009

ERROR[ORCAP-1615] - "Pin number is greater than 255"

Design Rules Check encountered a pin with a number greater than 255, while checking for SDT compatibility. All pins on parts of schematics designed for SDT cannot exceed numbers greater than 255. Change the number of the pin so that it is 255 or less.

Legacy Message ID: DRC0017

ERROR[ORCAP-1616] - "Reference is invalid for this part"

The reference for the part is invalid. For example, this occurs when a part reference like U?A has not been updated. Update the part reference.

Legacy Message ID: DRC0011

ERROR[ORCAP-1620] - "Port has a type which is inconsistent with other ports on the net"

One or more ports on the net are not consistent because all ports are not of the same type. Ensure all ports on the net are of the same type.

Legacy Message ID: DRC0003

ERROR[ORCAP-1621] - "This reference has already been assigned to a different package type."

The part's reference (such as R or U1) has been assigned to a different package type. For example, this warning occurs when a 74LS08 has been assigned a part reference U1B, but a 74LS32 part has been assigned a part reference U1A. Change the part's reference to an appropriate value, or use Update Part References to first reset, and then update the part references.

This warning also occurs when one part in a multi-part package is edited in place on the schematic. For example, this warning occurs when a part with part reference of U1D has been edited in place in the schematic, but U1A, U1B, and U1C have not. The edited part has lost its connection with the original library part. However, the other parts in the package have not lost this connection. Either use Update Part References to reset the edited part, or update the other parts to match the edited part.

This warning also occurs when one part in a multi-part package is edited and it results in an extra entry of that library part in the design cache. Either use Update Part References to reset the edited part, or remove the extra entry in the design cache.

Legacy Message ID: DRC0026

ERROR[ORCAP-1622] - "The schematic contains multiple pages and hierarchy"

Design Rules Check detected a hierarchy in your design, and a schematic with multiple pages, while checking for SDT compatibility. SDT designs must be flat (no hierarchy) to contain multiple pages, or contain exactly one page per schematic in a hierarchy. Remove the second schematic page, or eliminate the hierarchy in the design.

Legacy Message ID: DRC0022

ERROR[ORCAP-1623] - "The package contains different types of parts"

Design Rules Check detected a heterogeneous part while checking for SDT compatibility. Designs created for SDT cannot have heterogeneous parts. Replace the part with a homogeneous part.

Legacy Message ID: DRC0024

ERROR[ORCAP-1624] - "Pin number is not numeric "

Design Rules Check detected a pin number that isn't numeric, while checking for SDT compatibility. All pin numbers in schematics designed for SDT must be numeric. Change the pin number to a numeric value.

Legacy Message ID: DRC0018

ERROR[ORCAP-1625] - "Pin buses exist in the schematic"

Design Rules Check detected bus type pins while checking the design for SDT compatibility. SDT cannot handle parts with bus type pins. Change the pin to another type. If necessary, add pins to the part or hierarchical block.

Legacy Message ID: DRC0015

WARNING[ORCAP-1626] - "Bus width is not matching with the port Width"

This message is displayed when the width of a bus does not match the width of the port to which it is connected.

For example, this warning is displayed if you connect bus A[0:4] to the hierarchical pin A[0:5] of a hierarchical block that contains the port A[0:5] down the hierarchy.

Legacy Message ID: DRC0030

WARNING[ORCAP-1629] - "Multiple Hierarchical Ports of same name exist across Hierarchical Blocks."

Using multiple hierarchical ports of the same name in single schematics is perfectly legal. This causes no harm and does not create any ambiguity in terms of the final netlist. However, identifying multiple hierarchical ports will help to check those portions of designs where point-to-point connections are required. An example of such a scenario is the dedicated parallel interface between an FPGA and a FLASH memory.

Most users involved in large pin-count FPGAs hardware designs prefer to divide the FPGA part into several hierarchical blocks, making the part heterogeneous. Designers frequently place each block of the same part on a separate page.

When working on team designs, two designers may connect the same signal originating from /going to/ the upper level of the hierarchy to an I/O pin of the FPGA blocks on which they are working. This creates unintentional redundancy in connectivity that may lead to unnecessary PCB routes.

If two outputs are connected in this manner, Capture displays this warning. However, if the FPGA pins are of type input, passive or bidirectional, the DRC tool does not recognize any problem.

In a team design scenario, it is the team leader’s responsibility to ascertain whether this redundancy occurs in the project. This DRC check occurs when integrating pages and schematics created by team members into a final complex design.

Current designs may comprise hundreds of signals /wires/ and FPGA heterogeneous symbol containing hundreds of pins that may reside on many pages. In this case, the tasks of manually identifying and excluding redundant signal paths from such complex designs becomes extremely difficult and time consuming.

Legacy Message ID: DRC0042

ERROR[ORCAP-1631] - "Duplicate reference"

Design Rules Check encountered a duplicate reference. For example, this message occurs when two parts have a reference value of U1A. Increment one of the parts to a different member in the package (such as B or C), or change it to a different package (such as U2). Alternately, use Update Part References to first reset, and then update the part references.

Legacy Message ID: DRC0010

ERROR[ORCAP-1633] - "Cannot open the file for output"

This error arises when Express is unable to open the selected export file for output. Check that the file is not read-only.

Legacy Message ID: EXP0001

I[ORCAP-1635] - "No errors on Export Properties"

Export Properties successfully created the export property file.

Legacy Message ID: EXP0002

WARNING[ORCAP-1638] - "Port has no matching pin in part instance above"

Design Rules Check detected a hierarchical port on a child schematic with no matching port in the hierarchical block of the parent schematic. For example, this warning occurs when the child schematic has five hierarchical ports in the child schematic, but only four ports exist in the hierarchical block of the parent schematic. This warning also occurs when the names of the hierarchical port names don't share the same case. For example, the parent schematic hierarchial port may be "Out" but the matching child schematic port is "out." Check that all hierarchical ports in hierarchical blocks match up with one hierarchical port in the child schematic, and that these names are spelled exactly the same.

Legacy Message ID: DRC0013

ERROR[ORCAP-1647] - "Errors were detected while reading the design."

This message occurs when Capture cannot perform a paste successfully.

Legacy Message ID: CAP0050

ERROR[ORCAP-1648] - "Save As. Must Save As to a different file."

You have selected a part in PCB Editor that has no corresponding part in Capture. The part may have been deleted from Capture or added in PCB Editor.

Legacy Message ID: CAP0051

ERROR[ORCAP-1650] - "Unable to save ' %s'."

Capture was unable to save the specified file. For example, this error occurs when the file is open in another application. Close the file out of all other applications.

Legacy Message ID: DSM0006

ERROR[ORCAP-1672] - "Unable to save part."

Capture was unable to save the part. For example, this error occurs when the file is open in another application. Close the file out of other applications, and try saving the part again.

Legacy Message ID: CAP0003

ERROR[ORCAP-1674] - "Unable to create a new part."

Capture was unable to create a new part. For example, this error occurs when a part by the specified name already exists. Check the library if the name already exists in the library

Legacy Message ID: CAP0002

ERROR[ORCAP-1691] - "(Line %d) Part not found in design"

Import Properties detected a part in the export property file that is not in the design or library. For example, this error occurs if you delete a part from the design after generating the export property file, but before importing the file. It will also occur if you add an extra PART line to the file. Check that the export property file contains the same number of PART lines as there are parts in the design.

Legacy Message ID: IMP0007

ERROR[ORCAP-1692] - "(Line %d) Page not found"

Import Properties was unable to find the page in the design that corresponds to the schematic page listed in the export property file. For example, this error occurs when the export property file contains information about a second page in the schematic when the design contains only one page in the schematic. Check that the correct page is referenced in the export property file.

Legacy Message ID: IMP0001

ERROR[ORCAP-1700] - "(Line %d) Object ID column not found in HEADER line"

Import Properties encountered a missing ID column in the HEADER line. For example, this error occurs if the following line appears in the export property file:

"HEADER" "Part Reference" "Value" "1ST PART FIELD"

instead of the following line:

"HEADER" "ID" "Part Reference" "Value" "1ST PART FIELD"

Check that all HEADER lines have an ID column.

Legacy Message ID: IMP0003

ERROR[ORCAP-1702] - "(Line %d) No columns after object ID in HEADER line"

Import Properties detected no columns after the object ID column in the HEADER line of the export property file. For example, this error occurs if all of the columns after the ID column are deleted from the HEADER line. Check that the HEADER line has all of the required columns.

Legacy Message ID: IMP0004

ERROR[ORCAP-1704] - "(Line %d) Line has more fields than previous HEADER line"

This error message is flagged if a property value in your design ends with the " character. During the import process, the " character is treated as a delimiter for a field and hence reported as extra rows in the field. For example, a property called MyProp1 with a Value test3".

Legacy Message ID: IMP0018

ERROR[ORCAP-1725] - "Library file '%s' not found."

This error message appears when you select a component and use the Update Cache command and the source library is not added to your project.

To solve this error, make sure that the library where the selected component resides is added to the project before using the Update Cache command.

Legacy Message ID: CAP0031

ERROR[ORCAP-1727] - "Invalid scalar pin name specified. Cannot give a bus name to a scalar pin."

This error message appears when you specify a bus name (for example, AS[10:0]) for a Scalar pin in the Place Hierarchical Pin dialog box.

To solve this error, do not specify a bus name for a scalar pin.

Legacy Message ID: CAP0041

ERROR[ORCAP-1729] - "Invalid bus pin name specified. Use this format: busname[1..20]"

This error message appears when Capture encounters an invalid bus pin name in the design. Some of samples of a correct bus pin name are: DSP[1..20], A[1..5].

Legacy Message ID: CAP0001

ERROR[ORCAP-1733] - "Allegro footprint %s was not found in the search path."

This error message appears if the path to the library containing the footprint is not set in PCB Editor and OrCAD Capture. In PCB Editor set padpath and psmpath to the location of the library in the User Preferences Editor (Setup-User Preferences) under Library in the Paths category. In Capture, edit the Capture.ini file to add the library path in the Allegro Footprint section.

Legacy Message ID: CAP0095

ERROR[ORCAP-1734] - "Unable to open file %s."

This error arises when Extract PLD cannot open the specified file. This is typically due to a typographical error in the path or filename.

Legacy Message ID: EXT0012

ERROR[ORCAP-1735] - " Unable to create the report file."

This error arises when Cross Reference encounters a problem while trying to create a report file. Typical causes include invalid path, insufficient disk space, lack of write access, and empty floppy disk drive.

Legacy Message ID: EXT0011

ERROR[ORCAP-1736] - "Cannot close the report file."

This error arises when Extract PLD cannot close the report file. It may be caused by insufficient disk space or some other system error.

Legacy Message ID: EXT0019

ERROR[ORCAP-1737] - "A pin with the number '%s' already exists on the part. This message appears for the first duplicate, others may exist."

This warning message appears when Capture encounters two pins having the same pin number on a part.

Legacy Message ID: CAP0040

WARNING[ORCAP-1738] - "A pin with name '%s' is duplicated on the part."

This warning message appears when Capture encounters two pins having the same name on a part.

Legacy Message ID: CAP0045

ERROR[ORCAP-1749] - "You can specify another source library (.OLB) using Replace Cache"

This error message appears when you select a component and use the Update Cache command and the source library is not added to your project.

To solve this error, make sure that the library where the selected component resides is added to the project before using the Update Cache command.

Legacy Message ID: DSM0018

ERROR[ORCAP-1756] - "Unable to update occurrences."

Capture was unable to update occurrences. For example, this error occurs when you try to change from logical mode to physical mode while the file is open in another application. Close the file out of all other applications.

Legacy Message ID: CAP0033

ERROR[ORCAP-1758] - "Unable to save symbol."

Capture was unable to save the symbol. For example, this error occurs when the file is in use by another application. Close the file out of all other applications.

Legacy Message ID: CAP0026

ERROR[ORCAP-1759] - "Unable to save page."

Capture was unable to save the page. For example, this error occurs when the file is open in another application. Close the file out of all other applications, and try to save the page again.

Legacy Message ID: CAP0030

ERROR[ORCAP-1760] - "Unable to rename object."

This message occurs when Capture cannot rename a page. For example, might this message arises when you rename a schematic page from "PAGE1" to "X", and then from "X" to "page1". Save the design after you rename the schematic page the first time, but before you rename it the second time.

Legacy Message ID: DSM0007

ERROR[ORCAP-1768] - "Unable to allocate additional memory for the %s."

This error arises when Cross Reference cannot allocate additional conventional memory for the named data table. Free up memory by removing unneeded drivers, closing other applications, and so on.

Legacy Message ID: EXT0010

ERROR[ORCAP-1772] - "Unable to descend part."

Capture was unable to descend into the selected part or schematic. For example, this error occurs when you choose Descend Hierarchy when a hierarchical block is selected, but has no specified attached schematic. This error also occurs when the attached schematic is specified, but doesn't exist. Before descending hierarchy on a part or hierarchical block, check that it has an attached schematic, and that the attached schematic exists.

Legacy Message ID: CAP0017

ERROR[ORCAP-1775] - "Unable to create a new library symbol."

Capture was unable to create a new library symbol. For example, this error occurs when another symbol with the same name already exists in the active library. Consider changing the name of one of the symbols, or save the new symbol in a different library.

Legacy Message ID: CAP0008

ERROR[ORCAP-1776] - "Unexpected exception."

This error arises when the tool encounters a problem in the database.

Legacy Message ID: BOM0017

ERROR[ORCAP-1777] - "Part has Part Value but no Reference %s."

This error arises when the tool finds a part with a valid part value, but no part reference. Enter a part reference and rerun the tool.

Legacy Message ID: BOM0006

ERROR[ORCAP-1778] - "Reference Designator is longer than 24 characters."

This error arises when the combination of the part reference and package exceeds 24 characters. Check the length of the part reference. This error also arises when you have a hierarchical block set to primitive, or it's missing its attached implementation, and its name exceeds 24 characters.

Legacy Message ID: BOM0020

ERROR[ORCAP-1779] - "Part has Reference but no Part Value %s."

This error arises when the tool finds a part with a valid part reference, but no part value. Enter a part value and rerun the tool.

Legacy Message ID: BOM0005

ERROR[ORCAP-1780] - "The number of parts used is greater than the number of parts specified in the package."

This error arises when the tool while creating a report of unused parts, finds a part that is outside the limits for the number of parts in the package. For example, if U1 is a four-part package and the tool encounters the part reference U1H, it reports an error because it expects only U1A, U1B, U1C, and U1D. Eliminate the problem and rerun the tool.

Legacy Message ID: BOM0014

ERROR[ORCAP-1782] - "Unable to write to the report file."

This error arises when a system error prevents the tool from writing data to the report file.

Legacy Message ID: BOM0016

WARNING[ORCAP-1783] - "There is no Title Block for this design. Headers were omitted."

This warning arises when there is no title block for the design. Capture responds by omitting the headers from the Bill of Materials and Cross Reference reports. To include these headers, check the design for a title block.

Legacy Message ID: BOM0021

ERROR[ORCAP-1790] - "Unable to read from source file."

This error arises when the tool encounters a corrupted file, an unexpected EOF, or some other critical system error.

Legacy Message ID: BOM0007

ERROR[ORCAP-1791] - "No parts were found in the design."

This error arises when the tool finds no parts in the selected schematic pages.

Legacy Message ID: BOM0002

ERROR[ORCAP-1792] - "Missing open quote, line %s."

This error arises when the tool reads an include file and one of the lines is missing the opening quote. Add the mark and rerun the tool. Blank lines do not generate this message.

Legacy Message ID: BOM0008

ERROR[ORCAP-1793] - "Missing closing quote, line %s."

This error arises when the tool reads an include file and one of the lines is missing the closing quote. Add the mark and rerun the tool.

Legacy Message ID: BOM0009

ERROR[ORCAP-1794] - "Unable to close the include file."

This error arises when the tool is trying to close the include file and encounters a system critical error (such as having a floppy disk removed).

Legacy Message ID: BOM0015

ERROR[ORCAP-1795] - "Name is too long, line %s."

This error arises when the tool reads an include file and the name (typically the Part Value) field is longer than 255 characters. Reduce the length of the name field and rerun the tool.

Legacy Message ID: BOM0010

WARNING[ORCAP-1796] - "Include file match key was not found: %s, line %s."

This warning arises when a match key in the include file could not be located in the design.

Legacy Message ID: BOM0019

ERROR[ORCAP-1797] - "Line is too long, line %s."

This error arises when the tool reads an include file and a line is longer than 255 characters. Reduce the length of the line and rerun the tool.

Legacy Message ID: BOM0011

ERROR[ORCAP-1798] - "Line %s of the include file is incomplete, ignoring this line. "

This warning arises when the tool encounters an incomplete or incorrect line in the include file. For example, you will get this message if you have the quoted match string at the beginning of the line but not the include information.

Legacy Message ID: BOM0003

ERROR[ORCAP-1800] - "Duplicate Reference: %s."

This error arises when the tool, while creating a report of unused parts, finds a duplicate part reference. Eliminate the duplicate part reference and rerun the the tool.

Legacy Message ID: BOM0013

ERROR[ORCAP-1801] - "Part has duplicate Reference %s"

This error arises when the tool encounters a duplicate part reference while building its Part Table. Eliminate duplicate references and rerun the tool.

Legacy Message ID: BOM0004

ERROR[ORCAP-1802] - "Include file has duplicate match field: %s, line %s."

This warning arises when the tool encounters duplicate match keys in the include file.

Legacy Message ID: BOM0018

ERROR[ORCAP-1804] - "Invalid reference designator: %s. Valid formats are: U1A or U1-11."

This error arises when Bill of Materials encounters a part with an invalid reference designator. Check that all parts have valid reference designators. Capture accepts reference designators in the form of U1, U1A, and U1-11.

Legacy Message ID: BOM0022

WARNING[ORCAP-1829] - "Possible pin type conflict"

A pin type is not compatible with another pin on the same net as specified by the ERC Matrix. (You can set pin types using the ERC Matrix tab on the Design Rules Check dialog box.)

For example, the default settings for the ERC Matrix specify that output pins are incompatible with other output pins. Therefore, this message will appear when two output pins are connected.

This message also will appear if the pin or hierarchical port on a hierarchical block type is not compatible with a hierarchical port on the same net. Make sure that pins and hierarchical ports use appropriate pin types.

Legacy Message ID: DRC0004

WARNING[ORCAP-1831] - "Unconnected pin"

Design Rules Check encountered an unconnected pin. Either connect the pin to a wire, or place a No Connect on the pin.

Legacy Message ID: DRC0005

ERROR[ORCAP-1844] - "The tap may not be connected to the bus or the tap may be connected to a bus with a different base name."

" Design Rule Check detected a net physically connected to a bus with a different base name than the net itself or there may be some unconnected bus bits. To solve this problem, make sure that the Tap is property connected to the bus.

Legacy Message ID:DRC0039"

ERROR[ORCAP-2414] - "Visible unconnected Power Pins are connected to global nets"

The visible power pin on the part instance is not connected to a wire or other power object and hence has a default connection to a global net that has the same name as the power pin. You may override this default connection either by connecting the visible power pin to a wire or adding a wire between the pin and the power object.

This warning appears if you enable the Report visible unconnected power pins design rules check.

Legacy Message ID: DRC0038

WARNING[ORCAP-39026] - "Cleanup Schematic Duplicates found, need to be corrected."

Duplicate objects were detected in the SDT design. This situation arises when two parts are placed on the schematic, on top of the other. Use Design Rules Check to locate the duplicate parts, and delete the duplicates.

Legacy Message ID: CUP0006

WARNING[ORCAP-39027] - "Cleanup Schematic Non-Orthogonal Wires/Buses found, need to be checked."

This situation arises when there are non-orthogonal wires or buses in the design. SDT designs typically do not use non-orthogonal wires or buses in designs. Although Capture translates these objects, you should check that they appear as expected.

Legacy Message ID: CUP0007

WARNING[ORCAP-39028] - "Cleanup Schematic Off-grid objects found, need to be corrected."

Parts in the SDT design were placed off-grid. As a result, pins may not line up correctly. To fix this, use Design Rules Check with the Report off-grid objects option selected to locate the off-grid objects. Then, select and move the parts.

Legacy Message ID: CUP0008

ERROR[ORCAP-22001] - "Bus pin on '%s' is not supported at this time."

This error message appears when your design contains a bus pin. Currently, bus pins are not supported.

Legacy Message ID: MNL0015

WARNING[ORCAP-22002] - "Duplicate pin number '%s' on '%s'."

This warning message appears if you design has a pin with the same pin number assigned to another pin.

Legacy Message ID: MNL0016

ERROR[ORCAP-22003] - "'%s' has already been encountered. Design is not packaged."

This error message appears when the design has multiple parts with the same part reference.

Legacy Message ID: MNL0017

WARNING[ORCAP-22004] - "Skipping '%s' with 0 pins."

This warning message appears if your design contains a part with no pins.

Legacy Message ID: MNL0018

ERROR[ORCAP-22006] - "'%s.%s' is tied to nets '%s' and '%s'."

This error message appears when there is a conflict between an existing part reference value for a net with another net. To solve this error, make sure that the part reference value for a net is unique.

Legacy Message ID: MNL0019

ERROR[ORCAP-22007] - "Unable to validate Design"

This error message appears if the Layout netlister is unable to access or open the design for which you want to create a netlist. This message is flagged even if the design is open in the Capture window.

To solve this error, make sure that the design file is not corrupted.

Legacy Message ID: MNL0020

ERROR[ORCAP-22008] - "Unable to find root schematic."

This error message appears in Capture 9.2.2 or earlier releases for all those designs which did not have a root schematic folder specified in the project manager.

Legacy Message ID: MNL0021

ERROR[ORCAP-22010] - "Unable to Open MNL Output File - %s"

This error message appears when Capture is unable to create or open the netlist file (.MNL) specified in the Netlist File 1 text box.

To resolve this error make sure that the path where the netlist file (.MNL) is getting generated exists.

Legacy Message ID: MNL0022

ERROR[ORCAP-22011] - "%d errors found in file."

This error message appears only when errors are reported in the sessions log during netlist generation. This message lists the total number of errors reported in the sessions log.

To resolve this error, check the session log for errors and fix them in your design.

Legacy Message ID: MNL0023

ERROR[ORCAP-22014] - "Layout DB Error code %d, '%s'."

This error message appears due to some problem in the Layout database. Make sure that your design does not contain parts whose PCB Footprint value is greater than 255 characters.

For more information, see the OrCAD Layout Online Help.

Legacy Message ID: MNL0025

WARNING[ORCAP-22015] - "String too long -- will be truncated. Original string: %s Truncated string: %s"

This warning messages appears when the string size of is more than the limit.

Legacy Message ID: MNL0001

ERROR[ORCAP-22016] - "'Layout.ini' missing from dsn2mnl's directory."

This error message appears when the LAYOUT.INI file is not present in {your_install_dir}\tools\capture directory. A copy of LAYOUT.INI must be located in this directory in order for Capture to generate a netlist or perform forward annotation to Layout. The Layout netlist dynamic link library, dsn2mnl.dll, uses the Windows Registry and other search methods to locate LAYOUT.INI.

To resolve this error, copy the LAYOUT.INI from the Layout directory and paste it in {your_install_dir}\tools\capture directory.

Legacy Message ID: MNL0002

ERROR[ORCAP-22018] - "Unknown LAYERS name '%s' for net '%s' Please use standard three-letter nicknames (TOP, BOT, IN1, etc.)"

This error message appears when your design has a net with an incorrect layer name. The layer names must use the standard three-letter nicknames (like TOP, BOT, IN1, etc.). Layout supports up to 30 routing layers plus Layer "0" to signify all layers.

To resolve this error, specify the correct Layer name for the net.

Legacy Message ID: MNL0004

ERROR[ORCAP-22019] - "Unknown VIAPERNET name '%s' for net '%s' Use standard via names (VIA1, VIA2, etc.)"

This error message appears when your design has a net with an incorrect name for Via type. The names for the Via types must use the standard three-letter nicknames (like VIA1, VIA2, etc.). Layout supports up to 30 routing layers plus Layer "0" to signify all layers.

To resolve this error, specify the correct Via name for the net.

Legacy Message ID: MNL0005

ERROR[ORCAP-22020] - "Unknown WIDTHBYLAYER syntax ('%s') for net %s"

This error message appears when the WIDTHBYLAYER syntax for one or more layers is incorrect for a net. The WIDTHBYLAYER property defines the net width for one or more layers.

Example of a correct syntax would be:

"WidthByLayer" "TOP=7,BOT=12" 0

To resolve this error, specify the correct WIDTHBYLAYER syntax for the net.

Legacy Message ID: MNL0006

ERROR[ORCAP-22021] - "Unknown WIDTHBYLAYER layer name ('%s') for net '%s'. Use standard three-letter nicknames (TOP, BOT, IN1, etc.)"

This error message appears when your design has a net with an incorrect WIDTHBYLAYER property name. The WIDTHBYLAYER property defines the net width for one or more layers. The name for the WIDTHBYLAYER must use the standard three-letter nicknames like (TOP, BOT, IN1, etc.),

To resolve this error, specify the correct WIDTHBYLAYER name for the net.

Legacy Message ID: MNL0007

ERROR[ORCAP-22022] - "Bad WIDTHBYLAYER value ('%s') for net '%s'."

This error message appears when your design has a net with a WIDTHBYLAYER property value as "0". The WIDTHBYLAYER property defines the net width for one or more layers. The WIDTHBYLAYER value cannot be 0.

To resolve this error, specify a value > 0 or WIDTHBYLAYER for the net.

Legacy Message ID: MNL0008

WARNING[ORCAP-22024] - "NETWEIGHT must be between 0 and 100 for net '%s'."

This warning message appears when your design has a net with NETWEIGHT property value specified as -1 or greater than 100. The NETWEIGHT property is used to assign relative priority to the net. The default value is 50.

Legacy Message ID: MNL0010

WARNING[ORCAP-22025] - "NETGROUP must be between 0 and 255 for net '%s'."

This warning message appears when your design has a net with NETGROUP property value specified as -1 or greater than 255. The NETGROUP property is used to identify grouped nets.

Legacy Message ID: MNL0009

WARNING[ORCAP-22026] - "MINWIDTH is greater than nominal WIDTH for net '%s'."

This warning message appears when your design has a net with MINWIDTH property value greater than the WIDTH property value. The MINWIDTH property is used to set the minimum track width. The WIDTH property is used to track width value specified for MINWIDTH and MAXWIDTH properties.

Legacy Message ID: MNL0011

WARNING[ORCAP-22027] - "MAXWIDTH is less than nominal WIDTH for net '%s'."

This warning message appears when your design has a net with MAXWIDTH property value less than the WIDTH property value. The MAXWIDTH property is used set the maximum track width.The WIDTH property is used to track width value specified for MINWIDTH and MAXWIDTH properties.

Legacy Message ID: MNL0013

WARNING[ORCAP-22028] - "COMPGROUP must be between 0 and 255 for comp '%s'."

This warning message appears when a part in your design has a COMPGROUP property value specified as -1 or greater than 255. The COMPGROUP property is used to assign a part to a group for placement.

Legacy Message ID: MNL0014

WARNING[ORCAP-22029] - "MINWIDTH is greater than MAXWIDTH for net '%s'"

This warning message appears when your design has a net with MINWIDTH property value greater than the MAXWIDTH property value. The MINWIDTH property is used to set the minimum track width. The MAXWIDTH property is used set the maximum track width.

Legacy Message ID: MNL0012

ERROR[ORCAP-36001] - "Conflicting values of Source Part names found on %s(%s) and %s(%s) for part "%s"."

This situation should only occur if you use the DEVICE property. You can change one or more of the DEVICE properties to make the Part Name (primitive line) unique for each distinct part in the PSTCHIP.DAT file.

The part name, which is the value found between single quotation marks in primitive line of the PSTCHIP.DAT file, must be unique for every different type of part in the design.

The part name is just the value of the DEVICE property present. If there is no DEVICE property on the part, then the part name is a made by combining the values of the Source Package, PCB Footprint, Value, and other properties that may be found in the [ComponentDefinitionProps] section of the configuration file. The part name string is a concatenation of these properties, with each value separated by an underscore character.

For more information, see the OrCAD Capture User's Guide.

Legacy Message ID: ALG0011

ERROR[ORCAP-36002] - "Property "PCB Footprint" missing from instance %s: %s, %s %s."

A PCB Footprint (JEDEC_TYPE in PCB Editor) is required for all parts in PCB Editor. Therefore all parts without this property are listed before aborting the netlisting. You can add the PCB Footprint property by selecting the part listed, then choosing Edit Properties from the pop-up menu and placing a value, such as dip14_3, on the part.

Legacy Message ID: ALG0012

ERROR[ORCAP-36003] - "Conflicting values of following %s properties found on %s %s."

For multi-section parts, all sections must have the same values for the properties listed under [ComponentInstanceProps] in the configuration file. For example, ROOM is a component instance property, so if you add {ROOM} to the combined property string when you annotate, then sections with differing ROOM properties will not be packaged together.

You can separate or combine component instances in a multi-section parts just by specifying distinguishing properties in the combined property string. Check the configuration file to identify the component instance properties currently available.

For more information, see the OrCAD Capture User's Guide.

Legacy Message ID: ALG0013

ERROR[ORCAP-36004] - "Conflicting values of part name found on different sections of "%s". Conflicting values: "%s" "%s" Property values of "Device","PCB FootPrint", "Class" and "Value" should be identical on all sections of the part."

Property values of DEVICE, PCB Footprint, Class, Value, Source Package and other properties found in the [ComponentDefinitionProps] of the configuration file should be identical for all sections of the part. To correct the problem, make sure the part name is identical on all sections of a multi-section part.

The primitive line of PSTCHIP.DAT has a value in single quotation marks that is referred to as the part name. It is composed of the DEVICE property or PCB Footprint, Value, Source Package and

the values of other component definition properties.

There is a 31 characters limit for the part name. So, if the part name is not unique, the netlister generates a unique name by appending _1 (underscore plus the 1 character). This final digit is incremented until a unique name is formed.

For more information, see the OrCAD Capture User's Guide.

Legacy Message ID: ALG0014

WARNING[ORCAP-36005] - "Net "%s" is renamed to "%s"."

A net name alias has a length limit of 31 characters. If the name exceeds this limit, it will be automatically shortened to 31 characters. The rules for net names may be found in the PCB Editor topic in the OrCAD Capture User's Guide.

Legacy Message ID: ALG0015

WARNING[ORCAP-36006] - "Part Name "%s" is renamed to "%s"."

Part name has a length limit of 31 characters. Part name is either the DEVICE property value or a combination of Source Package, PCB Footprint and other properties found in the [ComponentDefinitionProps] section of the configuration file. The DEVICE property is only used if it is present on the part.

If the name exceeds the 31-character limit, the Value property is automatically shortened to obtain the 31 character limit. The rules for renaming parts may be found in the PCB Editor topic in the OrCAD Capture User's Guide.

Legacy Message ID: ALG0016

WARNING[ORCAP-36007] - "Ignoring new Component "%s" found on board."

This warning could be caused by a part that has been added on the PCB Editor board but which is not present on the Capture schematic. Another reason could be that design file/root schematic/hierarchical block name got changed after board file generation.

Legacy Message ID: ALG0017

WARNING[ORCAP-36008] - "Ignoring Component "%s" missing from board."

This warning could be caused by a part that was deleted from the PCB Editor board but which exists on the Capture schematic.

Legacy Message ID: ALG0018

WARNING[ORCAP-36009] - "Ignoring new Net "%s" found on board."

A net added on the PCB Editor board is not present on the Capture schematic. If a net name was changed on the schematic, then, when back annotating, the old net name is no longer found and is seen as a new net.

Deleting a net on the design and then adding it again will result in a newly-generated net name if there is not an alias. So, it is a good idea not to change your schematic design at all in Capture before back annotating from PCB Editor.

For more information, see the OrCAD Capture User's Guide.

Legacy Message ID: ALG0019

WARNING[ORCAP-36010] - "Ignoring Net "%s" missing from board."

A net exists on the Capture schematic but does not exist on the PC Editor board. A net might have been deleted on the PCB Editor board.

Legacy Message ID: ALG0020

ERROR[ORCAP-36011] - "Net "%s" renamed to "%s" on the board."

A schematic net has been renamed on the board. This is usually done by modifying the SPECCTRA netlist.

Legacy Message ID: ALG0021

ERROR[ORCAP-36012] - "Property "%s" on Pin "%s" of "%s" updated. Schematic Value : "%s" Board Value : "%s" "

Pin properties are not back annotated to Capture. This warning is issued so that you know a pin property has been changed in PCB Editor. You may or may not want to update the pin property manually in Capture to match.

Legacy Message ID: ALG0022

WARNING[ORCAP-36013] - "Property "%s" on Part "%s" is deleted from board."

A part property was deleted on the board in PCB Editor.

Legacy Message ID: ALG0023

WARNING[ORCAP-36014] - "Property "%s" on Net "%s" is deleted from board."

A net property was deleted from the board in PCB Editor.

Legacy Message ID: ALG0024

WARNING[ORCAP-36015] - "Illegal Reference change from "%s" on schematic to "%s" on board."

An error was encountered during back annotation. This error occurs when you change the format of a reference designator for a part in PCB Editor to an invalid format. Make sure you do not change the format for reference designators as <Alphabet(s)><Numeric><Alphabet(s) or <Alphabet(s)>-<Alphabet(s)>.

Legacy Message ID: ALG0025

WARNING[ORCAP-36016] - "Connectivity Change for Pin "%s" of "%s". Schematic Net: "%s" Board Net : "%s" "

A net name was changed in Capture without netlisting afterwards and importing the logic changes into PCB Editor. In this case, if you back annotate in Capture, Schematic Net and Board Net will have different names.

To avoid this warning, it is a good idea not to change your design in Capture until you are done back annotating changes from PCB Editor. Just deleting a net and re-drawing a net will result in a new generated net name.

Legacy Message ID: ALG0026

WARNING[ORCAP-36017] - "Test point on Net "%s" ignored for Part %s: %s, %s %s."

Either the pin number or node name was not available for this port. It is most likely considered a test point and therefore not part of the netlist.

Legacy Message ID: ALG0027

ERROR[ORCAP-36018] - "Aborting Netlisting... Please correct the above errors and retry."

There were fatal error(s) encountered during netlisting that caused the netlister to abort. See the Capture session log for details. There is also a netlist.log file in the same directory as the PST*.DAT files which contains netlisting information that gets written to the session log.

Legacy Message ID: ALG0028

ERROR[ORCAP-36019] - "Unable to open file "%s" for writing."

The listed file is set to read-only. If you want to use this file, you can change the file properties by unchecking the Read Only option.

Legacy Message ID: ALG0029

ERROR[ORCAP-36020] - "Unable to read design "%s"."

If the board specified in the PCB Editor Board File field of the Back Annotate dialog box doesn't exist in the path specified you will get this error. Check to make sure the .BRD file is there or browse to your file directories to be sure you have the right path.

For more information, see the OrCAD Capture User's Guide.

Legacy Message ID: ALG0030

ERROR[ORCAP-36021] - "Pin number "0" found on Pin "%s" of Package %s , %s: %s, %s %s. Pin number should be greater than "0"."

PCB Editor does not support pin numbers equal to 0 (zero). Change the pin number by editing the part, then the pin. All parts with pin numbers equal to zero are listed before the netlister aborts. Look in the session log for details about which pins must be changed.

Legacy Message ID: ALG0032

ERROR[ORCAP-36024] - "Unable to open file "%s" for reading."

This error message occurs when the configuration file or board file specified in the Create Netlist dialog box or the Back Annotate dialog box does not exist in the path specified or is corrupted. You can go to the Setup dialog box and click on the edit button to verify which problem you have. To do so, you browse to a valid configuration or board file.

A default configuration file (allegro.cfg) was installed with Capture in the Capture subdirectory. If you have deleted it you can recreate it by writing a text file with the contents from the configuration file Help topic and save the file with a .CFG extension.

Legacy Message ID: ALG0034

ERROR[ORCAP-36025] - "Aborting Swap file creation... Please correct the above errors and retry."

There were fatal error(s) encountered during swap (.SWP) file creation. See the Capture session log for details. There is also a swap.log text file that contains the same errors that were found in the session log at the time the last back annotation was run.

Legacy Message ID: ALG0035

ERROR[ORCAP-36026] - "Unable to read logical netlist data."

There was an error encountered while reading PST*.DAT files. This error is probably caused by invalid characters in the PST*.DAT files. You have probably modified your netlist files since importing logic to PCB Editor. Remember that even things like deleting a net that doesn't have a net alias and then re-drawing it will result in a new net name since net names are auto generated.

Legacy Message ID: ALG0036

ERROR[ORCAP-36027] - "Unable to read physical netlist data."

This error message appears when Capture encounters an error while reading the *VIEW.DAT files. The possible causes of this error are:

- Invalid character in *VIEW.DAT files
- *VIEW.DAT files are not stored in the same location as pst*.dat files
- PST*.dat and *VIEW.DAT files do not correspond to the same design
- Illegal characters in reference designators or net
- Changes made to a part in Capture design after board file creation. This can happen because

of running the Replace Cache or Edit part commands.

- One or more values of the component definition properties got changed after board file creation
- Some components got deleted from Capture design after board file creation
- Design file/root schematic/hierarchical block names got changed after board file creation
- Netlist files not readable because either the path to the file is long or has spelling errors.

For more information, see the OrCAD Capture User's Guide.

Legacy Message ID: ALG0037

ERROR[ORCAP-36028] - "Unable to create logical-physical difference data."

There was an error encountered while creating data for swap (.SWP) file writing. Back annotation generates new netlist files (PST*.DAT), compares them with the *VIEW.DAT board files, and then generates a swap (.SWP) file with the differences. Look for errors in swap.log netlist.log files or the in the session log, or consider reasons why one step in this back annotation process might have failed.

Legacy Message ID: ALG0038

ERROR[ORCAP-36029] - "No components to package in design."

Either no components are placed on the schematic or all placed components are to be excluded from the netlist. For example, if all components are assigned the PSpiceOnly property, there would be no netlist. To eliminate this error, you must use components besides those with just the PSpiceOnly property when creating an PCB Editor netlist.

Legacy Message ID: ALG0039

WARNING[ORCAP-36030] - "Value of Property "%s" missing from instance %s: %s, %s %s."

This error is reported when a property value needed for netlisting is NULL (the occurrence value is blank or empty but there is an instance value).

To fix the problem you should edit the part and enter an occurrence value for the property. If there is an instance value, you can also delete the occurrence value which will make the instance property "shine through" to the occurrence in the property editor because a child occurrence inherits the

property of its parent instance if no property value exists on the occurrence.

The yellow values are the occurrence values in the property editor. To make the instance shine through, you have to select the occurrence cell (appropriate row and column) in the property editor and then right mouse click and select delete property. Backspacing or just deleting the characters in the occurrence cell one at a time won't work.

Legacy Message ID: ALG0040

WARNING[ORCAP-36031] - "Part Reference missing from part with Value %s: %s, %s %s."

Part Reference is a combination of reference and designator. Make sure that the designator is not a question mark (?). If it is, you need to annotate your design. You may need to check occurrence mode when annotating if your instance designators have values but your occurrence designators do not.

Legacy Message ID: ALG0041

ERROR[ORCAP-36032] - "Duplicate Reference Designator %s: %s, %s %s."

To avoid this error, run the DRC report to check for duplicate Reference Designators before creating the PCB Editor netlist. Duplicate part references are reported in the session log. Proper annotation will avoid or fix this error. You can unconditionally annotate your design or manually change one of the duplicated part references.

Legacy Message ID: ALG0042

ERROR[ORCAP-36033] - "Net named "NC" found on the design. Please rename the net."

PSTXNET.DAT uses a net named nc to specify where all the unconnected or no-connect pins no-connect pins are for PCB Editor. So, do not create a net with an alias of NC or all those pins connected to the net will get a NC property on them in PCB Editor and therefore won't be a net there.

Legacy Message ID: ALG0043

ERROR[ORCAP-36034] - "Conflicting values of "Power Pin Visibility" found on different sections of %s: %s, %s %s"

Power pins should be either visible or invisible for all occurrences in a reference; mixed visibility is not allowed in a reference. You can fix this problem by selecting the part, choosing Edit Part from the pop-up menu, and making the power pins visible.

Legacy Message ID: ALG0044

ERROR[ORCAP-36035] - "Multiple pin %s's which have different nets connected for %s: %s, %s %s."

A part with duplicated pins, whether the part be on the same section or in a different section of a package, cannot be connected to different nets. For example, if you have a part with two power pins named VCC and you connect one of the pins to ground and the other to VCC or VEE then you get this error. Another reason could be that the design is not annotated.

Make sure the same net goes to all pins on the part having the same name. Remember that in Capture, two nets not visually connected together but with the same net alias name are connected electrically. Check all sections for multi-section parts, especially if the sections are on different pages.

Legacy Message ID: ALG0045

ERROR[ORCAP-36037] - "Duplicate Pin Number "%s" found on "%s" and "%s" on Package %s , %s: %s, %s %s. Please rename one of these."

Two pins cannot share the same pin number. To fix this problem, select the identified part, then choose Edit Part from the pop-up menu. Choose Package from the View menu, then select the desired component and change its pin numbers so they are unique. Alternatively you could change the part in the library and do a replace cache (with preserve option).

Legacy Message ID: ALG0046

WARNING[ORCAP-36038] - ""No_connect" property on Pin "%s" ignored for %s: %s, %s %s. Connecting pin to net "%s"."

A pin has the Is No Connect property enabled (checked) but there is a net connected to it. Either remove the net or disable the Is No Connect property.

Legacy Message ID: ALG0047

ERROR[ORCAP-36039] - "Conflicting values of "Power Pin Visibility" on "%s" and "%s" for %s."

All sections of a multi-section part must have identical Power Pin Visibility values. These values can be changed in the property editor.

Legacy Message ID: ALG0048

ERROR[ORCAP-36040] - "Pin Number "%s" specified in "NC" property %s %s of Package %s , %s: %s, %s %s."

If a net is attached to a pin number that is also listed as value of a NC property, then you get this error. The NC property value should be made up of pin numbers separated by commas. Here are three cases:

If you have a part with six pins and all of them are connected to nets and you also have an NC property with value of "6,7,8" then you will get an error for pin 6.

If the NC property has a value of "18,19,18" then you'll get an error for pin 18.

If "NC" has "3" where 3 is also the pin number on O you'll get an error for pin O.

Legacy Message ID: ALG0049

ERROR[ORCAP-36041] - "Duplicate Pin Name "%s" found on Package %s , %s: %s, %s %s. Please renumber one of these."

A component contains more than one pin with the same logical name. You can fix this problem by selecting the part and choosing Edit Part from the pop-up menu. Choose Package from the View menu, then Properties from the Edit menu. Change the value in the Name Normal column to the desired pin name.

This error occurs if you have multiple ground or power pins with the same name but different pin numbers, and you make the power pins visible on the component. The PST*.DAT file format does not currently support this condition. However, it allows duplicate pin-names for invisible Power pins. So, you need to make the power pins invisible or give each one a different name, such as GND1 and GND2 instead of just GND.

To update a large number of parts, run the library correction utility on library files, to correct duplicate pin names. You can then use these libraries in creating new designs. After correcting the libraries you will not encounter the ALG0050 error.

Legacy Message ID: ALG0050

ERROR[ORCAP-36045] - "All pins are power %s: %s, %s %s"

This error occurs if all pins are power pins (effectively a zero pin component where no pins are found in the Pin section of the part in PSTCHIP.DAT file). PCB Editor can't handle this kind of part yet. If you encounter this error, edit the part to contain at least one non-power type pin.

Legacy Message ID: ALG0053

ERROR[ORCAP-36046] - "Conflicting number of parts per package (%i) for %s: %s, %s %s"

All the sections of the package with the reference <Ref Des> are not the same. It appears that there is a section packaged incorrectly with the rest of the sections, possibly something like a 74LS00 got packaged with a 74LS04 gate.

Legacy Message ID: ALG0054

ERROR[ORCAP-36047] - "Pin "%s" is invisible on %s: %s, %s %s. Only invisible Power Pins are allowed."

Other than power pins, only parts with visible pins are allowed. Either remove the invisible pin, or make it visible.

Legacy Message ID: ALG0056

WARNING[ORCAP-36048] - "Ignoring Pin Name "%s" given to invisible NC pin on %s: %s, %s %s."

An invisible pin with a no connect property on it is named something other than NC.

Legacy Message ID: ALG0057

WARNING[ORCAP-36049] - "Ignoring following Component Definition properties since they also exists as Component Instance properties in %s."

The configuration file, ALLEGRO.CFG (or other user-specified file), has a property listed in both the [ComponentInstanceProps] and the [ComponentDefinitionProps] sections. The component definition instance will be used when netlisting, rather than component definition property.

Legacy Message ID: ALG0059

WARNING[ORCAP-36050] - "No pins are present in %s. Ignoring this component in netlist."

This warning appears when a design contains a component without any pins on it. Such components could be mechanical parts which do not have any pins on them. These components are ignored during netlist generation.

Legacy Message ID: ALG0060

ERROR[ORCAP-36051] - "Value for property %s contains carriage return for net %s."

This error occurs because the device value for the instance is the same as the schematic name.

Change the value of the DEVICE property on the instance or rename the schematic.

Legacy Message ID: ALG0063

ERROR[ORCAP-36052] - "Value for property %s contains carriage return for %s."

This error occurs because the device value for the instance is the same as the design name.

Change the value of the DEVICE property on the instance or rename the design.

Legacy Message ID: ALG0062

ERROR[ORCAP-36054] - "Schematic name is same as Device value for instance %s: %s, %s %s. Please change one of them."

The design name, schematic name, part name or property value has one of the following illegal characters. Please remove the illegal character.

- (
-)
- @
- :

[*Legacy Message ID: ALG0063*

WARNING[ORCAP-36060] - "Ignoring PIN_GROUP on pin "%s" as it is part of PACK_SHORT property for part "%s"."

If the pin_group property is added on any pin that is part of the PACK_SHORT property, it will be ignored while allegro netlisting.

Legacy Message ID: ALG0070

ERROR[ORCAP-36061] - "Syntax error found in PACK_SHORT property on part "%s". One or more parentheses are incorrectly placed."

Incorrect parentheses in the property value. E.g: (IN1,O4,O5) (IN2,O6.

Legacy Message ID: ALG0071

ERROR[ORCAP-36062] - "Syntax error found in PACK_SHORT property on part "%s". One or more commas are incorrectly placed."

Incorrect use of commas. E.g: (IN1,O4,O5) (IN2 O6)

Legacy Message ID: ALG0072

ERROR[ORCAP-36063] - "Duplicate pin "%s" found in PACK_SHORT property on part "%s". You need to uniquely specify the pins that you want to short."

Duplicate pin-name in PACK_SHORT property value. E.g = (IN1,O4,O5) (IN2,O4,O6)

Legacy Message ID: ALG0073

ERROR[ORCAP-36065] - "Pins "%s" and "%s" defined in PACK_SHORT property on part "%s" are connected to two different nets "%s" and "%s"."

Incorrect pin-name which doesnt exist on symbol. E.g = (IN1,O4,O5) (IN2,O6,O51)

Legacy Message ID: ALG0074

ERROR[ORCAP-36066] - "An invisible Power pin "%s" found in PACK_SHORT property on part "%s". Only visible pins can be part of PACK_SHORT property"

Invisible Power pins cant be part of PACK_SHORT property

Note: If you want to use Power/GND pins in PACK_SHORT property then you need to make them visible on the symbol and if you dont want to show them on the symbol, make them PACK_IGNORE.

Legacy Message ID: ALG0076

ERROR[ORCAP-36067] - "First pin "%s" found in PACK_SHORT property value "%s" is not visible on the symbol for part "%s".First pin should be visible on the symbol to evaluate net connectivity correctly."

First pin in PACK_SHORT property should be visible on the symbol to evaluate net connectivity correctly

Legacy Message ID: ALG0077

ERROR[ORCAP-36068] - "Identical Name property value "%s" found on Hierarchical block instances %s: %s, %s %s and %s: %s, %s %s. The value should be unique for correct netlisting."

This error appears in case the "Name" property value is identical for one or more instances of a hierarchical block referring to same schematic view. For example, if you have placed two instances of Halfadder block say H1 and H2 in your top level design, the Name property value should be unique for each of the instances say Name = HalfaddedI nst1 for H1 instance and Name = HalfaddedInst2 for H2 instance.

Legacy Message ID: ALG0078

ERROR[ORCAP-36070] - "Inconsistent pin-number values found for pin "%s" in Normal and Convert views of component instance %s: %s, %s %s ."

This error occurs if the pin numbers on the Normal and Convert views of a component instance do not match. To correct this error, you need to edit the pin-number values of pins of Convert view in sync with that of Normal view.

Legacy Message ID: ALG0080

ERROR[ORCAP-36071] - "Illegal character "%s" found in "%s" property for component instance %s: %s, %s %s ."

PCB Editor does not support Dots(.), Forward Slash(/) and White space in footprint names. The supported characters include Alphabets, Numerics, Underscore(_) and Hyphen(-).

Legacy Message ID: ALG0081

ERROR[ORCAP-36074] - "Unable to get information for %s from netlist file."

This error is most likely caused when a part in the physical layout is deleted from the schematic page.

Legacy Message ID: ALG0058

WARNING[ORCAP-17035] - "No physical wires found for signal %s. DIFFERENTIAL_PAIR property was not created."

This warning appears if the user tries to add a property on bus members whose wires do not exist in the Instance mode.

Legacy Message ID: DBO3853

WARNING[ORCAP-21004] - "Unable to load the primary SDT Configuration file: '%s'."

This error occurs when Capture attempts to read SDT.CFG and encounters a problem. Check that the SDT.CFG file exists. Also make sure it isn't corrupt. Sample SDT.CFG files can be found in SDT.CFG_files.

When translating an SDT design into Capture, an SDT.CFG file or an SDT.BCF file (Release IV) must exist in one of the following places:

- The current directory.
- The directory that contains the design.
- The directory specified by the ORCADPROJ environment variable.

When translating a Capture design into an SDT schematic, an SDT.CFG file must exist in the same directory as your design or in the directory specified by your ORCADPROJ environment variable.

Legacy Message ID: XLT0012

ERROR[ORCAP-21009] - "Translation cancelled."

Translation of the design or library was canceled before completion by the user. The design or library was not translated as a result.

Legacy Message ID: XLT0057

WARNING[ORCAP-21010] - "Window's temporary environment variable was not found. Using design directory."

This error occurs when Capture cannot find the Window's TEMP environment variable during translation. Capture creates its temporary file in the design's specified directory as a result. To correct this problem, check to see if the TEMP environment variable is specified in your AUTOEXEC.BAT file. If it isn't, add the following line and restart Windows:

SET TEMP=C:\WINDOWS\TEMP\

[*Legacy Message ID: XLT0062*

WARNING[ORCAP-21011] - "Window's environment variable '%s' is not set."

This error occurs when Capture requires the environment variable to be set. Check your AUTOEXEC.BAT file to see if the environment variable has been set. If the environment variable isn't set, add any necessary lines to the AUTOEXEC.BAT file, and restart Windows.

Legacy Message ID: XLT0063

WARNING[ORCAP-21013] - "No SDT or Capture libraries were found. Please check the SDT configuration file."

Capture could not find any of the libraries specified in the SDT.CFG file. Check that the 'PLIB =' line is set to the correct directory in the SDT.CFG file. If it is set correctly, then check that there is at least one library specified in the SDT.CFG, and that all libraries specified in the file are located in the directory indicated by the 'PLIB =' line.

Legacy Message ID: XLT0074

ERROR[ORCAP-21014] - "Skipped design without view '%s'"

This error occurs when Capture attempts to translate a design that has no schematics. Check the design for schematics.

Legacy Message ID: XLT0089

ERROR[ORCAP-21015] - "Skipped heterogeneous part '%s'."

This error occurs when Capture encounters a heterogeneous part during translation. Heterogeneous parts are not supported in SDT 386+. Capture doesn't translate the part. You should use Design Rules Check and specify Check SDT compatibility before translating a design or library back to SDT, and correct any incompatible objects.

Legacy Message ID: XLT0091

WARNING[ORCAP-21018] - "Invalid Library Prefix in SDT.CFG, directory '%s' does not exist."

This error occurs when Capture detects an invalid directory path in the SDT.CFG file. Check that the directory specified by the PLIB line in your SDT.CFG file exists on your system. This line should be set to the directory that contains your SDT libraries.

Legacy Message ID: XLT0101

ERROR[ORCAP-21022] - "'%s', the run time library, cannot be accessed or created."

This error arises when Capture cannot find or create CAPSYM.OLB.

Legacy Message ID: XLT0079

ERROR[ORCAP-21023] - "SDT Release IV translator cannot find required EXE file: '%s'"

This error occurs when Capture is unable to find the EXE file required to complete the translation. Locate the file and move it to the same directory as CAPTURE.EXE.

Legacy Message ID: XLT0080

ERROR[ORCAP-21024] - "SDT Release IV translator cannot find SDT.BCF or SDT.CFG files in the design directory or the TEMPLATE directory. Please provide one."

This error occurs when Capture cannot find either an SDT.BCF or SDT.CFG file in the design directory or the TEMPLATE directory. Capture requires one of these files for a sample during translation. Locate one of these files and copy it into either the design directory or the TEMPLATE (destination) directory.

Legacy Message ID: XLT0081

ERROR[ORCAP-21025] - "Cannot change directory to: '%s'"

This error arises during translation of an SDT Release IV design (or library), and Capture cannot save to the specified directory. Check the directory's properties to make sure it isn't a read-only directory.

Legacy Message ID: XLT0082

ERROR[ORCAP-21026] - "Cannot find: '%s'"

This error arises when Capture cannot find the file SDT.CFG.

Legacy Message ID: XLT0083

ERROR[ORCAP-21027] - "Cannot change to drive: '%s'"

This error occurs when Capture cannot change to the specified drive to save the design or library.

Legacy Message ID: XLT0084

ERROR[ORCAP-21028] - "Batch process failed. Please check the Session Log."

This error occurs when Capture is unable to translate the file due to a problem with one of the translation programs.

Legacy Message ID: XLT0086

ERROR[ORCAP-21029] - "Skipped design without root '%s'"

This error arises when Capture attempts to translate a design to SDT format, but the design has lost its root schematic. For example, this message is generated if you delete the root schematic, and then attempt to save the design in SDT format. Check that all designs have a root schematic before translating.

Legacy Message ID: XLT0088

WARNING[ORCAP-21055] - "File does not exist '%s'."

Capture could not find the specified file. If the library file exists, move it to the directory specified by the 'PLIB =' line in the SDT.CFG file. If the library does not exist, or is not used in the design, then remove the 'LIB =' line for that library.

Legacy Message ID: XLT0018

ERROR[ORCAP-21057] - "Decompiling the SDT/IV library file failed."

This error occurs when Capture is unable to translate the library back to a Release IV library. Translate the Capture library into an SDT 386+ library. Using the schematic, run DECOMP.EXE to obtain an ASCII library source file. Finally, run Release IV's COMPOSER.EXE to create the Release IV library.

Legacy Message ID: XLT0020

WARNING[ORCAP-21084] - "Creation of a new alias object '%s' at %.2f, %.2f failed, wire not found. Label converted to comment text."

A label in the SDT design wasn't connected to a wire. Labels are only translated as aliases, if there are wires under the labels. If a wire doesn't exist under the label, then the label is translated as text.

Legacy Message ID: XLT0048

WARNING[ORCAP-21091] - "Creation of a junction object at %.2f, %.2f failed, wire not found."

This situation arises when a junction wasn't placed on either a wire or a bus. Capture doesn't translate the junction. Junctions must be placed on nets in Capture.

Legacy Message ID: XLT0056

ERROR[ORCAP-21093] - "Cannot replace '%s', already in use."

This error occurs when Capture attempts to save the file while it is in use by another application. Close the file out of all other applications and try translating again.

Legacy Message ID: XLT0059

ERROR[ORCAP-21094] - "Cannot save '%s' onto itself."

This error occurs when Capture attempts to save a translated file using the same filename and path as the original design. Save the file under a different name, path, or both.

Legacy Message ID: XLT0060

WARNING[ORCAP-21096] - "Cannot find part '%s' at %.2f, %.2f."

This message arises when Capture cannot find a part while translating from SDT to Capture. For example, this message is generated when a part in an SDT file is corrupted. Check the files for corrupted parts. This situation may also arise when Capture cannot find the library containing the parts. This is related to [XLT0018].

Legacy Message ID: XLT0064

WARNING[ORCAP-21097] - "Bus entry was replaced with a bus wire at %.2f, %.2f to %.2f, %.2f."

This warning arises when a bus uses a bus entry to turn a corner in an SDT design. Capture replaces the bus entry with a bus.

Legacy Message ID: XLT0065

ERROR[ORCAP-21102] - "Translation failed."

One or more errors occurred during the translation, causing the translation to be unsuccessful. Check the session log for more information.

Legacy Message ID: XLT0070

ERROR[ORCAP-21103] - "Translation Notice. Error(s) encountered. Please check the Session Log."

One or more errors or warnings occurred during the translation. Check the session log for more information.

Legacy Message ID: XLT0071

WARNING[ORCAP-21104] - "Translation Notice. Warning(s) encountered. Please check the Session Log."

This situation arises when Capture encountered translation problems. You should check the session log for more information.

Legacy Message ID: XLT0072

WARNING[ORCAP-21105] - "Duplicate timestamp encountered on '%s' at %.2f, %.2f. A unique timestamp has been generated."

Two parts with identical timestamps were detected. Capture automatically creates a new timestamp for the second part. No action needs to be taken.

Legacy Message ID: XLT0075

ERROR[ORCAP-21120] - "Unrecoverable error: design lost its view '%s'"

This error arises when an internal problem occurs during translation. Try translating the design or library again.

Legacy Message ID: XLT0090

I[ORCAP-21121] - "PartField '%s' in SDT.CFG exceeds SDT length limit. String truncated."

This error occurs when the specified part field in the SDT.CFG file exceeds the limit set for SDT 386+. Capture truncates the part field to fit within the limit.

Legacy Message ID: XLT0092

WARNING[ORCAP-21122] - "A wire auto-connection occurred. Please examine point (%.2f, %.2f) on page: %s"

This warning occurs when Capture auto-connects a wire to another wire or part during translation. Check the indicated wire to see if the connection is correct.

Legacy Message ID: XLT0093

ERROR[ORCAP-21123] - "Release IV Library Translation failed. DOS Prompt did not log translation errors. Please run: COMP16.EXE %s %s"

This error arises when Capture fails to completely translate a library to SDT Release IV. Complete the translation by running COMP16.EXE on the specified file.

Legacy Message ID: XLT0094

WARNING[ORCAP-21124] - "Can't find source package %s for part at %s."

This warning occurs when Capture is unable to find one of the library files listed in the configuration file. Check all the schematic pages in the design to make sure the translation was complete. If parts are missing from the design, locate the indicated library, and move it to the specified directory.

Legacy Message ID: XLT0095

WARNING[ORCAP-21125] - "'%s' not found."

This error arises when Capture cannot find a library listed in the configuration file. Check SDT.CFG, PLDSDT.CFG, and VSTSDT.CFG (as appropriate) for the library. If it isn't present in the required configuration file, add it.

Legacy Message ID: XLT0096

WARNING[ORCAP-21133] - "Creation of a noconnect property for a pin at %.2f, %.2f failed, pin not found."

The SDT design contained a noconnect object that wasn't connected to a pin. In Capture, no connects are pin properties. In SDT, noconnects are objects. As a result, noconnects are not translated unless they are connected to the end of pins on parts.

Legacy Message ID: XLT0107

WARNING[ORCAP-21135] - "Title block text '%s' was altered by the conversion to the ANSI character set."

A character in the title block text text was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "micro" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0112

WARNING[ORCAP-21136] - "Sheet filename '%s' on sheet at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the sheet filename was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0113

WARNING[ORCAP-21137] - "Sheet name '%s' on sheet at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the sheet name was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0114

WARNING[ORCAP-21138] - "Part reference '%s' on part at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the part reference was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0115

WARNING[ORCAP-21139] - "Part value '%s' on part at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the part value was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0116

WARNING[ORCAP-21140] - "Part field string '%s' on part at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the part field was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0117

WARNING[ORCAP-21141] - "Part filename '%s' on part at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the part filename was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0118

WARNING[ORCAP-21142] - "Label '%s' at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the label was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0119

WARNING[ORCAP-21143] - "Text '%s' at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the text string was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0120

WARNING[ORCAP-21144] - "Module port '%s' at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the module port was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0121

WARNING[ORCAP-21145] - "Power object '%s' at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the power object was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0122

WARNING[ORCAP-21146] - "Property '%s' at %.2f, %.2f was altered by the conversion to the ANSI character set."

A character in the property was altered when Capture converted it from OEM to ANSI. For example, this warning occurs when the character "MICRO" is converted from OEM character 230 to ANSI character 181. Some characters won't convert. Check that the specified sting of text converted correctly. If the string did not convert correctly, change the characters as appropriate.

Legacy Message ID: XLT0123

I[ORCAP-21165] - "'%s' is expected to be in SDT 386+ format."

This error occurs when Capture expects the selected file to be in SDT 386+ format, but the file is of another format. Check to see if you selected the correct file, or that the file isn't corrupt.

Legacy Message ID: XLT0076

ERROR[ORCAP-21166] - "Failed allocating sufficient memory '%s'"

This error occurs when Capture fails to create a buffer for translating a design back to SDT, due to a lack of memory.

Legacy Message ID: XLT0087

5

ORDBLL Messages

ERROR[ORDBDLL-1012] - "SHARE.EXE not loaded or shared region is locked"

Capture was unable to perform an operation because SHARE.EXE was not loaded, or a shared region was locked. Make sure that you are using SHARE.EXE if you are running Windows 3.11. Also check that that any files Capture is using are not also open in another application. If a file is open in another application, close that file.

Legacy Message ID: DBO3211

ERROR[ORDBDLL-1023] - "Attempt to read file with invalid file format"

This error appears when a design created in Capture 10.0 or later version has pin name and number text movement and is opened in Capture 9.2.3 or earlier version. Make sure that you select the Remove Pin Name and Number Movement check box in the Save As dialog box while saving your design in Capture 10.0 or later versions. When you save your design with this check box selected, Capture removes all the pin name and number movements you have done in the design.

Legacy Message ID: DBO3220

ERROR[ORDBDLL-1024] - "The Design has no specified Root View"

Capture was unable to complete the operation because the design has no specified Root View. for example, this error occurs if you delete the root schematic in logical mode and then attempt to change to physical mode. Make sure that a schematic has been assigned as the root schematic before attempting to change to physical mode.

Legacy Message ID: DBO3221

ERROR[ORDBDLL-1026] - "A Package with this name already exists"

Capture was unable to complete the operation because a package with the specified name already exists in the active library. Try using a different name, or saving the part to a different library.

Legacy Message ID: DBO3223

ERROR[ORDBDLL-1027] - "A View with this name already exists "

Capture was unable to complete the operation because a view with the specified name already exists. For example, this error occurs when you attempt to rename a schematic using a name already used by another schematic in the design.

Legacy Message ID: DBO3224

ERROR[ORDBDLL-1029] - "A Symbol with this name already exists"

Capture was unable to complete the operation because a symbol with the specified name already exists in the active library. Try using a different name for the symbol or save it to a different library.

Legacy Message ID: DBO3226

ERROR[ORDBDLL-1049] - "The specified Part is incompatible with the Device."

This warning message is displayed if a part is instantiated with two different reference designators. This is invalid in Capture. For example, a part MyPart is instantiated with two different reference designators; H1 and HT4. To solve this problem, copy the part to a new library from the Design Cache and then use Replace Cache to update the cache.

Legacy Message ID: DBO3246

ERROR[ORDBDLL-1051] - "Uprev Conversion Cancelled."

Capture was unable to complete the operation because a page with the specified name already exists. For example, this error occurs when you attempt to rename a schematic page to a name already in use by another page in the schematic.

Legacy Message ID: DBO3248

I[ORDBDLL-1123] - "Devices existing in the original package do not exist in its replacement. Instances of these devices have been reassigned to the 1st device in the package. References may now be duplicated."

This warning message appears, If you place all the sections of a part from a library, which has Parts per Package greater than 1 and then reduce the Parts per Package in the library. On using the Update Cache command, this warning message appears.

Legacy Message ID: DBO3881

ERROR[ORDBDLL-1125] - "The part being pasted is a different version than is currently in use in this design. Update or replace the parts in the cache to bring these versions in sync."

This error message appears when you copy and paste components between designs that are common to the two designs, but some of the parts are not up to date.

To resolve this error, make sure that the components you are pasting is up-to-date, by using the Update Cache command.

Legacy Message ID: DBO3883

ERROR[ORDBDLL-1219] - "Part Occurrence for %s Not Found; Please check if there is any recursive reference. "

This error message comes up when during incremental annotation Capture encounters parts (components) in your design, which are already annotated with part references that are outside the range specified in the grid. Make sure that the part reference range does not conflict with the parts already annotated in the design.

Legacy Message ID: ANN0010

WARNING[ORDBDLL-1223] - "Component %s%s has different common pin connection for two instances and hence packaged separately. "

When a common pin of two instantiated sections of a part has different net connections, Capture packages the two instances separately to eliminate design errors. This warning can be ignored if the connections are done intentionally.

Legacy Message ID: ANN0009

WARNING[ORDBDLL-1225] - "Properties specified in combined property string not found on part, '%s'. Using default combine property string {Value} for this part."

This warning message comes up when the properties specified in the combined property string are not found in a part while performing annotation. Capture assumes default combined property string in such case. User should check the combined property string that was put while annotating.

Legacy Message ID: ANN0006

WARNING[ORDBDLL-1226] - "Warning limit exceeded. Please check combined property string for invalid specification. "

This warning message comes up when there are too many warning messages, and the number exceeds the limit. Currently the limit is 10. User needs to check the messages and rectify. One situation while this message comes up is if the user has specified some properties as part of combined property string, and they are not available in any of the parts in the design.

Legacy Message ID: ANN0007

6

ORNET Messages

ERROR[ORNET-1006] - "Netlist failed or may be unusable."

Capture was unsuccessful in generating a netlist.

Legacy Message ID: NET0011

I[ORNET-1011] - "Invalid hierarchy type."

The check for a logical or physical design type failed. This error can occur when the design is corrupt.

Legacy Message ID: NET0014

ERROR[ORNET-1017] - "Unconnected pin, no FLOAT property or FLOAT = e"

This error message appears when you create a PSpice netlist for a design that contains unconnected pins and don't have the FLOAT property or FLOAT = e set in the User Properties dialog box.

To resolve this error, make sure that you specify FLOAT = e or FLOAT = u (unmodeled) in the User Properties dialog box.

Legacy Message ID: NET0071

WARNING[ORNET-1018] - "Connection to unmodeled pin"

This warning message appears when you create a PSpice netlist for a design that contains an unmodeled pin and it appears the PSpice template too.

It is recommended that you remove any unmodeled pins appearing in the PSpice template.

Legacy Message ID: NET0072

ERROR[ORNET-1020] - "Encountered reserved word '%s'"

This error arises when the netlister encounters the use of a reserved word for an object name in the schematic. Change the object's name.

Legacy Message ID: NET0048

ERROR[ORNET-1021] - "Design does not have a root schematic."

This error arises when the netlister cannot find the root schematic. For example, this message is generated if the root schematic is deleted from the design prior to creating the netlist. Choose Make Root Schematic from the Design menu in the design manager to set one of the schematics to the root.

Legacy Message ID: NET0050

ERROR[ORNET-1022] - "Duplicate reference found '%s'."

This error arises when the netlister encounters a duplicate reference in the design. Check the design for duplicate part references using Design Rules Check before creating the netlist.

Legacy Message ID: NET0051

ERROR[ORNET-1023] - "Part %s does not have a Value or Schematic (primitive hierarchical block). Make sure the Combine String is valid."

This error arises when a primitive hierarchical block is connected to a wire or net in a design. Set the hierarchical block's primitivity either to Default or No.

Legacy Message ID: NET0052

ERROR[ORNET-1024] - "Unnamed bus for pin %s. This will be unconnected in the netlist!"

This error arises when a bus doesn't have a name, or a valid range. This error can occur when an SDT design containing an unnamed bus is translated into Capture and then netlisted.

Legacy Message ID: NET0053

ERROR[ORNET-1026] - "Part %s of type %s is packaged incorrectly with parts of another type in the %s package."

This error arises when the netlister encounters one type of part that is packaged with another type of part. For example, this error is generated if a 74LS00 and a 74LS04 were both given a part reference of U1A.

Legacy Message ID: NET0054

ERROR[ORNET-1027] - "Illegal character in string: %s."

This error arises when the netlister encounters an invalid character in a particular string. For example, this error is generated if the character "#" appears in the name of a part while creating a Verilog netlist. Find and remove the illegal character.

Legacy Message ID: NET0055

ERROR[ORNET-1028] - "Unable to delete file: %s. Read-Only file, file is in use by another application, or possibly a complex hierarchical design in LOGICAL mode."

This message occurs when you cancel the netlist using the Cancel button on the Generate Netlist dialog box.

Legacy Message ID: NET0057

ERROR[ORNET-1047] - "Unable to open %s."

Capture could not open the specified netlist file with its associated editor.

Legacy Message ID: NET0017

ERROR[ORNET-1048] - "Design is not annotated."

Capture discovered that one or more parts in the design did not have their part references updated. For example, this error may occur if you update part references in logical mode, but not in physical mode, and then attempt to create a netlist in physical mode. Update the part references in the same mode for which you intend to create a netlist.

Legacy Message ID: NET0018

ERROR[ORNET-1049] - "Empty device designator encountered. Aborting."

The entire design is not annotated.

Legacy Message ID: NET0019

ERROR[ORNET-1050] - "Memory exhausted while loading parts."

Capture ran out of memory while building the part table for netlisting.

Legacy Message ID: NET0020

ERROR[ORNET-1051] - "Cannot get part."

The netlister was unable to get a part instance while creating is packing table. This error can occur when the design file is corrupt.

Legacy Message ID: NET0021

ERROR[ORNET-1052] - "Cannot get unique part."

A database error or corrupted design file prevented Capture from getting a unique part while building the part table.

Legacy Message ID: NET0022

ERROR[ORNET-1053] - "Unrecognized string id."

A database error, corrupted design file, or changed string ID prevented the netlist formatter from updating part properties.

Legacy Message ID: NET0023

ERROR[ORNET-1054] - "Invalid child."

A database problem occurred when Capture retrieved the child instance and received an unexpected result.

Legacy Message ID: NET0025

ERROR[ORNET-1055] - "Problem while retrieving part declarations."

Capture encountered a corrupted part instance or similar database problem.

Legacy Message ID: NET0026

ERROR[ORNET-1056] - "Problem getting child name."

Capture encountered an instance of a child lacking an associated name. Check to see if you have a part non-primitive part without an attached schematic.

Legacy Message ID: NET0027

ERROR[ORNET-1057] - "Unable to push onto sheet-stack."

Capture ran out of system memory while creating the stack of sheet names.

Legacy Message ID: NET0028

ERROR[ORNET-1060] - "Unable to load design hierarchy."

The design failed to identify if it was in logical or physical mode. Check to see if the design file is corrupt.

Legacy Message ID: NET0029

ERROR[ORNET-1061] - "Unable to update flat nets."

The database failed to produce a flat net. Check to see if the design file is corrupt.

Legacy Message ID: NET0030

ERROR[ORNET-1062] - "Wrong object type for netlisting."

The netlist formatter encountered an invalid object type. Check to see if you are netlisting a design and not a library. Also, check to see if the design file is corrupt.

Legacy Message ID: NET0031

ERROR[ORNET-1063] - "Unable to create header structure."

The netlist formatter encountered a problem with a title block. Check to see if the title block exists, and that it is complete.

Legacy Message ID: NET0032

ERROR[ORNET-1064] - "Unable to create globals table."

Capture ran out of memory while creating the table of global nets.

Legacy Message ID: NET0033

ERROR[ORNET-1065] - "Initialization aborted."

Capture was unable to complete the netlist.

Legacy Message ID: NET0034

ERROR[ORNET-1066] - "Unable to load table of globals."

Capture encountered a memory or database error while trying to load parts from the database into the part map. Check to see if the design file is corrupt.

Legacy Message ID: NET0036

ERROR[ORNET-1067] - "Unable to get next module port."

Capture encountered a problem while traversing the list of module ports in the design. Check to see if the design file is corrupt.

Legacy Message ID: NET0037

ERROR[ORNET-1069] - "Unable to get next port on node."

Capture encountered a problem while getting nets, due to a database problem or system error. Check to see if the design file is corrupt.

Legacy Message ID: NET0039

ERROR[ORNET-1070] - "Unable to get next pin on node."

Capture encountered a problem while getting nets, due to a database problem or system error. Check to see if the design file is corrupt.

Legacy Message ID: NET0040

ERROR[ORNET-1071] - "Unable to get property value."

The database is corrupt or is missing a property. For example, this error would occur if your PCB Footprint property combine string was {PCB Footprint}{Foo} where Foo didn't exist. Check to see if all part properties exist, and if the design file is corrupt.

Legacy Message ID: NET0041

ERROR[ORNET-1072] - "Unable to get next global."

Capture failed to get the next global net from the database, and the database didn't indicate it was at the end of the list. Check to see if the design file is corrupt.

Legacy Message ID: NET0042

ERROR[ORNET-1073] - "Unable to get next flat net."

Capture failed to get the next global net from the database, and the database didn't indicate it was at the end of the list. Check to see if the design file is corrupt.

Legacy Message ID: NET0043

ERROR[ORNET-1074] - "Unable to get next port on current flat net."

Capture failed to get the next global net from the database, and the database didn't indicate it was at the end of the list. Check to see if the design file is corrupt.

Legacy Message ID: NET0044

ERROR[ORNET-1090] - "Unable to create parts table."

This error arises when Capture tries to allocate more memory for the part information table than what is available. Try closing down other applications that are running.

Legacy Message ID: NET0046

WARNING[ORNET-1100] - "Ambiguous bus range specified. %s and %s on the same net."

Capture detected a mismatch in pin type within a hierarchy. For example, this message occurs when you have a bus connected to a hierarchical pin on a hierarchical block, but a wire connected to the corresponding hierarchical port in the schematic below. Create the netlist again using the option to make everything scalars. Alternately, change either the hierarchical port or the hierarchical pin so that the corresponding nets connect to the same pin type.

Legacy Message ID: NET0068

ERROR[ORNET-1101] - "Scalar/Bus conflict detected within hierarchy for net %s."

You can draw a hierarchical block with bussed pins and draw the underlying schematic with non-bussed (scalar) ports. In this case, you must select the option to output buses as scalars so that the VHDL netlister produces a correct netlist. If this error occurs when you use the Capture Build function, the VHDL netlister automatically re-runs with the option to output buses as scalars selected.

This error might also occur when both hierarchical block and underlying schematic are drawn with bussed pins and ports, but you have a syntax error in a bus name, such as "A[0..5]]" (here there is an extra "]" in the bus name).

Legacy Message ID: NET0070

ERROR[ORNET-1144] - "Invalid netlist format DLL file."

Capture detected an invalid netlist format .DLL file. Check that you have the correct version of the .DLL file and that it isn't corrupt. If in doubt, reinstall Capture.

Legacy Message ID: CAP0021

ERROR[ORNET-1214] - "A file must be selected in order to perform back annotation."

This error arises when a swap file has not been selected in the Gate and Pin Swap dialog box. Select a swap file before choosing OK in the dialog box.

Legacy Message ID: GAT0022