Parametric Harness Layout
Menu and Technical Reference

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Preface

Parametric Harness Layout Menu and Technical Reference gives you a quick reference for designing and manipulating harness models in the Parametric environment. It provides a reference to the menus and property sheets on the Graphical User Interface.

Parametric Harness Layout provides functionality that facilitates the modeling of harness components and channels in the Parametric environment.

Related Documents

The following documents may be helpful as you use *Parametric Harness Layout Menu and Technical Reference*:

• *Explicit Harness Layout Menu and Technical Reference*

Book Conventions

The following table illustrates and explains conventions used in writing about CADDS applications.

Convention	Example	Explanation
Menu selections and options	List Section option, Specify Layer field	Indicates a selection you must make from a menu or property sheet or a text field that you must fill in.
User-selected graphic location	X, d_1 or P1	Marks a location or entity selection in graphic examples.

User input in CADDS text fields and on any command line	`cvaec.hd.data.param`
	
	`tar -xvf /dev/rst0`
		Enter the text in a CADDS text field or on any command line.

System output	`Binary transfer complete.`	Indicates system responses in the CADDS text window or on any command line.
Variable in user input	`tar -cvf /dev/rst0` filename	Replace the variable with an appropriate substitute; for example, replace filename with an actual file name.
Variable in text	tagname	Indicates a variable that requires an appropriate substitute when used in a real operation; for example, replace tagname with an actual tag name.
CADDS commands and modifiers	INSERT LINE TANTO	Shows CADDS commands and modifiers as they appear in the command line interface.
Text string	`"SRFGROUPA"` or `'SRFGROUPA'`	Shows text strings. You must enclose text string with single or double quotation marks.
Integer	n	Supply an integer for the *n.*
Real number	x	Supply a real number for the *x.*
#	`# mkdir /cdrom`	Indicates the root (superuser) prompt on command lines.
%	`% rlogin` remote_system_name `-l root`	Indicates the C shell prompt on command lines.
$	`$ rlogin` remote_system_name `-l root`	Indicates the Bourne shell prompt on command lines.

Window Managers and the User Interface

According to the window manager that you use, the look and feel of the user interface in CADDS can change. Refer to the following table:

Look and Feel of User Interface Elements

User Interface Element	Common Desktop Environment (CDE) on Solaris, HP, DEC, and IBM	Window Manager Other Than CDE on Solaris, HP, DEC, IBM, SGI, and NT
Option button	ON — Round, filled in the center OFF — Round, empty	ON — Diamond, filled OFF — Diamond, empty
Toggle key	ON — Square with a check mark OFF — Square, empty	ON — Square, filled OFF — Square, empty

Online User Documentation

Online documentation for each book is provided in HTML if the documentation CD-ROM is installed. You can view the online documentation in the following ways:

- From an HTML browser
- From the Information Access button on the CADDS desktop or the Local Data Manager (LDM)

Please note: The LDM is valid only for standalone CADDS.

You can also view the online documentation directly from the CD-ROM without installing it.

From an HTML Browser:

1. Navigate to the directory where the documents are installed. For example,

   `/usr/apl/cadds/data/html/htmldoc/` (UNIX)

   `Drive:\usr\apl\cadds\data\html\htmldoc\` (Windows NT)

2. Click `mainmenu.html`. A list of available CADDS documentation appears.

3. Click the book title you want to view.

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.

2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.

3. Choose DOCUMENTATION. A list of available CADDS documentation appears.

4. Click the book title you want to view.

From the Documentation CD-ROM:

1. Mount the documentation CD-ROM.

2. Point your browser to:

   CDROM_mount_point/htmldoc/mainmenu.html (UNIX)

   CDROM_Drive:\htmldoc\mainmenu.html (Windows NT)

Online Command Help

You can view the online command help directly from the CADDS desktop in the following ways:

- From the Information Access button on the CADDS desktop or the LDM

- From the command line

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.

2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.

3. Choose COMMAND HELP. The Command Help property sheet opens displaying a list of verb-noun combinations of commands.

From the Command Line: Type the exclamation mark (!) to display online documentation before typing the verb-noun combination as follows:

        #01#!INSERT LINE

Printing Documentation

A PDF (Portable Document Format) file is included on the CD-ROM for each online book. See the first page of each online book for the document number referenced in the PDF file name. Check with your system administrator if you need more information.

You must have Acrobat Reader installed to view and print PDF files.

The default documentation directories are:

- /usr/apl/cadds/data/html/pdf/doc_number.pdf (UNIX)

- CDROM_Drive:\usr\apl\cadds\data\html\pdf\doc_number.pdf (Windows NT)

Resources and Services

For resources and services to help you with PTC (Parametric Technology Corporation) software products, see the *PTC Customer Service Guide*. It includes instructions for using the World Wide Web or fax transmissions for customer support.

Documentation Comments

PTC welcomes your suggestions and comments. You can send feedback in the following ways:

• Send comments electronically to `doc-webhelp@ptc.com.`

• Fill out and mail the PTC Documentation Survey located in the *PTC Customer Service Guide.*

Chapter 1   # Overview

This chapter describes how you can access the Graphical User Interface (GUI) for Harness Layout in the parametric environment of CADDS. The topics discussed in this chapter are

- Introduction
- Accessing Harness Layout Task Set
- Harness Product Flow Chart
- Property Sheets

Introduction

Currently, CADDS applications such as Harness Layout, HVAC and Piping perform routing and component insertion in the Explicit environment of CADDS using nodal entities. These applications have their own user interfaces and ways of modeling in CADDS. All these applications have their own limitations in terms of routing, component insertion and editing capabilities.

Harness Layout in Parametric provides an unified and generalized functionality to perform routing, component insertion and modification in the Parametric environment. It reduces the efforts required to route, insert components and make changes to the model or associated model data.

Harness Layout provides you with functionality that can be divided into the following modules.

Initializing Harness Layout

This functionality allows you to initialize a version and initiate a Harness Layout session. You can associate a netlist file, parameter file and wirespec file with the initialized version. The parameter and wirespec files are mandatory for setting up the parameters required for the harness session whereas the netlist file can be loaded optionally.

For details on initializing a harness layout, refer to Chapter 2, "Initializing Harness Layout".

Deleting Harness Version

This functionality allows you to delete an existing harness version. Deleting the harness version removes all the version related information from the system.

For details on deleting a harness version, refer to Chapter 7, "Deleting Harness Version".

Displaying Version Status

This functionality allows you to display the details of the active version in a status window on the screen. The status window displays the name of the activated version, the names of the parameter and wirespec files and the name of the netlist file, if specified for the currently active harness version.

For details on displaying version status, refer to Chapter 6, "Displaying Harness Status".

Selecting Harness Attributes

This functionality allows to select various Harness Layout modeling attributes. The default values for these attributes are based on the values specified in the parameter file, specified in the currently active harness version. These attribute values become the global defaults through-out the current session and are saved in the part database.

For details on selecting harness attributes, refer to Chapter 3, "Selecting Harness Attributes".

Inserting Components

This functionality allows you to insert harness components such as connectors, junctions and clips, into the harness model, and associate various attributes with these components.

These harness components are inserted from a user specified library, which is selected using the Select Harness Attributes functionality. Library components are created as CADDS features and stored in user defined component libraries.

For details on inserting components, refer to Chapter 4, "Modeling a Harness Part".

Editing Components

This functionality allows you to edit harness components created using the Insert Component functionality. You can edit various component attribute values, like the connector name, part number, and the tangency attributes as applicable to the component being edited.

For details on editing components, refer to Chapter 4, "Modeling a Harness Part".

Routing Channels

This functionality allows you to route channels in the Parametric environment.

For details on routing channels, refer to Chapter 4, "Modeling a Harness Part".

Editing Channels

This functionality allows you to edit the existing channel parameters. You can edit parameters for a single channel or a group of channels. The parameters that can be edited include the cross section shape, pertinent linear parameters such as the width, height, diameter and other channel related parameters like the maximum number of wires, segregation class, splicing, priority factor.

Channel associativity with connectors, junctions and clips is maintained during channel editing.

For details on editing channels, refer to Chapter 4, "Modeling a Harness Part".

Creating a Display Representation

This functionality allows you to create a surface or solid representation of selected channels in the model using the cross-section related attributes defined on the channel.

For details on creating a display representation, refer to Chapter 5, "Displaying and Verifying Components and Channels".

Listing Components

This functionality allows you to list connectors, clips or junctions components present in a specified component library. You can verify if a particular component exists in a specified component library or list all the components of a particular type in the specified component library.

For details on listing components, refer to Chapter 5, "Displaying and Verifying Components and Channels".

Verifying Components and Channels

This functionality allows you to display and verify the harness related attributes of selected harness components and channels.

For details on verifying components and channels, refer to Chapter 5, "Displaying and Verifying Components and Channels".

Saving a Harness Session

This functionality allows you to save a harness session while continuing to work in the same session. You can save the currently active version and the related harness model information along with the active part database.

For details on saving a harness session, refer to Chapter 8, "Saving and Exiting a Harness Session".

Exiting a Harness Version/Session

This functionality allows you to exit an active harness version/session.

For details on exiting a harness version/session, refer to Chapter 8, "Saving and Exiting a Harness Session".

Converting a Parametric Model to Explicit

Channel routing and component insertion is done in the Parametric environment whereas other existing Harness Layout features, such as the Constraints Based Router and manufacturing modules, are run in the Explicit environment.

This functionality converts Parametric channels to Explicit Nlines, and Parametric components to Explicit Cnodes or Nfigs.

For details on converting a Parametric model to Explicit, refer to Chapter 9, "Converting a Parametric Model to Explicit".

Highlighting Cables and Components

This functionality allows you to highlight cables and other harness components such as nets, wires, connectors, clips and splices in the CADDS model.

For details on highlighting cables and components, refer to Chapter 9, "Converting a Parametric Model to Explicit".

Interference Detection

You can detect interference between channels, harness components and any other valid objects, by using the standard CADDS interference detection commands. Identify the objects for interference detection by selecting them on the screen.

Accessing Harness Layout Task Set

The CADDS 5 Parametrics as well as Explicit environments have four user interface locations.

- The Environment Management menu on the top bar
- The Environment Globals I menu on the top bar
- The Environment Access menu on the side bar
- The Task Set Access menu on the side bar
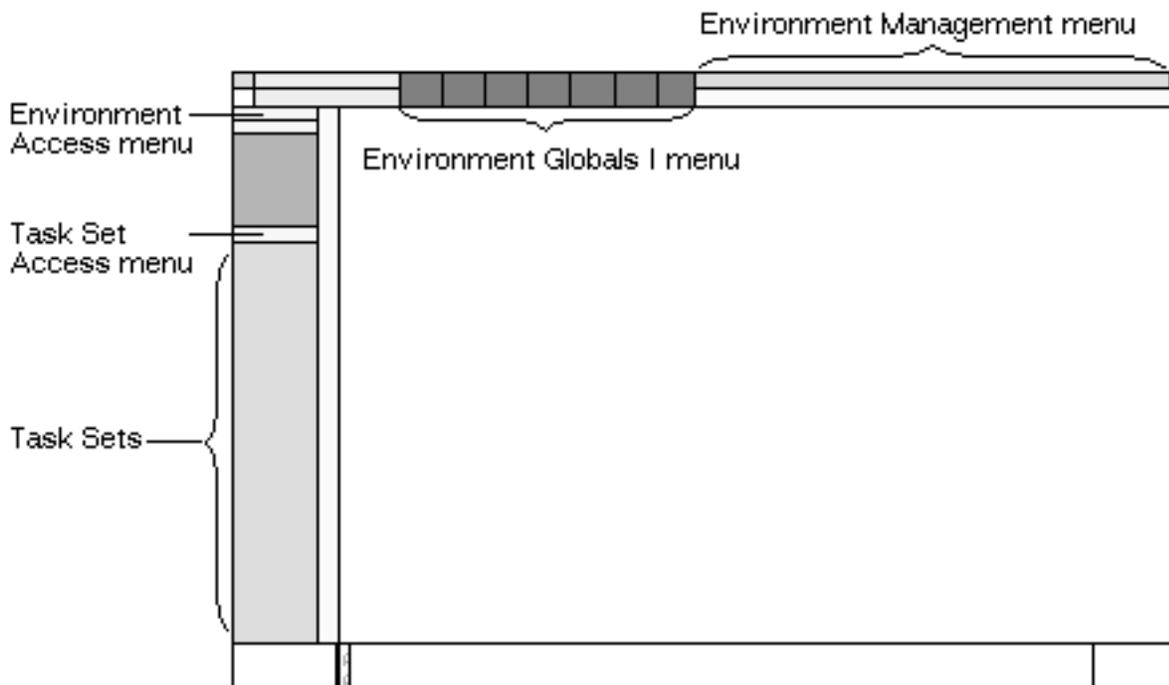
You can access the Harness Design environment from the Environment Access menu and the Task Set Access menu, in both Parametrics as well as Explicit environment.

Accessing Harness Options in Parametric Environment

To access harness layout options available in CADDS Parametric environment

1. Choose Parametric from the Environment Access menu to access Parametric environment of CADDS.

2. Click MODEL. The following menu appears.
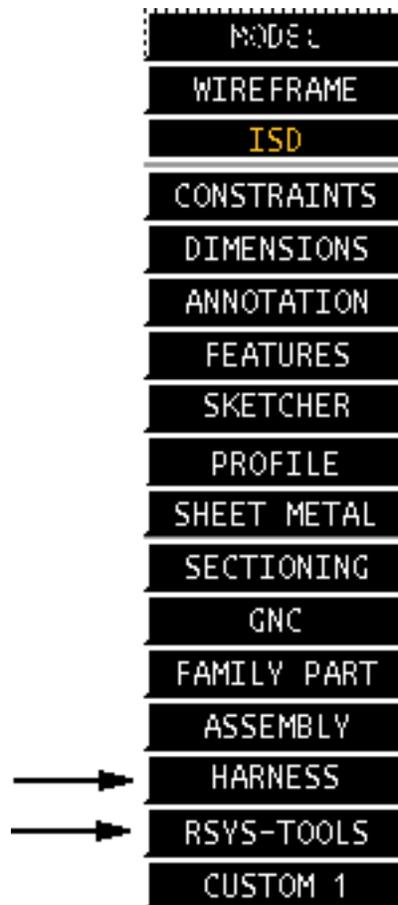
3. Choose HARNESS to access the Harness Layout environment. The Harness task set, as shown in Figure 1-1, "Harness task set," on page 1-8, appears.

   OR

   Choose RSYS-TOOLS to access the Rsys Toolbox. The Rsys toolbox allows you to insert Connection points while creating a Harness Design component. For more details on inserting Connection points, refer to "Inserting Connection Points" on page 4-28.

The Harness Layout task set

The Parametric Harness Layout task set contains the following options. Each of these options are described in detail in the following chapters.

Figure 1-1   Harness task set
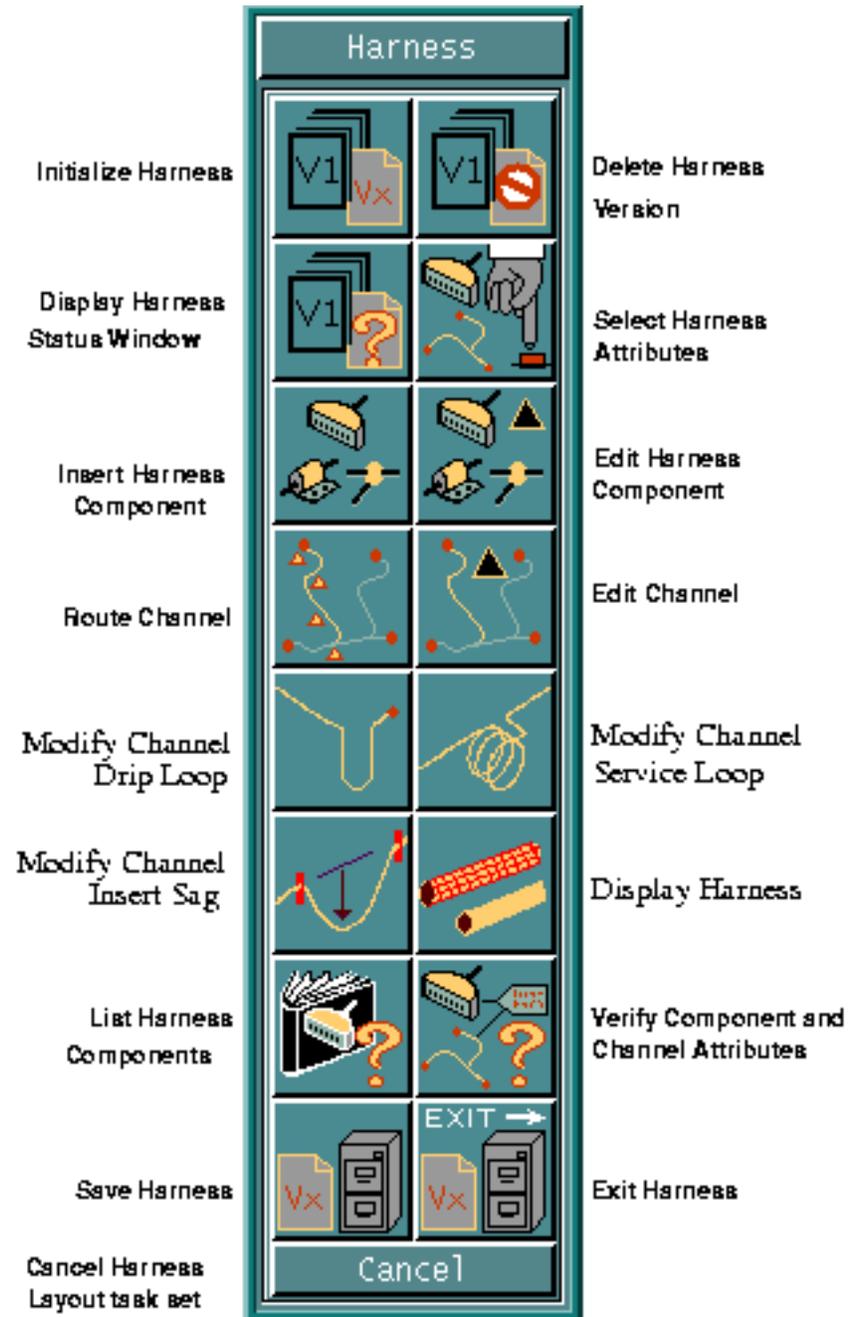
4. Choose the required option from the Harness task set. A brief description of the various options is given below.

Initialize Harness:

Initializes a version and initiates a harness layout session.

Delete Harness Version:

Deletes an existing harness version.

Display Harness Status Window:

Displays various details of the active version.

Select Harness Attributes:

Selects various Harness Layout modeling attributes.

Insert Harness Component:

Inserts harness components such as connectors, junctions and clips into the harness model.

Edit Harness Component:

Edits harness components created using the Insert Harness Component option.

Route Channel:

Routes a harness channel along selected locations and associates it with selected harness components.

Edit Channel:

Edits the attributes of an existing harness channel or a group of harness channels.

Modify Channel Drip Loop:

Inserts a Drip loop in a harness channel.

Modify Channel Service Loop:

Inserts a Service loop in a channel.

Creating a Display Representation:

Creates surface or solid display representations of selected channels in the model using cross-section related attributes defined on the channel.

List Harness Components:

Lists connectors, clips or junction or all components present in a specified component library.

Verify Component and Channel Attributes:

Displays and Verifies harness related attributes of selected harness components and channels.

Save Harness:

Saves the active harness session.

Exit Harness:

Exits Harness Layout with an option to save the harness session.

Accessing Harness Options in Explicit Environment

To access harness layout options available in CADDS Explicit environment

1. Choose Parametric from the Environment Access menu.

2. Click MODELING. The following menu appears.

```
   MODELING
   ANALYSIS
   PLASTICS
   STRESS
   SYSTEMS
   THERMAL
   ENG. PROD.
   CONTROL PIPE
→  HARNESS
   HVAC
   PIPING
   STRUCTURAL
   SUPPORTS
   ELEC CABLE
   Q. A.
   DESIGN VER.
   DMIS
   INSPECTION
```
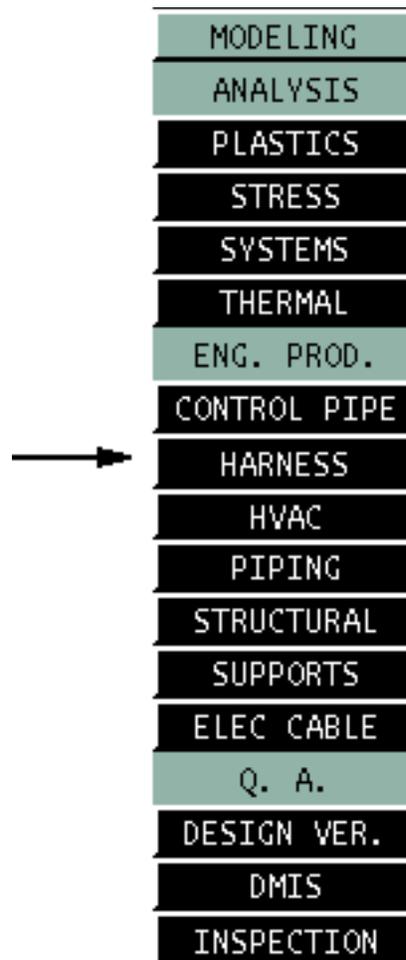
3. Choose Harness to enter the Harness Design environment.

   OR

   Use the ENTER HARNESS command in the CADDS text window to enter the Harness Design environment.

The Design task set appears. Use the Convert to Explicit option to convert a harness model from Parametrics to Explicit environment.

4. Choose the Router option from the Design task set to display the Router task set which contains the option for highlighting the harness.

Harness Product Flow Chart

This flowchart shows the typical sequence of tasks in the Parametrics environment as well as the Explicit environment while designing a harness model.
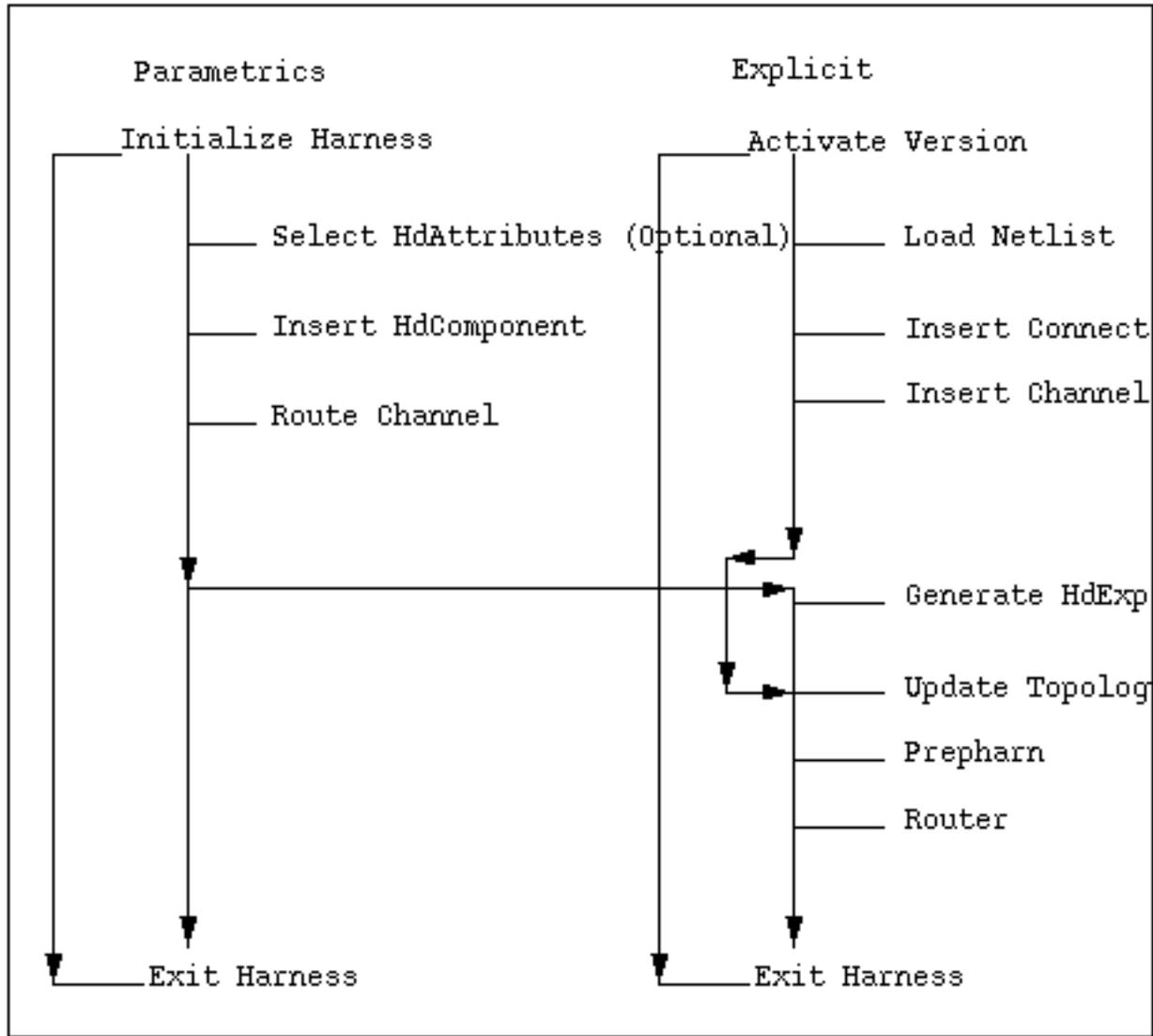
Property Sheets

This section describes how to use the special buttons provided on the property sheets. These buttons are used to display a scrolling list of valid selections for a field or to display a property sheet with additional options.

Please note: Basic information about using CADDS 5 property sheets is in the *Explicit Modeling User Guide.*

List Button

Harness property sheets have a feature called the List button.

Some fields on the property sheet are provided with a scrolling list of valid selections for that field. To access such a list, a list button is provided next to the field.

For example, the following is a field on a property sheet:

Click the list button to display a scrolling list of valid component library names as shown below. When you choose a component library name from the list, and click the Accept button, that component library name is automatically displayed in the field to the left of the List button.

For example, the following is the Component Library scroll list that is displayed when you click the list button next to the Component Library Name field.
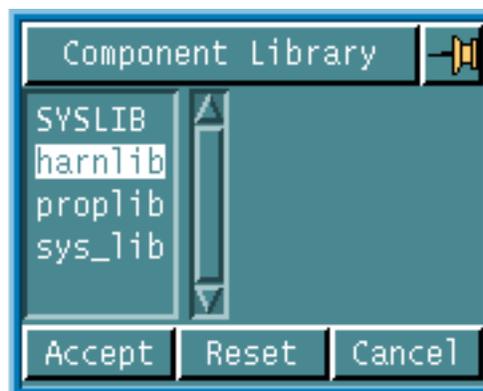
Initializing Harness Layout

Chapter 2

This chapter describes the initialization of a Harness Layout session. This option is available from the Harness task set in the parametric environment. The tasks discussed in this chapter are

- Initializing a Harness Layout Session

Initializing a Harness Layout Session

Choose the Initialize Harness option from the Harness task set to initialize a harness layout session. The Initialize Harness property sheet, as shown in Figure 2-1, "Initialize Harness Property Sheet," on page 2-3, appears.

OR

Use the INITIALIZE HARNESS command in the CADDS text window to initialize a harness layout session. See the online help for further details.

The Initialize Harness option enables you to start a harness layout session associated with a new or an existing version.

A version is a directory that stores the information related to a harness layout session. An existing version is a previously defined and saved state of the harness according to the topology and parameters existing at the time of saving. The version is saved in the

```
<parts_directory>/<part_name>/<version_name>
```

directory, when the you save the harness. A version must be active in any harness layout session.

The Initialize Harness option activates the specified version and loads the wirespec file, the parameter file, and optionally the netlist file into memory.

- The wirespec file contains wire definitions and information related to manufacturing. For a sample wirespec file, refer to Appendix B of the *Explicit Harness Layout Menu and Technical Reference*.

- The parameter file contains the default settings for component layers, colors, distances and directory paths. For a sample parameter file, refer to Appendix B of the *Explicit Harness Layout Menu and Technical Reference*.

- The netlist file (with a .wir extension) is created by Electrical Design Entry (EDE). It contains all the routing information of components, nets, wires, cables and segregation.

The default wirespec and param files supplied are located in the
/usr/apl/cadds/data/cvaec/hd/data/_bcd directory. Harness Layout
searches the CV_PATH and accepts the first wirespec and param files found in
this directory irrespective of whether the netlist file has been specified or not. If the
netlist file is not specified, harness layout assumes that routing is done in the
conceptual modeling mode.

These files are later used for other harness layout tasks like inserting harness
components and routing channels. Any or all these files can be reloaded during a
same session using the Initialize Harness option. You can change any of these files
during the same session by entering the new names in the appropriate fields in the
Initialize Harness Property sheet.

Please note: You must initialize the harness session on the Parametric side
only and not the Explicit side.

Please note: Only one version can be active at a time. To change the version,
you must exit the active version.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Initialize Harness option from the Harness task set. The Initialize
   Harness property sheet appears.

Figure 2-1   Initialize Harness Property Sheet
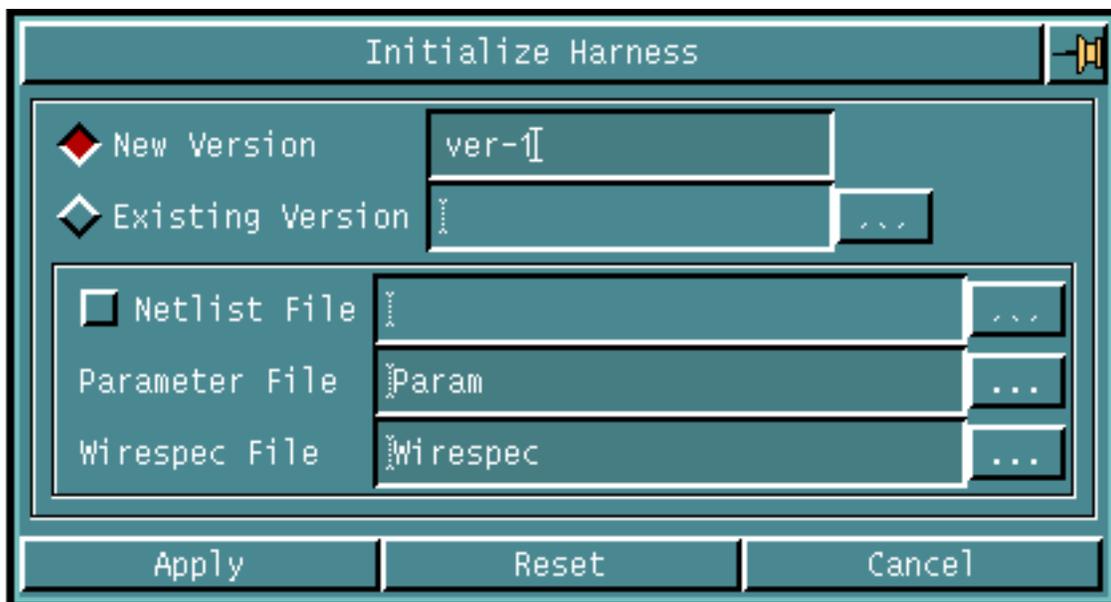
Procedure

You can specify the following

- Version Name
- Netlist File
- Parameter File
- Wirespec File

Version Name:

Specify the version name for the harness layout session.

- Click the New Version option to specify a new version name for the harness layout session. Only alphanumeric characters are allowed in the version name.

  OR

- Click the Existing Version option to activate an existing version. Choose a name of an existing version from the list that appears.

Netlist File:

Specify the name of the Netlist File or click the options button against the field to display the File Name Editor property sheet.

- Enter the name of the netlist file to be used in the field.

  OR

- Specify the netlist file name using the File Name Editor property sheet. See section "Using the File Name Editor Property Sheet" for details.

Parameter File:

Specify the name of the Parameter File or click the options button against the field to display the File Name Editor property sheet.

- Enter the name of the parameter file to be used in the field.

  OR

- Specify the parameter file name using the File Name Editor property sheet. See section "Using the File Name Editor Property Sheet" for details.

Please note: If the parameter file specified above does not exist, an error message is issued and the first parameter file with the name param found in the CV_PATH is used instead.

Wirespec File:

Specify the name of the Wirespec File or click the options button against the field to display the File Name Editor property sheet.

- Enter the name of the wirespec file to be used in the field.

  OR

- Specify the wirespec file name using the File Name Editor property sheet. See section "Using the File Name Editor Property Sheet" for details.

Please note: If the wirespec file specified above does not exist, an error message is issued and the first wirespec file with the name `wirespec` found in the `CV_PATH` is used instead.

Using the File Name Editor Property Sheet:

When you click the options button against the following fields,

- Netlist File
- Parameter File
- Wirespec File

the File Name Editor property sheet, as shown below, appears.

Figure 2-2   File Name Editor Property Sheet
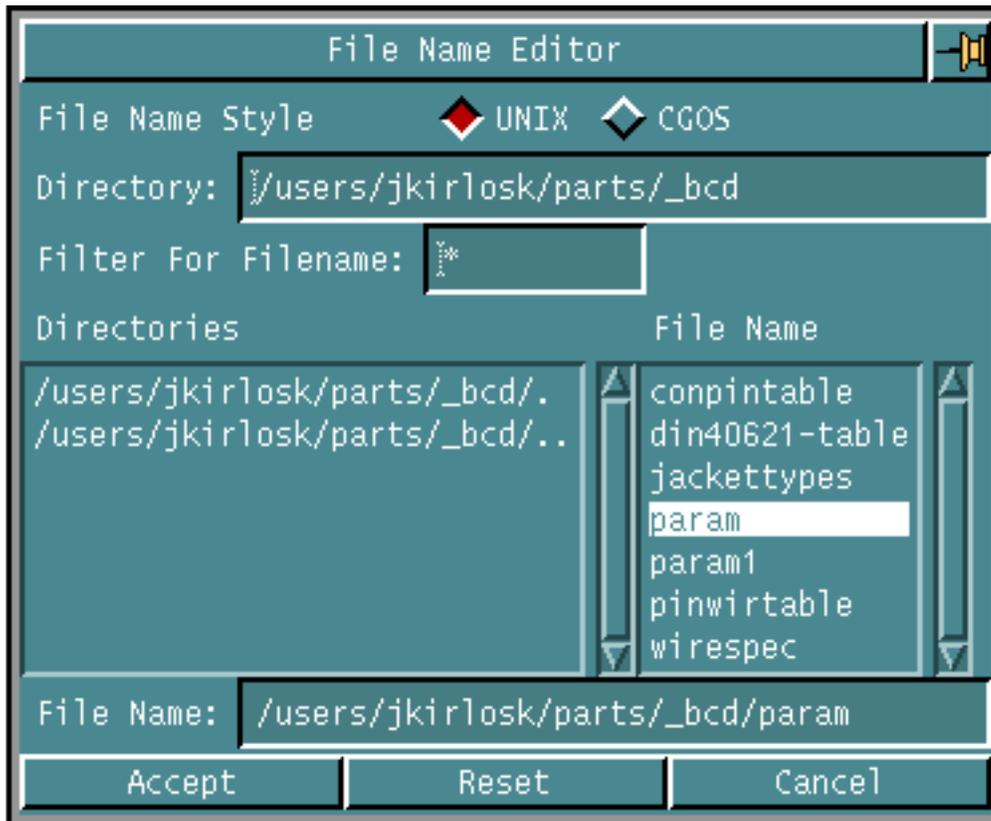
You can use this property sheet to specify the above file names.

- Specify one of the File Name Styles to be used, CGOS or UNIX for the required file.

- Specify the Directory in which the required file exists.

- Specify a suitable Filter for displaying files in a directory.

- Scroll through the directory listing that appears to select the required directory. The name of the directory selected from the scroll list is displayed in the Directory field. You can also type the directory name in the field.

- Scroll through the file listing that appears to select the required file. The name of the file selected from the scroll list is displayed in the File Name field. You can also type the file name in the field.

  If the file name specified is a relative path name, the editor loads the absolute path name from CVPATH. For example, if 'cvaec/hd/data/param' is specified in the parameter file field, the editor loads the /usr/apl/cadds/data/cvaec/hd/data/_bcd directory, lists all the files in that directory in the file name scroll list, highlights the param file and displays cvaec/hd/data/param in the File Name field.

Example

The following command activates a new version VER1 and loads the specified netlist, param and wirespec files.

```
INITIALIZE HARNESS VERSION VER1 NETLISTFILE
=USERS.ANKUMAR.PARTS.TAIL.CKT@WIR WIRESPECFILE
=USERS.ANKUMAR.PARTS.&BCD.WIRESPEC PARAMFILE
=USERS.ANKUMAR.PARTS.&BCD.PARAM

Activating new version VER1...
Processing Param file:=USERS.ANKUMAR.PARTS.&BCD.PARAM
Processing Wirespec file:=USERS.ANKUMAR.PARTS.&BCD.WIRESPEC
Processing Netlist file: =USERS.ANKUMAR.PARTS.TAIL.CKT@WIR
```

Chapter 3 # Selecting Harness Attributes

This chapter describes how to select various harness attributes.

- Selecting Harness Attributes

Selecting Harness Attributes

Choose the Select Harness Attributes option from the Harness task set to edit default centerline and component attributes for a harness layout session. The Harness Layout Attributes property sheet, as shown in Figure 3-1, "Harness Layout Attributes Property Sheet," on page 3-3, appears.

OR

Use the SELECT HDATTRIBUTES command in the CADDS text window to edit the default harness attributes. See the online help for further details on the command-line options.

This option allows you to edit default values of harness centerline and component attributes related to the Parametric environment.

When a harness layout session is initialized, the default attribute values are read from the parameter file. You can edit the attributes using this option and save them using the Save Harness option. These attributes are stored under the `version` directory associated with the part.

The values set using this option are used as the global default attribute values.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Select Harness Attributes option from the Harness task set. The Harness Layout Attributes property sheet appears.

Figure 3-1   Harness Layout Attributes Property Sheet
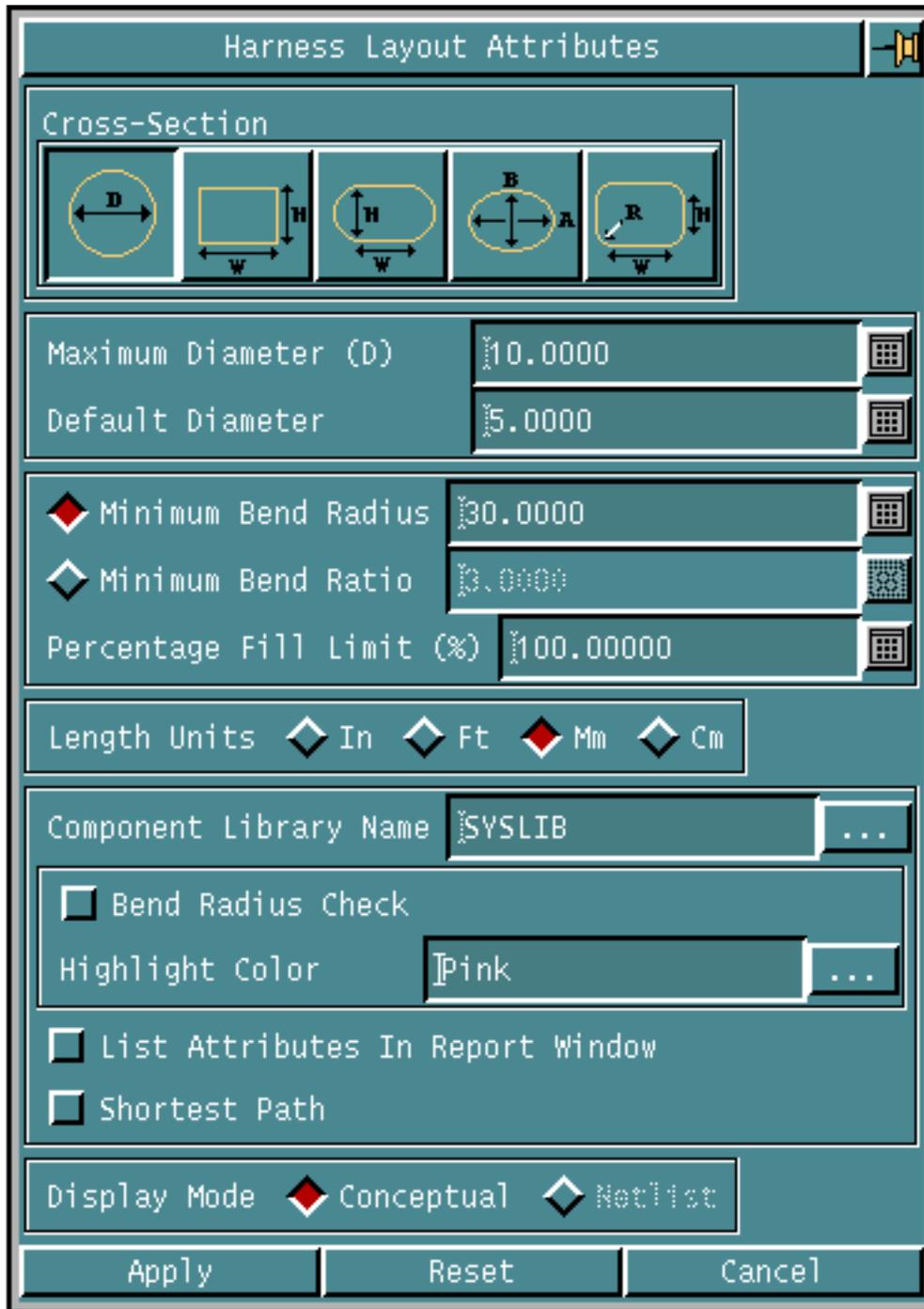
Procedure

You can specify the following attributes. The default attribute values from the parameter file are displayed in this property sheet.

- Attributes for a Circular Cross Section

  - Maximum Diameter: Specify the maximum diameter of the channel. The bundle diameter of the wires routed in the channel must not exceed the maximum diameter.

  - Default Diameter: Specify the default or initial diameter of the channel. This is useful for reserving channel volume prior to wire routing. It has no effect on the actual diameter of the channel once the wires are routed. This diameter is used during interference detection, shading, and HLR.

- Attributes for a Rectangular Cross Section

  - Height: Specify the height of the channel. The bundle height of the wires routed in the channel should not exceed this value.

  - Width: Specify the width of the channel. The bundle width of the wires routed in the channel should not exceed this value.

- Attributes for a Rounded Rectangular Cross Section

  - Height: Specify the height of the channel.

  - Width: Specify the width of the channel excluding the value of the rounded edges.

- Attributes for a Elliptical Cross Section

  - Major Axis: Specify the length of the major axis of the channel.

  - Minor Axis: Specify the length of the minor axis of the channel.

- Attributes for a Filleted Rectangle Cross Section

  - Height: Specify the height of the channel.

  - Width: Specify the width of the channel.

  - Fillet Radius: Specify the fillet radius to be used for a Filleted Rectangular Cross-Section.

- Component Library Name — Enter the Component Library Name to be used for the harness layout session in the field.

  OR

  Click the list button to display a list of existing feature libraries.

Figure 3-2   Component Library Scroll List
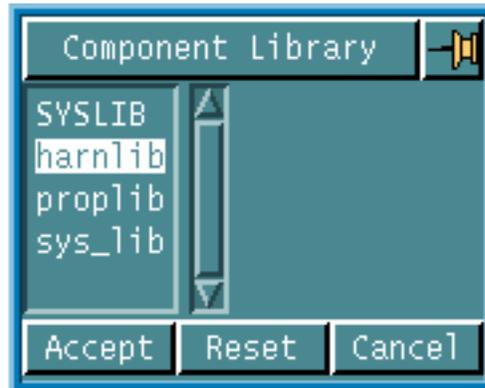
Choose a component library from the list. The default component library is
SYSLIB.

• Percentage Fill — Enter the fill limit for the channel. Percentage fill is the
percentage of the available cross-sectional area within a channel that can be
filled by the routed harness. The total cross section of all the wires should not
exceed the available cross section area of the channel. The percentage fill limit
cannot be greater than 100.

• List Attributes in Report Window — Click List Attributes to display the
parametric harness attributes and their global default values in the report
window.

• Highlight Color — Select a color for highlighting a channel if it fails the bend
radius check while routing.

• Display Mode — Specify the default modeling mode to be used for inserting
and editing components.

    • Conceptual - In the conceptual modeling mode, the netlist file is not
    referred to while inserting or editing harness components. If a netlist file
    name is specified, it is ignored.

    • Netlist - In netlist modeling mode, the netlist file is referred for inserting and
    editing harness components. This mode can be selected only if a netlist file
    name is specified in the currently active version.

Modeling a Harness Part

Chapter 4

This chapter describes the modeling of a harness layout part. This includes insertion and editing of harness components, and routing of channels and editing them. These options are available from the Harness task set in the Parametric environment. The topics discussed in this chapter are

- Overview
- Inserting Harness Components
- Inserting Connectors
- Inserting Break Connectors
- Inserting a Junction
- Inserting Clips
- Inserting Connection Points
- Editing Harness Components
- Routing Channels
- Editing a Channel
- Inserting a Drip Loop
- Inserting a Service Loop
- Inserting a Sag

Overview

The modeling tasks in harness layout include

- Inserting Harness Components

    You can insert the following harness components

    - Connectors

    - Junctions

    - Clips

- Editing Harness Components

- Routing Channels

- Editing Channels

All the above options are available from the Harness task set.

Inserting Harness Components

Choose the Insert Harness Component option from the Harness task set to insert a component into a harness model. Choosing this option displays the Harness Components palette.

OR

Use the INSERT HDCOMPONENT command in the CADDS text window to insert a harness component. See the on-line help for further details on the command-line options.

You can insert the following harness components into a model using an available component library

- Connectors:

    Connectors are electro-mechanical components used to connect two or more channels.

- Break Connectors:

    Break Connectors are electro-mechanical components used to split a complex wire harness into smaller harnesses, making it easier to manufacture and connect in an assembly.

    Break Connectors exist in pair of a male connector and a female connector. This male-female connector pair is inserted with one command.

- Junctions:

    Junctions are logical components that represent a channel junction or branch.

- Clips:

    Clips are mechanical fastening devices associated to channels.

The above components are represented by CADDS Parametrics features and a default set is supplied as part of the Harness Layout component library. You can insert components from the supplied component library or optionally, create your own library of components. For details see Appendix A, "Creating a Component Library".

You can insert the components both before and after routing the channels. In both the cases the components are associated to appropriate channels. A component can be moved or positioned relative to other geometry using the CHANGE PARAMETER command available in the Parametric environment.

Please note: A channel is associated to a component if the channel is inserted first and the channel is routed to the inserted component. If the channel is routed first and the component is inserted later, you must edit the channel and end it on the component to ensure that the channel gets associated with the component.

You can insert the harness component either in the conceptual mode or in the netlist mode. In conceptual mode the netlist file is not activated before inserting the components and you can specify any name for the inserted component. In the netlist mode, only the components that are defined in the netlist file can be inserted.

The attributes of a component can be edited using the Edit Harness Components option.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Insert Harness Component option from the Harness task set. The Harness Components palette appears.

Figure 4-1   Harness Components Palette
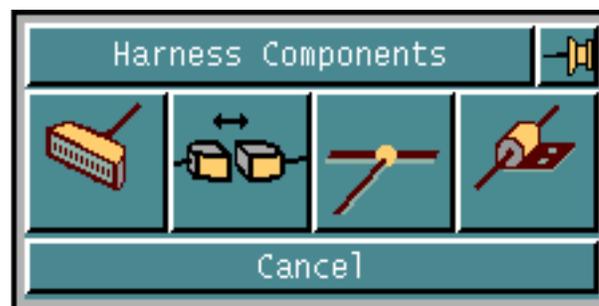
Procedure

You can use the options on the Harness Components palette for

- Inserting Connectors
- Inserting Break Connectors
- Inserting Junctions
- Inserting Clips

Inserting Connectors

Choose the Insert Connector option from the Insert Components palette to insert a connector into a harness model. Choosing this option displays the Insert Connector property sheet.

OR

Use the INSERT HDCOMPONENT command in the CADDS text window to insert a connector. See the online help for further details on the command-line options.

If the netlist is not loaded before inserting the connector, then only conceptual modeling mode is active. If the netlist is loaded, you can insert a connector in either conceptual or netlist mode.

Using this Option

1. Choose the Insert Connector option from the Harness Components palette. Depending on the modeling mode specified using the Select Attributes option, the Insert Connector property sheet appears.

Procedure

You can specify the following

- Connector Name
- Connector Part Name
- Part Number
- Orientation
- Modeling Mode
- Application and User Defined Attributes

Connector Name:

Connector name is the name by which the connector is referred to. This name can be different from the connector part name. This connector name is stored as a text string on the attribute compname attached to the component once it is inserted into the CADDS model.

In the Netlist modeling mode, since an active netlist file is accessible, you can select connector names from the currently active netlist file. You can do this by selecting the connector name from the scroll list or enter a name in the Connector Name field.

In the Conceptual mode, you cannot select connector names from the netlist file (the Connector Name scroll list does not appear). You have to type a name in the Connector Name field.

• Enter the reference name for the connector in the Connector Name field. In the netlist mode, click the list button to display a list of available connector names and then select the required connector name from the list.

Figure 4-2   Connector Names Scroll List
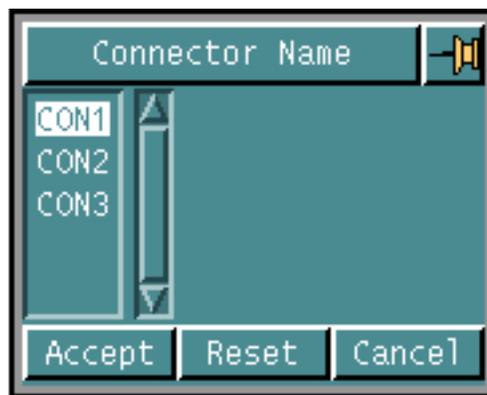
Connector Part Name:

The Connector Part Name is the connector part to be inserted from the component library. The component library name is specified in the Component Library Name field as set in the Harness Layout Attributes Property Sheet.

The connector part name is associated with a connector name in the netlist file. In netlist modeling mode, when you select a connector name specified in the netlist, the Connector Part Name field is automatically filled with the associated connector part name defined in the netlist.

• Enter the connector part name in the Connector Part Name field. Click the list button to display a list of available connector part names.

• Enter the connector part name in the Connector Part Name field or click the list button (in the netlist mode) to display a list of available connector part names and then select the required connector part name from the list. This list does not appear in the Conceptual mode.

Figure 4-3   Connector Part Name Scroll List
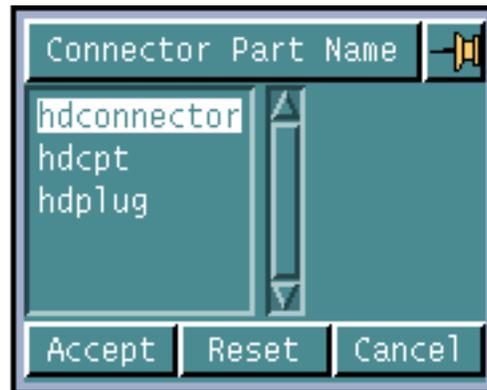
Part Number:

The component part number is like a stock number, library number or the manufacturer's part number. Its value is stored in the `partno` attribute on the component.

The default value is specified on the connector component during its creation.

*   Enter a Part Number for the connector being inserted in the field. The part number provided in the netlist is displayed by default. You can override this and enter another value if you wish.

Orientation:

Specify the orientation for the connector using one of the following options.

*   Click Cplane and specify the construction plane name in the field for insertion of the connector, or click the list button to display the Cplane scroll list.   The active Cplane is the default insertion plane. The X, Y and Z axes of the connector are defined when teaching the feature that represents the connector in the model. The X, Y and Z axes of the connector are aligned with the X, Y and Z axes of the selected Cplane.

Figure 4-4   Cplane Scroll List
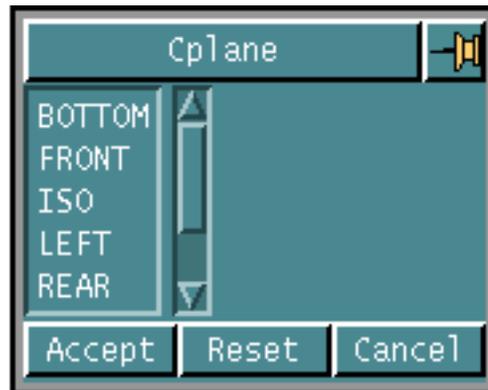
Choose one of the cplanes from the list. The connector is oriented with respect to the specified cplane.

OR

- Click 3 Points to define a Cplane for the connector by selecting three points. The first selection indicates the origin of the Cplane, the second selection indicates the direction of the X-axis, and the third selection indicates the direction of the Y-axis. The connector is inserted relative to the this runtime Cplane. The X, Y and Z axes of the connector are aligned with the X, Y and Z axes of the runtime Cplane.

Please note: Since a Cplane defined using 3 Points method does not have any name, it can not be referenced for subsequent insertions of components.

Modeling Mode:

Specify one of the following modeling modes for the insertion of a connector. A `cflag` logical attribute is stored on the connector to indicate the mode in which the connector was created.

- Click Conceptual Modeling to insert a connector without referencing the netlist file. The `cflag` is set to true in this mode.

OR

- Click Netlist Modeling to insert a connector that is defined in the active netlist file. The `cflag` is set to false in this mode.

Application and User Defined Attributes:

Specify the attributes for the connector to be inserted. All the attributes for the specified connector, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list to display the String field.

Figure 4-5   String Field

Enter the new value for the selected attribute in the String field.

You can specify the following for the specified attribute

- Name: The name for the attribute

- Type: The type of the attribute. For example, real, integer, length and so on.

- Value: The value of the attribute.

- Units: The units for the attribute, such as mm, inch and so on.

- Condition: The condition (>, < or =) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Apply:

Click Apply to display the Apply menu.

Figure 4-6   Apply Menu

You can:

- Click Abort to abort the command.

- Click Rotate to rotate the Cplane. The Axis of Rotation property sheet appears.

Figure 4-7   Axis of Rotation Property Sheet
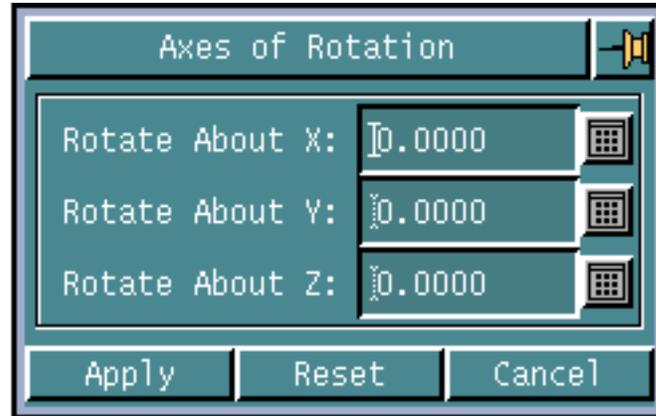
Specify the angles of rotation about X, Y and Z axes, with respect to the active Cplane.

- Click Origin to specify the insertion point for the connector to be inserted.

Please note: Use the Rotate option before using the Origin option.

- Once the origin is specified, the Display option can be used. Click Display to display the connector geometry at the insertion point or click Go to insert the connector and end the task.

Example

Inserting a connector with the Library Part Name, "PLUG", connector name, "CON1", and part number, "ESC4" using the TOP CPlane, in conceptual modeling mode:

INSERT HDCOMPONENT PLUG **APPLI** NOAPPLY **NAME** CON1 **PARTNO** ESC4 **ORIENTATION CPNAME** TOP **CFLAG** TRUE **ORIGIN** Dig1 **GO**

Inserting Break Connectors
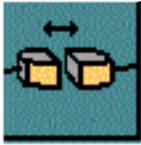
Choose the Insert Break Connector option from the Insert Components palette to insert a break connector into a harness model. Choosing this option displays the Insert Break Connector property sheet.

OR

Use the INSERT HDCOMPONENT command in the CADDS text window to insert a connector. See the on-line help for further details on the command-line options.

The Harness Components Library specified using the Select Attributes option is automatically activated on choosing this option.

A default break connector, hdbreakconnector is supplied as part of the Harness Layout component library, harnlib. You can insert break connectors from this library or from a user-defined library of components.

Using this Option

1. Choose the Insert Break Connector option from the Harness Components palette. Depending on the modeling mode specified using the Select Attributes option, the Insert Break Connector property sheet appears.

Procedure

You can specify the following

* Break Connector Part Name
* Break Connector Name
* Part Number
* Orientation
* Insertion
* Modeling Mode
* Application and User Defined Attributes

Break Connector Part Name:

Specify the name of the break connector part to be inserted.

• Enter the break connector part name in the Break Connector Part Name field or click the list button to display a list of available connector names from the active Harness Component Library and then select the required the part name from the list.

Figure 4-8   Break Connector Part Name Scroll List
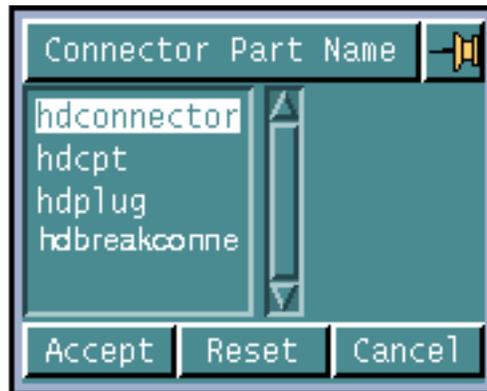
On selecting a valid break connector part name, the break connector attributes appear in the User Defined Attributes scroll list.

Break Connector Name:

Break Connectors consist of male and female connector components. The male and female connector components have their own connector names.

• Male Connector Name: This is the reference designator or reference name of the male component of the break connector. The male connector name is stored as a text string on the attribute malecompname attached to the component once it is inserted into the CADDS model.

• Female Connector Name: This is the reference designator or reference name of the female component of the break connector. The female connector name is stored as a text string on the attribute femalecompname attached to the component once it is inserted into the CADDS model.

In the Netlist Modeling mode:

• Enter the male or female connector name directly.

  OR

• Click the list button to display a list of connector names found in the netlist but not yet inserted into the model, as shown in the following figure. Select the required connector name from the list.

Figure 4-9   Connector Names Scroll List
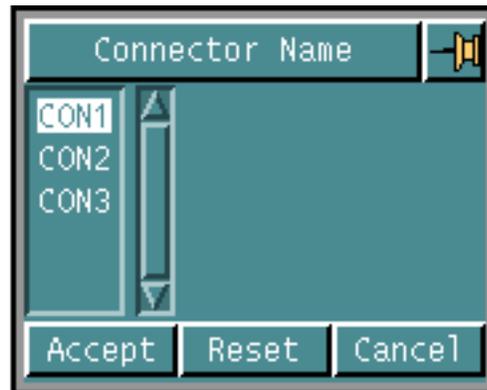
In the Conceptual Modeling mode, the scroll list is not displayed for selection.

Part Number : Apart number is like a stock number, library number or manufacturer's part number of the male connector in the break connector. The Male or Female components of a break connector have different part numbers.

In the Netlist Modeling mode, the part number from the netlist is displayed automatically. You can override this and enter another value.

• Male Part Number: Enter a Part Number for the male component of the connector being inserted. This value is stored with the `malepartno` attribute on the component.

• Female Part Number: Enter a Part Number for the female component of the connector being inserted. This value is stored with the `femalepartno` attribute on the component.

Orientation:

Specify the orientation for the connector using one of the following options. This option is valid only in Free Space Insertion mode.

• Click Cplane and enter the name of the CPlane or construction plane or click the list button to display the Cplane scroll list. The inserted component is oriented with respect to the specified CPlane. The default insertion plane is the active CPlane.

Figure 4-10 Cplane Scroll List
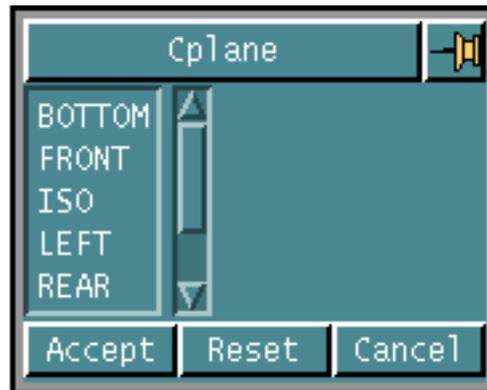
Choose one of the Cplanes from the list. The connector is oriented with respect to the specified cplane.

OR

- Click 3 Points to define a Cplane for the connector by selecting three points. The first selection indicates the origin of the CPlane, the second selection indicates the direction of the X-axis, and the third selection indicates the direction of the Y-axis.

Please note: Since a Cplane defined using 3 Points method does not have any name, it can not be referenced for subsequent insertions of components.

Insertion:

Specify whether the break connectors must be inserted on an existing channel or in free space.

- In-line: Specifies that the break connector must be inserted on an existing harness channel, breaking the channel into two channels. Since the break connector is aligned along the direction of the channel at the insertion point of the break connector, the Cplane definition is not considered in this mode. The channel is oriented in the direction of the channel selected, and it satisfies the tangency and direction constraints defined on the connect points of the break connector.

- Free Space: Specifies that the break connector must be inserted in free space using the Cplane selected or defined.

Modeling Mode:

Specify one of the following modeling modes for the insertion of a connector. A `cflag` logical attribute is stored on the connector to indicate the mode in which the connector was created.

The modeling mode set using the Select Attributes option is used as the default modeling mode.

- Click Conceptual Modeling to insert a break connector without referencing the netlist file. The `cflag` is set to true in this mode.

  OR

- Click Netlist Modeling to insert a connector that is defined in the active netlist file. The `cflag` is set to false in this mode.

Application and User Defined Attributes:

Specify the attributes for the connector to be inserted. All the attributes for the specified connector, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list. For string attributes, the following property sheet appears.

Figure 4-11 String Field

Enter the new value for the selected attribute in the String field.

You can specify the following for the specified attribute

- Name: The name for the attribute
- Type: The type of the attribute. For example, real, integer, length and so on.
- Value: The value of the attribute.
- Units: The units for the attribute, such as mm, inch and so on.
- Condition: The condition ($>$, $<$ or $=$) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Apply:

Click Apply to display the Apply menu. The Apply menu changes depending upon the options selected in the property sheet. It guides you to select various options depending upon whether the break connector is being inserted in free space or in-line.

You can:

- Click Abort to abort the command.
- Click Rotate to rotate the Cplane. The Axis of Rotation property sheet appears.

Figure 4-12 Axis of Rotation Property Sheet
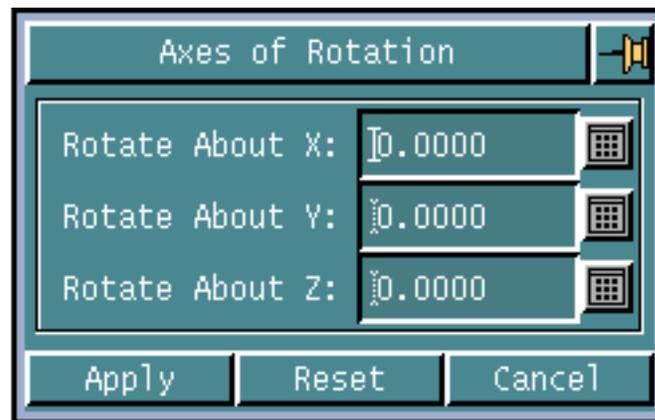
Specify the angles of rotation about X, Y and Z axes, with respect to the active Cplane.

- Click Channel to insert a break connector on an existing harness channel, splitting the harness channel into two sub-channels. This option can be selected only if the In-Line option is selected.
- Click Origin to specify the insertion point for the connector to be inserted.

Please note: Use the Rotate option before using the Origin option.

- Click Display to display the break connector geometry at the insertion point.
- Click Go to create the break connector in the database and end the task.

Example

The command to insert a break connector hdbreakconnector in-line, on an existing channel is as follows:

```
INSERT HDCOMPONENT hdbreakconnector APPLI Break MALECOMPNAME
B-CON1-A FEMALECOMPNAME BCON1-B MALEPARTNO ESC4001_A FEMALEPARTNO
ESC4001_B CFLAG true WORKPIECE <dig1 on or near channel> ORIGIN
<dig2 on channel> GO
```

The inserted break connector splits the channel into two channels as shown below:

Figure 4-13 Break Connector Inserted In-Line
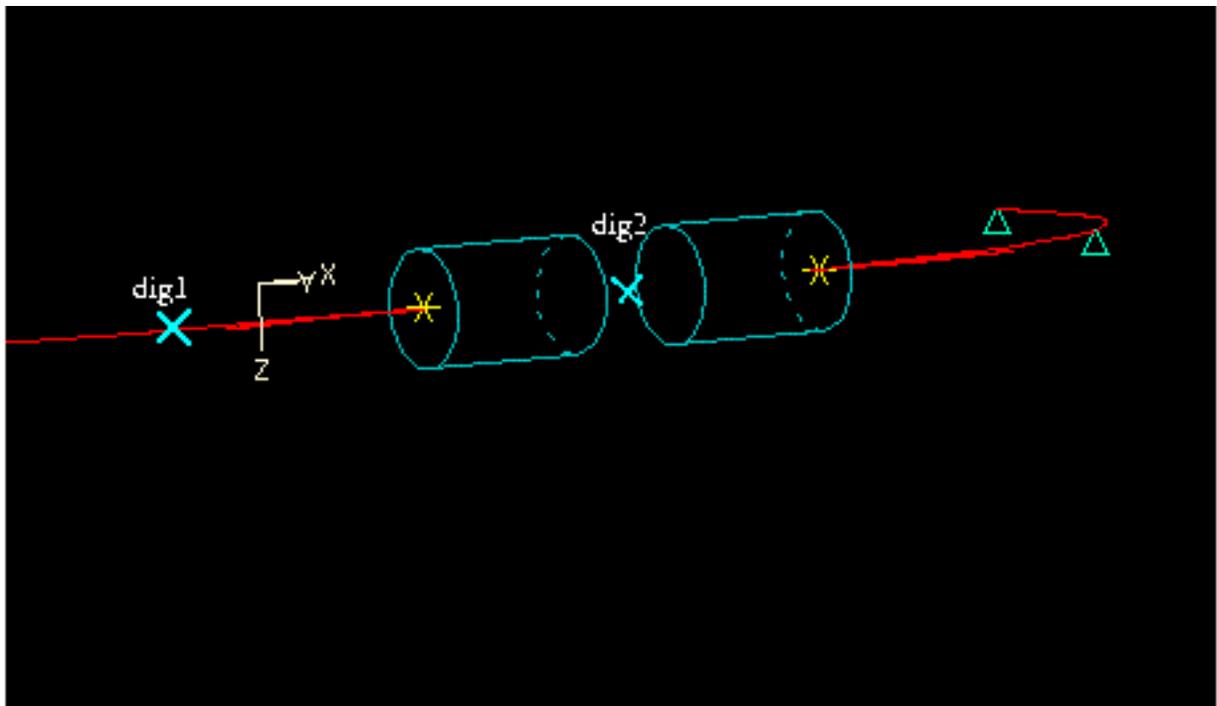
Inserting a Junction

Choose the Insert Junction option from the Insert Components palette to insert a junction into a harness model.Choosing this option displays the Insert Junction property sheet.

OR

Use the INSERT HDCOMPONENT command in the CADDS text window to insert a junction. See the online help for further details on the command-line options.

Using this Option

1. Choose the Insert Junction option from the Harness Components palette. The Insert Junction property sheet appears.

Procedure

You can specify the following in the Insert Junction property sheet:

• Junction Part Name

• Orientation

• Application and User-defined Attributes

Junction Part Name:

The Junction Part Name specifies the name of the junction part to be inserted from the component library. The component library name is specified in the Component Library Name field in the Harness Layout Attributes Property Sheet.The Junction Part Name has to be specified by the user since it is not specified in the netlist.

• Enter the Junction Part Name in the field or click the list button to display a list of valid junctions from the active harness component library and then select the required part name from the list.

Figure 4-14 Junction Part Names Scroll List
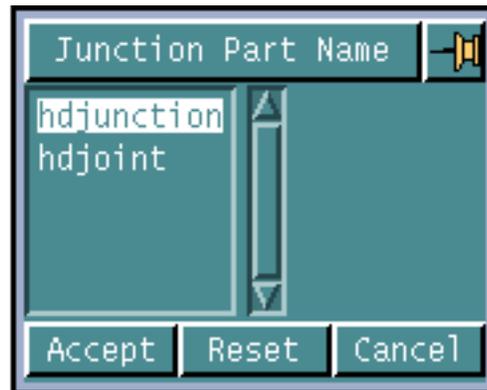
Orientation:

Specify the orientation for the junction using one of the following options.

• Click Cplane and specify the construction plane name in the field for insertion of the junction, or click the list button to display the Cplane scroll list and then select the required Cplane from the list.

Figure 4-15 Cplane Scroll List
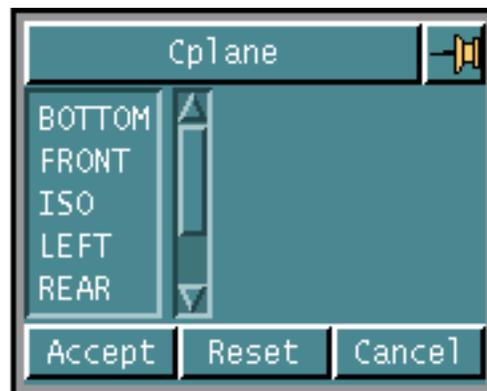
The junction is oriented with respect to the specified cplane. The active Cplane is the default Cplane.

OR

• Click 3 Points to define a Cplane for the junction by selecting three points. The first selection indicates the origin of the Cplane, the second selection indicates the direction of the X-axis, and the third selection indicates the direction of the Y-axis.

Please note: Since a Cplane defined using 3 Points method does not have any name, it can not be referenced for subsequent insertions of components.

Application and User-defined Attributes:

Specify the attributes for the junction to be inserted. All the attributes for the specified junction, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list. For string attributes, the following property sheet appears.

Figure 4-16 String Field

Enter the new value for the selected attribute in the String field.

You can specify the following for the specified attribute

- Name: The name for the attribute
- Type: The type of the attribute. For example, real, integer, length and so on.
- Value: The value of the attribute.
- Units: The units for the attribute, such as mm, inch and so on.
- Condition: The condition (>, < or =) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Apply:

Click Apply to display the Apply menu.

You can

- Click Abort to abort the command.
- Click Rotate to rotate the Cplane. The Axis of Rotation property sheet appears.

Figure 4-17 Axis of Rotation Property Sheet
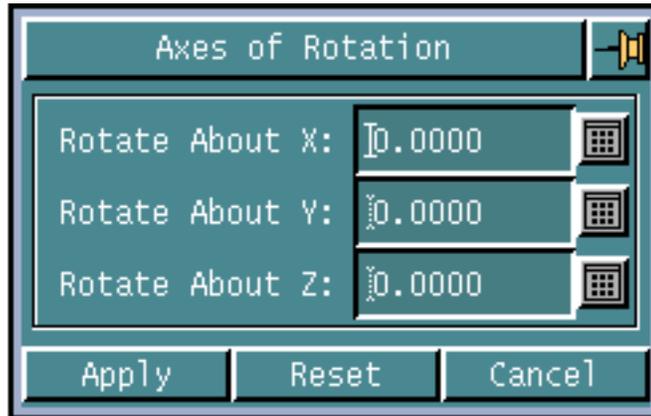
Specify the angles of rotation about X, Y and Z axes, with respect to the active Cplane.

- Click Origin to specify the insertion point for the connector to be inserted.

Please note: Rotate the Cplane before inserting the component. Use the Rotate option before you use the Origin option.

- Once the origin is specified, the Display option can be used. Click Display to display the junction geometry at the insertion point or click Go to insert the junction and end the task.

Example

Inserting a junction with the Library Part Name, "JUNCT" using the TOP CPlane:

INSERT HDCOMPONENT JUNCT **APPLI** NOAPPLY **CPNAME** TOP **ORIGIN** Dig1 **GO**

The inserted junction using interactive shading is displayed on the screen as shown below.
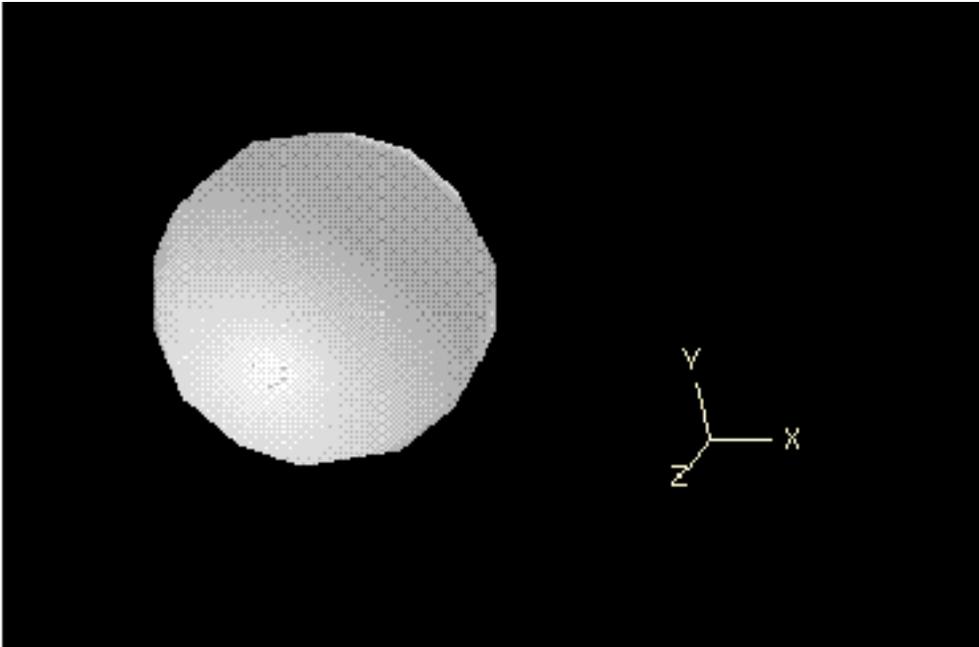
Inserting Clips

Choose the Insert Clip option from the Insert Components palette to insert a clip into a harness model. Choosing this option displays the Insert Clip property sheet.

OR

Use the INSERT HDCOMPONENT command in the CADDS text window to insert a clip. See the online help for further details on the command-line options.

Using this Option

1. Choose the Insert Clip option from the Harness Components palette. The Insert Clip property sheet appears.

Procedure

You can specify the following

- Clip Name
- Clip Part Name
- Part Number
- Orientation
- Application and User-defined Attributes

Clip Name:

Clip name is the name by which the clip is referred to. This name can be different from the library part name. The clip name specified on the clip part is displayed by default. You can change this name if you wish to. The clip name is stored as a text string on the attribute compname attached to the component once it is inserted into the CADDS model.

The clip name is not specified in the netlist.

- Enter the reference name for the clip in the Clip Name field. You can use the default value stored on the clip or type in the value in the field.

Clip Part Name:

The Clip Part Name specifies the name of the clip part to be inserted from the component library.

- Enter the clip part name in the Clip Part Name field or click the list button to display the Clip Part Name scroll list and then select the required part name from the scroll list.

Figure 4-18 Clip Part Name Scroll List
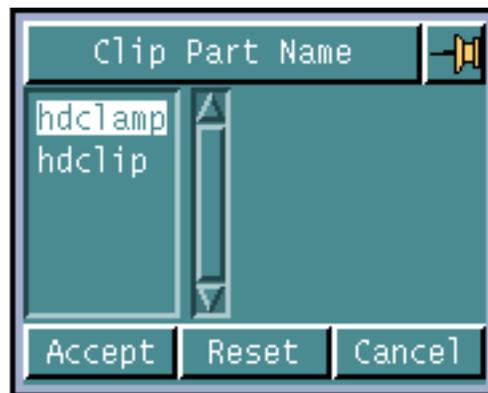
Part Number:

The component part number is like a stock number, library number or manufacturer's part number. Its value is stored in the `partno` attribute on the component.

- Enter a Part Number for the clip being inserted in the field. You can use the default value stored on the clip or type a new value in the field.

Orientation:

Specify the orientation for the clip using one of the following options.

- Click Cplane and specify the construction plane name in the field for insertion of the clip, or click the list button to display the Cplane scroll list.

Figure 4-19   Cplane Scroll List
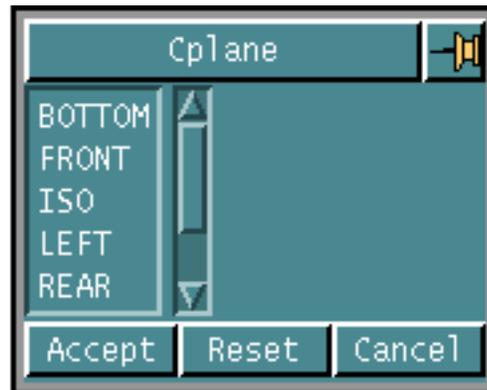
Choose one of the cplanes from the list. The clip is oriented with respect to the specified cplane. The active Cplane is the default Cplane.
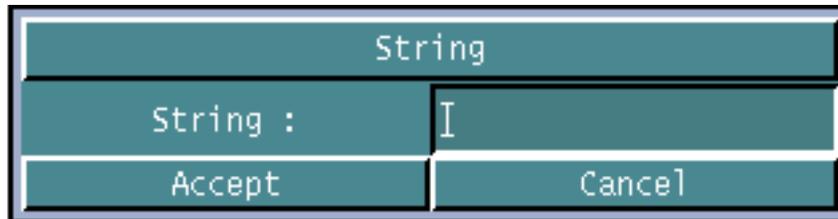
OR

- Click 3 Points to define a Cplane for the clip by selecting three points. The first selection indicates the origin of the Cplane, the second selection indicates the direction of the X-axis, and the third selection indicates the direction of the Y-axis.

Please note: Since a Cplane defined using 3 Points method does not have any name, it can not be referenced for subsequent insertions of components.

Application and User-defined Attributes:

Specify the attributes for the clip to be inserted. All the attributes for the specified clip, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list. For string attributes, the following property sheet appears.

Figure 4-20   String Field

Enter the new value for the selected attribute in the String field.

You can specify the following for the specified attribute

- Name: The name for the attribute

- Type: The type of the attribute. For example, real, integer, length and so on.

- Value: The value of the attribute.

- Units: The units for the attribute, such as mm, inch and so on.

- Condition: The condition (>, < or =) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Apply:

Click Apply to display the Apply menu.

You can:

- Click Abort to abort the command.

- Click Rotate to rotate the Cplane. The Axis of Rotation property sheet appears.

Figure 4-21 Axis of Rotation Property Sheet
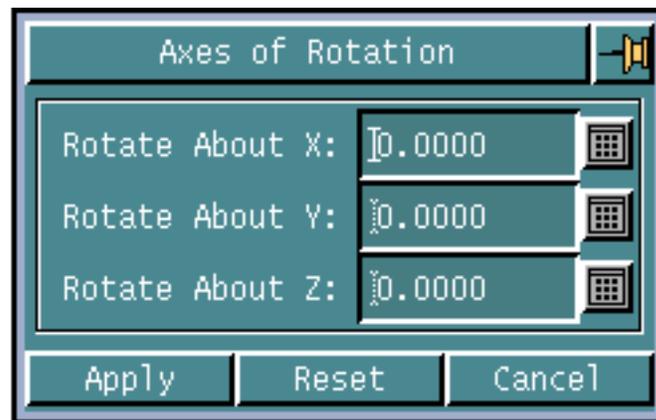
Specify the angles of rotation about X, Y and Z axes, with respect to the active Cplane.

- Click Origin to specify the insertion point for the connector to be inserted.

Please note: Use the Rotate option before using the Origin option.

- Once the origin is specified, the Display option can be used. Click Display to display the clip geometry at the insertion point or click Go to insert the clip and end the task.

Example

Inserting a clip with the Library Part Name, "CLIP", clip name, "CLIP1", and part number, "ESC444" using the TOP CPlane:

INSERT HDCOMPONENT CLIP **APPLI** NOAPPLY **NAME** CLIP1 **PARTNO** ESC444 **ORIENTATION CPNAME** TOP **ORIGIN** Dig1 **GO**

The inserted clip using interactive shading is displayed on the screen as shown below
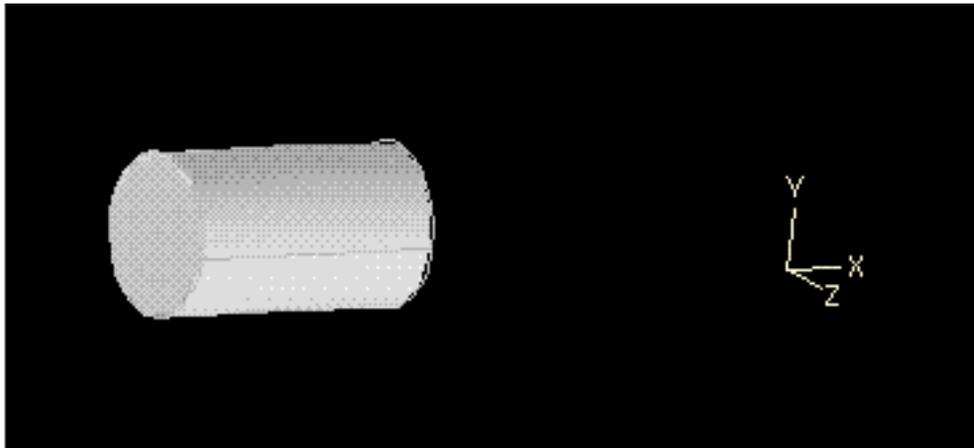
Inserting Connection Points

Choose the RSYS-TOOLS option in the Parametric environment to insert connection points while creating a Harness Design component.

OR

Use the INSERT CONNECTPOINT command in the CADDS text window to insert a connection point. See the online help for further details on the command-line options.

The connection point can be parametrically associated with harness components (features representing connectors or clips) and used for channel routing.

Connection points associated with connectors and junctions serve as origination/termination points of channels for the ROUTE CHANNEL command. Connection points associated with clips serve as reference points through which channels can be routed.

A multiple connect point junction consists of one center connection point and any number of reference connection points. The tangency and direction constraints are defined on the reference connect points. A channel always terminates or originates on the center connect points, optionally passing through the selected reference connect point satisfying the tangency and direction constraints defined on the reference connect point.

If the connection point is associated with a feature, you can change the parameter using the Change Feature menu.

Using this Option

1. Choose RSYS-TOOLS from the Parametric environment. The Rsys Toolbox, as shown below, appears.

Figure 4-22 Rsys Toolbox
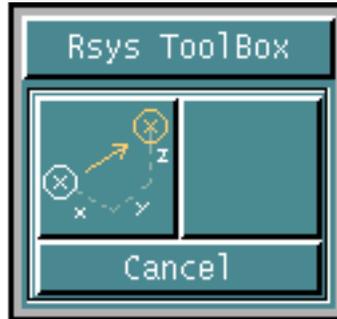
Procedure

1. Click the Insert Connect Point icon. The Connect Point Insertion Menu appears.

Figure 4-23 Connect Point Insertion Menu
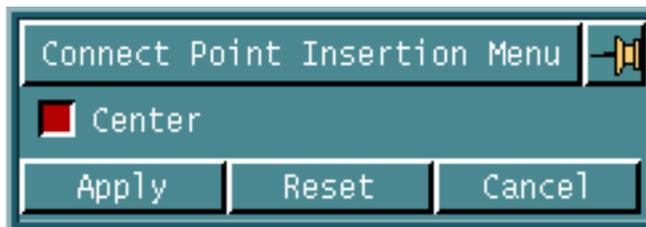
2. Click Center to specify the location of the center connect point.

3. Click Apply.

4. Select the location of the center connect point on the screen. You can use options like END, MID, INTOF, ORG, and NEAR to associate the location with a part/feature.

5. Specify the direction of the connect point. This location can be parameterized with respect to the first location. (for example, you can use the DXYZ option on the coordinate keypad).

   You can change the parameter of the connect point using the EDIT HDCOMPONENT or the CHANGE PARAMETER command.

Please note: This option must be used while creating a feature only and all harness features must have Connection points.

Editing Harness Components

Choose the Edit Harness Component option from the Harness task set to edit the attributes of an existing component in a harness model. Choosing this option displays a property sheet for editing various attributes of the selected harness component.

OR

Use the EDIT HDCOMPONENT command in the CADDS text window to edit attributes of an existing harness component. See the on-line help for further details on the command-line options.

You can edit the attributes of the following harness components

- Connectors
- Break Connectors
- Junctions
- Clips

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Edit Harness Component option from the Harness task set.

3. Select one of the following existing harness components for editing from the harness model

   - Connector
   - Break Connector
   - Junction
   - Clip

   Depending on the component you select, a property sheet for editing that component appears.

   If you select a connector for editing, the Edit Connector Attributes property sheet appears. For details, refer to "Editing a Connector" on page 4-32.

   If you select a break connector for editing, the Edit Break Connector Attributes property sheet appears. For details, refer to "Editing a Connector" on page 4-32.

If you select a junction for editing, the Edit Junction Attributes property sheet appears. For details, refer to "Editing a Junction" on page 4-38.

If you select a clip for editing, the Edit Clip Attributes property sheet appears. For details, refer to "Editing a Clip" on page 4-40.

Editing a Connector

If you select a connector from the harness model, the Edit Connector Attributes property sheet appears.

Procedure

You can edit the following

- Connector Name

- Part Number

- Modeling Mode

- Application and User Defined Attributes

Connector Name:

Connector name is the name by which the connector is referred to. This name can be different from the library part name. The connector name is defined during connector insertion into the model, and is stored in the `compname` attribute on the connector. In conceptual mode you can specify any name for the connector. In case of netlist mode, you need to choose one from the names available in the netlist file that is not yet used in the current model.

- Enter a new name for the connector in the Connector Name field. In case of netlist mode, click the list button to display a list of available connector names and then select the required name from the list.

Figure 4-24 Connector Names Scroll List
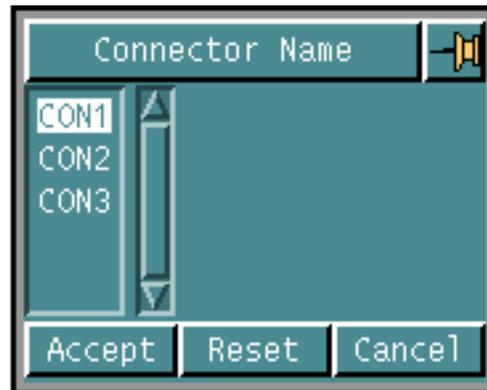
Please note: Only names of connectors that have not yet been inserted in the model appear in this list.

Part Number:

The component part number is like a stock number, library number or manufacturer's part number. Its value is stored in the `partno` attribute on the component.

*   Enter a new Part Number for the connector in the field. The part number is specified in the netlist. You can override this and enter another value if you wish.

Modeling Mode:

Specify one of the following modeling types for the connector. This flag is stored as a logical value on the attribute cflag attached to the connector.

*   Click Conceptual Modeling to change over to the conceptual mode. In this mode you can edit a connector name without referencing the netlist file.

    OR

*   Click Netlist Modeling to change over to the netlist mode. In this mode you can edit only such a connector name, which is defined in active netlist file.

Application and User Defined Attributes:

Edit the attributes for the selected connector. All the attributes for the selected connector, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list. For string attributes, the following property sheet appears.

Figure 4-25 String Field

Enter the new value for the selected attribute in the String field.

You can edit the following for the specified attribute

- Name: The name for the attribute

- Type: The type of the attribute. For example, real, integer, length and so on.

- Value: The value of the attribute.

- Units: The units for the attribute, such as mm, inch and so on.

- Condition: The condition (>, < or =) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Example

Changing a connector inserted in conceptual mode to the netlist mode and changing its connector name and part number.

- **EDIT HDCOMPONENT NAME** CON1 **PARTNO** ESC45 **CFLAG** False **GO**

Editing a Break Connector

If you select a break connector from the harness model, the Edit Break Connector Attributes property sheet appears. Depending upon the mode of insertion, the Edit Break Connector Attributes property sheet is displayed.

Procedure

You can edit the following

- Break Connector Name

- Part Number

- Modeling Mode

- Application and User Defined Attributes

Break Connector Name:

Break Connectors consist of male and female connector components. The male and female connector components have their own connector names. These connector names are defined while inserting the break connector and are stored on the break connector.

- Male Connector Name: Enter the new name for the male component of a break connector.

- Female Connector Name: Enter the new name for the female component of a break connector.

In the Netlist Modeling mode:

- Edit the male or female connector name directly.

  OR

- Click the list button to displays a list of connector names found in the netlist but not yet inserted into the model, as shown in the following figure. Choose a new connector name from the list.

Figure 4-26 Break Connector Names Scroll List
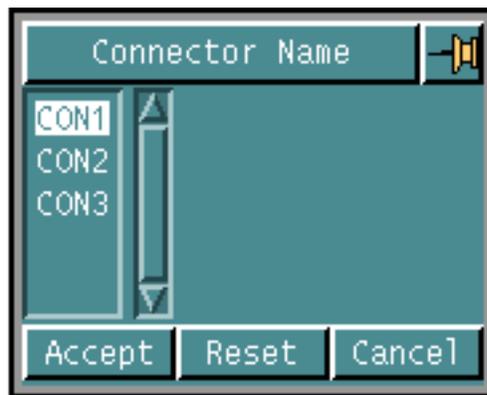
In the Conceptual Modeling mode, the scroll list is not displayed for selection.

Part Number :

Break Connectors consist of male and female connector components. Each component has its own part number.

- Male Part Number: The Male part number is like a stock number, library number or manufacturers part number of the male connector in the break connector and its value is stored with the `malepartno` attribute on the component.

- Female Part Number: The Female Part Number is like a stock number, library number or manufacturers part number of the female connector in the break connector and its value is stored with the `femalepartno` attribute on the component.

The default value is specified on the break connector component during its creation.

- Enter a Part Number for the connector being inserted in the field. In the Netlist Modeling mode, the part number from the netlist is displayed automatically. You can override this and enter another value. In the Conceptual Modeling mode, enter the new value of the part number.

Modeling Mode:

Specify one of the following modeling types for the connector. This flag is stored as a logical value on the attribute `cflag` attached to the connector.

- Click Conceptual Modeling to change over to the conceptual mode. In this mode you can edit a break connector name without referencing the netlist file.

  OR

- Click Netlist Modeling to change over to the netlist mode. In this mode you can edit only such a break connector name, which is defined in active netlist file.

Highlight:

Click Highlight to highlight the male or female connector of a break connector.

- Male Connector: Highlights the male component of the break connector component.

- Female Connector: Highlights the female component of the break connector component.

The male and female components of a break connector are never highlighted together at the same time.

Swap Attributes:

Click Swap Attributes to swap the male and female connector name and part number attributes of a break connector.

Swap Geometry:

Click Swap Geometry to swap the male and female connectors of a break connector.

Application and User Defined Attributes:

Click Application and User Defined Attributes to edit the attributes of the selected connector. All the attributes for the selected connector, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list. For string attributes, the following property sheet appears.

Figure 4-27 String Field

Enter the new value for the selected attribute in the String field.

You can edit the following for the specified attribute

- Name: The name for the attribute

- Type: The type of the attribute. For example, real, integer, length and so on.

- Value: The value of the attribute.

- Units: The units for the attribute, such as mm, inch and so on.

- Condition: The condition ($>$, $<$ or $=$) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Apply:

Click Apply to execute the command. The attributes of the selected component are updated with the new attribute values specified in the Edit Break Connector property sheet.

Example

If Netlist Modeling is active and the selected break connector name is not found in the active netlist, edit the break connector name to one contained in the active netlist as follows:

```
EDIT HDCOMPONENT * MALECOMPNAME BCON1-A MALEPARTNO ESC4001_B
FEMALECOMPNAME BCON1-B FEMALEPARTNO ESC4001_B CFLAG False GO
```

Editing a Junction

If you select a junction from the harness model, the Edit Junction Attributes property sheet appears.

Procedure

You can edit the following

• Junction Part Name

• Orientation

• Application and User-defined Attributes

Junction Part Name:

The Junction Part Name specifies the name of the junction part to be inserted from the component library. The component library name is specified in the Component Library Name field in the Harness Layout Attributes Property Sheet.

• Enter the Junction Part Name in the field or click the list button to display a list of valid junctions from the active harness component library and then select the required name from the list.

Figure 4-28 Junction Part Names Scroll List
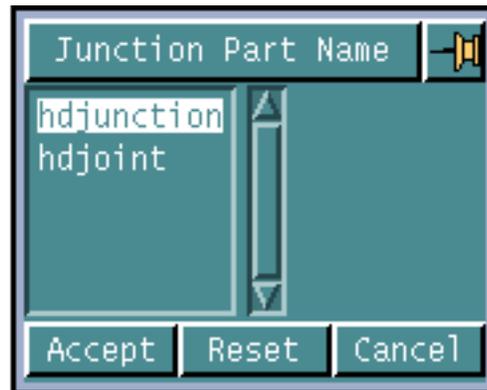
Orientation:

Specify the orientation for the junction using one of the following options.

- Click Cplane and specify the construction plane name in the field for insertion of the junction, or click the list button to display the Cplane scroll list and then select the required Cplane from the list.

Figure 4-29 Cplane Scroll List
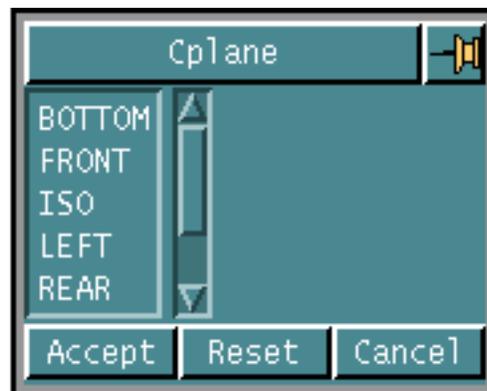
The junction is oriented with respect to the specified cplane. The active Cplane is the default Cplane.

OR

- Click 3 Points to define a Cplane for the junction by selecting three points. The first selection indicates the origin of the Cplane, the second selection indicates the direction of the X-axis, and the third selection indicates the direction of the Y-axis.

Please note: Since a Cplane defined using 3 Points method does not have any name, it can not be referenced for subsequent insertions of components.

Application and User-defined Attributes:

You can edit the attributes of the selected junction. All the attributes for the selected junction, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list. For string attributes, the following property sheet appears.

Figure 4-30 String Field

Enter the new value for the selected attribute in the String field.

You can specify the following for the specified attribute

- Name: The name for the attribute

- Type: The type of the attribute. For example, real, integer, length and so on.

- Value: The value of the attribute.

- Units: The units for the attribute, such as mm, inch and so on.

- Condition: The condition ($>$, $<$ or $=$) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Editing a Clip

If you select a clip from the harness model, the Edit Clip Attributes property sheet appears.

Procedure

You can edit the following

- Clip Name

- Part Number

- Attributes

Clip Name:

Clip name is the name by which the clip is referred to. This name can be different from the library part name. The clip name specified on the clip part is displayed by default. You can change this name if you wish to. The clip name is stored as an attribute `compname` on the clip during clip insertion into the model.

• Enter a new name for the clip in the Clip Name field.

Part Number:

The component part number is like a stock number, library number or manufacturer's part number. Its value is stored in the `partno` attribute on the component.

• Enter a new Part Number for the selected clip in the field.

Attributes:

Edit the attributes for the selected clip. All the attributes for the selected clip, that you have defined, are listed on the property sheet. To change the value of any of these attributes, choose the attribute from the list. For string attributes, the following property sheet appears.

Figure 4-31 String Field

Enter the new value for the selected attribute in the String field.

You can specify the following for the specified attribute

• Name: The name for the attribute

• Type: The type of the attribute. For example, real, integer, length and so on.

• Value: The value of the attribute.

• Units: The units for the attribute, such as mm, inch and so on.

• Condition: The condition ($>$, $<$ or $=$) with respect to the default value of the attribute. That is, if ">" is specified as the condition, then the value entered for the attribute must be greater than the default value, and so on.

Example

Changing the clip name and part number of a clip. The clip name is not defined in the netlist.

EDIT HDCOMPONENT NAME clip1 **PARTNO** ESC4001 **GO**

Routing Channels

Choose the Route Channel option from the Harness task set to route channels based on specified locations and/or harness components. The Route Channel menu appears.

OR

Use the ROUTE CHANNEL command in the CADDS text window to route a channel. See the online help for further details on the command-line options.

This option allows you to route channels based on specified locations and/or harness components.

During location based routing, the command routes the channel through the specified locations.

In case of an existing channel, the command connects to the end of the channel or branch based on the options selected. As the channel is being routed, straight line segments appear on the screen.

You can route a channel such that it

*   passes through the selected points in free space or with reference to other entities.

*   originates or terminates at the selected points in free space or with reference to other entities.

*   originates or terminates at the connection point of the selected connector, while maintaining the tangency and direction considerations for that connector.

*   originates or terminates at the connection point of the selected junction.

*   passes through the selected clips while maintaining the tangency conditions associated with the clips.

*   originates or terminates at the selected clip.

    If a junction is selected before a clip, the channel does not get routed automatically. This is because a clip is not a mandatory end point for a channel. You can select any number of clips after starting the channel on a junction. If the channel ends on the first clip selected, then you cannot select any more clips.

If a clip is selected before a junction, the channel is routed automatically. This is because a junction is a mandatory end point for a channel. After selecting a junction, you cannot select any other object.

- originates or terminates at an existing channel by automatically creating a branch. You can also specify the distance of this branch point from a reference point. This reference point could be:

    - Any harness components such as a connector, clip, or junction.

    - The start or end of the selected channel.

    - A channel branch location.

    - A parametric point created using parametric modifiers.

- originates or terminates at the end of a selected channel by automatically joining the new channel with the existing one, and thus extending the old channel.

The entire channel is created as a polycurve consisting of nsplines. The channels are created as soon as you select the free points. At the same time, the minimum bend radius check takes place and if the check fails, the channel is highlighted. The highlight color is set in the SELECT HDATTRIBUTES command. At this point, you can:

- correct the violation by deleting the previously selected free points and selecting new locations

    OR

- accept the minimum bend radius violation.

    Alternatively, you could abort the command completely.

The shape of the channel changes as more input points are selected. The channel remains highlighted until the bend radius check is satisfied.

Please note:

- A connector without any connection points is treated as an invalid connector. An error message is displayed if you select such a connector while routing a channel.

- If you use the netlist file while routing channels, and specify one of the following as a termination point for the channel, the channel being routed is automatically created and the command is terminated.

    - Connector

    - Junction

    - End of an existing channel

- Branch from another channel

- For parts having channels without the Pfill attribute, the Tarelen property instance on the channel must be replaced by the modified channel class. To replace the property instance:

  a. Detach the property using the DETACH PROPERTY command as follows:

  ```
  DETACH PROPERTY ENTITY <dig1>
  ```

  where **<dig1>** is the channel whose property needs to be updated.

  b. Attach the new property class using the INSERT PROPERTY command as follows:

  ```
  INSERT PROPERTY CHANNEL ATTACHTO ENTITY <dig1>
  ```

  where **<dig1>** is the channel whose property needs to be updated.

  c. Remove all unattached properties from the database using the REMOVE PROPERTY command as follows:

  ```
  REMOVE PROPERTY UNATTACHED REPORT
  ```

The channel is created on the layer specified by parameter 9 in the `param` file.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Route Channel option from the Harness task set. The Route Channel menu appears.

Figure 4-32 Route Channel menu
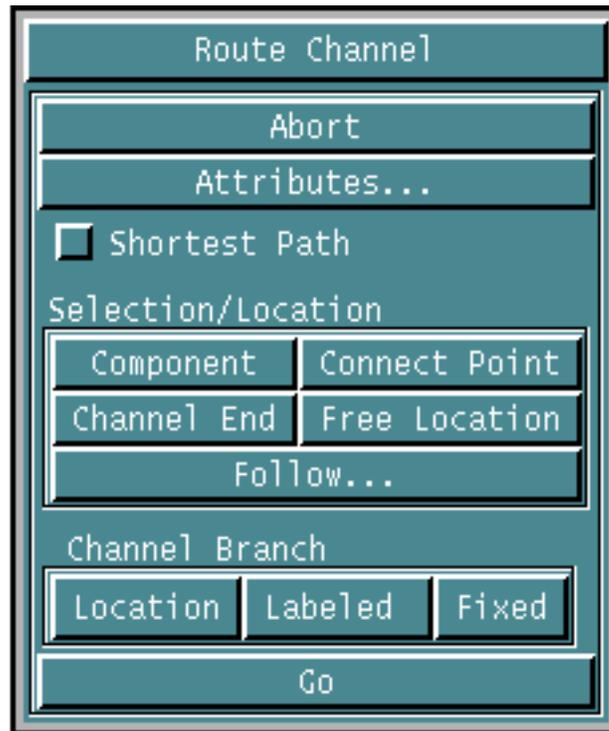
Procedure

You can perform the following tasks:

- Abort - Abort channel routing.

- Attributes - Specify channel attributes.

- Shortest Path - Route along the shortest path.

- Component - Select a component for routing.

- Connect Point - Select a specific connect point on a component for routing.

- Channel End - Select end of channel for routing.

- Free Location - Specify free locations for routing.

- Follow - Route a channel along an existing curve, channel, or surface.

- Location **-** Select a reference point near a channel for branch point routing.

- Labeled **-** Specify branch location distance from the channel end or the selected component.

- Fixed **-** Enables the user to specify a branch at a fixed location on a channel.

- Go - Complete routing

Abort:

Click Abort to abort the channel routing operation.

Attributes:

Click Attributes to specify various channel attributes using the Route Channel Attributes property sheet that appears.

Depending on the default cross-section and attributes selected in the Harness Layout Attributes property sheet, the appropriates attributes are displayed. For more details, refer to "Specifying Route Channel Attributes" on page 4-53.

Shortest Path:

Click Shortest Path to route along the shortest path between two components, satisfying minimum bend radius, tangency and direction conditions defined on the components.

Select two harness components without specifying any free locations between them. A channel is routed between these two components using the shortest path, satisfying minimum bend radius, length of the direction vector, tangency conditions, and direction conditions defined on the components. The shortest path is determined by following the specified direction vector length, a minimum bend radius tangent to the direction vector and a straight segment between the two selected components.

This kind of routing is used to route a channel, tangential to the direction vector defined on the connect point of one component, directly to another component, while using the specified minimum bend radius at bends.

Component:

Click Component to select a harness component as the start or end point for the channel being routed.

The following error message is displayed if the selected components are too close to be connected because the specified minimum bend radius is too large:

```
Error: Channel Creation failed. Please increase minimum
bend radius <value> or adjust distance between selected
components.
```

If this error message appears:

- change the values of the minimum bend radius using the Attributes option on the Route Channel menu

  OR

- change the tangency conditions on the components

  OR

- increase the distance between the selected components and create the channel again.

Connect Point:

Click Connect Point to select a specific connect point on a component. This connect point is used for routing the channel.

This is very helpful in cases where the component has multiple connect points (like a clip). In such a case, you can select the connect point graphically on the screen. In case of a clip, the selected connect point is used as entry point (the point at which the channel enters the clip) into the clip.

Channel End:

Click Channel End to select end of channel to join to.

Free Location:

Click Free Locations to select free locations through which the channel being routed passes, starts or ends.

Follow:

Click Follow to route a channel along an existing curve, channel, or surface. The Follow Channel menu appears. For details, refer to "Routing a Channel Along a Curve, Channel or Surface" on page 4-51.

Location:

Click Location to create a branch point using a parameterized reference point selected in free space.

Select a reference point near the required location of the branch. Select an existing channel to place the branch point.

A channel branch is routed from the selected channel to the intersection of the perpendicular dropped from the reference point of the selected channel to the selected channel. This reference location marker can be used to move the branch location. If the perpendicular from the reference point does not intersect the selected channel, the branch is created at the channel end located nearest to the reference point.

Labeled:

Click Labeled to create a channel branch at a specified distance from the channel end or harness component. The Channel Branch property sheet, as shown in the following figure, appears.

Figure 4-33 Channel Branch property sheet
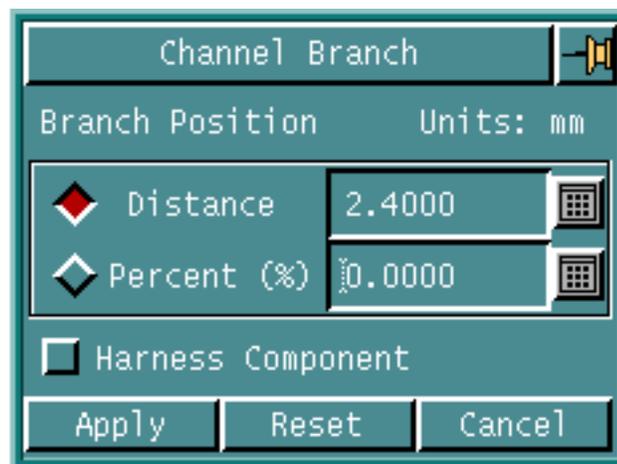
- Distance - Parameterizes the branch position in terms of the specified distance from a channel end or harness component. This distance must be lesser than the length of the selected channel and must be positive.

  - If the Distance option is selected and the Harness Component option is not selected:

    Select an existing channel on which the branch point is to be created. The command determines the channel end that is nearer to the selected point. The branch is routed on the selected channel at the distance specified from the channel end.

  - If the Distance option is selected and the Harness Component option is also selected:

    Select an existing harness component (connector, clip, or junction) and then select the channel on which the selected component is located. The branch is routed on the selected channel at the specified distance from the selected component. The connect point on the component closest to the selected channel is used as the reference point.

- Percent - Parameterizes the branch point position as a percentage of the total length of the selected channel. This value must be positive and between 0 and 1.

Select an existing channel on which the branch is to be routed. The branch is routed at the specified distance (corresponding to the specified percentage of the total length of the channel) from the start of the channel. The start of the channel is determined by the way in which the channel was routed for the first time.

This option is deactivated if the Harness Component option is selected.

- Harness Component - Allows you to specify the branch position at a fixed distance from any harness component such as a clip, connector or junction component.

  Routing a channel branch, results in the original channel being split into two channel segments and a channel branch being created. Additional channel branches can be routed from any of these channels using this functionality.

  This option is deactivated if the Percent option is selected.

Please note: If you have already issued this command, you can use the Replay History and Edit Operation options in the Parametric environment to edit the command and then specify the parameterised position for a particular branch.

Fixed:

Click Fixed to specify a branch at a fixed location on a channel. This fixed location is not parameterized.

Click Apply and select a location on an existing channel. The branch is routed at the selected location, the position of which is fixed.

Go:

Click Go to create the new channel and terminate the routing process.

Routing a Channel Along a Curve, Channel or Surface

To route a channel along a curve, or channel:

1. Click Follow on the Route Channel menu. The Follow Channel menu appears.

- Abort - Aborts the command.

- Curve- Selects the curve along which the channel is to be routed.

- Harness Channel - Selects the harness channel along which the channel is to be routed.

- Start Location - Specifies the start location on the selected curve or channel. The channel to be routed starts following the selected curve or channel from this point.

- End Location - Specifies the end location on the selected curve or channel. The channel to be routed stops following the selected curve or channel at this point.

- Offset Attributes - Displays the Channel Offset Attributes property sheet and allows you to specify offset attributes.
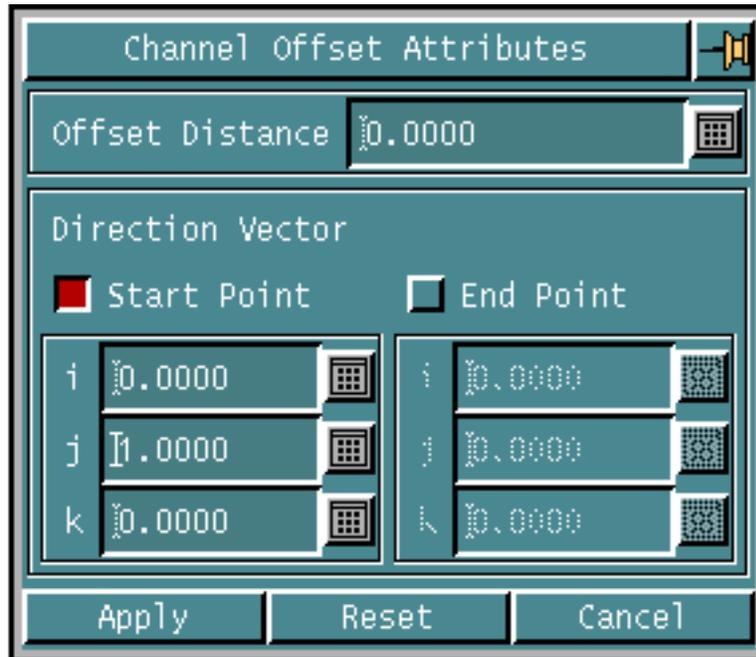
- Offset Distance - Specifies the distance separating the centerline of the channel being routed and the centerline of channel or curve being followed. Specify a value larger than or equal to the default value. The default value is calculated as:

```
default value = radius of the selected curve or harness
channel + radius of the smallest circle that encloses
the cross section of the channel being routed.
```

Please note: The offset distance must not be less than the default value. If you specify the offset distance less than the default value, the default offset value is used when the channel is routed.

The radius of the smallest circle for the various cross sections is as follows:

Table 4-1   Radius for various cross section

Type of Cross Section	Radius of the Smallest Circle
Circular	Actual radius of the bundle formed by routed wires.
Rectangular	Half the length of the diagonal.
Elliptical	Major Axis.
Rounded Rectangular	Half the length of the diagonal.
Filleted Rectangular	Half the length of the diagonal.

- Direction Vector - Specifies the direction of the offset points. The default direction is -Y. Click Start Point to specify the starting point location and click End Point to specify the end point location.

  The starting point of the channel being routed is located along this direction vector at a distance specified by the Offset Distance. The offset channel geometry is created from this starting point, based on the direction vector and the shape of the channel being followed. All the offset points are created using this direction vector. The direction can also be specified by selecting two locations in free space or with reference to other entities in the model.

Routing a Channel Along a Surface:

To route a channel along a surface:

1. Create the surface along which the channel needs to be routed.

2. Create a reference curve to be used to create the guide curve.

3. Create the guide curve by projecting the reference curve on the surface, using the DROP ENTITY command. This guide curve lies on the surface.

4. Click Follow and route a channel along the guide curve. Select the Curve option of the Follow Channel Menu while routing along this guide curve.

5. The user can specify an offset distance and offset direction during channel routing. The routed channel will be associated to the guide curve, which in turn is associated with the surface.

6. Specifying an offset distance of 0 will create the channel on the surface.

Specifying Route Channel Attributes

Depending on the default cross-section and attributes selected in the Harness Layout Attributes property sheet, the appropriates attributes are displayed.

The Route Channel Attributes property sheet for a circular cross-section is shown in the following figure.

Figure 4-34 Route Channel Attributes Property Sheet - Circular Cross-Section
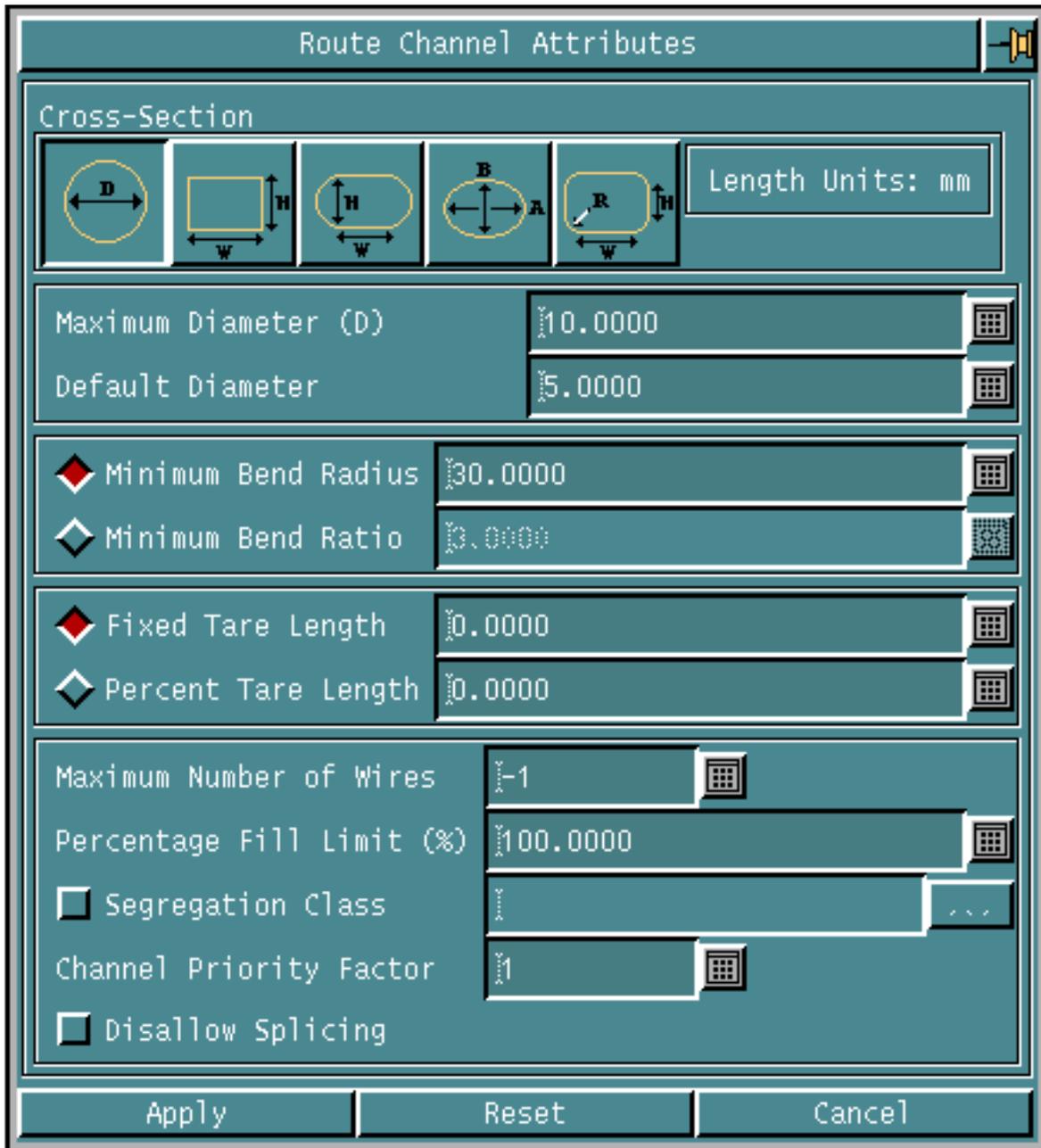
Cross Section: Specify the cross-section of the channel to be routed. The cross-section related attributes displayed in the property sheet will change depending on the type selected.

- Attributes for a Circular Cross Section

  - Maximum Diameter: Specify the Maximum diameter of the channel. This value is stored as an attribute `cirmaxdia` on the channel.

- • Default diameter: Specify the default or initial diameter of the channel. This is useful for approximating channel size requirements prior to wire routing. It has no effect on the actual diameter of the channel. The default diameter is used for HLR, IFC, shading, drawing generation, and clash detection purposes if the channel contains no wires. The actual diameter value is used if the channel contains wires.This value is stored as an attribute `cirdefdia` on the channel.

- • Attributes for a Rectangular Cross Section

  - • Height: Specify the maximum height of the channel. This value is stored as an attribute `recheight` on the channel
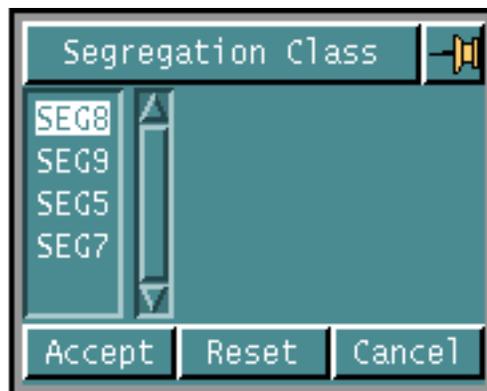
  - • Width: Specify the maximum width of the channel. This value is stored as an attribute `recwidth` on the channel.

- • Attributes for a Rounded Rectangular Cross Section

  - • Height: Specify the maximum height of the channel. This value is treated as the diameter of the rounded edges and is stored as an attribute `roundrecheight` on the channel.

  - • Width: Specify the maximum width of the channel excluding the value of the rounded edges. This value is stored as an attribute `roundrecwidth` on the channel.

- • Attributes for a Elliptical Cross Section

  - • Major Axis: Specify the length of the major axis of the channel. This value is stored as an attribute `ellmajordia` on the channel.

  - • Minor Axis: Specify the length of the minor axis of the channel. This value is stored as an attribute `ellminordia` on the channel.

- • Attributes for a Filleted Rectangle Cross Section

  - • Height: Specify the maximum height of the channel. This value is stored as an attribute `fillrecheight` on the channel.

  - • Width: Specify the maximum width of the channel. This value is stored as an attribute `fillrecwidth` on the channel.

  - • Fillet Radius: Specify the fillet radius to be used at the corner of the Filleted Rectangular Cross-Section. This value is stored as an attribute `fillrecrad` on the channel.

- • Attributes Common to all Cross Sections

  - • Minimum Bend Radius: Specify the minimum bend radius of the channel. The actual channel bend radius must be more than this value and is checked using the channel diameter. This value is stored as an attribute `cirminbendrad` on the channel.

- Minimum Bend Ratio: Specify the minimum bend ratio of the channel. The minimum bend radius is then calculated as the specified minimum bend ratio times the actual diameter of the channel. If no actual diameter is available, then the default diameter is used. During channel routing with no wires routed, the default diameter is used. This value is stored as an attribute `cirminrtod` on the channel.

- Fixed Tare Length: Specify the additional length of the wire that is to be considered during report generation. The tare length is added to the total wire length for tolerance and/or sag considerations. This value is stored as an attribute `fixedtarelen` on the channel.

- Percent Tare Length: Specify the additional length of the wire as a percent of the total length that is to be considered during report generation. The tare length is added to the total wire length for tolerance and/or sag considerations. This value is stored as an attribute `percenttarelen` on the channel.

- Maximum Number of Wires: Specify the maximum number of wires allowed to be routed in the channel segment. This value is stored as an attribute `maxwires` on the channel. To specify that there is no constraint on the number of wires, enter -1 or any other negative value.

- Percentage Fill Limit: Enter the percentage fill limit for wire routing. This checks and disallows wire routing through channel that are already filled to or beyond the specified percentage.

- Segregation Class: Specify the channel segregation class by selecting a class from the Segragation Class Scroll list. You can select more than one segregation class.

Segregation classes are used to determine which wires can or cannot be routed along a channel segment. Specifying a segregation class on a channel segment affects the automatic routing of wires. Wire segregation classes are defined in the netlist file. Only wires with a matching segregation class can be routed along a channel with the same segregation class. This value is stored as an attribute `segclass` on the channel.

- Channel Priority Factor: Specify wire routing priorities. The actual channel length is multiplied by the channel priority factor to determine a weighted channel length. This weighted channel length is used as the length to determine the shortest path by the wire router. The wires are then routed using the shortest possible weighted lengths. For example, if a wire could be routed within two possible channels, the wire will be routed within the channel that has the shortest weighted length. This value is stored as an attribute `priority` on the channel.

- Disallow Splicing: Specify whether splices are to be inserted into wires within that channel during wire routing. This value is stored as an attribute `nosplice` on the channel.

Examples

The following examples illustrate various features of the ROUTE CHANNEL command.

Example 1

Routing a channel between 2 connectors through 2 free locations.

```
ROUTE CHANNEL dig1 FREE dig2 dig3 COMPONENT dig4
```

Here dig1 is on connector CON1, dig2 and dig3 are in free space, and dig4 is on connector CON2.

Figure 4-35 Channel with circular cross-section
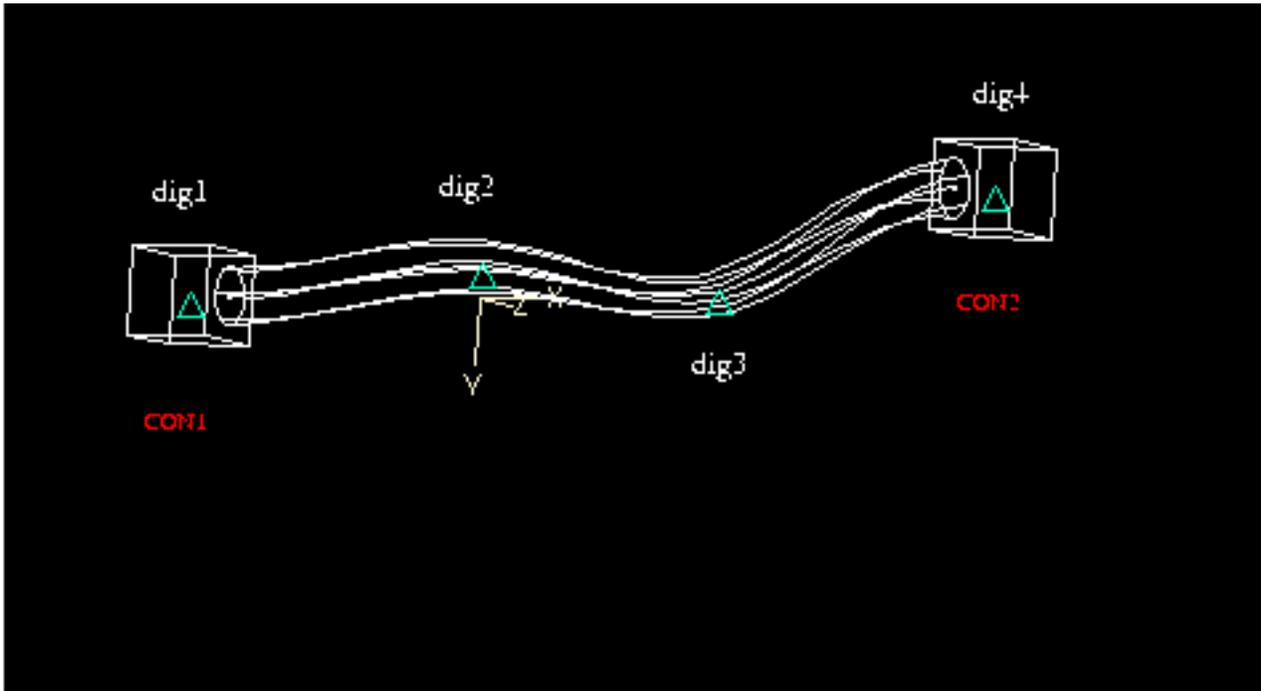
Example 2

The following example creates a channel branch at a distance of 2.775 units from connector CON1.

```
ROUTE CHANNEL FREE <dig1> <dig2> BRANCH DISTANCE 2.775
<select the channel connected to CON1 dig3> GO
```

Figure 4-36 Channel Branch At A Fixed Distance From Connector
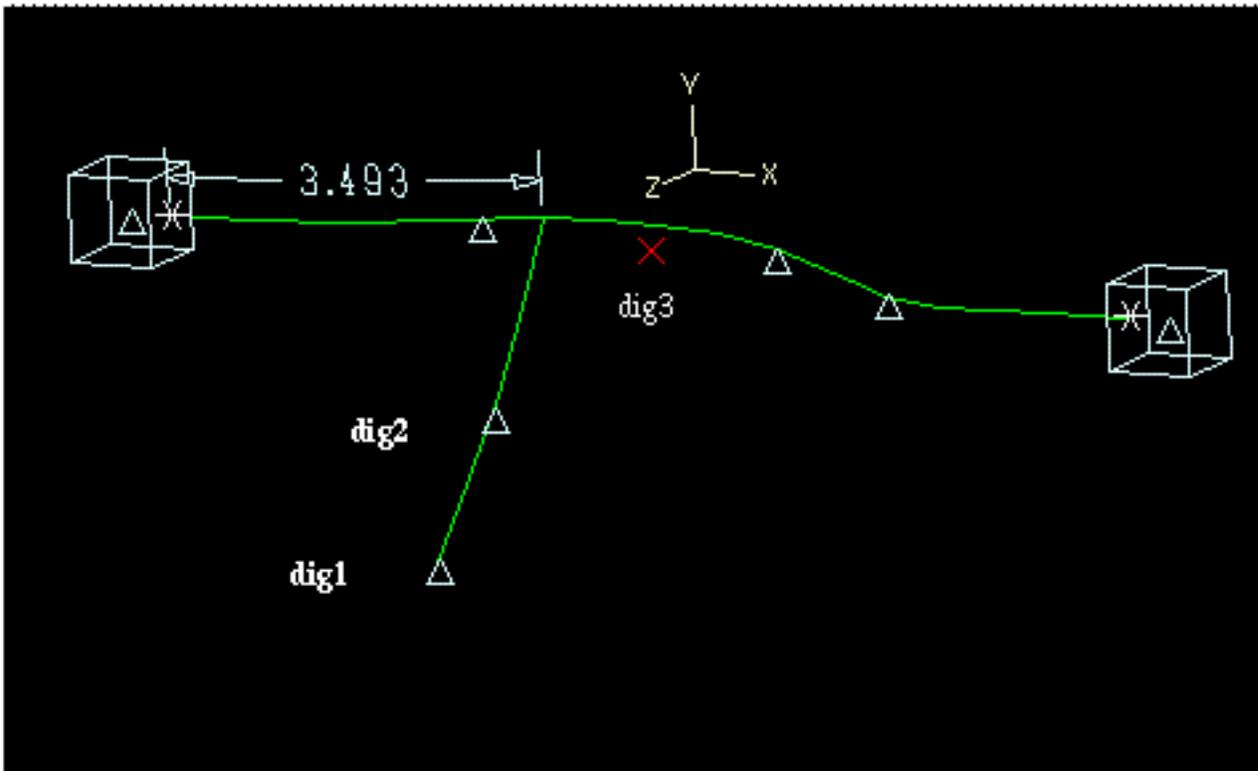
Please note: The branch distance specified in the command was 2.775 but the distance parameter appears as 3.493 in the figure above. The value appearing in the figure always corresponds to the straight line distance whereas the value specified in the command is the distance along the harness channel. If the CHANGE PARAMETER command is used and the distance parameter is selected, the correct value of 2.775 appears in the calculator menu.

Example 3

The following example creates a channel using the shortest path between a connector CON1 and a CLIP.

```
ROUTE CHANNEL component <select CON1 dig1> <select CLIP dig2> GO
```

Similarly, a channel is created between connector CON2 and a CLIP.

```
ROUTE CHANNEL component <select CON2 dig3> <select CLIP dig4> GO
```

The results of both the above commands are shown in the following figure.

Figure 4-37 Channel Created Using Shortest Path Routing
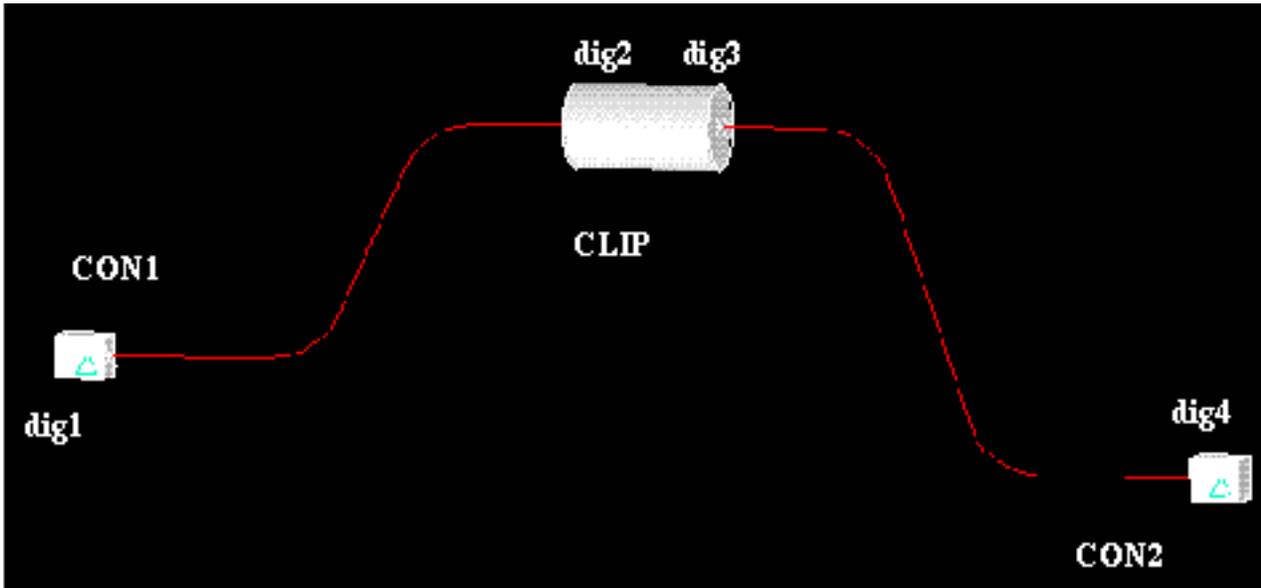
Example 4

The following example shows a 3 connect point junction. The channel is routed by selecting a connector, followed by one free location and then selecting the reference connect point shown. The tangency direction specified on the Reference Connect Point is in the -X direction.

```
ROUTE CHANNEL <select connector CON1 dig1> free <dig2>
Connectpoint <select the reference connect point dig3> GO
```

Figure 4-38 Routing A Channel To A Multiple Connect Point Junction
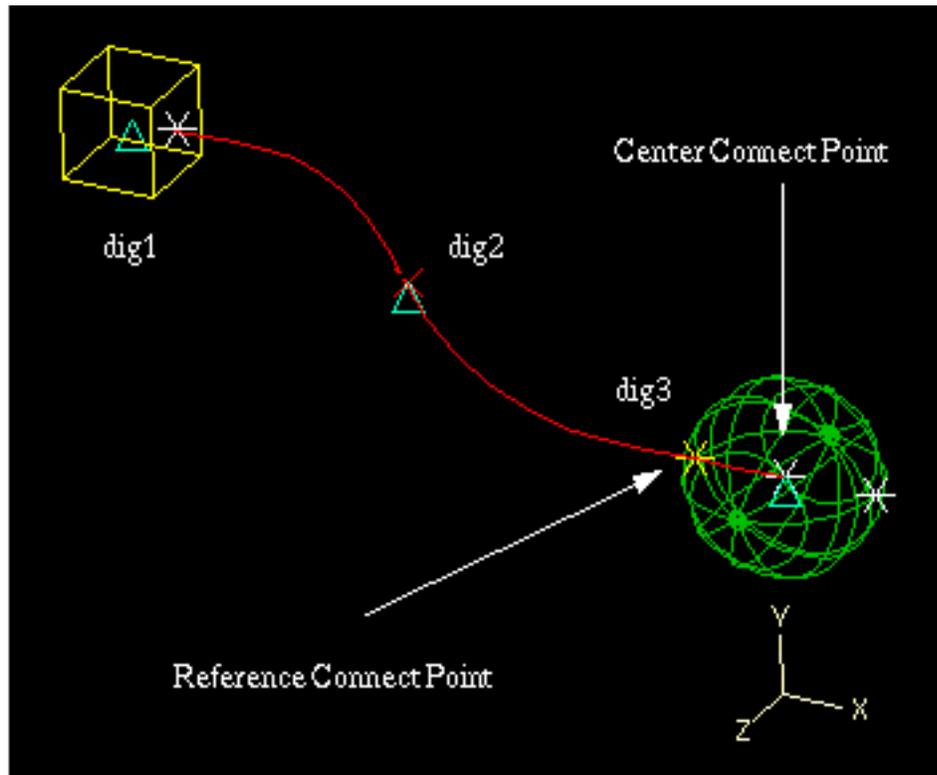
The channel is routed satisfying this tangency and direction constraint on the
Reference Connect Point and terminates on the center Connect Point.

Example 5

The following example routes a channel, that starts from connector CON1, passes
through two free points, follows an existing channel from start point to end point
and terminates on connector CON2, using an offset distance value of 2 and +Y
offset direction.

ROUTE CHANNEL component dig0 free dig1 dig2 **follow** dig3 **startpt**
dig4 **endpt** dig5 **offset_distance** 2 **direction** [0,0,0] [0,1,0]
end_follow free dig6 **component** dig7 **GO**

Figure 4-39 Channel Routing Following Existing Harness Channel
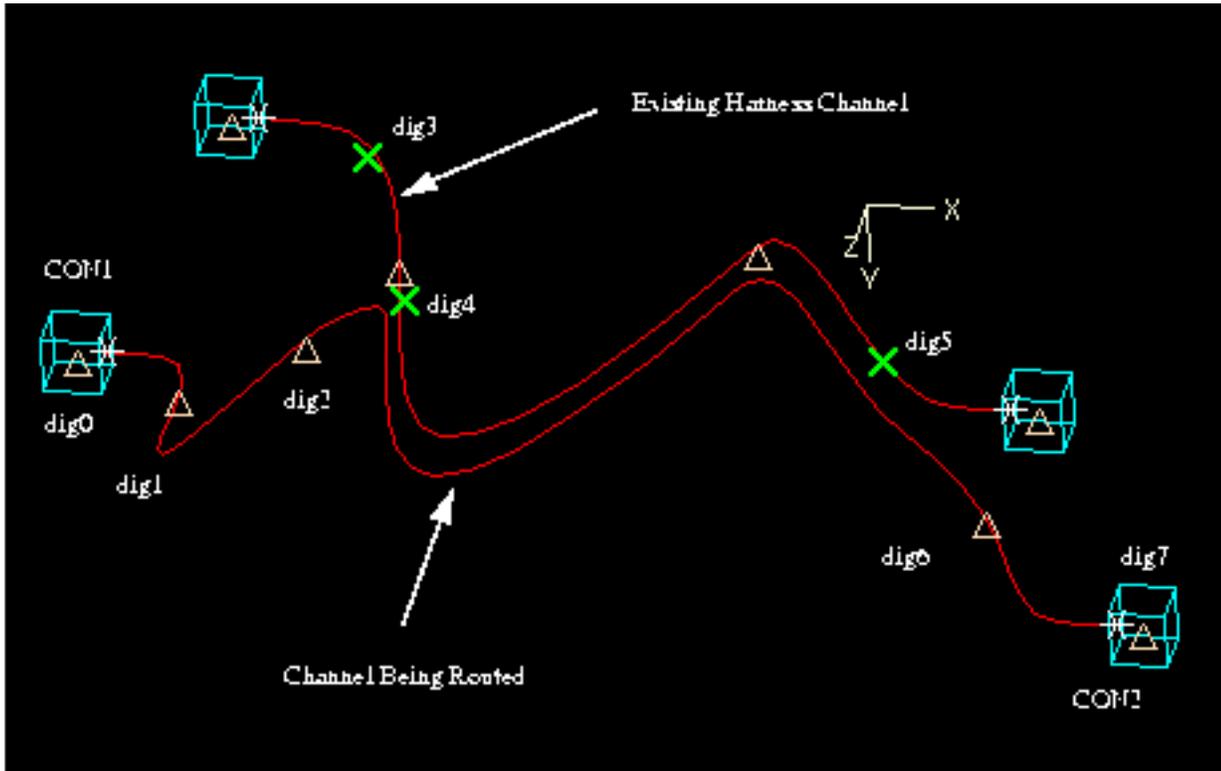
Example 6

The following example shows the steps to be followed to route along a surface with an offset distance of 5 units.

The guide curve is created by projecting the reference curve on the surface using the following command:

```
DROP ENTITY <select the reference curve> <select the surface> GO
```

The channel is routed on the surface at an offset distance of 5 units using the following command:

ROUTE CHANNEL component dig1 **follow harn_channel** dig2 **startpoint** dig3 **endpoint** dig4 **offset_distance** 5 **offset_direction vector** [0,0,1] [0,0,0] **end_follow component** dig5 **GO**

The result of these two commands is shown in the following figure.

Figure 4-40 Routing Along Surface
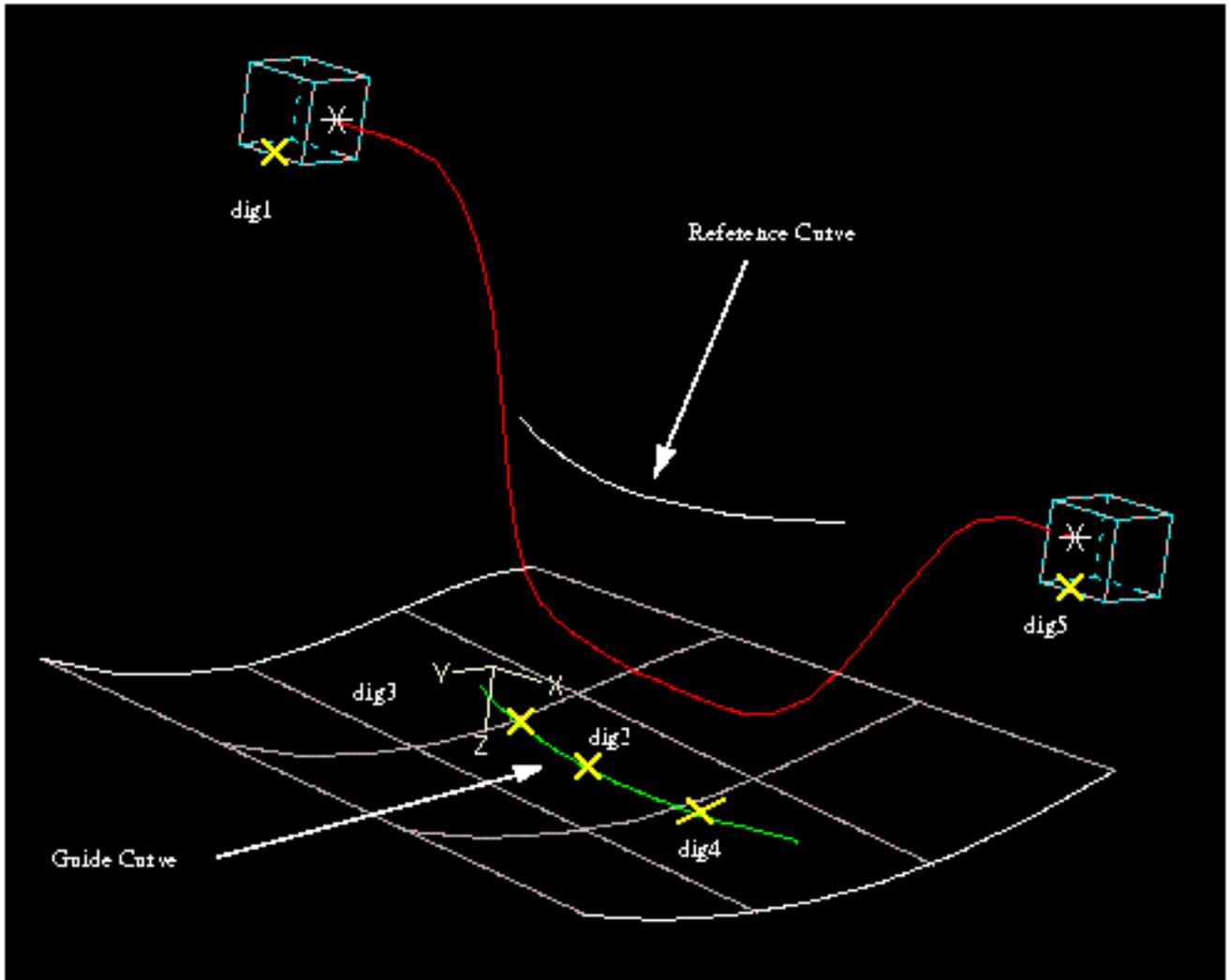
Editing a Channel

Choose the Edit Channel option from the Harness task set to edit the attributes of an existing harness channel or a group of channels. The Edit Channel property sheet, as shown in Figure 4-42 on page 4-66, appears.

OR

Use the EDIT CHANNEL command in the CADDS text window to edit a channel or a group of channels. See the online help for further details on the command-line options.

The attributes to be edited depend on the cross-section type of the channel. The EDIT CHANNEL command changes the attributes of all the channels selected in the group to the cross-section that was selected in the command.

If you select a group of channels for editing, consisting of channels with both rectangular and circular cross-section, you can specify new dimensional attributes for both kinds of channels. The new attributes specified for circular cross-section are applied to all the channels with circular cross-section in the group, and the attributes specified for rectangular cross-section are applied to all the channels with rectangular cross-section in the group.

You can also select a group of channels having both rectangular and circular cross-section and convert all the channels to channels with rectangular or circular cross-section.

Please note: For parts having channels without the Pfill attribute, the Tarelen property instance on the channel must be replaced by the modified channel class. To replace the property instance:

1. Detach the property using the DETACH PROPERTY command as follows:

    DETACH PROPERTY ENTITY <dig1>

    where <dig1> is the channel whose property needs to be updated.

2. Attach the new property class using the INSERT PROPERTY command as follows:

    INSERT PROPERTY CHANNEL ATTACHTO ENTITY <dig1>

    where <dig1> is the channel whose property needs to be updated.

3. Remove all unattached properties from the database using the REMOVE PROPERTY command as follows:

```
REMOVE PROPERTY UNATTACHED REPORT
```

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Edit Channel option from the Harness task set. The Select Channels menu appears.

Figure 4-41 Select Channels Menu

3. Select the channel to be modified from the model and then click Selection Done. The Edit Channel property sheet appears. The attributes that can be edited depend on the type of cross section.

Figure 4-42 Edit Channel Property Sheet - Circular cross section
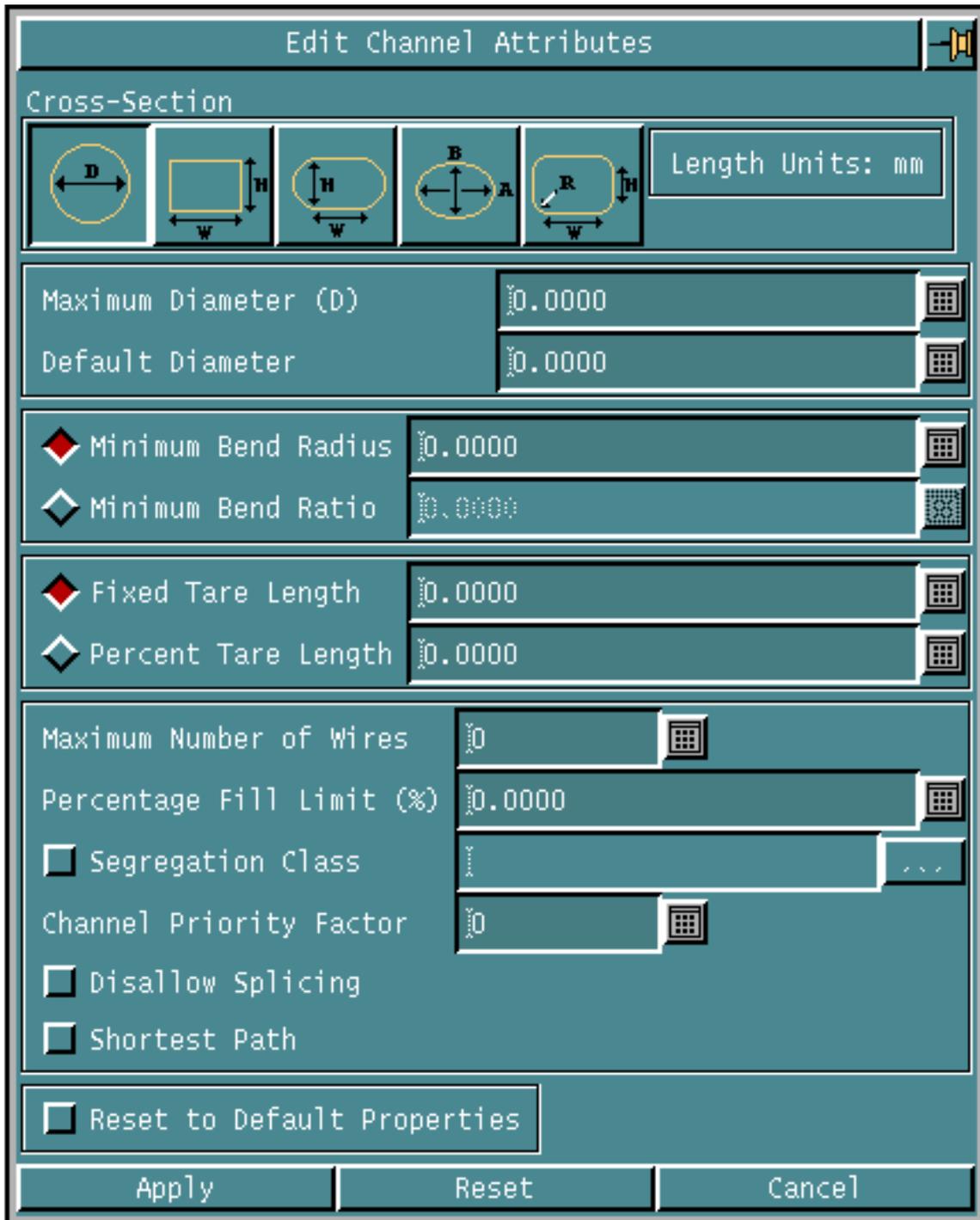
Procedure

You can edit the following attributes of the selected channels:

- Attributes for a Circular Cross Section

  - Maximum Diameter: Specify the maximum diameter of the channel. This value is stored as an attribute `cirmaxdia` on the channel.

  - Default diameter: Specify the default or initial diameter of the channel. This is useful for approximating channel size requirements prior to wire routing. It has no effect on the actual diameter of the channel. The default diameter is used for HLR, IFC, shading, drawing generation, and clash detection purposes if the channel contains no wires. The actual diameter value is used if the channel contains wires.This value is stored as an attribute `cirdefdia` on the channel.

- Attributes for a Rectangular Cross Section

  - Width: Specify the width of the channel. This value is stored as an attribute `recwidth` on the channel.

  - Height: Specify the height of the channel. This value is stored as an attribute `recheight` on the channel

- Attributes for a Rounded Rectangular Cross Section

  - Height: Specify the height of the channel. This value is treated as the diameter of the rounded edges and is stored as an attribute `roundrecheight` on the channel.

  - Width: Specify the width of the channel excluding the value of the rounded edges. This value is stored as an attribute `roundrecwidth` on the channel.

- Attributes for a Elliptical Cross Section

  - Major Axis: Specify the length of the major axis of the channel. This value is stored as an attribute `ellmajordia` on the channel.

  - Minor Axis: Specify the length of the minor axis of the channel. This value is stored as an attribute `ellminordia` on the channel.

- Attributes for a Filleted Rectangle Cross Section

  - Height: Specify the height of the channel. This value is stored as an attribute `fillrecheight` on the channel.

  - Width: Specify the width of the channel. This value is stored as an attribute `fillrecwidth` on the channel.

  - Fillet Radius: Specify the fillet radius to be used at the corner of the Filleted Rectangular Cross-Section. This value is stored as an attribute `fillrecrad` on the channel.

- Attributes Common to all Cross Sections

  - *Minimum Bend Radius*: Specify the minimum bend radius of the channel. The actual channel bend radius must be more than this value and is checked using the channel diameter. This value is stored as an attribute `cirminbendrad` on the channel.

  - *Minimum Bend Ratio*: Specify the minimum bend ratio of the channel. The minimum bend radius is then calculated as the specified minimum bend ratio times the actual diameter of the channel. If no actual diameter is available, then the default diameter is used. During channel routing with no wires routed, the default diameter is used. This value is stored as an attribute `cirminrtod` on the channel.

  - *Fixed Tare Length*: Specify the additional length of the wire that is to be considered during report generation. The tare length is added to the total wire length for tolerance and/or sag considerations. This value is stored as an attribute `fixedtarelen` on the channel.

  - *Percent Tare Length*: Specify the additional length of the wire as a percent of the total length that is to be considered during report generation. The tare length is added to the total wire length for tolerance and/or sag considerations. This value is stored as an attribute `percenttarelen` on the channel.

  - *Maximum Number of Wires*: Specify the maximum number of wires allowed to be routed in the channel segment. This value is stored as an attribute `maxwires` on the channel. To specify that there is no constraint on the number of wires, enter -1 or any other negative value.

  - *Percentage Fill Limit*: Enter the percentage fill limit for wire routing. This checks and disallows wire routing through channel that are already filled to or beyond the specified percentage.

- *Segregation Class*: Specify the channel segregation class by selecting a class from the Segragation Class Scroll list. You can select more than one segregation class.

Segregation classes are used to determine which wires can or cannot be routed along a channel segment. Specifying a segregation class on a channel segment affects the automatic routing of wires. Wire segregation classes are defined in the netlist file. Only wires with a matching segregation class can be routed along a channel with the same segregation class. This value is stored as an attribute `segclass` on the channel.

- Channel Priority Factor: Specify wire routing priorities. The actual channel length is multiplied by the channel priority factor to determine a weighted channel length. This weighted channel length is used as the length to determine the shortest path by the wire router. The wires are then routed using the shortest possible weighted lengths. For example, if a wire could be routed within two possible channels, the wire will be routed within the channel that has the shortest weighted length. This value is stored as an attribute `priority` on the channel.

- Disallow Splicing: Specify whether splices are to be inserted into wires within that channel during wire routing. This value is stored as an attribute `nosplice` on the channel.

- Shortest Path: Specify that the channel is to be routed along the shortest path between two components, satisfying minimum bend radius, tangency and direction conditions defined on the components.

  The length of the direction vector is also satisfied by the created channel. The shortest path is determined by following the specified direction vector length, a minimum bend radius bend tangent to the direction vector and a straight segment between the two selected components.

  This kind of routing is used to route a channel, tangential to the direction vector defined on the connect point of one component, directly to another component, while using the specified minimum bend radius at bends.

Inserting a Drip Loop

Choose the Modify Channel Drip_Loop option from the Harness task set to insert Drip Loops on a specified channel.The property sheet for inserting a Drip Loop, as shown in Figure 4-45, "Property Sheet for Inserting a Drip Loop," on page 4-72, appears.

OR

Use the MODIFY CHANNEL DRIPLOOP command in the CADDS text window to insert Drip Loops on a specified channel. See the online help for further details on the command-line options.

This option allows you to add Drip Loops on a channel. Various sub-options under this option allow you to specify parameters like the depth of the loop, the radius and the direction. The command then creates the shape for the Drip loop as a part of the channel shape.

Drip Loops are "U" shaped loops in a channel that enable condensed moisture on the channel to drip off before it reaches the connector at the end. The following figure shows an example of a Drip Loop in a channel.

Figure 4-43 Drip Loop in a channel
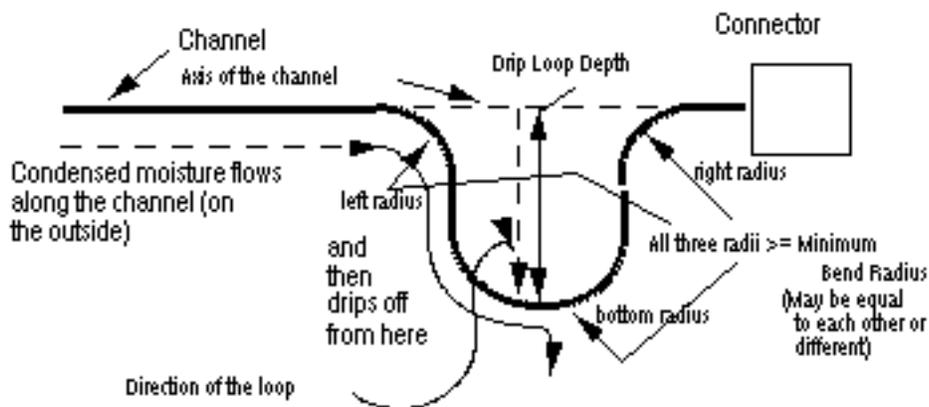
The command inserts the Drip Loop on the selected channel at the specified location. The specified location may be taken as the start point or the end point of the Drip Loop depending on which side (along the channel axis at that point) the Drip Loop is best created. It is taken as the start point but on doing so, if the Drip Loop goes beyond the channel, the specified location is taken as the end point. The portion of the channel that had spanned the start point and end point of the newly

inserted Drip Loop is deleted, and the channel assumes a new shape (the original channel shape and the Drip Loop shape). The following figure illustrates the new channel shape.

Figure 4-44 Original channel along with the Drip Loop
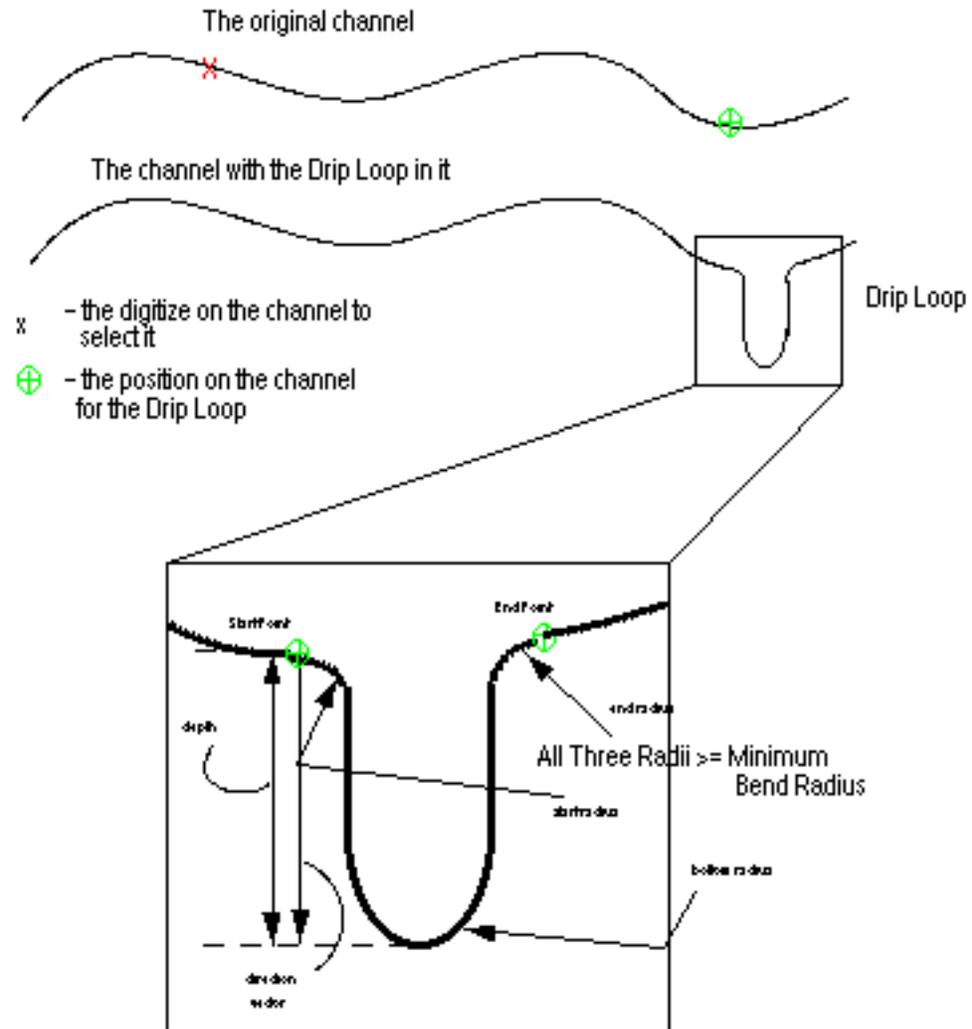
Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Modify Channel Drip_Loop option from the Harness task set. The property sheet for inserting a Drip Loop appears.

Figure 4-45 Property Sheet for Inserting a Drip Loop
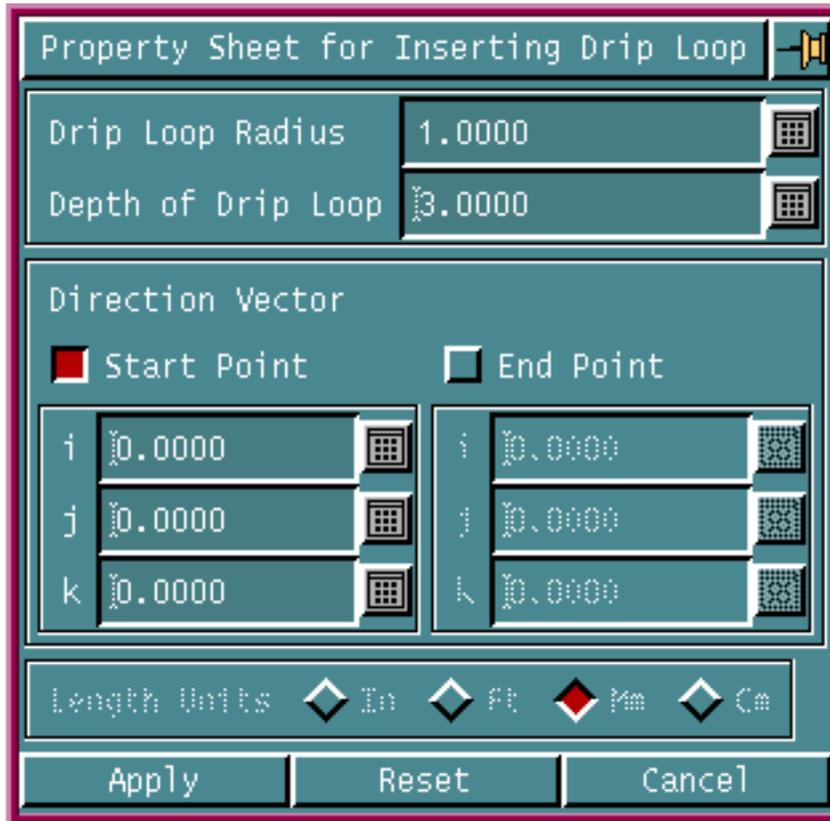
Procedure

1. Enter the bottom radius for the Drip Loop in Drip Loop Radius. This radius must be greater than the minimum bend radius of the selected channel.

2. Enter the depth of the Drip Loop in Depth of Drip Loop.

3. Enter the vector direction for the Drip loop by specify the start point and the end point for the Drip Loop in Start Point and End Point respectively. Either of the start point or end point of the vector can be [0,0,0] but not both.

    This option is mandatory. If the direction is not specified, the Drip Loop will not be created.

4. Click Apply. The Drip Loop pulldown menu, as shown in the following figure, appears.

Figure 4-46 Drip Loop pulldown menu

5. Click Select a Channel to select a channel on which the Drip Loop is to be inserted.

6. Click Select Location on Channel to specify the location on the selected channel at which the Drip loop is to be inserted. Use the Mid or Near options on the side bar to specify the location.

Please note: The last valid (lying on the channel) location selected is used as the Drip Loop location. To change the location, select a valid location again.

7. Click Go to execute the command.

Inserting a Service Loop

Choose the Modify Channel Service_Loop option from the Harness task set to insert Service Loops on a specified channel.The property sheet for inserting a Service Loop, as shown in Figure 4-49, "Property sheet for inserting a Service loop," on page 4-77, appears.

OR

Use the MODIFY CHANNEL SERVICELOOP command in the CADDS text window to insert Service Loops on a specified channel. See the online help for further details on the command-line options.

This option allows you to add Service Loops on a channel. Various sub-options under this option allow you to specify parameters like the depth of the loop, the radius and the direction. The command then creates the shape for the Service loop as a part of the channel shape.

Service Loops are typically coils (helices) in a channel. They are normally used for adding extra length to any wire passing through that channel. This is generally used for maintenance purposes where the connecting equipment is very difficult to reach or in a cramped position.

These coils may be circular, oval or figure-of-eight shaped. CADDS 5 Revision 8.0 supports circular coils only. These Service Loops may be parallel to any plane, vertical, horizontal or any other. The following figure shows typical service loops parallel to horizontal and vertical planes.

Figure 4-47 Typical Service Loops
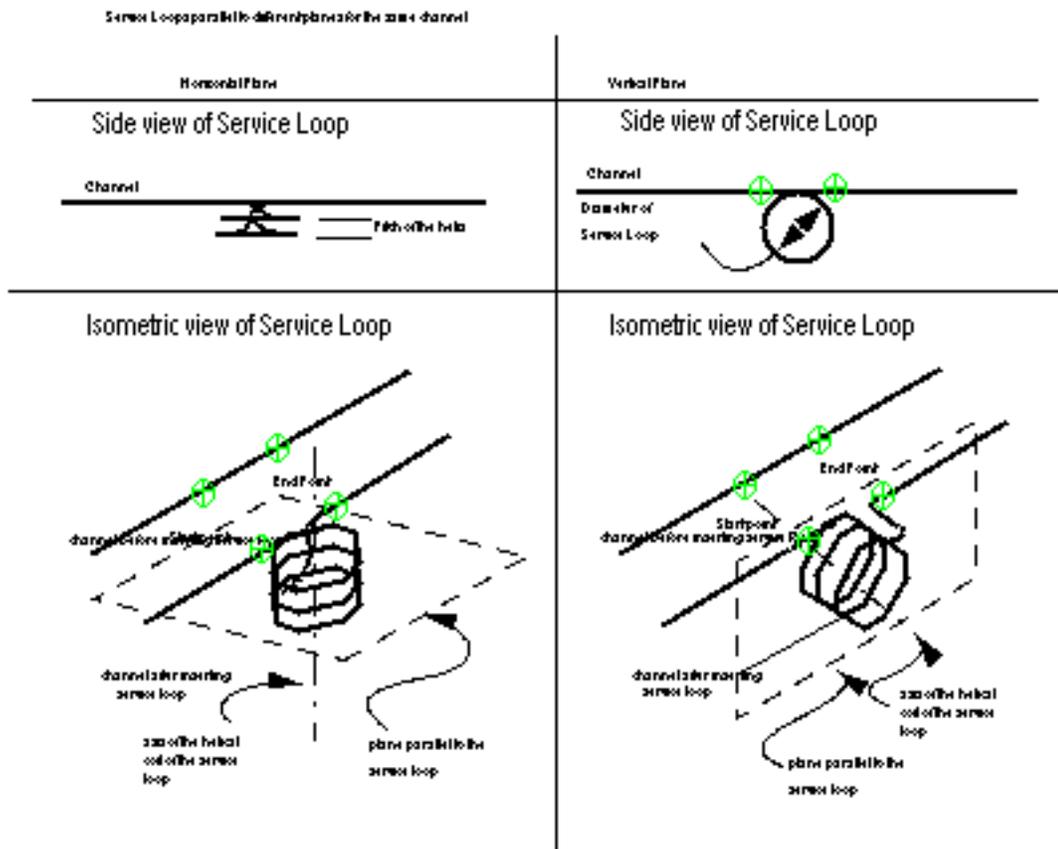
The command inserts the Service Loop on the selected channel at the specified locations. The portion of the channel that spanned the start point and end point of the newly inserted Service Loop is deleted and the channel assumes a new shape (the original channel shape and the Service loop shape). The following figure illustrates the new channel shape.

Figure 4-48 Original channel along with the Service Loop
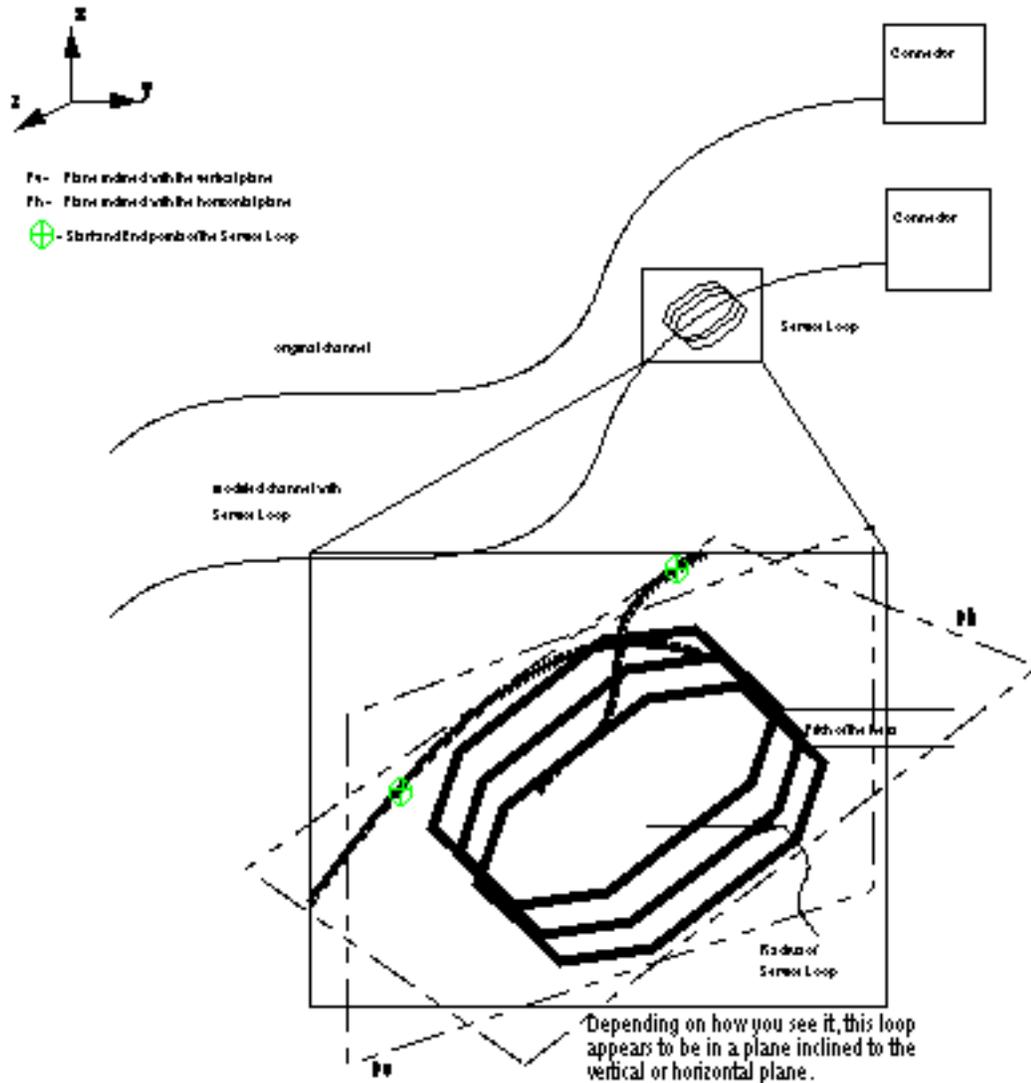
Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Modify Channel Service_Loop option from the Harness task set. The property sheet for inserting a Service Loop appears.

Figure 4-49 Property sheet for inserting a Service loop
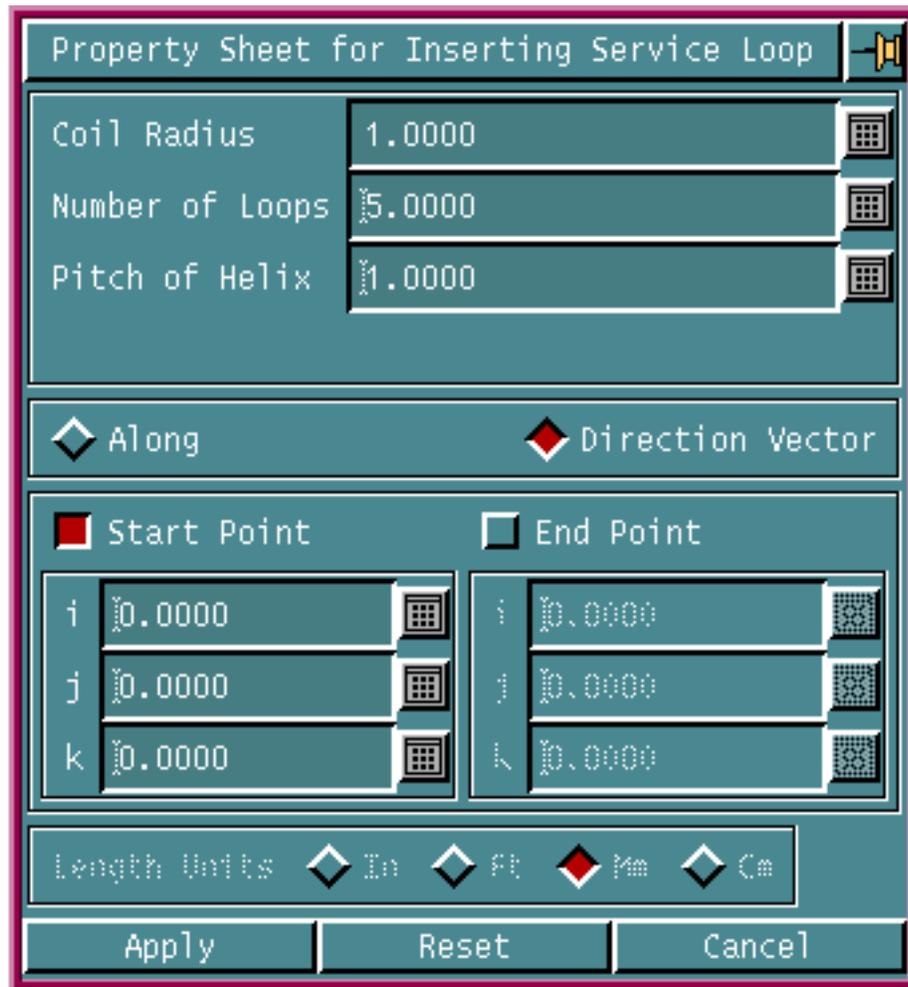
Procedure

1. Enter the radius for the Service loop in Coil Radius. This radius must be greater than the minimum bend radius of the selected channel.

2. Enter the number of loops for the Service loop in Number of Loops.

3. Enter the pitch of a helical Service loop in Pitch of Helix.

4. Click on one of the following:

- Along: Specify that the Service Loop must be created along the channel axis from the start point to the end point.

- Direction: Enter the vector direction to be used for the axis of the helical Service loop. Specify the start point and end point for the Service Loop in Start Point and End Point respectively.

5. Click Apply. The Service Loop pulldown menu, as shown in the following figure, appears.

Figure 4-50 Service Loop pulldown menu

Service Loop
Abort
Select a Channel
Select Two Locations on Channel
Go

6. Click Select a Channel to select the channel on which the Service loop is to be inserted.

7. Click Select Two Locations on Channel to select two locations on the selected channel between which the Service loop is to be inserted. The first location is the start point of the Service Loop and the second location is the end point of the Service Loop. Use the Mid or Near options on the side bar to specify the location.

Please note: The first valid location selected is used as the start point of the Service Loop. To change the start point location, backspace all the locations in the command window and then select the start point location again.

The last valid location is used as the end point of the Service Loop. To change the end point location, select a location again and this is used as the end point of the Service Loop.

8. Click Go to execute the command.

Inserting a Sag
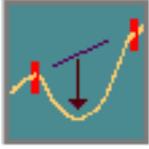
Choose the Create Sag option from the Harness task set to insert a sag between harness components like clips, connectors, and junctions. The Create Sag menu appears.

OR

Use the MODIFY CHANNEL SAG command in the CADDS text window to insert a sag between harness components like clips, connectors, and junctions. See the on-line help for further details on the command-line options.

A channel created between harness components tends to sag because of the channel weight. The shape of the sag depends on the distance between the support points, relative height difference between the support points, and the channel weight.

The sag shape generated considers tangency directions at the components and minimum bend radius constraints defined on the channel.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Create Sag option from the Harness task set. The Create Sag menu appears.

Figure 4-51 Create Sag Menu

Procedure

1. Click Select A Channel to select the harness channel segment on which the sag is to be created.

2. Select the channel.

3. Click Select Two ConnectPoints to select two component connect points between which the sag is to be created.

4. Select two connect points connected to the selected harness channel.

Please note: The sag profile is not be created if:

- any other components exist in between the selected connect points on the selected channel.

   OR

- less than two components are selected.

   OR

- the selected connect points are not connected to the selected channel.

5. Click Sag Attributes if you want to modify sag attributes. The Create Channel Sag property sheet appears.

   If this option is not selected, the sag is created with the default attribute values.

Figure 4-52 Create Channel Sag Property Sheet

You can specify the following:

- Allowable Tension - Specify the allowable tension at the support points. The profile of the sag is determined by the sag constant. The maximum allowable tension is determined by the manufacturing data. The sag constant is the ratio of the horizontal component of the tension to the mass per unit length.

- Mass Per Unit Length - Specify the channel weight per unit length. The default value is read from parameter 90 of the `param` file and corresponds to the mass per unit length with no wires routed in the channel. If the channel contains routed wires, then the weight of all these wires per unit length is added to the channel unit weight and displayed in this field.

- Maximum Allowable Sag Depth - Specify the maximum allowable depth of the sag. The sag is not created if the calculated sag depth exceeds this value.

- Maximum Allowable Channel Length - Specify the maximum allowable channel length of the sag. The sag is not created if the calculated length of the sag exceeds this value.

- Direction Of Gravity Vector - Specify the vector direction in which the sag is to be created. The sag profile is created in the gravity direction. The vector joining the start point and end point specifies the direction of gravity for the sag profile. The default gravity direction is -Y. You can specify these two points by selecting two locations on reference entities or in free space.

  - Start Point - Specifies the start point (x,y,z) of the direction vector.

  - End Point - Specifies the end point (x,y,z) of the direction vector.

- Weight Units - Specifies the current harness weight units. This is a display only field and can be set in the SELECT HDATTRIBUTES command.

  - Kg - The current harness weight units are set to kilogram.

  - Lb - The current harness weight units are set to pounds.

- Length Units **-** Specifies the current harness length units. This is a display only field and can be set in the SELECT HDATTRIBUTES command.

  - in - The current harness units are set to Inches.

  - ft. - The current harness units are set to feet.

  - mm - The current harness units are set to millimeters.

  - cm - The current harness units are set to centimeters.

  - mt **-** The current harness units are set to meters.

  - km - The current harness units are set to kilometers.

6. Click Go to execute the command.

The sag profile is created on the selected channel, satisfying the conditions specified in the Channel Sag property sheet. The profile is created in one plane passing through the support points.

Example
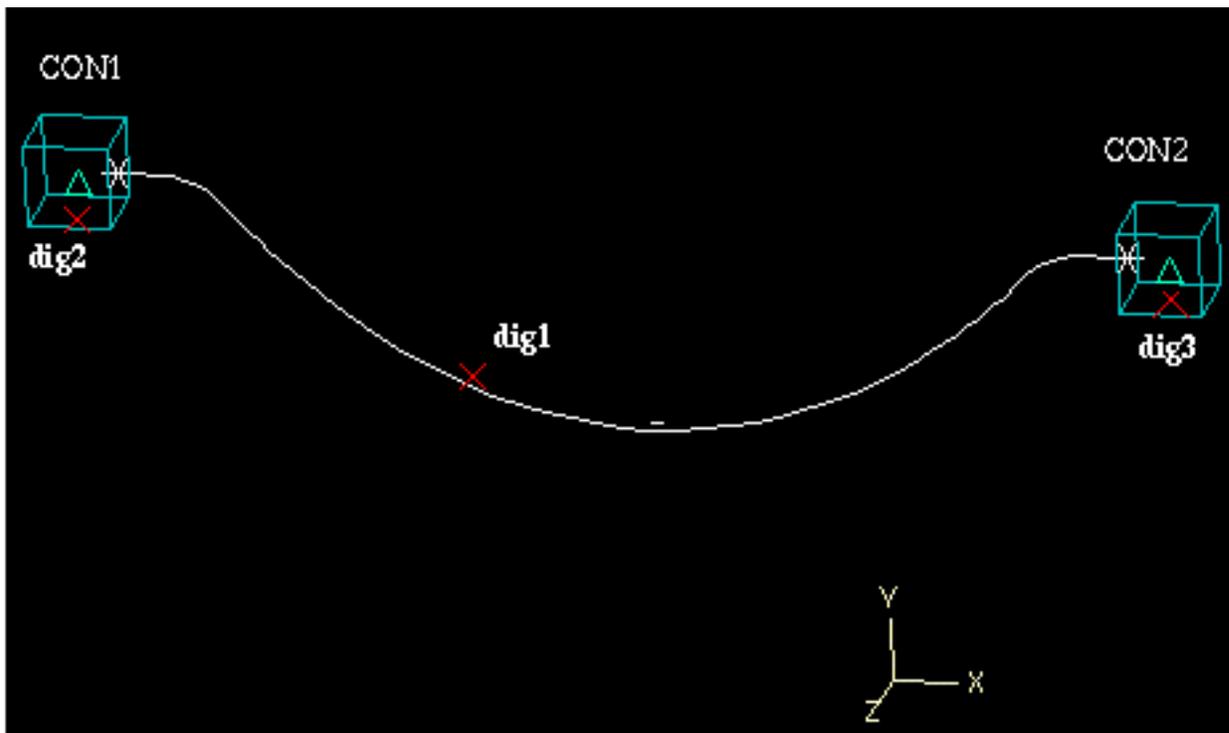
The command to create a channel sag between connectors CON1 and CON2 along the -Y direction using an allowable tension of 200 is shown below:

```
MODIFY CHANNEL SAG ON <dig1 on channel> CONNECTPOINT <dig2 on CON1>
<dig3 on CON2> TENSION 0.02 DIRECTION [0,0,0] [0,1,0] GO
```

The following figure shows the sag created by the above command:

Figure 4-53   Channel Sag Between Two Connectors

Chapter 5

Displaying and Verifying Components and Channels

This chapter describes how to create a display representation for a channel, list components in a specified library and verify inserted components in the harness model. These options are available from the Harness Layout task set in the parametric environment. The tasks described in this chapter are

- Creating a Display Representation
- Listing Harness Components
- Verifying Components and Channels

Creating a Display Representation

Choose the Shade Channels option from the Harness task set to create a surface or solid display representation of selected channels in the model using the cross-section attribute values defined on the channel. The Shade Channels menu appears.

OR

Use the GENERATE SHAPE command in the CADDS text window to create the surface or solid representation. See the online help for further details on the command-line options.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Shade Channels option from the Harness task set. The Shade Channels menu appears.

Figure 5-1   Shade Channels Menu

Procedure

You can specify the following

- Abort - Aborts the command.

- Solid - Selects the solid display mode for representing channels.

- Surface - Selects the surface display mode for representing channels.

- Go - Executes the command.

A surface or solid representation of each selected harness channel is created based on the cross-section attributes defined on each channel.

The surface and solid representations are associated to the respective channels. If the channel geometry or cross-section attributes are modified, the surface and solid representations are recreated and displayed after regeneration.

Please note: Use the RENDER VIEW command to shade and view the channels in the shaded mode. If the Interactive Display mode is enabled, channels are automatically shaded when the display representation is created.

For more details on the Interactive Display mode, refer to the *Parametric Modeling User Guide and Menu Reference*.
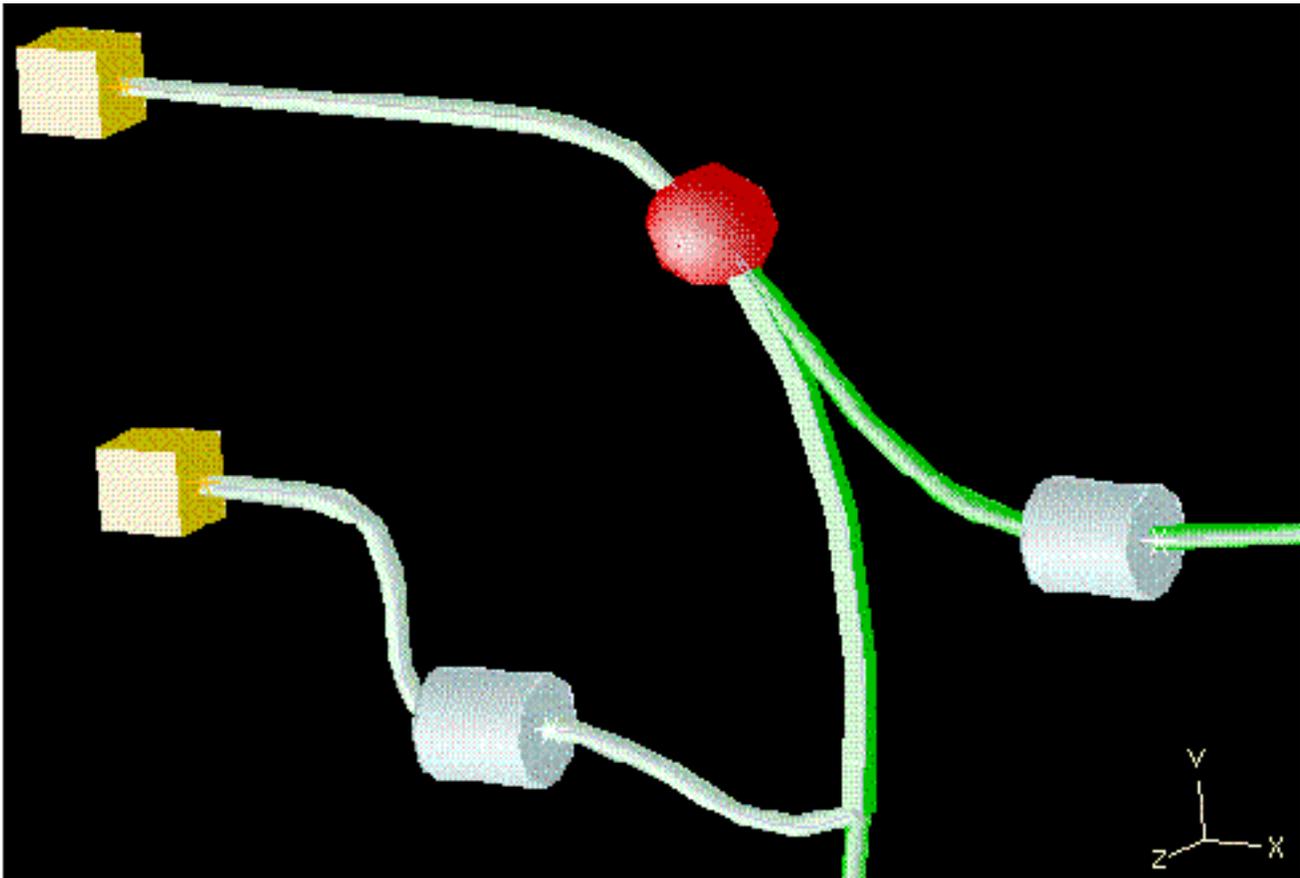
Example

The following figures give you an example of a solid representation of channels.

Figure 5-2   Solid Representation of Harness Channels
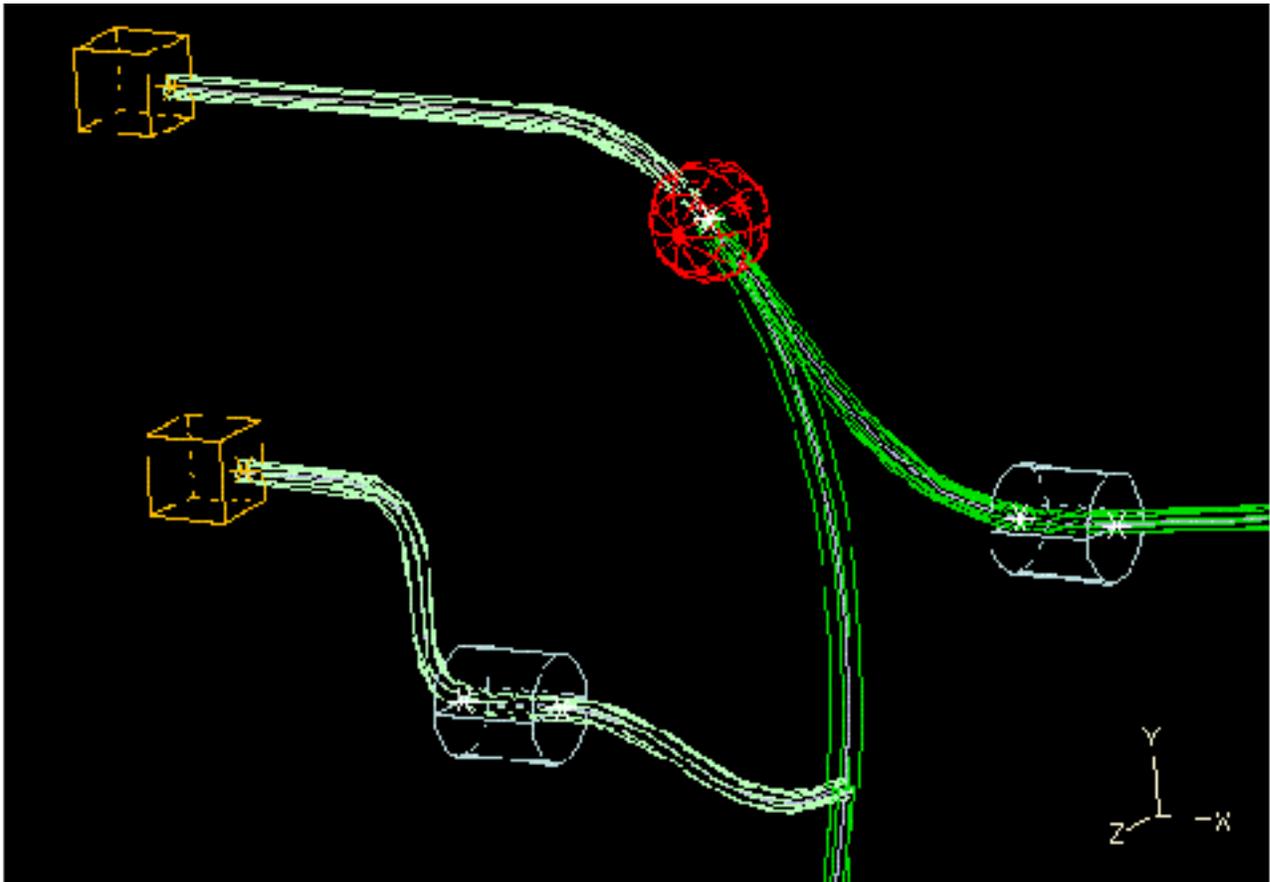
Figure 5-3   Solid Representation of Harness Channels with Shading Enabled

Listing Harness Components

Choose the List Harness Components option from the Harness task set to list all the connectors, junctions or clips in a specified component library and verify whether a component exists in the specified library. The List Harness Components property sheet appears.

OR

Use the LIST HDCOMPONENT command in the CADDS text window to insert a harness component. See the online help for further details on the command-line options.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the List Harness Components option from the Harness task set. The List Harness Components property sheet appears.

**Figure 5-4   List Harness Components property sheet**
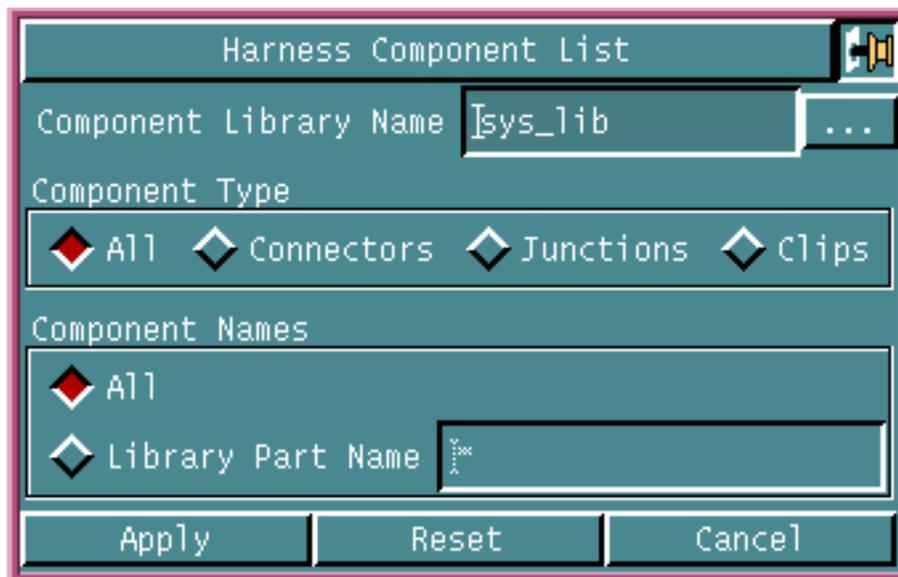
Procedure

You can specify the following

- Component Library Name
- Component Type to be listed
- Component names

Component Library Name:

- Enter the name of the harness layout component library located in the `features` directory, as specified during library creation, in the Component Library Name field or click the list button to display a list of component library names available. Various harness components are stored in this library.

Figure 5-5   Component Library Scroll List
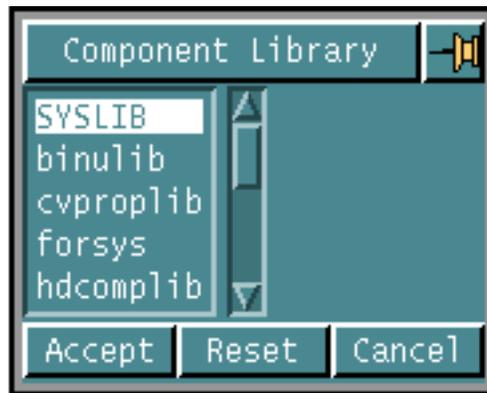
Choose a component library name from the list.

Component Type to be listed:

Select the component type to be listed.

- All - Selects all types of components (connector, clip and junction components) from the library for listing.
- Connectors- Selects connector components from the library for listing.
- Junctions - Selects junction components from the library for listing.
- Clips - Selects clip components from the library for listing.

Component names:

Select the component in the component library to be listed.

- All - Specify that all components of the selected types should be listed.

- Library Part Name - Specify the name of the component in the component library to be listed. Wildcard character * can be used to list a group of components together. For example, if hd* is specified in the Library part Name field, all the components of the specified type with their names starting with 'hd' are listed.

The results of the command are displayed in a report window.

Verifying Components and Channels

Choose the Verify Harness Components and Channels option from the Harness task set to verify whether a component or a channel is a valid harness object and list its attributes in a report window. It also allows you to perform bend radius check on the channels.

OR

Use the VERIFY HDOBJECT command in the CADDS text window to verify whether a component or channel is a valid harness object and list its attributes in a report window. See the online help for further details on the command-line options.

All entities created by the INSERT HDCOMPONENT command are considered valid Harness components.

All entities created by the ROUTE CHANNEL and MODIFY CHANNEL commands are considered valid harness channels.

All channels created by the INSERT HDCOMPONENT command while inserting a break connector are also considered valid harness channels.

This command also allows you to perform bend radius check on the channels. If the minimum bend radius of the channel is less than the user-specified minimum bend radius, indicator graphics are displayed at the location where the violation occurred.

Please note: The bend radius check works only when an harness is active.

Using this Option

1. Choose Harness from the task set Access menu.
2. Choose the Verify Harness Components and Channels option from the Harness task set.

Figure 5-6   Verify Hdobjects list

- Click Properties to see the attributes of a valid harness object. Select an harness object. If the selected component is a valid harness object, the attributes of the object are displayed in the report window. You can select any number of harness objects in the same command. The results keep appearing in the scrollable report window.

- Click Bend Radius Check to perform the bend radius check on channels.

  - Click Channel to select the a channel manually for bend radius checking.

  - Click All Channels to select all the channels for bend radius checking.

Example

If you select a harness connector, the report window displays the attributes of the selected connector as follows:

```
****************************************************************
       The selected entity is a harness connector.

String Compname     = CON1
String Partno       = ESC100
String Equipno      = TAIL200
Length length       = 5.000000e-01 = inches
Length breadth      = 5.000000e-01 = inches
Length width        = 5.000000e-01 = inches
Length dx           = 1.000000e+00 = inches
Length dy           = 0.000000e+00 = inches
Length dz           = 0.000000e+00 = inches
Logical Cflag       = True


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

Chapter 6

Displaying Harness Status

This chapter describes how to display the harness status information. This option is available from the Harness task set in the parametric environment. The tasks discussed in this chapter are

• Displaying Harness Status

Displaying Harness Status

Choose the Display Harness Status option to display the Harness status. The status window displays the active version name, wirespec, parameter and netlist file names that were specified in the INITIALIZE HARNESS command.

Using this option

1. Choose Harness from the task set Access menu.

2. Choose the Display Harness Status Window option from the Harness task set.

   The Harness Layout Status Window appears.

Figure 6-1   Harness Layout Status Window

This window displays the version name, netlist, parameter and wirespec filenames. This is a read-only window.

When a new version is initialized using Initialize Harness option, the status window is updated automatically if it is open. Similarly if an active version is exited using the Exit Harness option, the information in the status window is cleared.

3. Click C to make the Harness Layout Status Window disappear from the screen. The Harness Layout Status Window remain on the screen till the end of the harness session or until the user clicks C from the Window.

Chapter 7 # Deleting Harness Version

This chapter describes options for deleting a harness version. This option is available from the Harness task set in the Parametric environment. The tasks discussed in this chapter are

• Deleting Harness Version

Deleting Harness Version

Choose the Delete Harness Version option from the Harness task set to delete an harness layout version. The Delete Version property sheet appears.

OR

Use the DELETE HARNESS command in the CADDS text window to delete an existing harness layout version. See the online help for further details on the command-line options.
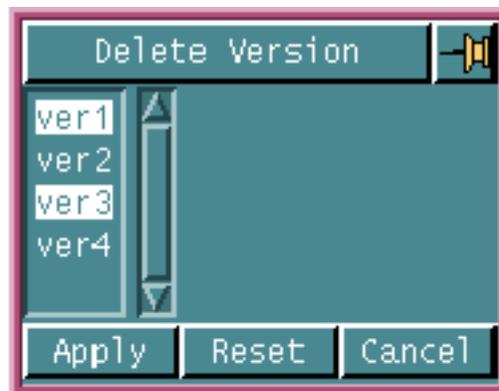
The Delete Harness option enables you to delete an existing version stored under the `<parts_directory>/<part_name>`.

Please note: You cannot delete the active version.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Delete Harness Version option from the Harness task set. The Delete Version property sheet appears.

Figure 7-1   Delete Version Property Sheet

3. Select the version to be deleted. You can also select multiple version names.

4. Click Apply.

   The selected versions are deleted.

Please note: When a version is deleted all information related to that version (for example, wire routing, splicing, harness segment, wire routing log) is deleted.

Chapter 8

Saving and Exiting a Harness Session

This chapter describes saving a harness part and exiting from a active harness layout session. These options are available from the Harness task set in the Parametric environment. The tasks discussed in this chapter are

- Saving a Harness Session
- Exiting a Harness Version/Session

Saving a Harness Session

Choose the Save Harness Session option from the Harness task set to save the active harness session.

Alternatively, you can save the harness session using the FILE PART and EXIT PART options provided on the top bar. You can also use the FILE PART command from the command-line to save the harness session. Similarly, you can use the EXIT PART FILE command from the command line to save the harness session before exiting the current part.

A harness session consists of two types of closely related information.

- harness related information stored in a harness database (the version, topology etc.) and

- geometry information stored in the CADDS part database.

The Save Harness Session option saves both the harness and part database information in a synchronized way.

This option enables you to save the active harness version related information in the `<parts_directory>/<part_name>/<version_name>` directory in a file called `harness-data` and the `part` database under the part directory.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Save Harness Session option from the Harness task set. The Save Harness property sheet appears.

Figure 8-1   Save Harness property sheet

3. Click SAVE to save the active harness session information.

   Information related to the active harness session is saved.

Examples

Example 1

The command line syntax to save the harness version **ver1** and part **wing1**, without exiting the part is as follows:

FILE PART

Example 2

The CADDS command line syntax to save the harness version **ver1** and part **wing1**, and exit the part, is as follows:

EXIT PART FILE

Exiting a Harness Version/Session

Choose the Exit Harness option from the Harness task set to exit the active harness layout version.

OR

Use the EXIT HARNESS command to exit the active harness layout session. See the online help for further details on the command-line options.

You can also use the EXIT PART QUIT option to exit the harness layout version and exit the active part.

A harness session consists of two types of closely related information.

- harness related information stored in a harness database (the version, topology etc.) and

- geometry information stored in the CADDS part database.

The Exit Harness task also has an option to save the active harness session before exiting the harness version and maintains both the harness and part database information in a synchronized way.

Using this Option

1. Choose Harness from the task set Access menu.

2. Choose the Exit Harness option from the Harness task set. The Exit Harness alert menu appears.

Figure 8-2   Exit Harness alert menu

3. Choose YES to save the changes to the harness version and part database and then exit the harness version.

OR

4. Choose NO to exit the harness version without saving the active harness version and part information.

5. Choose Cancel to cancel the Exit Harness alert menu.

Examples

Example 1

The command line syntax to save the harness version **ver1** and part **wing1**, and exit the harness version, is as follows:

EXIT HARNESS FILE

Example 2

The command line syntax to exit the harness version without saving is:

EXIT HARNESS QUIT

Chapter 9

Converting a Parametric Model to Explicit

Channel routing is performed in the Parametric environment whereas wire routing is performed in the Explicit environment.

This chapter gives an overview of the options for converting a harness part from the Parametrics to Explicit environment after completing channel routing and component insertion.It also gives you an overview of the option for highlighting a harness. These options are available from the Design task set and the Router task set in the explicit environment respectively. The tasks described in this chapter are

• Converting a Harness Model to Explicit

• Highlighting Harness Components

Converting a Harness Model to Explicit

Choose the Convert Harness to Explicit option from the Harness Design task set in CADDS Explicit environment to convert the harness part to the explicit environment. The Parametric Harness Converter property sheet appears.

For more details on the Convert Harness to Explicit option, refer to the *Explicit Harness Layout Menu and Technical Reference*.

OR

Use the GENERATE HDEXPLICIT command in the CADDS text window to convert a harness part from Parametric to Explicit. See the online help for further details on the command-line options.

Highlighting Harness Components

Choose the Highlight Harness option from the Wire Router task set in CADDS Explicit environment to highlight various harness components. This option is active only in the CADDS Explicit environment.

For more details on the Highlight Harness option, refer to Chapter 4, "Constraints-based Router Task Set Overlay" of the *Explicit Harness Layout Menu and Technical Reference*.

OR

Use the HIGHLIGHT HARNESS command in the CADDS text window to initialize a harness layout session. See the online help for further details on the command-line options.

Use this option to highlight the following components in a harness model or part

- Nets
- Wires
- Cables
- Connectors
- Junctions
- Splices
- Clips
- Channels

Please note: You must convert the harness part created to Explicit environment using Generate Harness to Explicit option. See "Converting a Harness Model to Explicit" on page 9-2.

Creating a Component Library

Harness components are created as features using the feature creation capability of the CADDS Parametric Modeler. These harness components can be created and stored in an user specified library. This user-specified library can co-exist along with the library supplied with the Parametric Modeler.

This appendix describes the procedure for creating a harness component library. It also describes how to create and store user-defined harness components in a user specified library. For detailed information on feature creation and storage, refer to the *Feature Based Modeling User Guide and Menu Reference*. The topics discussed here are

- Overview
- Creating a Component Library
- Defining a Feature
- Teaching a Feature

Overview

The creation of an user defined harness component library involves the following steps:

1. Creating the Component Library

   The user creates a new library and assigns it a name This library co-exists along with the Parametric Modeler's system supplied SYSLIB library.

2. Defining the Feature

   Features are composed of geometric and attribute information. This step involves defining attribute information using a hierarchical class structure that supports attribute inheritance from super classes. For details refer to the *Feature Based Modeling User Guide and Menu Reference.*

3. Teaching the Feature

   In this step, the geometry of the component is created using the Parametric Modeler and the attributes defined in above step are associated to the geometry and stored in the specified library.

   This completes the creation of the feature.

Creating a Component Library

To create a component library:

1. Choose FEATURES/PROPERTIES mode from the LDM.

2. Choose LIBRARY MANAGEMENT > ADD LIBRARY from the top menu. The Add Library property sheet appears.
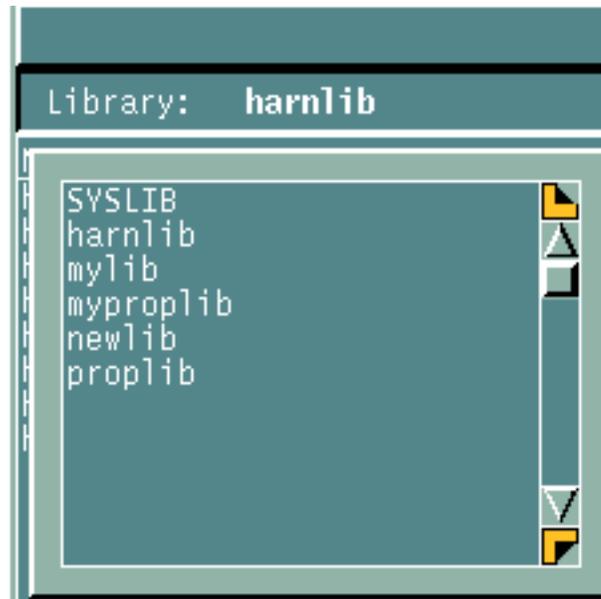
Figure A-1   Add Library property sheet

3. Enter the library name in the Library field.

4. Click Apply.

The component library is created.

Defining a Feature

To define a feature:

1. Choose LDM Mode: FEATURES/PROPERTIES from the top menu.

2. Choose Active Mode: DEFINE from the top menu.

3. Click on the Library field. A scroll list appears displaying a list of library names.

4. Select a user-defined library name from the list.

5. Choose the DEFINE NEW FEATURES/PROPERTY option from the lower half of the screen. The Define Feature/Property property sheet appears.

Figure A-2   Define Feature/Property property sheet

6. Choose the FEATURE option. The Define Feature property sheet appears.

7. Enter the name of the component in the Feature field. Feature Names are case sensitive.

8. Add a static attribute called HDTYPE as follows:

   a. Click the + icon in the Static Attributes section. The Add Static Attribute property sheet appears.

   b. Enter **HDTYPE** in the Name field.

   c. Select String from the Type scroll list.

   d. Enter **connector**, **junction** or **clip** in the Default field.

   e. Click Apply.

The HDTYPE attribute cannot be modified once the feature is inserted into the harness model. Features with an HDTYPE attribute value of connector, junction or clip are identified as valid harness components by the harness application.

9. Use the Attributes option to define various dynamic attributes for the component and assign them default values, for example, "Partno" with a default value "ESC0030". As these are normal attributes, they can be modified even after the feature has been inserted into the harness model.

The following table presents the attribute names and default values for the various attributes that must be created for the appropriate harness components:

Table A-1   Attribute Names and default values

Attribute	Type	Default value	Component
Compname	String	...	connectors, clips
Partno	String	...	connectors, clips
Cflag	Logical	True	connectors
dx	Length	1.0	connectors, clips, junctions
dy	Length	0.0	connectors, clips, junctions
dz	Length	0.0	connectors, clips, junctions

10. Click Apply.
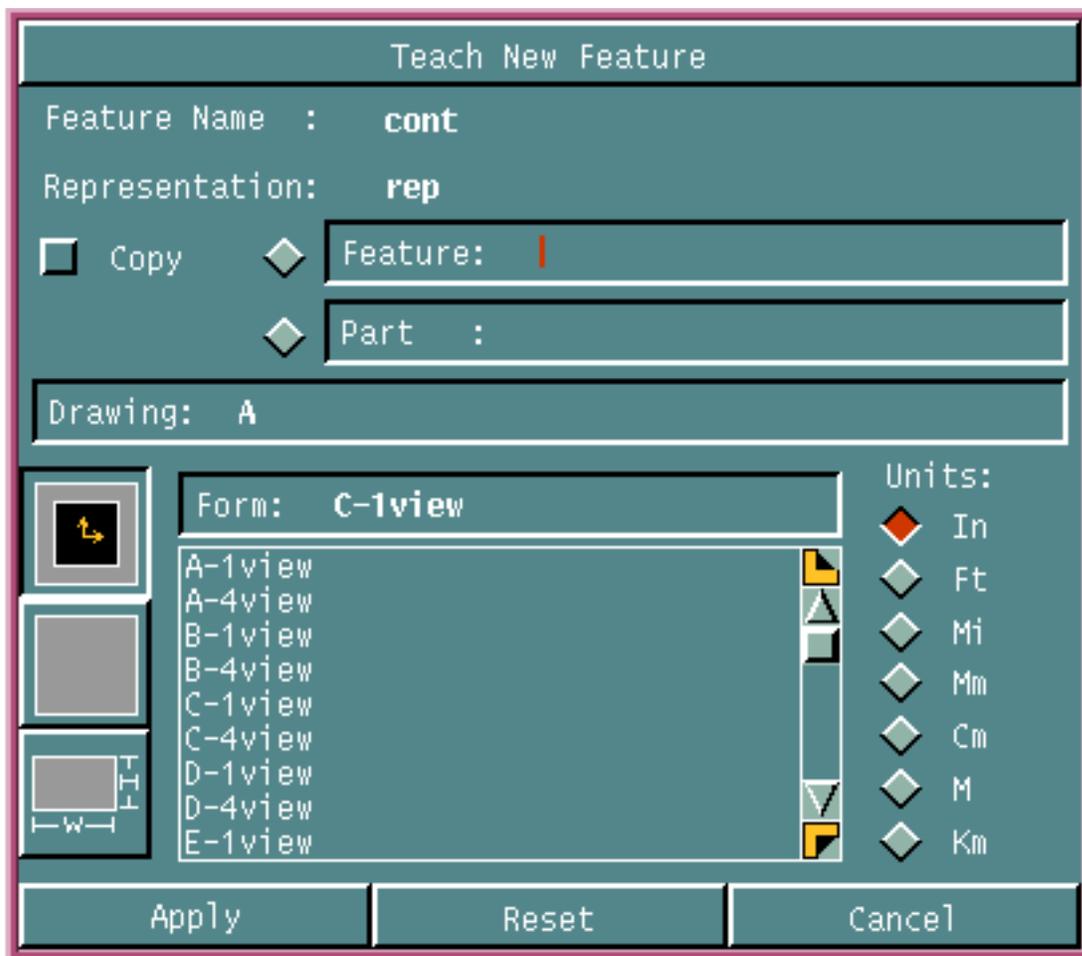
A new feature class is created.

Teaching a Feature

Follow this procedure to teach a feature:

1. Choose FEATURES/PROPERTIES mode from the LDM.

2. Set the active mode to TEACH.

**3.** Choose a defined feature class.

The Teach New Feature property sheet appears.

Figure A-3   Teach New Feature property sheet

4. Fill in this property sheet in the same way as you do when creating a part in CADDS and click Apply.

5. Create the required feature geometry using the standard commands supported by the Parametric Modeling mode.

6. Using the command line option, enter the INSERT CONNECTPOINT command. This command prompts for 2 locations.

   a. Specify the location of the connection point using the [XYZ] option on the Coordinate keypad.

   b. Specify the vector for the tangency direction of the channel that will connect to this point during channel routing using the [DXYZ] option on the Coordinate key-pad.

7. Enter the FEATURES mode. The Features task set appears.
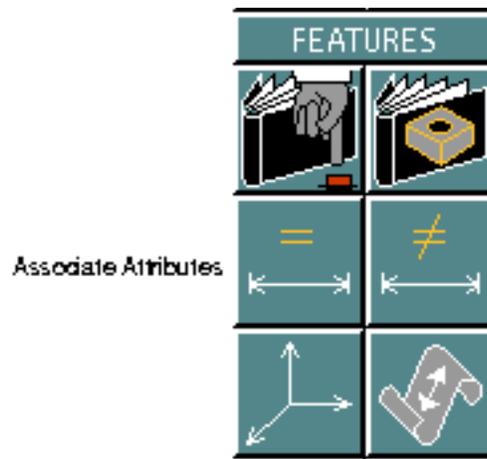
8. Choose the Associate Attributes option. The Associate Attribute property sheet appears.

---

9. Click Workpiece and then select the connect-point geometry on the screen. Similarly, you can select other attributes from the scroll list and attach them to the new geometry.

10. File the part.

The newly created feature is saved in the active library.

Glossary

Some of the terms used in this book and their definitions are described here in alphabetical order.

- Glossary of terms

bundle

> A bundle is a logical grouping of routed wires used as a manufacturing aid.

centerline

> Centerlines are single line representations for pipes, ducts, tubing, cable and channel or harness pathways.

channels

> A path through which wires are ultimately routed. Each channel ends at a junction or a connector.

Centerline Router

> The Centerline Router consists of the INSERT/ CHANGE CHANNEL, INSERT/ CHANGE HDCONNECTOR and INSERT JUNCTION commands in the Object Model and is used to route harness centerlines and insert connectors.

CL Router

> The CL Router is a CVMAC based tool used to route channels in the Harness Layout package.

CBR

> The Constraint Based Router is part of the Object Model and is used to do shortest path and constraint based wire routing, based on an input from the netlist and harness channels/centerlines.

Conceptual Modeling

> In this modeling mode, a component is inserted in the harness model without referring to the `Netlist` file for information.

Connector

> A connector is an electro-mechanical component on which a harness wire starts or terminates.

Clip

> A Clip is a mechanical component attached to an equipment or structure, used to hold the wires of a harness.

Connect Point

> The harness originates from or terminates on the connect point of the connector.

CPL

> Construction Plane or CPlane.

channel segment

> The part of a channel between any two components.

Drip loops

> Drip Loops are "U" shaped loops in a channel that enable condensed moisture on the channel to drip off before it reaches the connector at the end.

EDE

> Electrical Design Entry is a schematic capture package which allows the user to route wires, cables and insert components to form circuits. EDE also creates the netlist file which provides the wire routing information.

harness

> A harness is a complete assembly of wires, cables, and connectors, all wrapped and tied together. It is ready for installation in the equipment (such as a vehicle or aircraft). The complete harness can be made up of many interconnected harness subassemblies. Each of these are referred to as a subharness.

Harness Layout

> The Harness Layout package allows the user to model, route and generate manufacturing information for wire harness. The package includes commands to route channel, insert components, route wires and macros to generate the manufacturing information and drawings.

Harness Unfolding

> Transforms or unfolds the 3-d harness channels and creates 2-d drawings for harness manufacturing.

HLR

> Hidden Line Removal.

IFC

> Interference Checking

Junction

> A junction is a logical component that represents a channel junction or branch, which has significance in wire routing. It is not a physical component.

Mating Point

> The point at which a connector is attached to another connector or a socket on equipment, is referred to as the mating point of the connector.

minimum bend radius

> The minimum bend which a wire, cable, harness, or other conductor can form without causing damage to the conductor, insulator, or shielding.

Nodal Nsplines

> Nodal Nsplines are Nsplines with connectivity and will be used as the entity representing centerlines.

New Nurbs

> The New Nurbs package is an easy to use graphical user interface to create and modify NURB surfaces and Nsplines in the Parametric environment.

Netlist file

> The netlist file is an ASCII file created by Electrical Design Entry (EDE) schematic package that contains all routing information of wires, connectors, cables and components forming the circuit.

Netlist Modeling

> In this modeling mode, a component is inserted in a harness model using the information in the Netlist file.

net

> The net is a list of connections for a given signal e.g. GND.

Object Model

> The Object Model of the Harness Layout package is an Object oriented representation of the various harness components (channel, connectors, etc.) and includes the commands for channel routing, connector insertion and wire routing.

Part Number

> The manufacturer'specification number assigned to a component.

route

> A route is a centerline defining the 3 dimensional path of an acceptable grouping of wires which form a bundle or harness.

Service loops

> Service Loops are typically coils (helices) in a channel. They are normally used for adding extra length to any wire passing through that channel. This is generally used for maintenance purposes where the connecting equipment is very difficult to reach or in a cramped position.

signal

> A signal is a link at the same potential between two or more electrical connections.

Splice

A splice is an electro-mechanical component that is used to connect two or more wires, electrically.

wire

A wire belongs to a net and is routed from a start point to end point.

Wire Router

The wire router is a set of tools used for the routing of wires through channels. It is CVMAC based and is part of the Harness Layout package.

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