DRAFT COPY

USERS GUIDE

TO INTERFACE BETWEEN

SUPERMAX E-CAD VERSION 3.0

AND

WORKVIEW SERIES II VERSION 4.1

Dansk Data Elektronik A/s
Supermax EDA Division

Contents.

1. Overview

This interface is designed to allow for the interchange of design information between Viewlogic Systems Workview 4.1 series II software and the Supermax E-CAD interactive print layout system, version 3.0, hereafter referred to as E-CAD.

It is assumed that any design information converted by this interface has originated as a schematic generated from within Workview 4.0 or 4.1.

The following documentation assumes the user is familiar with both E-CAD and with Workview version 4.0 or 4.1.

Refer to the Release Notes for the installation procedure.

2. From Workview to Supermax E-CAD

With the aid of this interface a design can be transferred from Workview to E-CAD, with or without component preplacement by Viewplace. The system allows the user to add E-CAD specific wirelist options from within Viewdraw, preplace the design, and then transfer the design from Workview to E-CAD.

2.1 Adding E-CAD wirelist options

In order to add wirelist options to a net from within Workview the user must select a net, and then either type in the macro command **mex wl001**, or choose the same command via the menu. In the second case the menu selection is Window Open Utility E-CAD Net_attr.

When the above macro or menu command is selected, Workview will apply all sixteen E-CAD wirelist options to the net, and give them a default value of **UNSPECIFIED.** A pop up menu will then appear, showing all the wirelist options available, and their default values. Unless these default attribute values are changed to values accepted by E-CAD, running the Viewlogic to E-CAD part of the interface will not pass the attributes through to E-CAD's wirelist file, even though they have been written into the Viewdraw's database.

Workview will not accept the same attribute being applied to the same portion of the same net, though it will accept the same attributes being applied to different segments of the same net. Consequently, if after already having applied wirelist options to a segment of a net the user tries to repeat the procedure on the same segment of the net, then Workview will accept the instruction, and display the wirelist options applied to that net in a pop up menu, but it will not add any more wirelist options to the net.

If, however, after already having applied wirelist options to a segment of a net, the user repeats the procedure on another segment of the same net (for example, if a net has a junction from which the net connects to more than one component pin) then Workview will accept the newly applied wirelist options, and will then display a pop up menu that shows the originally applied wirelist options plus those that have just been applied.

It should be noted that since E-CAD accepts only one occurance of a specific wirelist option per net, applying the same wirelist options to a net more than once would be a wasted action, since E-CAD will read the first occurance and then ignore further occurances of the same. Aside from this, Workview will not allow the changing of attribute values in instances where the same attribute is applied to a net more than once. What this means is that although Workview will allow the user to apply a set of wirelist options to the same net more than once, he or she will not then be able to change their values from **UNSPECIFIED** to whatever is desired.

Because Workview treats wirelist options as attributes, it allows the user to add wirelist options to any selectable object within a project. However, since within E-CAD wirelist options are only applied to nets, the interface will, when transferring a design to E-CAD, ignore any wirelist option applied to an object other than a net and output a warning message similar to :

WARNING: Wirelist options applied to component R1 will be ignored

Additionally, the interface will ignore any wirelist options applied to a net that have a value equal to **UNSPECIFIED.** In this case however, there will be no warning message.

Note: Wirelist options can only be added to the design from within Viewdraw. They can not be added to the design via a unix shell command, and from within E-CAD only existing wirelist options, as specified from within Workview, can be changed.

Within Viewdraw all wirelist options have the prefix "**wl_**". Consequently, any "wirelist options" attributes added by the user cannot be prefixed with the same.

The transfer process strips the "**wl_**" prefix from any wirelist options specified from within Workview, and changes all upper case lettering within a wirelist name and wirelist value to lowercase.

see also: Supermax E-CAD Reference Manual
Supermax E-CAD Pre Post Processing Manual

2.1.1 Adding Component Attributes

Supermax E-CAD handles attributes by assigning special lays, where the attribute information can be stored. The attribute lays that are of interest when transferring data from Workview to E-CAD are:

article lay	: article name
block lay	: block attribute
heat lay	: power consumption
height lay	: component height
lambdalay	: heat transmission
comment lay	: non specific component attributes

In detail:

Article:

If for a particular component in Workview no article name has been specified, then the interface will use the component device name as the default article name. If a part descriptor number (p/d_num) is specified, then that will be used as the article name instead of the device name. If both article name and p/d_num are specified, then the interface will pass the specified article name through to E-CAD, and pass the p/d_num to E-CAD as a comment assigned to that particluar component.

Block:

Assigning a block attribute to several components in workview, where the value of the block attribute is the same for aech of the components will allow E-CAD to treat these components as one unit, that is, as a supercomponent.

The attributes are specified as **BLOCK=VALUE.**

Heat:

A thermo attribute, giving the power consumption of the component. Units are in milliwatts.

The attribute is specified as **MWATT=VALUE.**

Height:

An attribute giving the height of the component. Can be entered with or without units.

The attribute is specified as **HEIGHT=VALUE.**

Lambda:

A thermo attribute, giving the ability of the component (and of the pcb below it) to cunduct heat away from itself. Units are in 1/(ohm).

The attribute is specified as **LAMBDA=VALUE.**

Comment:

If a component in Workview has attributes assigned to it that are not recognized by E-CAD, then they will be transferred to E-CAD as comments in the following manner.

comment='**ATTRIBUTE_NAME=ATTRIBUTE_VALUE**".

see also: Supermax E-CAD Reference Manual, **sysparm** pages.

2.2 From Workview to E-CAD

The transfer of data from Workview to E-CAD can be done in any of four different ways.

From within Workview menu system
Using a Workview macro command
Using a unix shell command
From within E-CAD's help menu

When using the relevant unix shell command, or the E-CAD help menu, the full path name of the project must be given. If, when using Workview's menu system, or when using the relevant Workview menu command, the user is prompted for the project name, only the project name itself, and not the "path name" should be entered. This is because Workview automatically assigns the full path of the currently specified project to the name entered by the user.

When a project is transferred from Workview to E-CAD, the output is placed in the **/usr/ipl/owlsave** directory, with the default output file name being **project.iwl**, where **project** is the name of the project. If E-CAD is not located on the same machine as Workview, or the Supermax E-CAD **iwl** wirelist check program fails to execute properly, the output file will be given the name **project.preiwl**. This file can then be checked for syntactic correctness at a later date, by entering the command:

iwl -iy **project.preiwl** -cwl **project.iwl**

The output file obtained after having run the **iwl** program is a file that conforms to the Supermax E-CAD wirelist file format specifications, as defined in the Supermax E-CAD Pre Post Processing Manual. If Supermax E-CAD is installed on the same machine as Workview then the interface will automatically run the **iwl** program, place the resultant **project.iwl** file in the **/usr/ipl/owlsave** directory.

see also: Supermax E-CAD Reference Manual
Supermax E-CAD Pre Post Processing Manual

2.2.1 From Viewdraw to E-CAD

2.2.1.1 Workview menu selection

By selecting **Window** **Open** **Utilities** **E-CAD** **V1toE-CAD** the user can transfer Workview's currently specified project to E-CAD's data format. If no project is specified the user is prompted to enter project name. In this case a full path name is not required, as Workview automatically assigns the full path to the project name entered by the user. If no project name is entered, or the name is invalid, the system produces appropriate error messages and exits from the procedure.

2.2.1.2 Workview macro command

The user types in **mex V12E-CAD** followed by the project name if no project is specified within Workview. As with the menu command, the user should not enter full path name, as this is assigned by Workview to the name entered by the user.

If no project name is specified, or if the user enters an invalid project name then appropriate error messages appear, and the system exits the procedure.

2.2.1.3 Unix command

To transfer a design from within a unix shell, type **vl2ipl.sh**, followed by the name of the project. The user must enter the full path name of the project. if no project name is entered, or if the project name is invalid (for example not a full path name) then appropriate error messages appear, and the system exits the procedure.

2.2.1.4 The E-CAD Help Menu

This option is only available if E-CAD is installed on the same machine as Workview.

To transfer a design using E-CAD's help menu type capital H, and then <return>. This then brings the E-CAD help menu onto the screen. The user enters 'a' to convert alien netlists, and then '1' to convert netlists from Viewlogic's netlist format to that used by E-CAD. When that is done the screen prompts for the full path name of the project to be converted. This full path name is expected to be the name of the project, as referred to by Workview when accessing that project. If no project name is entered, or if the entry is inavalid, then the system will display appropriate error messages and quit the conversion procedure.

2.2.2 Viewdraw to E-CAD, with Viewplace

The procedure followed when converting a Viewlogic netlist with preplaced components is exactly the same as above, except that the design must have been updated from Viewplace before the conversion procedure is begun, in order for the preplacement data to be included in the transfer.

Workview 4.0 has defined only a limited number of component types. Supermax E-CAD is a print layout system, which means by definition that it must have a large library of component package types available to the print designer. These different package types are selected by setting the component package type attributes within Workview to a particular package type, which must then be listed in a cross reference table with the name : **/usr/ipl/tables/iplvl.lib.**

If a new component package type is added to the E-CAD componet library, then it should be entered into the look up table, and a corresponding Workview name should be entered as well. This will then allow for this package type to be selected from within Workview. The format to be used is

typ='Viewlogic pkg type' types/'E-CAD type'

If a component type is referred to within Workview which does not exist in E-CAD's component library, then the corresponding "types/'E-CAD type'" entry should be left blank. This will then result in the Workview package type being transferred over to E-CAD's netlist, and the designer can decide from there what should be done.

in iplvl.lib should be left blank, so that the Viewlogic pkg type is listed instead. The resultant E-CAD netlist file must then be manually altered before E-CAD can process the component(s) in question.

When transferring from Viewplace to E-CAD, the user must create a board symbol to be used for the pre placement process. Once the design has been converted to E-CAD's wirelist format, the user must, from within E-CAD, define the x and y offsets so that the board reference is not destroyed.

Note:

This version of the interface does not accomodate double sided preplacement.

When designing component types for use in Viewplace it is important to use the same origin and rotation in the corresponding E-CAD type.

The present release of Workview only supports symbol pkg-type attributes in Viewplace. This means that Viewplace will only support 28 different types of components, and is in fact only useful as an indication of component placement.

A sample **iplvl.lib** file supplied with this interface is shown on the following page. This listing shows some of the pkg type symbol_attributes included in Viewlogics standard library.

```
##############################################################################
# file: /usr/ipl/tables/iplv1.lib                                            #
#                                                                            #
#                                                                            #
#                                                                            #
#                                                                            #
#                        Supermax E-CAD version 3.0                          #
#                  Copyright Dansk Data Elektronik A/S 1991                  #
##############################################################################


typ=DIP                 types/ic/
typ=MOS
typ=8PDIP               types/ic/DIP8-rca
typ=14PDIP              types/ic/DIP14-mot
typ=16PDIP              types/ic/DIP16-1-mot
typ=18PDIP              types/ic/DIP18-mot
typ=20PDIP              types/ic/DIP20-1-mot
typ=22PDIP              types/ic/DIP22-rca
typ=24PDIP              types/ic/DIP24-mot
typ=CAP                 types/ic/cap/3M1100w30h80
typ=RES                 types/res/04Md25p09
typ=CONN                types/con/
typ=SMD4                types/smd/
typ=SMD5                types/smd/
typ=TO-5                types/tra/TO-5-ebc
typ=TO-18               types/tra/TO-18-ebc
typ=TO-92               types/tra/TO-92-ebc
typ=TO-202              types/tra/TO-202-123-ver
typ=TO-204              types/tra/
typ=TO-220              types/tra/TO-220-bce-hor
typ=DO-35               types/dio/
typ=DO-41               types/dio/
typ=59-03               types/dio/
typ=HC                  types/cry/HC-18T
typ=HO8A                types/cry/
typ=HO8C                types/cry/
typ=94Z                 types/trf/
typ=BATTERY             types/mec/
```

2.3 Quick Guide (from within Workview)

To transfer a design to E-CAD without pre-placement

Enter the design into Workview

select File Write

select Window Open Utility E-CAD VltoE-CAD

To pre place components first

(to create Viewplace database)

Enter the design into Workview

select File Write

select Window Open Utility Vplace Create

from shell enter **vlattrib project name**

select Window Close

(to create a board outline)

select Window Open Viewplac Symbol
(Enter name, specify window size, and symbol size, create box, and/or create line, for board outline. If no board outline is created then viewplace has no reference co-ordinates, and preplacement will be meaningless.)

select Change Block Type Brdoutlin
(This changes the above specified block to a board outline)

select **File** **Write** to ensure Viewplace database is updated

select **Window** **Open** **Viewplac** **Board**
(Enter project name, enter board outline name if this is unspecified, specify window size.)

select **Place** **All** to place all components

select **File** **Write** to ensure database is updated

select **Window** **Open** **Utility** **Viewplac** **Vdraw**
(To transfer preplacement information back to Viewdraw.)

select **Window** **Open** **Utility** **E-CAD** **VltoE-CAD**

3.0 From Supermax E-CAD to Workview

Once a design has been laid out in E-CAD, some of the information relating to the layout can be back annotated to Workview. This can be done by back annotating placement information to Viewplace, or by back annotating attribute changes, gate and pin swaps to Viewdraw.

Data can be transferred in any one of four ways.

From within the Workview menu system
Using a Workview macro command
Using a unix shell command
From within the E-CAD Help Menu

When using the relevant unix shell command, or E-CAD's help menu, the full path name of the project must be given. No path name is required when using Workview's menu system, or when using Workview's macro commands, as long as there is a currently specified project

Any one of the following four procedures may be used

3.0.1 Workview menu selection

By selecting **Window Open Utility E-CAD E-CADtoVd** or **Window Open Utility E-CAD E-CADtoVp** the user can transfer design information from E-CAD to either Viewplace, or to Viewdraw. Selecting **E-CADtoVd** uses as input the back annotation file that can be generated from within E-CAD. Selecting **E-CADtoVp** uses as input the job file saved by E-CAD, which is located under the directory **/usr/ipl/iplsave.**

3.0.2 Workview macro command

The user types in **mex E-CAD2Vd** or **mex E-CAD2Vp**, followed by the project name, if no current project name is specified within Workview. If no project name is entered, or if it is invalid, then appropriate error messages will appear, and the conversion procedure will be halted.

3.0.3 Unix shell command

The user types in either **ipl2vp.sh** or **ipl2vl.sh** , as the case may be, followed by the full path name of the project. If no project name is given, or if the name given is invalid, then appropriate error messages appear, and the system exits the procedure.

3.0.4 Supermax E-CAD Help Menu

This option is only available if E-CAD is installed on the same machine as Workview.

The user types 'H' and then <return> to bring the menu onto the screen. By selecting 'a', for alien netlists, and then 'L',to convert netlists to Viewlogic's format, a menu is displayed on the screen and the user is prompted to select conversion to either Viewdraw or Viewplace, and to enter the appropriate E-CAD job file, or E-CAD back annotation file name. When this is entered the system reads the input file from the Supermax E-CAD jobfile directory (that is the **/usr/ipl/iplsave** directory) and sends the output to the specified directory.

3.1 From E-CAD to Viewplace

When back annotating from E-CAD to Viewplace, the interface uses the E-CAD job file, **project.ipl** as the default input file, and the **project.brd** file, located in the **/brd** sub directory of the the project directory, as the output file. If the project is stored under another name than **project.ipl**, and the system cannot find this file, the user will be prompted to enter another input file name for the project. It is necessary to save the project from within E-CAD before converting back to E-CAD, to ensure that the input file is up to date.

It is possible for the user to back annotate information to Viewplace without having previously created a Viewplace database. Because of this flexiblity, the interface creates a default board size for this particular process.

The default board, which has the same dimensions as the pcbmax coordinates within E-CAD, is called **e-cad.1**, and is located in the **/brd sym** sub directory of the project directory. This file is written to every time a design is back annotated to Viewplace, and is likely to change every time this happens. The user should therefore use Workview's commands to change the board symbol name for every design every time back annotation to Viewplace has taken place.

The default input file **project.ipl** is typically located in the **/usr/ipl/iplsave** directory of the machine on which E-CAD is installed.

3.2 From E-CAD to Viewdraw

Back annotation from E-CAD to Viewdraw allows the user to update Viewdraw with attribute **changes** made to the design, and with gate and pin swaps. This version of the interface does **NOT** allow the user to add or delete components or nets from the design, nor to add or delete attributes.

The following changes can be back annotated to Viewlogic:

renaming components.
renaming groups.
gateswapping, by using the "auto GATEswop" and the "man GATEswop" commands.
pinswapping, by using the "auto PINswop" and the "man PINswop" commands

NOTE:

The swapping of gates between components will only be back annotated to Workview if all the gates to be swapped are already included in the Workview schematic.

The default input file for this transfer is the **project.ipl.w** file, created by enabling back annotation from within E-CAD, and then saving the E-CAD job file. The output file is the **project.baf** file, the location of which is specified by the full path name for the project. The system will first check for the existance of the **project.ipl.w** file, and if it is not found, will prompt the user to enter the file name to be used.

The default input, **project.ipl.w** is typically stored in the **/usr/ipl/iplsave** directory of the machine on which E-CAD is installed.

3.3 Quick Guide

From E-CAD to Viewplace or Viewdraw

From the main menu of E-CAD the user

select I/O menu

select backanno (for back annotation to Viewdraw)

select save pcb

If E-CAD and Workview are installed on the same machine, the user can then back annotate the design using the help menu. If this is not the case, the user must transfer the saved file to the machine on which Workview is installed, and save it in the directory specified when running iplvl.

From within Workview (menu selections)

select Window Open Utility E-CAD E-CADtoVd

or

select Window Open Utility E-CAD E-CADtoVp

From within Workview (macro commands)

type **mex E-CAD2Vd**

or

type **mex E-CAD2Vp**

If no project has been selected from within Workview, the user is required to enter the project name.

From within unix shell

type **ipl2vl.sh** followed by **(full path) project name**

or

type **ipl2vp.sh** followed by **(full path) project name**

If no project name is given the user will be prompted for one. If the project name entered refers to irrelevent files, or ones that the user does not have access rights to, then appropriate error messages will appear.

Interfacet inkludere følgende:

```
 /usr/ipl/wv_if/vl2ipl
        "       ipl2vl
        "       ipl2vp
        "       iplvl.sh
        "       vl2ipl.sh
        "       ipl2vl.sh
        "       ipl2vp.sh
/usr/ipl/wv_if/sub/vl2ipl.awk
/usr/ipl/wv_if/sub/vl2ipl1.awk
/usr/ipl/wv_if/sub/ipl2vl.awk
/usr/ipl/wv_if/sub/ipl2vp.awk

/usr/ipl/tables/iplvl.lib

/usr/ipl/wv_if/workview/standard/wl001.mac
                                  E-CAD2VL.mac
                                  E-CAD2VP.mac
                                  VL2E-CAD.mac
                                  viewdraw.mnl
                                  utils.mnl
                                  net.flt

/usr/ipl/prepostpro/help/vl2ipl.sh
                         vl2ipl.def
                         vl2ipl.hlp
                         ipl2vl.sh
                         ipl2vl.def
                         ipl2vl.hlp

/usr/ipl/prepostpro/view2ipl
                    ipl2view


/usr/ipl/prepostpro/help/contoothn.def
```