Sectioning User Guide

CADDS® 5 15.0

DOC40076-007

Parametric Technology Corporation

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Sectioning in the Explicit Environment

Preface

Sectioning User Guide describes in detail how to use the Sectioning tools in both the Parametric and Explicit environments.

Related Documents

The following documents may be helpful as you use *Sectioning User Guide*:

- *Design and Drafting User Guide and Menu Reference*
- *Explicit Solid Modeling User Guide*
- *Parametric Modeling User Guide and Menu Reference*
- *Parametric Modeler Interface Guide for MEDUSA*

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.

Convention	Example	Explanation
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 and HP	Window Manager Other Than CDE on Solaris, HP, and Windows
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)
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)

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)

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 electronically to `doc-webhelp@ptc.com`.

Chapter 1

Sectioning in the Parametric Environment

This chapter introduces Sectioning and describes how to generate section views from within the Parametric environment.

- Overview of Sectioning
- Creating Entities for Sectioning
- The Sectioning Task Menu
- Using Sectioning Task Menu Options
- Sample Session: Sectioning

Overview of Sectioning

Sectioning as a tool can be applied to either the modeling or drafting environments. As a modeling aid it helps the engineer/designer with design enhancement (sections used as a profile generation mechanism) or conceptualization (cutaways and callouts). Used in the drafting world, sections convey hidden or internal model data in drawing form to manufacturing.

By using the Sectioning Task menu in the Parametric environment you can carry out sectioning and crosshatching operations interactively on a part.

From the Sectioning Task menu you can also preview the section views produced before processing them through the back interface. The sectioning process has no permanent effect on the model created. Only sectioned views of the model are displayed and they can be canceled at any time.

Sectioned views may be transferred to MEDUSA using one of the MEDUSA interfaces, namely the back interface. For further details, refer to the *Parametric Modeler Interface Guide for MEDUSA*.

Entities for Sectioning Operations

When you carry out sectioning operations on one or more entities, you use the following two types of solid entities:

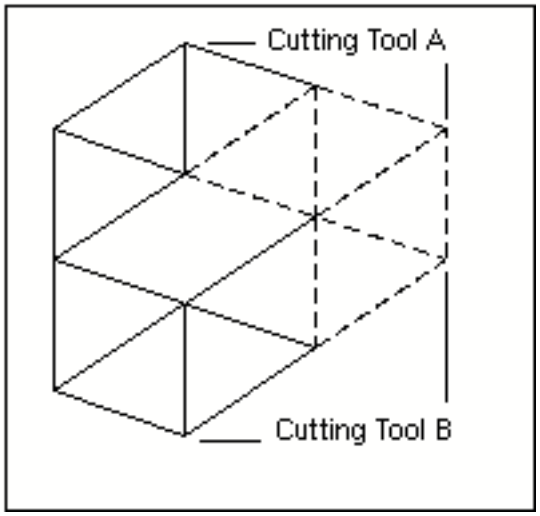
- Section cutting tools
- Sectioned entities

Section Cutting Tools

All the cutting faces of the solid entity to be used as a section cutting tool must be planar for this entity to be a valid cutting tool. For example, curved faces such as those on a cylinder or a sphere are not valid cutting tools.

Also if two section cutting tools overlap each other, as shown in Figure 1-1, each tool will cut the entity independently from the other (see the example on page 1-6). To cut the entity with both tools simultaneously you must merge the two cutting tools with a Boolean operation.

Figure 1-1 Section Cutting Tools Overlapping Each Other



You can create cutting tools using:

- Primitives such as boxes
- Solids created from sweep operations
- Solids created from Boolean operations

Sectioned Entities

A sectioned entity is any solid entity in the part, of which a section view is required. You can use any type of solid to represent the entity to be sectioned.

Overview of the Procedure for Sectioning

To produce and transfer sectioned views, follow these steps:

1. Within the Parametric environment, create the entities to be sectioned.
2. Create the solid entities to represent the section cutting tools by using the standard modeling tools available in the Parametric environment.
3. Carry out sectioning using the Sectioning Task menu options, as follows:
   a. Identify (label) a solid entity as a section cutting tool in the Parametric Modeler database by choosing the Label/Unlabel Tool icon.
   b. Select (define) the elements for sectioning (entities and view) by choosing the Define/Undefine Section icon.

c. Display the section views by choosing the Rendering icon.

d. You can also add crosshatching to the sections by selecting the Crosshatching icon. This must be done after defining the section, but before rendering it.

The steps listed above are detailed in the section “Sample Session: Sectioning” later in this chapter.

Transferring Sectioning Information Using the Back Interface

You can transfer sectioning and crosshatching information to MEDUSA sheets by using the back interface command-line options. One of these options allows you to specify new crosshatching parameters and overrides any parameter values specified when using the Sectioning Task menu. For further information on the back interface sectioning command-line options, see the *Parametric Modeler Interface Guide for MEDUSA*.

Please note: If you transfer the sectioned views to MEDUSA using one of the MEDUSA interfaces, you can crosshatch the section planes either automatically using the back interface or at a later date once they have been transferred to the MEDUSA sheet. If you choose the second method you must crosshatch using MEDUSA 2D Design.

Creating Entities for Sectioning

There are many methods available to create section cutting tools. The major criterion is that the cutting faces are planar. Examples of methods that produce valid cutting tools are explained in this section.

You can create section cutting tools using:

- Primitives such as boxes
- Solids created from sweep operations
- Solids created from Boolean operations

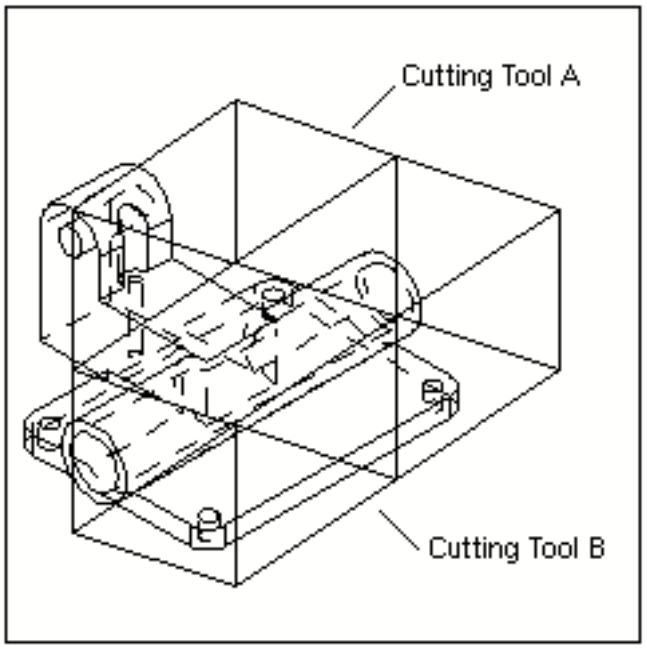
Creating Cutting Tools Using Primitives

You can create section cutting tools using primitives such as boxes.

If you create two boxes that overlap, as shown in the following figure, you obtain two section views: one from Cutting Tool A in the front view and one from Cutting Tool B in the side view.

To cut the object with both tools simultaneously, you must merge the two solids using a Boolean operation.

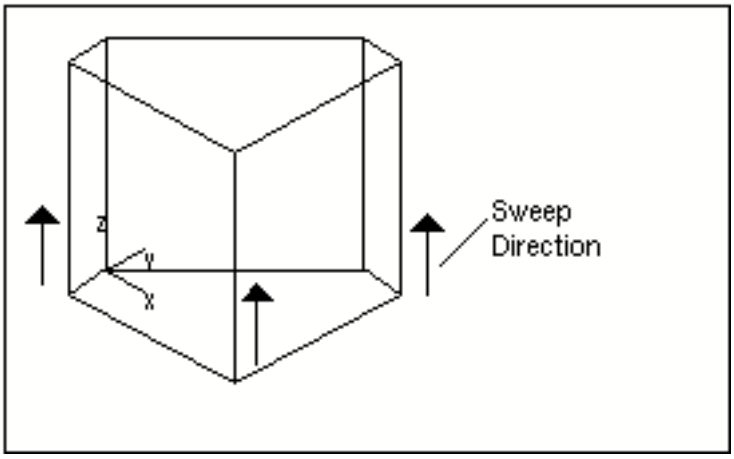
Figure 1-2 Creating Cutting Tools Using Boxes



Creating Cutting Tools by Conducting Sweep Operations

You can also produce cutting tools by conducting sweep operations. Create a linear polycurve and then produce the required solid by sweeping the curve in a direction normal to the plane. An example is shown below.

Figure 1-3 Creating Cutting Tools by Sweeping a Curve



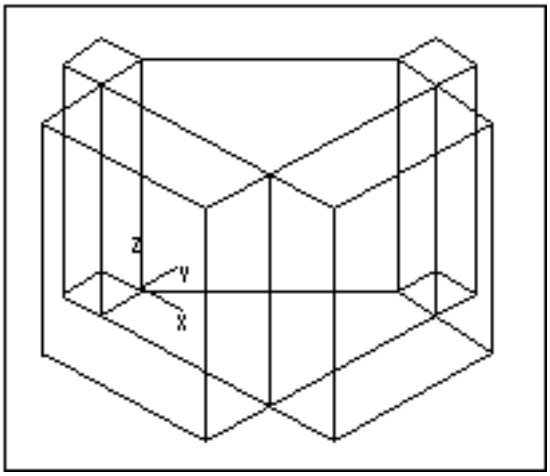
By conducting linear sweeps, you can also produce cutting tools of a more complex shape. An example is given below.

1. Create a solid similar to the one shown in the above figure.
2. Sweep a face to a specified depth in a direction normal to the plane using the Move Face Sweep option. A single object is created and hence Boolean operations need not be conducted.

For more details on the Move Face Sweep option, see Chapter 8 of the *Parametric Modeling User Guide and Menu Reference*.

If you sweep several faces you can create a more complex cutting tool, as shown in Figure 1-4.

Figure 1-4 Creating Cutting Tools by Sweeping Several Faces



Creating Cutting Tools by Conducting Boolean Operations

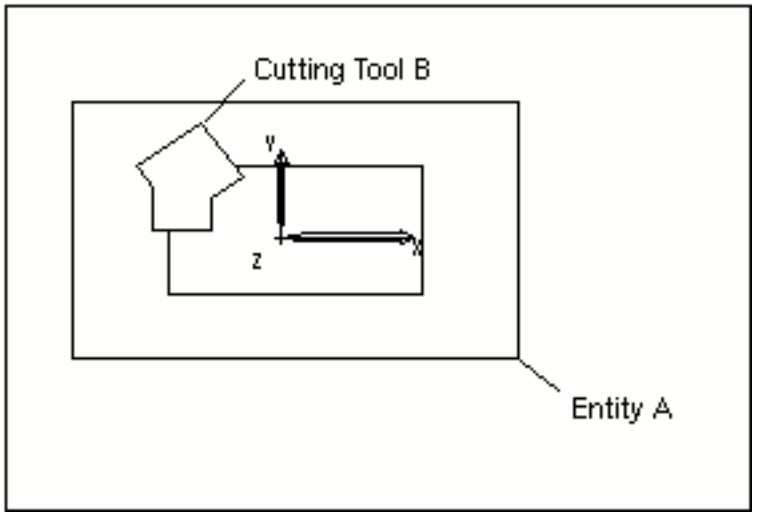
To perform a sectioning operation on one object using multiple section cutting tools, join the section cutting tools together using a Boolean operation before performing the sectioning operations.

If you need to use many cutting tools, you may find it useful to place them on a separate layer so that you can exclude them at any time. This allows you to make these cutting tools invisible when needed. You can also control the visibility of the cutting tools from the sectioning menus.

If the relevant section cutting tools intersect, you can perform Boolean operations easily using the standard CADDS commands.

For example, as shown in Figure 1-5, Cutting Tool B was created by performing a Boolean union between two intersecting solids. Entity A is the entity to be cut.

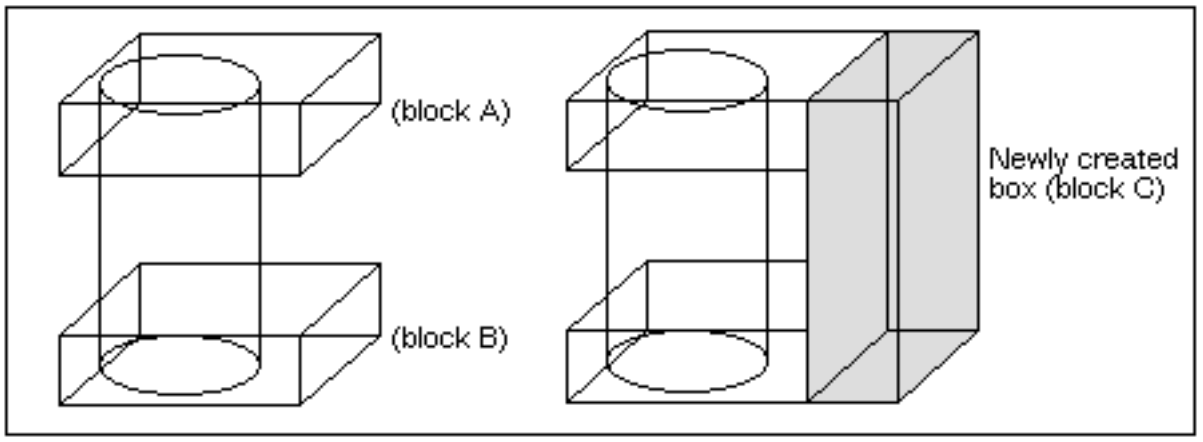
Figure 1-5 Example



If the relevant section cutting tools do not intersect, create a third section cutting tool and then boolean all the three section cutting tools together.

For example, as shown in the following figure, blocks A and B do not intersect. A new block, block C, was created to link them and then Boolean addition operation was performed on all three blocks.

Figure 1-6 Example



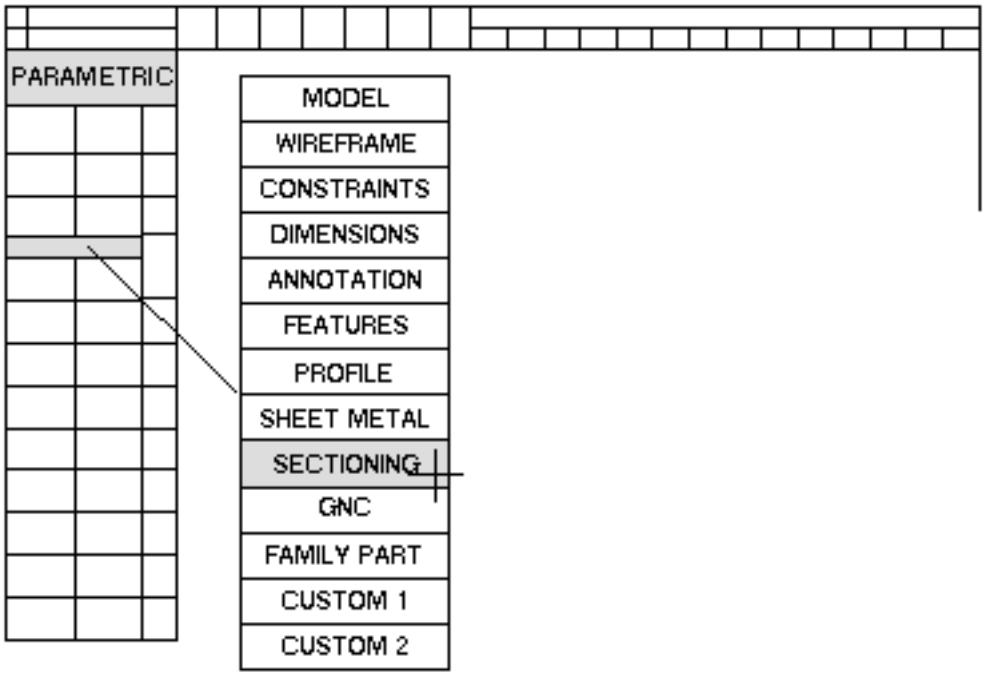
The Sectioning Task Menu

To perform sectioning and crosshatching operations, choose the options from the Sectioning Task menu.

Accessing the Sectioning Task Menu

To start any sectioning operation, choose the Sectioning menu tab from the Task Set menu shown in the following figure. This displays the Sectioning Task menu.

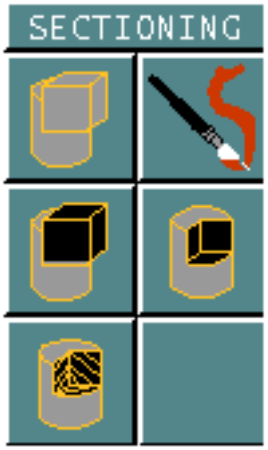
Figure 1-7 Accessing the Sectioning Task Set Menu



The Sectioning Task Menu

The Sectioning Task menu, as shown below, contains five options. Each option displays a property sheet and has a corresponding command.

Figure 1-8 The Sectioning Task Set Menu Options



The Sectioning Task Menu Options

This section briefly describes the options available on the Sectioning Task menu. Each option is described in detail later in this chapter.

Label/Unlabel Tool

The Label/Unlabel Tool allows you to identify by name a solid entity as a section cutting tool in the Parametric Modeler database. It also allows you to revert the section cutting tool back to a normal solid entity.

Show/Unshow Tool

The Show/Unshow Tool controls the display of section cutting tools. It allows you to blank, unblank, or highlight these cutting tools.

Crosshatching

The Crosshatching icon defines the crosshatching requirements. It allows you to specify crosshatching parameters like angle and spacing.

Rendering

The Rendering icon displays the section views with shading or hidden lines.

Define/Undefine Section

Define/Undefine Section allows you to specify the elements for sectioning (entities and view). Undefine Section allows you to undo this operation.

Using Sectioning Task Menu Options

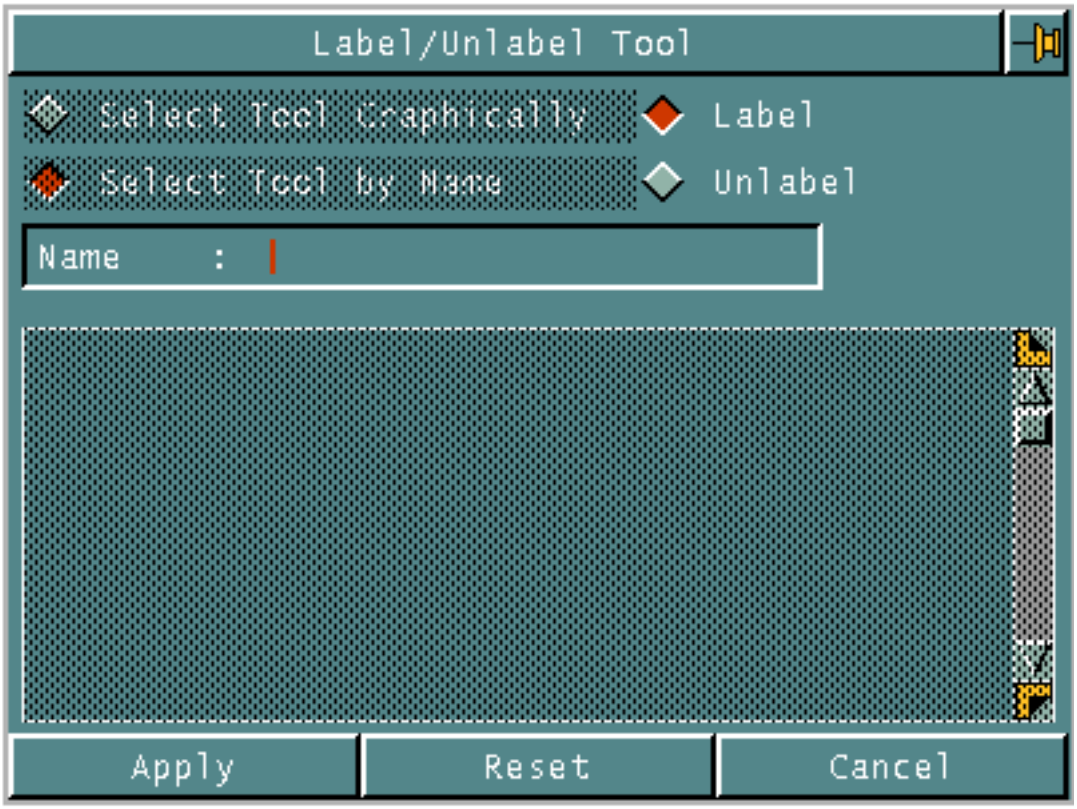
This section fully describes the options that are available from the Sectioning Task menu. An example is provided if appropriate.

Label/Unlabel Tool

The Label Tool identifies by name a solid entity as a section cutting tool in the Parametric Modeler database. The Unlabel Tool allows the section cutting tool to revert to a normal solid entity.

To label or unlabel a solid entity, choose the Label/Unlabel Tool icon from the Sectioning Task set menu. The Label/Unlabel Tool property sheet, as shown below, appears. The Label option is chosen by default.

Figure 1-9 Label/Unlabel Tool Property Sheet



The Label option allows you to identify by name the solid entity that will be used as the section cutting tool. This step is compulsory for any subsequent sectioning operation. An entity can only be used as a section cutting tool in a sectioning operation, if it is first named as such in the database.

The Unlabel option allows you to revert the named section cutting tool to a normal solid entity in the database.

Please note: If your database already contains entities identified as section cutting tools, they are included in the list when the property sheet appears.

Labeling a Solid Entity

To label a solid entity, follow these steps:

1. Choose the Label option from the Label/Unlabel Tool property sheet.
2. Enter a name for the section cutting tool in the Name field.
3. Choose Apply.
4. When prompted, select a solid entity.

   The entity is highlighted when it is selected.

Please note: When the Label option is chosen, the Select Tool Graphically and Select Tool by Name options that allow you to select a tool are no longer active.

When you complete the operation, the property sheet disappears. The solid entity that you have named is now identified as a section cutting tool in the database. If the selected solid entity was already named as a cutting tool, it is now renamed with the name you specified. This does not affect any section or crosshatch information already associated with the cutting tool but only renames this tool.

If you want a section tool to revert to a normal solid entity in the database, use the Unlabel option.

Please note: To be able to merge two section cutting tools by a Boolean operation you must first ensure that they are no longer named as section cutting tools using the Unlabel option. You can then rename the single entity as a section cutting tool using the Label option.

Unlabeling Cutting Tools

To unlabel a solid entity, follow these steps:

1. Choose the Unlabel option from the Label/Unlabel Tool property sheet.
2. Choose the Select Tool Graphically or Select Tool by Name option.
   - If you want to select the cutting tool graphically, choose Apply and then select the required solid entity.
   - If you want to select the tool by name, choose the required name from the list. When this name appears in the Name field, choose Apply.

In the database, the solid entity selected is no longer identified as a cutting tool. All the section view definitions that were related to this cutting tool are also deleted.

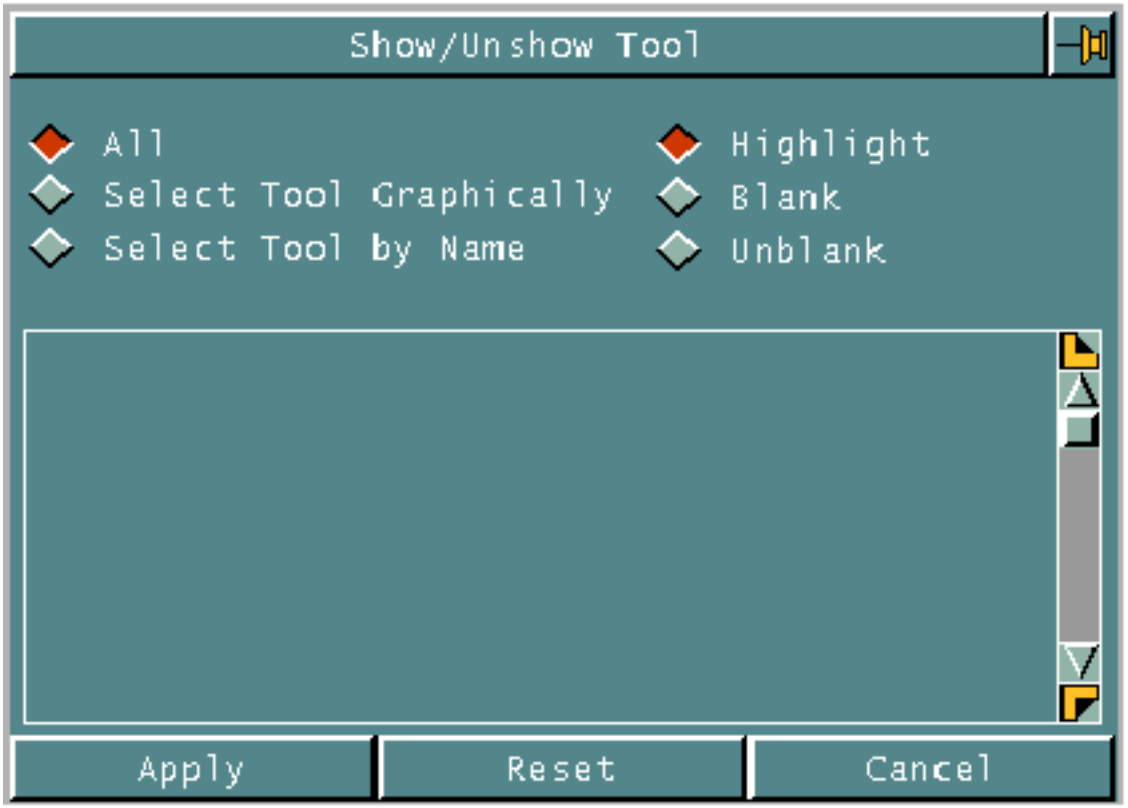
Please note: When you use this option, the solid entity identified as a cutting tool is not deleted from the database. It only reverts from a section tool to a normal solid object. If you delete a solid entity that is defined as a cutting tool, you also delete associated sectioning definitions from the database.

Show/Unshow Tool

The Show/Unshow Tool controls the display of section cutting tools. It allows you to blank, unblank, or highlight these cutting tools.

To control the display of section cutting tools, select the Show/Unshow Tool icon from the Sectioning Task set menu. The Show/Unshow Tool property sheet, as shown in Figure 1-10, appears. The All and the Highlight options are chosen by default.

Figure 1-10 Show/Unshow Tool Property Sheet



Highlighting Cutting Tools

To highlight all objects identified as cutting tools, follow these steps:

1. Choose the Highlight option from the Show/Unshow property sheet.
2. Choose the All option.
3. Choose Apply.

Alternatively, you can highlight individual cutting tools as follows:

1. Choose the Highlight option.
2. Choose the Select Tool Graphically or Select Tool by Name option.
   - If you want to select the cutting tool graphically, choose Apply and then select the required solid entity.
   - If you want to select the tool by name, choose the required name from the list. When this name appears in the Name field, choose Apply.

Controlling the Visibility of Cutting Tools

By using Show/Unshow Tool, you can also make the section cutting tools visible or invisible.

To control the visibility of all the cutting tools, follow these steps:

1. Choose the Blank or Unblank option as appropriate.
2. Choose the All option.
3. Choose Apply.

Alternatively, you can control the visibility of individual cutting tools.
To make a cutting tool visible, follow these steps:

1. Choose the Blank option.
2. Choose the Select Tool Graphically or Select Tool by Name option.
   - If you want to select the cutting tool graphically, chose Apply and then select the required solid entity.
   - If you want to select the tool by name, select the required name from the list. When this name is displayed in the Name field, choose Apply.

The cutting tool whose name is specified is made invisible. If you want to make it visible again, you can use the same procedure with the Unblank option chosen.

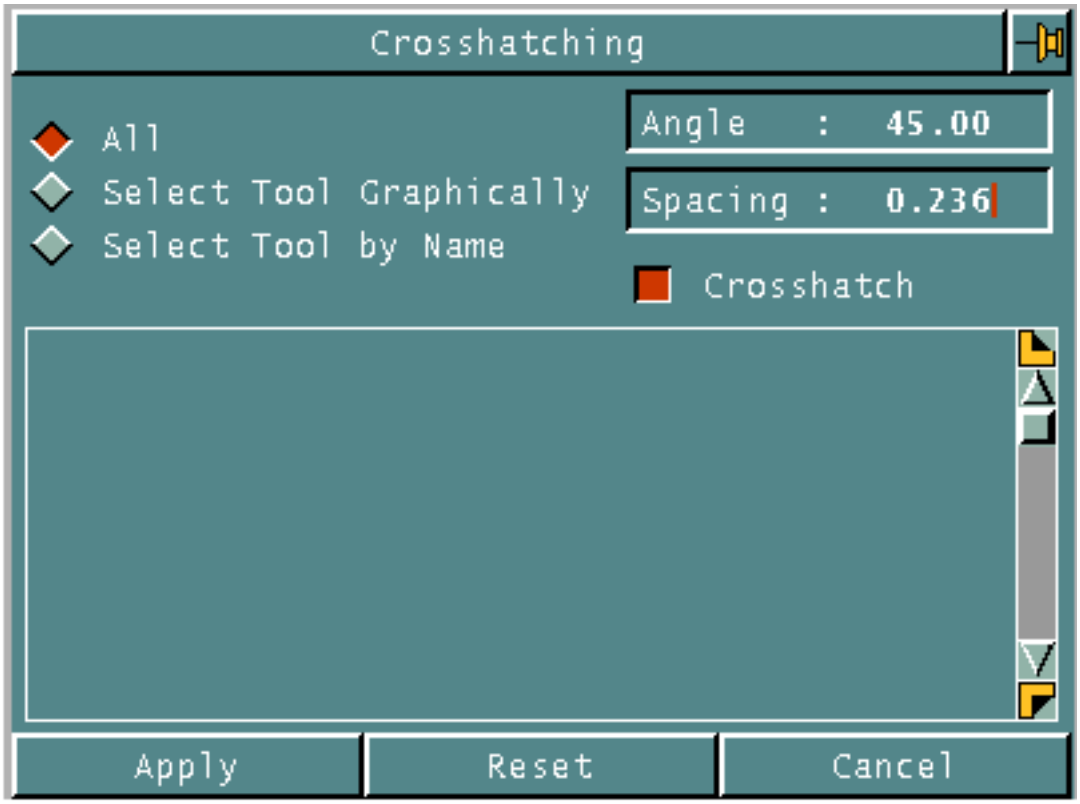
Please note: The Unblank option does not allow you to select the cutting tool graphically. Use the Select Tool by Name option and then choose the required name from the list.

Crosshatching

The Crosshatching icon allows you to specify the crosshatching requirements for the sections defined.

To specify the crosshatching requirements, choose the Crosshatching icon from the Sectioning Task set menu. The Crosshatching property sheet, as shown in Figure 1-11, appears. The All and the Crosshatch options are chosen by default.

Figure 1-11 Crosshatching Property Sheet



Crosshatching Parameters

To define crosshatching parameters for sectioned entities, follow these steps:

1. Change the Angle and Spacing parameters to suit your requirements or use the default parameters as shown:
   - Angle 45.0
   - Spacing 0.236 in (imperial) or 6 mm (metric)
2. Select the cutting tool that will use the crosshatching specified, either graphically or by name.

   If you want to select the cutting tool graphically, choose Apply and then select the required solid entity used as a cutting tool.

   If you want to select the tool by name, choose the required name from the list and then choose Apply.
3. Choose the Render Section icon to display the crosshatching.

Rendering

The Rendering icon displays the section views. When you choose it, a pulldown menu appears with these two options:

- Hidden Lines
- Shade

You must choose an option and then select the required view(s). The sectioned entities are displayed in the view(s) selected.

Displaying Wireframe Sections

If you choose the Hidden Lines option, wireframe sections are displayed in the view(s) selected. Crosshatching is also displayed if you use the Crosshatching option.

Displaying Shaded Sections

If you choose the Shade option, the sections appear shaded in the view(s) selected. Crosshatching is also displayed if it has been selected.

Redisplaying the Original Views

To redisplay the whole drawing with the original views, that is, without sections, press RETURN at the end of the command line. Choosing another menu option has the same effect and reinstates the original views.

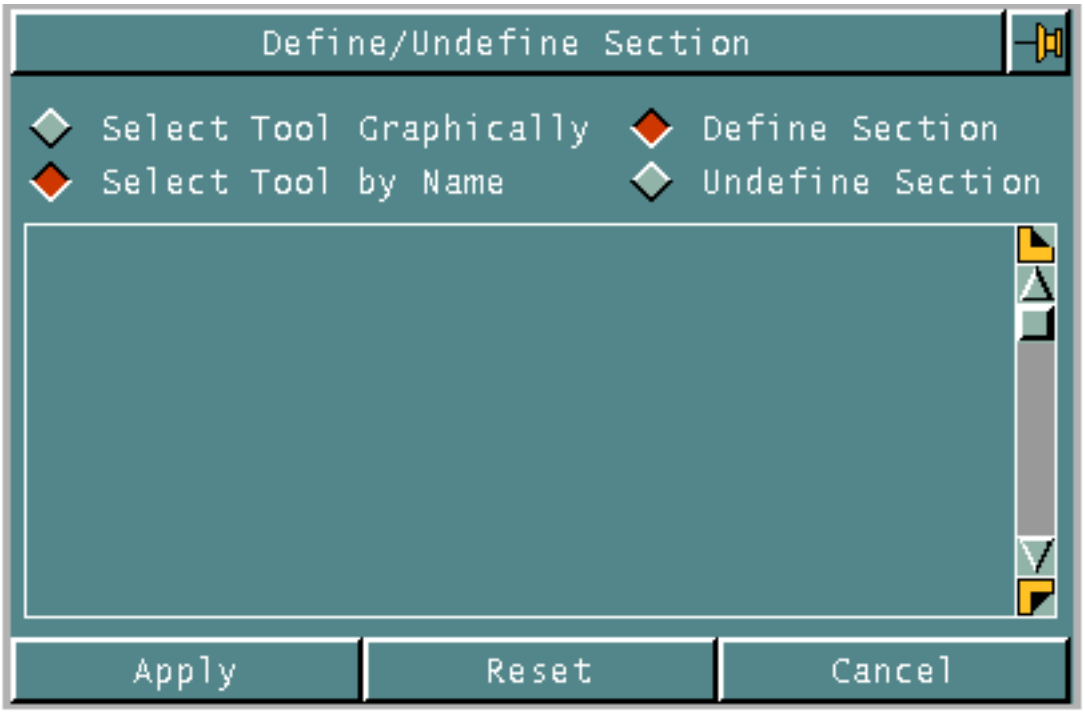
Currently, sections do not render on the HP platform.

Define/Undefine Section

Define Section allows you to specify the elements for sectioning (entities and view). Undefine Section allows you to undo this operation.

To define or undefine a section cutting tool, choose the Define/Undefine Section icon from the Sectioning Task set menu. The Define/Undefine Section property sheet, as shown in Figure 1-12, appears. The Define Section and the Select Tool by Name options are chosen by default.

Figure 1-12 Define/Undefine Section Property Sheet



The Define Section option allows you to specify which section cutting tool cuts which solid object in which view.
The Undefine Section option allows you to undo this operation, that is, cancel each sectioning operation stored in your database.

Defining a Section

To define a section, follow these steps:

1. Choose the Define Section option.
2. Select the Select Tool Graphically or Select Tool by Name option.
   - If you want to select the cutting tool graphically, choose Apply and then select the required solid entity.
   - If you want to select the tool by name, select the required name from the list and then choose Apply.

     In both cases, the Done option appears.

3. From any view, select the entities to be sectioned. The entities are highlighted when selected. These are the entities that will be sectioned by the cutting tool in the view that you have specified.

Please note: When defining section cutting tools, make sure that they do not have faces that are coincident with any of the faces of the section object, as this can cause problems later.

4. Select Done when you have selected all the entities to be sectioned.

Undefining a Section

To undefine a section, follow these steps:

1. Choose the Undefine Section option.
2. Choose the Select Tool Graphically or Select Tool by Name option.

   If you want to select the cutting tool graphically, choose Apply and then select the required solid entity.

   If you want to select the tool by name, select the required name from the list.

   In both cases, the Done option appears.
3. Select the entities that were previously sectioned.

   The entities are highlighted as you select them.
4. Choose Done to complete the procedure.

Please note: A quick way to undefine sections, if you are unsure about the views or entities that have been used, is to unlabel the section cutting tool. All the sections defined using this section cutting tool are canceled.

Sample Session: Sectioning

This section shows a simple example of how to create a sectional view using the Sectioning Task menu options.

Overview

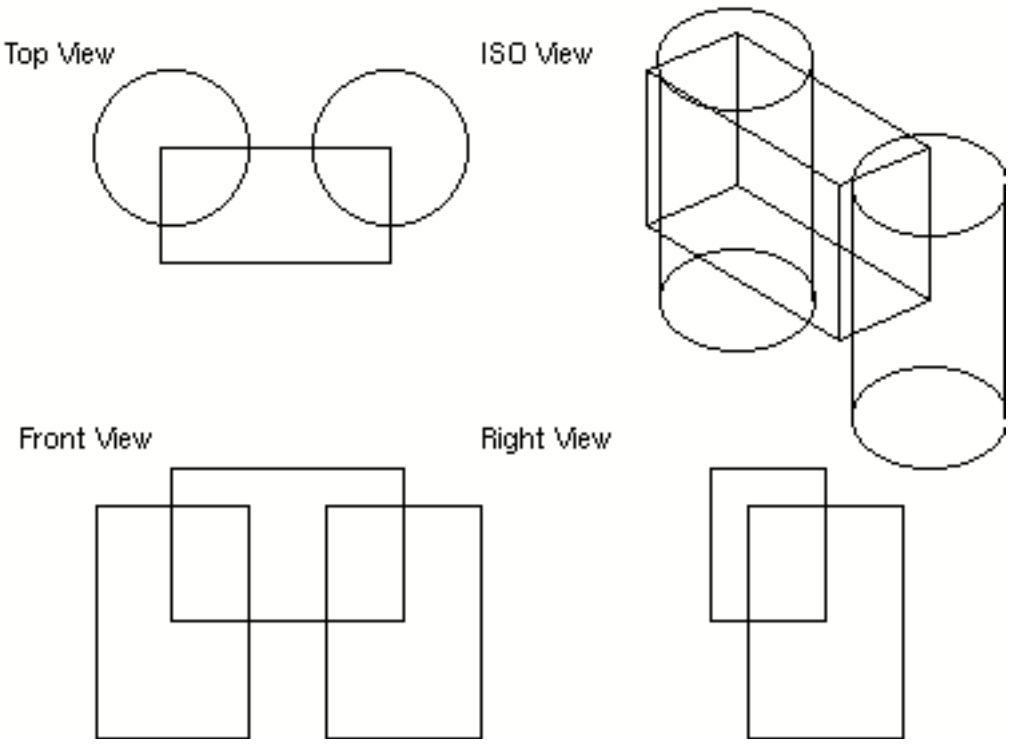
Two cylinders and a box are created. The box is made to overlap both cylinders (see the following figure). The cylinders and the box are normal Parametric Modeler solid entities.

To cut the two cylinders with the box and display the section views generated, follow these steps:

1. Name the section cutting tool (box), by choosing the Label/Unlabel Tool icon.
2. Select the entities for sectioning (cylinder and box) and a view, by choosing the Define/Undefine Section icon.
3. Display the section in the views selected by choosing the Rendering icon.

You can also crosshatch the sections by choosing the Crosshatching icon.

Figure 1-13 Sample Session: Sectioning



Naming the Section Cutting Tool

You first name and select the entity to be identified as a section cutting tool in the database, follow these steps:

1. Choose the Label/Unlabel Tool icon.
2. Choose the Label option, then type a cutting tool name into the Name field. For this exercise, enter the name `sec-AA` and then choose Apply.
3. Select the cutting tool by graphically selecting the required solid entity (the box in this example). The box is highlighted. When you complete the operation, the property sheet disappears.

   To select a cutting tool, graphically select the required entity or use any other valid method of selection.

The box is now a section cutting tool named sec-AA in the Parametric Modeler database.

Producing the Sections

To produce the sections, you must then select the entities used in the sectioning operations (cylinder and box) and a view. To display the sections produced, in the view selected, use the Rendering option.

To produce the sections, follow these steps:

1. Choose the Define/Undefine Section icon. The section cutting tool, sec-AA, that you have just named appears in the list.
2. Select the cutting tool (box in this example) by name as follows:
   a. Make sure that the Select Tool by Name option is chosen. This is the default setting.
   b. Choose sec-AA from the list and then choose Apply.

   Alternatively:

   a. Choose the Select Tool Graphically option.
   b. Choose Apply and then graphically select the cutting tool (box in this example). The box is highlighted.

   In both cases, when you choose the Apply option, the Done option appears.
3. Select the view (ISO view in this example) in which the sections will be displayed.

4. From any view box, select the solid entity to cut through (leftmost cylinder in this example). The cylinder is highlighted in all the view boxes.
5. Choose Done.

In this example, we have selected the isometric view and the leftmost cylinder.

As part of the same exercise, repeat the above procedure but this time, select the front view and both the cylinders.

Displaying the Sections

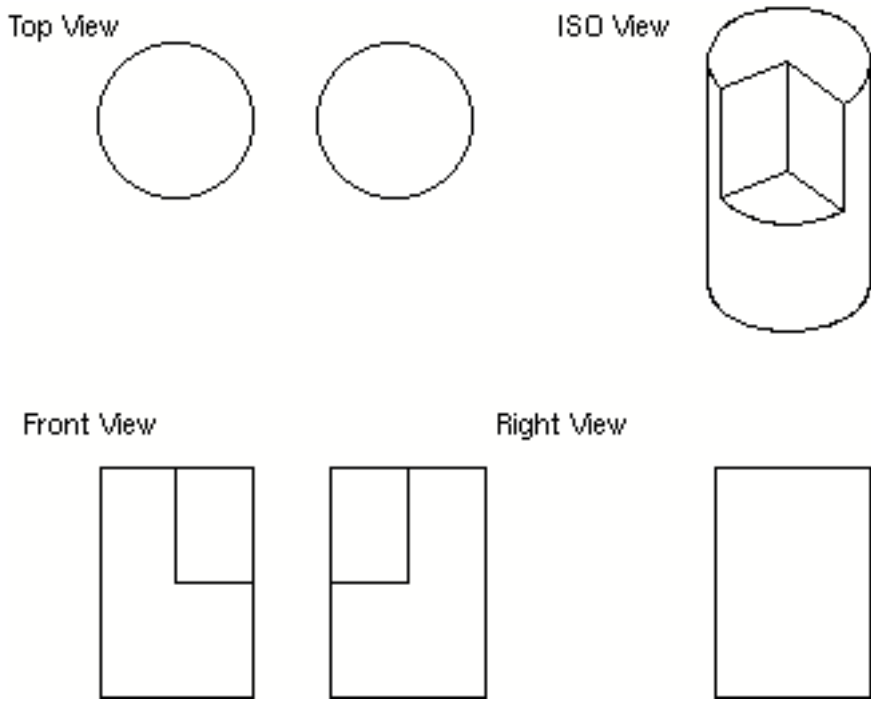
To display the section views, choose the Rendering icon. Two options are available on this pulldown menu:

- Hidden Lines (for a wireframe representation)
- Shade (for a shaded representation)

You must choose an option and then select the view(s) defined.

The following figure shows the cylinders sectioned with the section cutting tool sec-AA (box).

Figure 1-14 Sample Session: Sectioning



When the isometric view and then the front view are selected, the leftmost cylinder appears sectioned in the isometric view and both cylinders appear sectioned in the front view.

To redisplay the whole drawing with the original views, that is, without sections, press RETURN at the end of the command line. Choosing another Rendering option has the same effect and reinstates the original views.

Crosshatching the Sections

Once you produce the sections using the Define option, you can crosshatch them by using the Crosshatching option as follows.

Please note: The crosshatching is only displayed when you choose the Rendering option from the Sectioning Task menu.

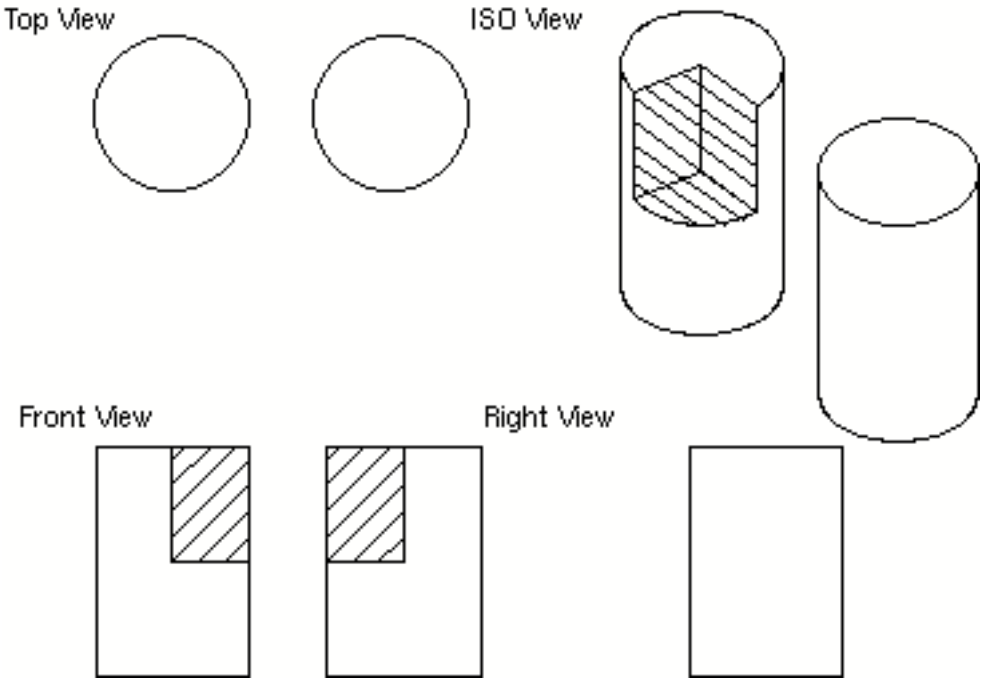
1. Choose the Crosshatching icon.
2. Change the Angle and Spacing parameters to suit your requirements or use the default parameters as shown:
   - Angle 45 degrees
   - Spacing 0.236 in (imperial) or 6 mm (metric)

   In this example, the default values were used for the crosshatching parameters.
3. Select the cutting tool (box in this example) using one of the following:
   - By choosing sec-AA from the list, then selecting Apply.
   - By choosing the Select Tool Graphically option and then selecting the cutting tool entity (box in this example).
4. Choose the Rendering icon to display the crosshatched section views. Two options are available on this pulldown menu:
   - Hidden Lines (for a wireframe representation)
   - Shade (for a shaded representation)

   You must choose an option and then select the views required.

Figure 1-15 shows the cylinders sectioned with the section cutting tool sec-AA (box). The Crosshatching option is chosen.

Figure 1-15 Sample Session: Sectioning



When the isometric view and then the front view are selected, the leftmost cylinder appears sectioned in the isometric view and both cylinders appear sectioned in the front view. The sections are crosshatched.

To redisplay the whole drawing with the original views, that is, without sections, press RETURN at the end of the command line. Choosing another Rendering option has the same effect and reinstates the original views.

Chapter 2

Sectioning in the Explicit Environment

This chapter describes in detail how Sectioning as a tool can be applied to the Explicit environment.

- The Sectioning Menu
- The Sectioning Menu Options
- Define Section
- Examples of Defining Sections
- Undefine Section
- Example of Undefining a Section
- Regenerate Section
- Define View Clipping
- Example of Defining a View Clipping
- Revise View Clipping
- Generate Cross Sections Using Solids
- Example of Generating Cross Sections Using Solids
- Generate Cross Sections Using Strings
- Example of Generating Cross Sections Using Strings
- Generate Cross Sections Using Surfaces
- Boolean Operations
- Example of Intersecting Solids
- Multiple Boolean Operations

The Sectioning Menu

To perform sectioning operations in the drafting environment, choose the options from the Sectioning menu.

Accessing the Sectioning Menu

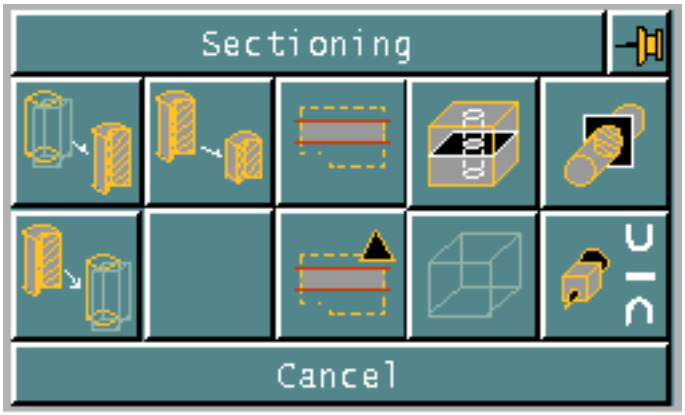
To start any sectioning operation, follow these steps:

1. Choose the Drafting option from the Task set menu. The Drafting Task set menu appears.
2. Choose the Section icon from the Drafting Task set menu. The Sectioning menu appears as shown in the following figure.

The Sectioning Menu

The Sectioning menu has nine options that allow you to section a model.

Figure 2-1 The Sectioning Menu



The Sectioning Menu Options

This section briefly describes the options available on the Sectioning menu Each option is described in detail later in this chapter.

There are a many ways to section a model. Each way depends on the type of model entities you are working with.

Define Section

Define Section allows you to create nondestructive sections. Nondestructive sections are sections where the sectioner and the sectionee(s) remain intact even after the sectioning operation. The view, label, and geometry data related to the created section are added to the current database and displayed.

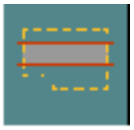
Undefine Section

Undefine Section removes sections generated by the Define Section command. All the display views, crosshatch curves, section boundaries, limblines (in the case of CUTAWAY sections), and section labels created during the Define Section command operation are removed. Undefine Section operates on one view at a time.

Regenerate Section

Regenerate Section updates sections created by the Define Section command. Any modifications made to the sectioner and the sectionee(s), can be reflected in the current database and displayed by executing the Regenerate Section command. The selected display views, crosshatching curves, section boundaries, limblines (in the case of CUTAWAY sections), and section labels are updated accordingly.

Define View Clipping



Define View Clipping allows you to define a view for z-clipping or perspective. You can clip from the front, back, front and back, or take a slice from the model geometry. Unlike folding an existing view, this method does not always automatically align the new view.

Revise View Clipping

Revise View Clipping allows you to revise a view for z-clipping or perspective. You can clip from the front, back, front and back, or take a slice from the model geometry.

Generate Cross Sections Using Solids

Generate Cross Sections Using Solids allows you to create a cross section of a solid using a plane or another solid.

Generate Cross Sections Using Strings

Generate Cross Sections Using Strings allows you create a cross section by creating lines at the intersection of a user-defined plane and three-dimensional model graphics. You can use multiple cutting planes.

The Generate Cross Sections from Strings option can be used on CADDS 4X parts (3D Model graphics generated from the Project Outline command) which have been converted to CADDS 5 parts.

Generate Cross Sections Using Surfaces

Generate Cross Sections Using Surfaces allows you to create cross sections or intersects curves, surfaces (including planes), Tsurfaces, and solids (by creating Nsplines, B-splines, strings, or Cpoints) using parallel planes, any selected planar entities or with planes defined along an spine curve.

Boolean Operations

Boolean Operations allows you to create new solids or change existing solids by:

- Joining two or more solids together
- Subtracting the volume of one solid from another
- Creating a new solid from the common volume between two intersecting solids

Define Section

You can create nondestructive sections using the Sectioning menu in the Drafting environment. Nondestructive sections are sections where the sectioner and the sectionee(s) remain intact even after the sectioning operation. The view, label, and geometry data related to the created section are added to the current database and displayed.

You can create nondestructive sections by using:

- A linear curve, string, plane, or solid as a sectioner
- Well-formed solids, Nfigures, or Nlines with the appropriate AEC properties as sectionee(s)

Types of Sections

You can create the following types of sections using the Sectioning menu. These three options are mutually exclusive. A Simple section is created by default.

Simple

A Simple section comprises of intersection boundaries and crosshatching.

Cutaway

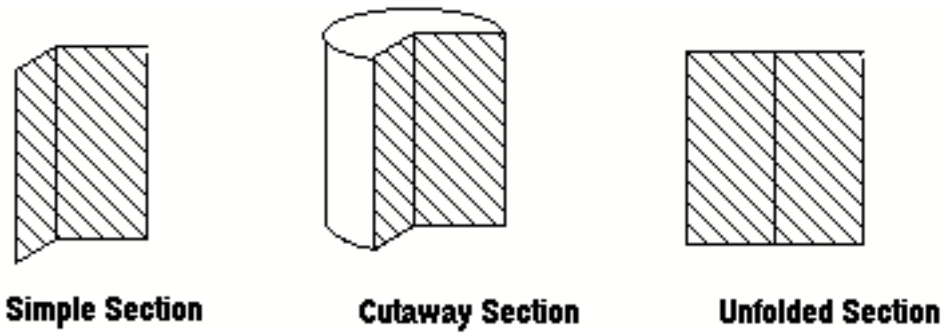
A Cutaway section comprises intersection boundaries and crosshatching, with the hidden lines either blanked or dashed on the resulting sectionee(s).

Unfolded

An Unfolded section comprises of intersection boundaries and crosshatching. The planar faces created by the sectioning operation are unfolded along the common edge until they become coplanar.

Figure 2-2 gives you an example of the three different types of sections.

Figure 2-2 Examples of Different Types of Sections



Data Required for Defining a Section

To define a section, you need the following data:

- Sectioner: The sectioner could be a linear curve, string, plane, or a solid.
- Sectionee(s): The sectionee(s) could be a well-formed solid, Nfigure, or an Nline with the appropriate AEC properties.
- Definition View: The Definition View is the view in which the sectioning operation is defined.
- Display View: The Display View is the view in which the sectioning results are displayed.

  By default, the definition view is selected as the display view. However, if the View Name option on the Define Section View property sheet is used, a new display view can be created. The sectioner and the sectionee(s) are blanked in the display view.

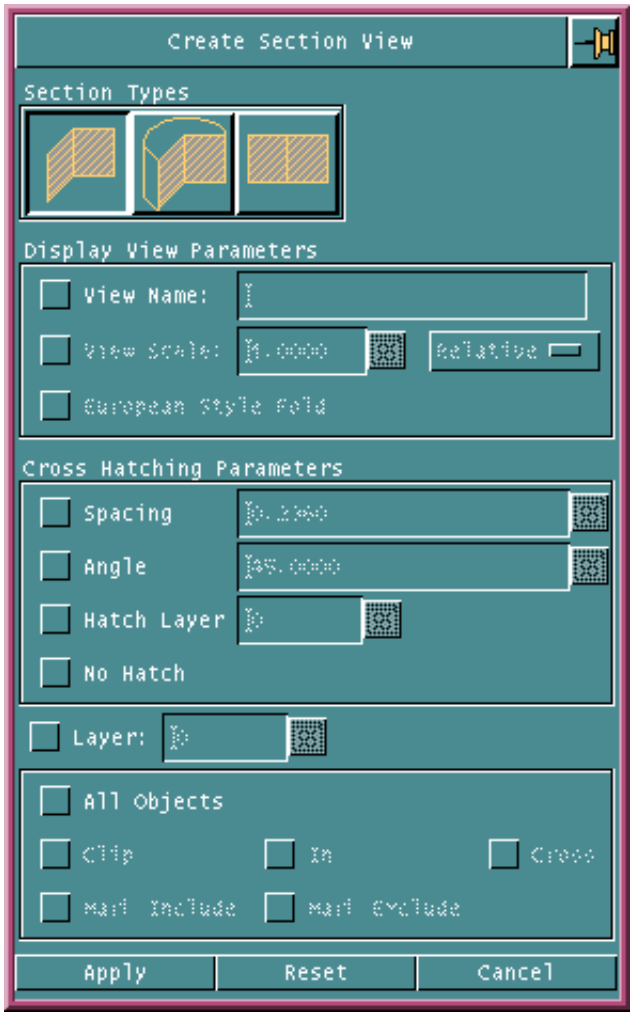
  The sectioner and the sectionee(s) can be used in more than one sectioning operation. In particular, a definition view can be used for more than one sectioning operation. However, a display view can contain the results of only one sectioning operation.

Defining a Section

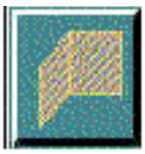
To define a section, follow these steps:

1. Select the Define Section icon from the Sectioning menu. The Create Section View property sheet appears.

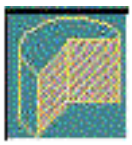
Figure 2-3 Create Section View Property Sheet



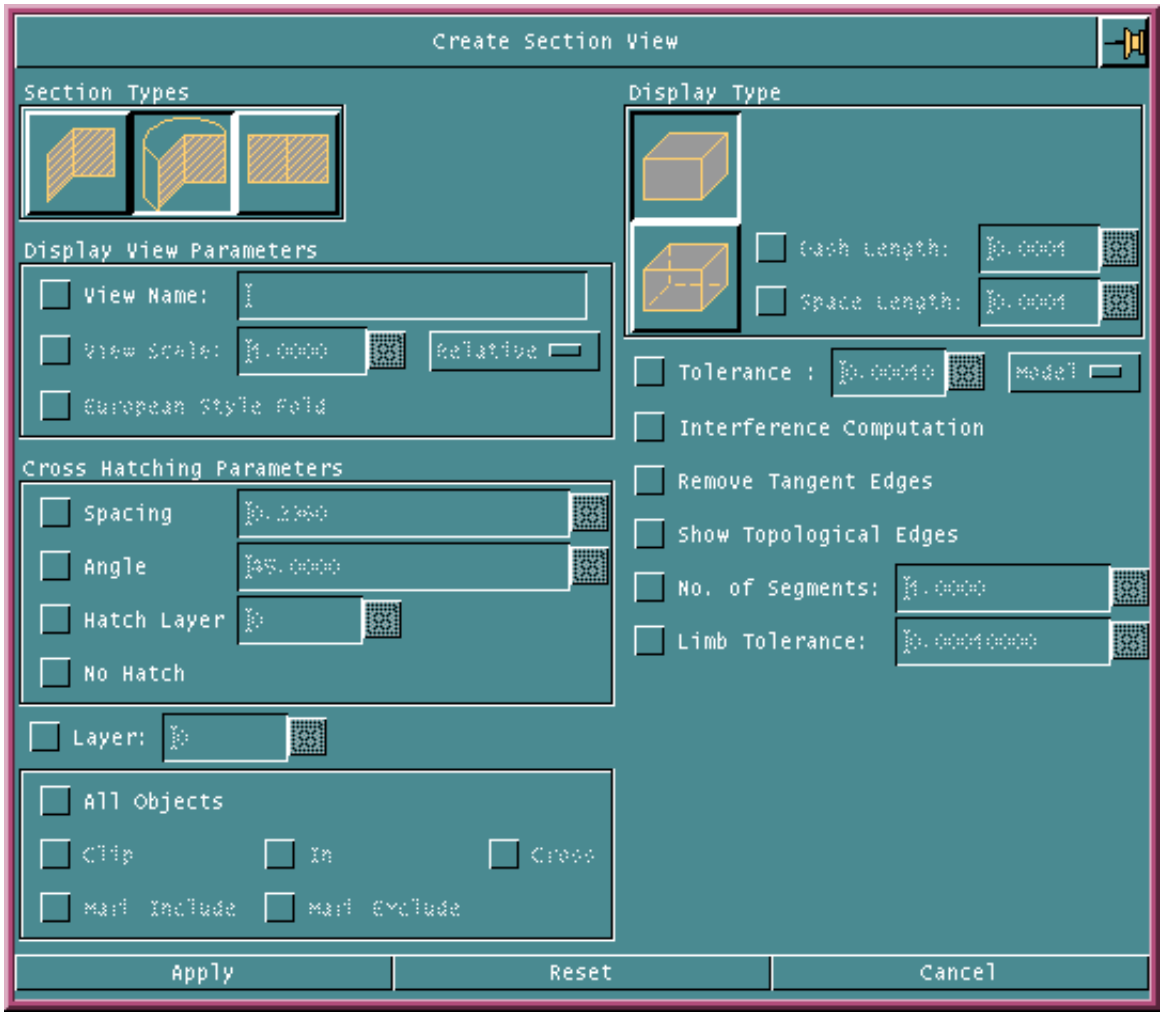
2. Select the Section Type that you want to create.



A Simple Section is created by default.



3. Choose this option to create a Cutaway Section. The Display Type options are available for selection. Use these options to specify the HLR properties for the cutaway section. On choosing this option the Sectioning menu appears as follows:



The options available are as follows:

a. Choose this option to blank the hidden lines. This is the default.

b. Choose this option to display the Hidden Lines as dashed lines.

Specify the length of the dashes in the Dash Length field.

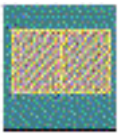
Specify the spacings between the dashes in the Space Length field.

Please note: The Blank and Dash options are mutually exclusive.

- Specify the tolerance in the Tolerance field.
- This tolerance is used in the HLR computations such as curve approximations and curve-curve intersections.
- Choose the Interference option to obtain better HLR results when the objects are interfering with each other.
- Choose the Remove Tangent Edges option to suppress all the tangent cutaway edges.
- Choose the Show Topological Edges to process all the topological cutaway edges for HLR.

Please note: The Tangency and Showtopoedges options are mutually exclusive.

- Choose the No. of Segments option to obtain better results for HLR even with reduced performance. Specify the number of segments in the field provided.
- Choose the Limb Tolerance option to specify the tolerance for generating limbs.



4. Choose this option to create an unfolded section.
5. Specify the display view parameters.

   Use the View Name option to create a new display view. If you want the definition view to be selected as the display view, do not use the View Name option and go to step 5. By default, the definition view is selected as the display view.

   If you have selected the View Name option, follow these steps:

   a. Choose View Name and enter a name for the section label and display view being created in the View Name field.

      If the View Name option is used, a new Cplane and display view will be created. The results of the new sections defined will be displayed in this view. The sectioner and the sectionee(s) are blanked in the display view.

   b. Choose View Scale and enter the display view scale value in the View Scale field.

      The view scale can be Relative or Absolute.

      The Relative option specifies that the display view scale entered is relative to the definition view.

      The Absolute option specifies that the display view scale value entered is the absolute scale value for the display view.

The Relative and Absolute options are mutually exclusive. If neither is selected, then the default used is RELSCALE = 1.0.

c. Choose European Style Fold if you want to reverse the view orientation.

6. Specify the Cross Hatching Parameters.

a. Choose Spacing and enter the spacing between the crosshatch lines in the Spacing field. The default is 0.2360mm.

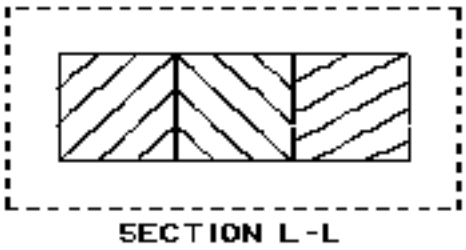
The sectioning operation results in the generation of crosshatch lines, with at least one line displayed inside the section boundary.

b. Choose Angle and enter the angle between the crosshatch lines and the x-axis in the Angle field.

The default is 45 degrees for the first sectionee in the sectioning list. The angles for the subsequent sectionee's are assigned different values. If a section boundary from any other sectionee is found adjacent to the existing section boundary. Its negative value, in this case angle -45 is assigned to the adjacent boundary.

For example Figure 2-4 explains how the system creates the crosshatch parameters for a part or assembly consisting of three objects or components kept adjacent to each other.

Figure 2-4 Example of the Angle Crosshatch Parameter



c. Choose Htch Layer and specify the layer on which the crosshatches are to be created. If the layer is not specified the crosshatches share the respective sectionee's layer.

d. Choose No hatch if you want the section boundaries to be generated without any crosshatching.

7. Choose Layer and enter the layer number in the Layer field. The section entities are created on this layer. The active layer is the default layer.

8. Choose All Objects if you want to select all the viewed solids as sectionees. Modifiers can be used for this option as follows:

- Choose Clip to select all those objects within and crossing the clip boundary. You can further modify the Clip option as follows:

  — Choose In along with Clip to select all those objects that are lying inside the clip boundary in the view.

  — Choose Cross to select objects which are crossing the clip boundary in the view.

  — Choose Clip, In, and Cross to select all objects which are lying inside or crossing the clip boundary in the view (same as Clip).

- Choose Mark Include to select all those objects as sectionees which are marked in the view.
- Choose Mark Exclude to select all those objects as sectionees which are unmarked in the view.

Please note: Mark Include and Mark Exclude are mutually exclusive.

9. Click Apply. The following menu appears:

10. Select the Definition View.
11. If you have used the View Name, select the display view.
12. Select the model entity to be used as the sectioner.
13. Select the model entity or entities to be used as the sectionee(s).
    If you have used All, omit this step, as all the model entity or entities are automatically selected as sectionee(s).
14. Choose Done.

Results of Sectioning

The results of sectioning are displayed only in the Display View. If the View Name option is chosen, the display view is generated and a new Cplane is created. The display view contains crosshatched section boundaries and limb line data (in the case of CUTAWAY sections). This view is coplanar with the first intersection face/surface generated by the sectioning operation.

The section label which is created exists in the definition view and is associated with the sectioner. If the sectioner is a curve/string, then the section label is placed at the end vertices of the sectioner. If the sectioner is a solid, then the section label

is placed in the definition view either horizontally or vertically across the sectioner. This is based upon the angle of the vector described by the Definition and display view's origin.

If the sectioner is a linear curve or string geometry, then a solid is created as a sectioner. To generate this runtime solid, the linear curve or string is projected on the definition view's xy-plane. A profile is constructed using the projected curve or string and the model extents of the sectionee(s). This profile is swept based on the model extents of the sectionee(s) along the z-axis of the definition view.

Restrictions

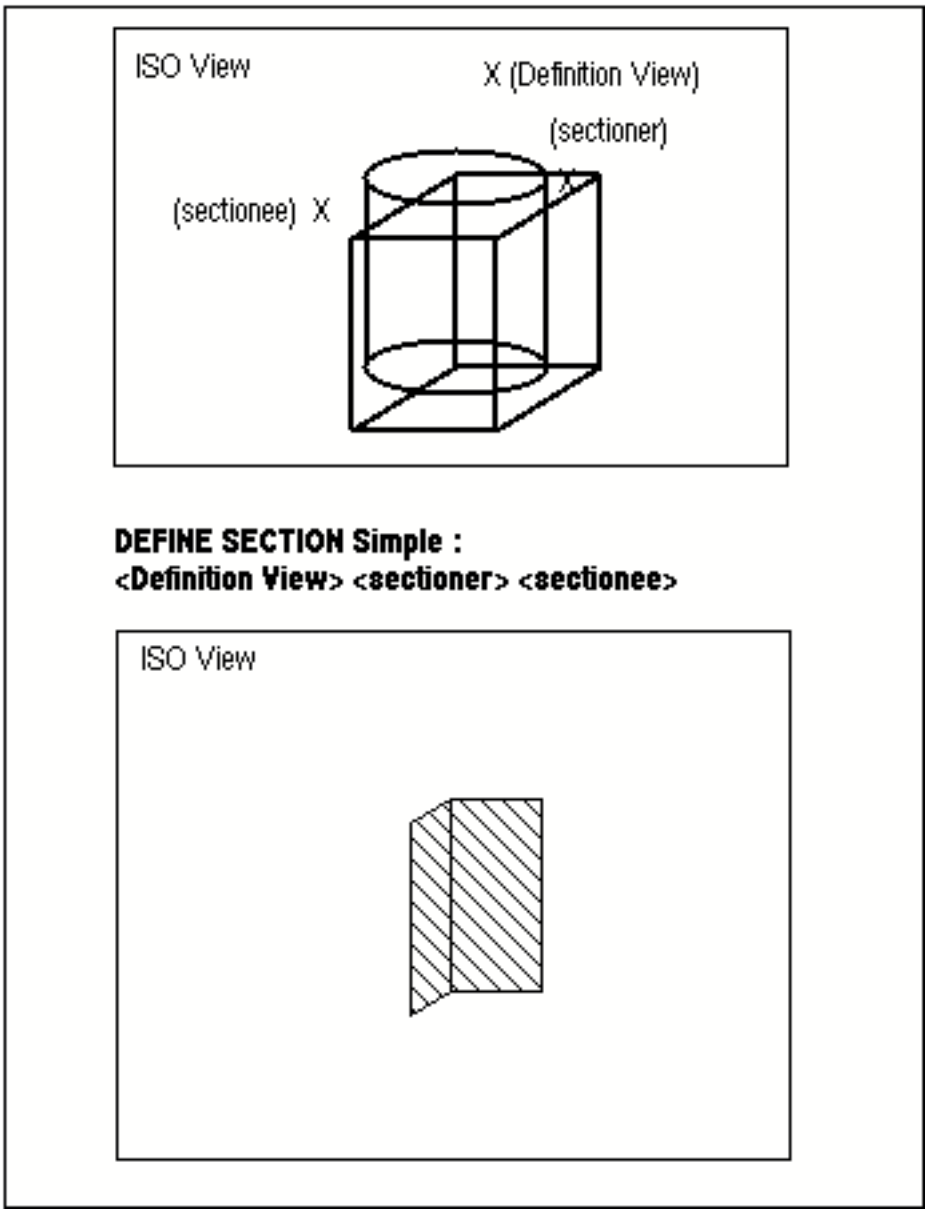
- There can be only one section defined per display view.
- If you choose the section type as Unfolded, then the linear sectioner selected should not have more than two segments.
- The sectionee(s) should not be selected from the display view.
- Define Section cannot be used in MDRAW.
- If there are more than two adjacent section boundaries, only one of them will have opposite angles while the others will have an angle change.

Examples of Defining Sections

DEFINE SECTION Simple in Isometric View

An example of a Define Section Simple operation in Isometric View is shown below.

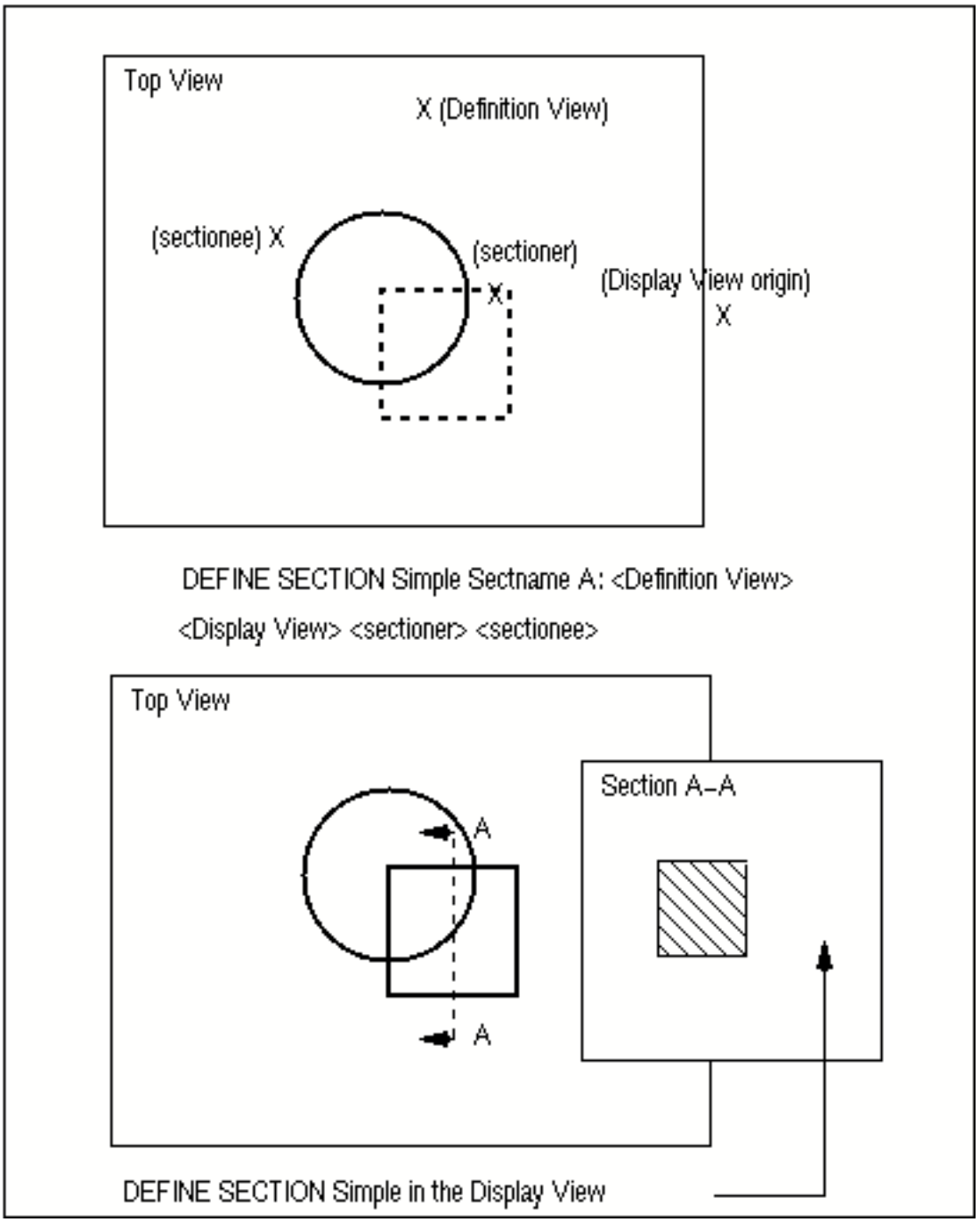
Figure 2-5 Example: Define Section Simple in ISO View:



DEFINE SECTION Simple in Top View

An example of a Define Section Simple operation in Top View with the display view specified is shown below.

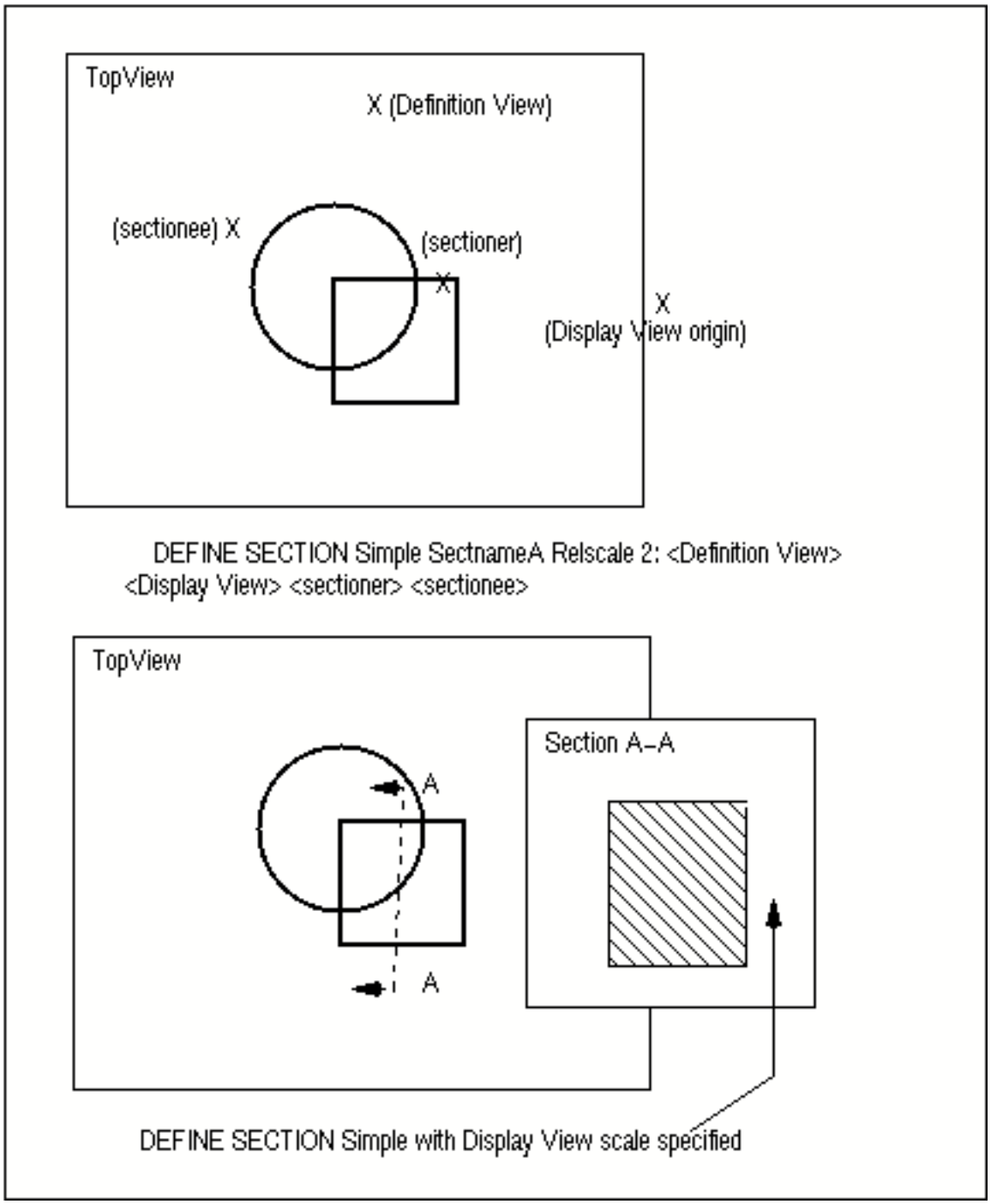
Figure 2-6 Example: Define Section Simple in Top View



DEFINE SECTION Simple - Another Top View

An example of a Define Section Simple operation in Top View with the display view size specified relative to the definition view is shown below.

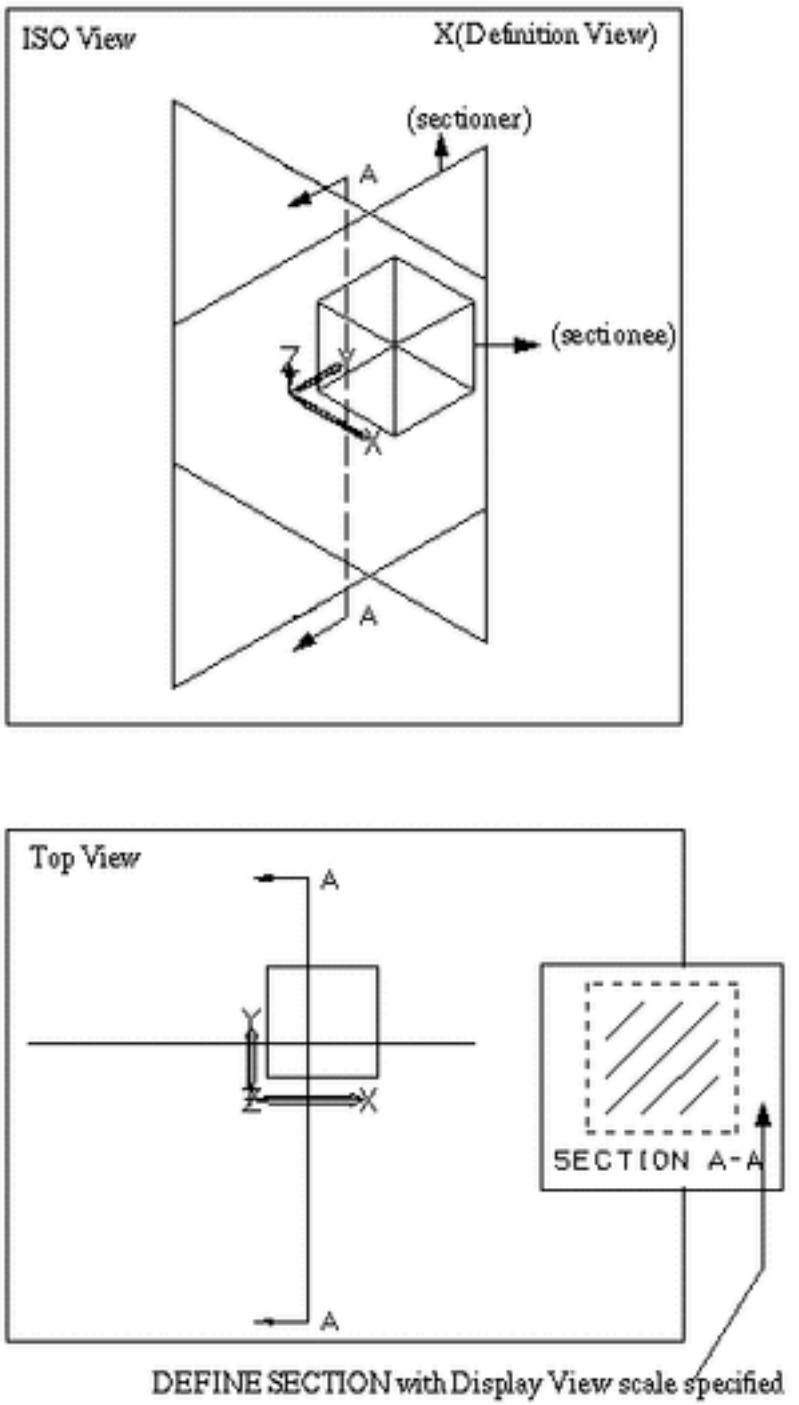
Figure 2-7 Define Section Simple in Top View



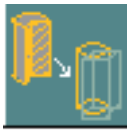
DEFINE SECTION Using Planes

An example of a Define Section operation, using a plane as a sectioner, in the Isometric View is shown in the following figure.

Figure 2-8 Define Section Using Plane in Isometric View



Undefine Section



You can remove the sections generated by the Define Section command by choosing the Undefine Section icon from the Sectioning menu. display views, crosshatch curves, section boundaries, limblines (in the case of CUTAWAY sections) and section labels created during the Define Section command operation are removed.

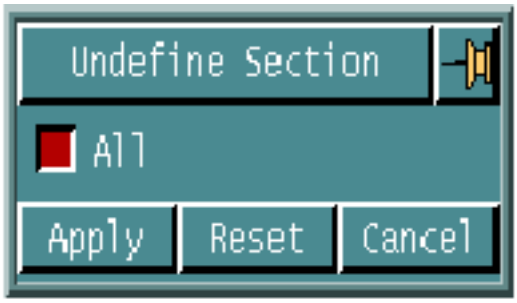
If the display view is the same as the definition view, the operands that were blanked during the sectioning operation are unblanked.

If the display view is not the same as the definition view, it is deleted.

To undefine a section, follow these steps:

1. Choose the Undefine Section icon from the Sectioning menu. The following property sheet appears:

Figure 2-9 Undefine Section Property Sheet



2. Choose All to undefine all the section views in the active drawing. You can interactively select multiple views to be undefined by leaving this option unselected.
3. Click Apply.

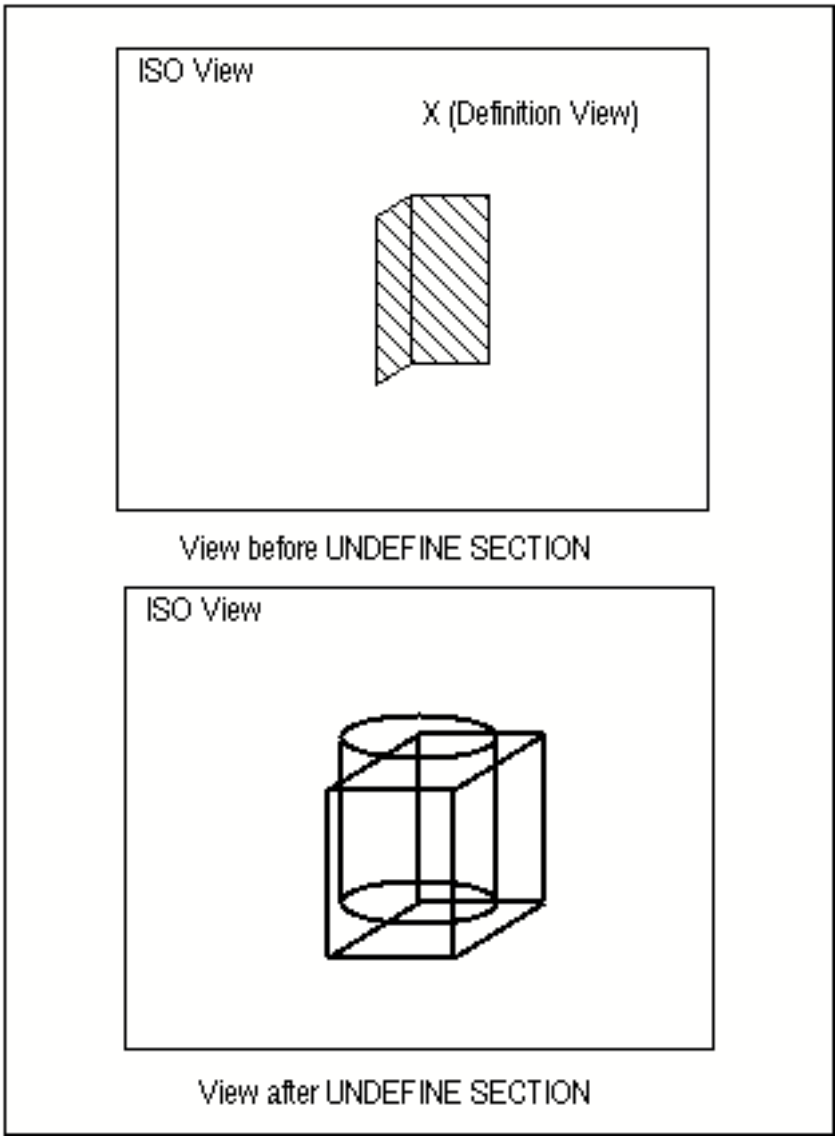
Restriction

The Section View must be deleted only with the Undefine Section command. Define Section creates many entities and builds many associations to tie the Section View together. Currently, only the sectioning commands understand these new entities and associations fully. If any other command such as Delete View or Delete Entity is used, it may corrupt the section entity relationships or the Section View. Although the section entities may still remain in the database and are usable, the sectioning commands no longer work on these sections, and you are left with a database that is difficult to clean up manually.

Example of Undefining a Section

An example of an Undefine Section Simple operation in Isometric View is shown below.

Figure 2-10 Example: Undefine Section Simple in ISO View



Regenerate Section

When changes are made to the sectioner, the sectionee(s), or both, the sectioning results that were generated by the Define Section command are not updated automatically. To see the changes, use Regenerate Section. This command preserves the changes that you made to the crosshatch parameters such as angle and spacing.

For the display view, if you have changed view parameters such as viewclip parameters, view location, or view scale, the sectioning results may not reflect the changes made to the view parameters. In this case also, to see the changes, use Regenerate Section.

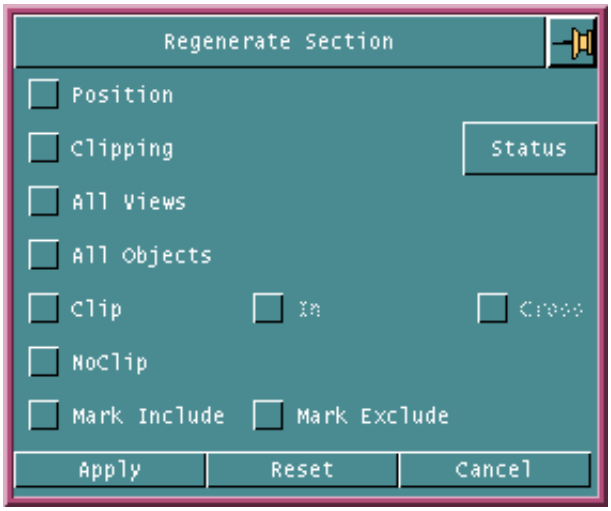
Regenerate Section recomputes section boundaries (and Hidden Line Removal results in the case of CUTAWAY sections) to reflect all changes to the sectioner or the sectionee(s) that have occurred since the last execution of Define Section or Regenerate Section. If you use either the Clipping or the Position option, the display view boundaries are also recomputed. The database and the display are updated accordingly.

To regenerate the sections for all the currently viewed solids, use the All Objects option for Regenerate Section. This results in all the viewed solids to be considered as sectionees during regeneration, irrespective of whether they were sectionees when the Define Section command was used.

To regenerate a section, follow these steps:

1. Choose the Regenerate Section icon from the Sectioning menu. The Regenerate Section property sheet appears.

Figure 2-11 Regenerate Section Property Sheet



2. Choose Position to align the display view with the alignment direction defined by the Define Section command (shown by arrows on the screen).
3. Choose Clipping to clip the display view so that it closely fits the sectioning results.
4. Choose All Views to regenerate all the sections in the current drawing. If changes are made to the sectioner, the sectionee(s), or both, regenerate the section using the Regenerate Section command to see the changes. If the sectioner is missing or corrupted, the following error message appears at the command line:

```
SECTIONER WAS MISSING OR CORRUPTED IN VIEW A
Please type 0 to postpone regeneration (default)
type 1 to reselect sectioner (Viewed sectioner not
allowed)
```

   Type the following to decide regeneration:

   - Type 0 to abort regeneration.
   - Type 1 to reselect a new sectioner and continue regeneration.

5. Select All Objects to select all the currently viewed entities as sectionees. If you do not select All Objects CADDS will by default apply regeneration to the entities stored in the database when previously selected for either sectioning or regeneration. The modifiers Clip, Mark Include, and Mark Include are available for the All Objects option.

- Select All Objects with Clip to choose objects which are within or on the clip boundary.
- Select All Objects with Clip and In to choose as sectionees all objects that are currently viewed solids, if they are lying inside the clip boundary.
- Select All Objects with Clip, In, and Cross to select all objects which are lying inside or crossing the clip boundary in the view (same as Clip).

6. Select Clip to select all objects within and crossing the clip boundary from among the entities which were previously chosen as sectionees. You can further modify the Clip option as follows:
   - Select In to select all objects lying inside the clip boundary from among the entities which were previously chosen as sectionees i.e. the intersection of all entities lying inside the Clip boundary and the entities which were earlier chosen as sectionees.
   - Select Cross to select objects that cross the clip boundary in the view from among the entities which were previously chosen as sectionees.
   - Select Clip, In, and Cross to select all objects which are lying inside or crossing the clip boundary in the view from among the entities which were previously chosen as sectionees (same as Clip).

Please note: In case the viewed solid was a sectionee during DEFINE SECTION then the crosshatching parameters associated with the sectionee are respected by Regenerate Section. If the viewed solid was not a sectionee when using DEFINE SECTION then the default crosshatch parameters are associated with the sectionee.

7. Select the NoClip modifier to override the Clip modifier of the previously used DEFINE SECTION or REGENERATE SECTION command. If the user executes REGENERATE SECTION NOCLIP All Objects, then all entities would be processed.

Please note: Clip and NoClip are mutually exclusive. The objects selected by the Clip modifier are present in the database until you replace them with new objects.

8. Choose Mark Include to select all those objects as sectionees which are marked in the view.
9. Choose Mark Exclude to select all those objects as sectionees which are unmarked in the view.

Please note: Mark Include and Mark Exclude are mutually exclusive.

10. Click Status to view the following information in the CADDS text window:

- View name
- Section type (simple, unfolded, or cutaway)
- Hatch flag (hatched or not)
- Hatched layer value
- Hatch angle
- Section layer
- Number of sectionees
- Clip modifier used

Please note: The Status button allows you to view the various modifiers used in the current section definition. The history of the previous section is not stored.

11. Click Apply.

12. Select the display view to be regenerated.

Please note: If either the Clipping or Position option is used, the Display View boundaries are also recomputed. The database and display are updated accordingly.

If some of the objects were within the view during regeneration and are outside the view during the update, then these entities will not be sectioned and will also be removed from the database during update.

Restriction

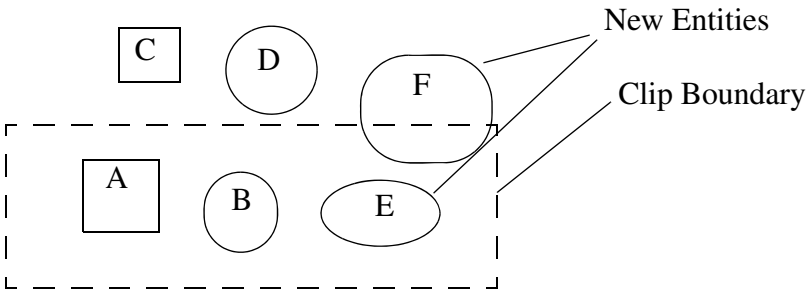
The regeneration cannot attach remote section tool. The section tool must be local to enable regeneration.

If any of the section's operands have been deleted or modified in such a way that it cannot be accessed, the section with all its dependent data is removed.

Example for Modifier Behavior

Consider a drawing view consisting of initially four entities: A, B, C and D and one sectioner. Entities A and B are within the clip boundary. Run the DEFINE SECTION or REGENERATE SECTION command on these entities. Now create the entities F and E as shown in the following figure.

Figure 2-12 Example for Regenerate Section Modifier Behavior



Entities F and E are not in the database for sectioning as they were created after running the sectioning or regenerate command. Now, running the command REGENERATE SECTION with various modifiers will show behaviors respectively as follows:

All Objects: The entities A, B, C, D, E and F are selected as sectionees.

All Objects with Clip: The entities A, B, E, and F are selected as sectionees.

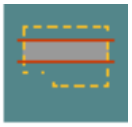
All Objects with Clip and In: The entities within the clip boundary A, B, and E are selected as sectionees.

All Objects with Clip, In, and Cross: The entities within the clip boundary A, B, E, and F are selected as sectionees.

Clip and In: The entities A and B are selected as sectionees.

Clip and Cross: The entities A, and B are selected as sectionees.

Define View Clipping

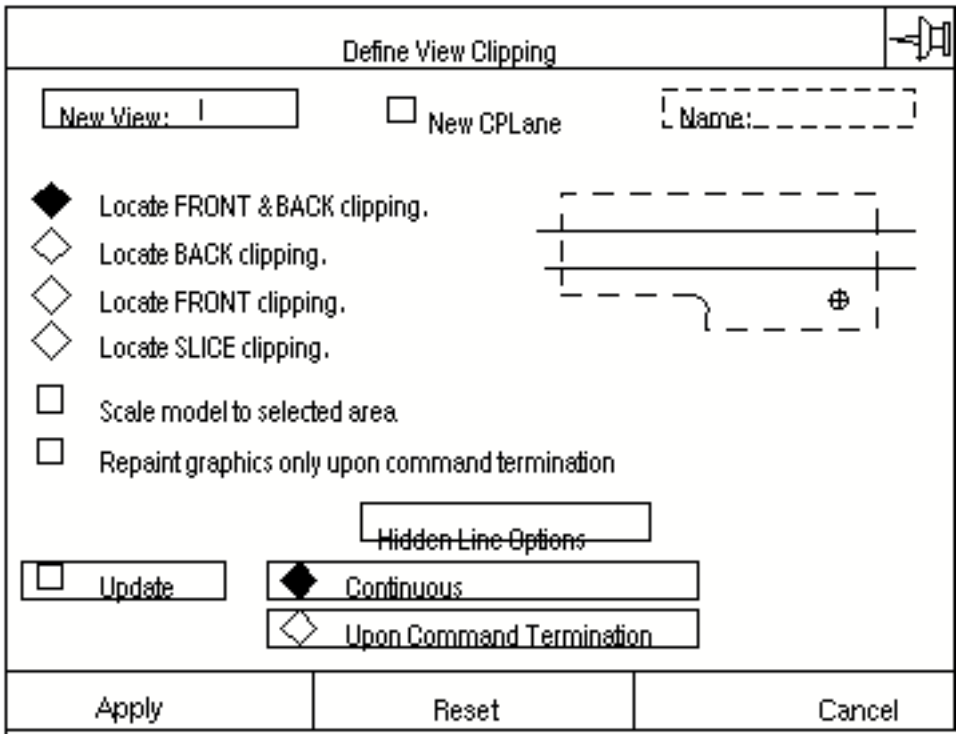


Once the model database is passed to the drafter, it may be necessary to create additional views in order to do the job. Creating a view by using the z-depth clipping allows you more options than traditional methods. You can also create standard CADDS views using the Define View Clipping command.

To create a view using the z-depth clipping, follow these steps:

1. Choose the Define View Clipping icon from the Sectioning menu. The Define View Clipping property sheet, as shown below, appears.

Figure 2-13 Define View Clipping Property Sheet



2. Enter a name for the new view being created in the New View field. The name of the view can be of up to 20 characters in length.
3. Specify the Cplane for the view orientation. You can create a new Cplane or use an existing Cplane.

   To create a new Cplane, choose New CPLane and enter a name for the new Cplane in the Name field.

   To use an existing Cplane, choose the Cplane from the Name pulldown menu. Do not choose New CPLane.

4. Specify the type of clipping by choosing one of the following:
   - Locate FRONT & BACK clipping

     Specifies that both the z-front plane clipping and z-back plane clipping are specified in getdata. Locate FRONT & BACK clipping is selected by default.

     If Locate FRONT & BACK clipping is not chosen, it specifies that neither z-back plane nor z-front plane clipping is specified in getdata.
   - Locate BACK clipping

     Specifies that only the z-back plane clipping will be specified in getdata.

     If Locate BACK clipping is not selected, it specifies that no z-back plane clipping is specified in getdata.
   - Locate FRONT clipping

     Specifies that only the z-front plane clipping is specified in getdata.

     If Locate FRONT clipping is not selected, it specifies that no z-front plane clipping is specified in getdata.
   - Locate SLICE clipping

     This option specifies that both the z-front plane and z-back plane clippings are defined with the same getdata specifications.
5. Choose Scale Model to selected area to scale the view to fit the model to view.
6. Click Repaint graphics only upon command termination to update the graphics for each set of selection.

   If this option is not chosen, the view is updated only when you enter a semicolon (;).
7. Click Apply.
8. If you have chosen the New CPLane option, specify three model space locations to define the Cplane. The first location specifies the origin of the Cplane, the second location specifies the x-vector direction, and the third location specifies the y-vector direction.
9. Specify the view location and clipping boundaries by following these steps:
   a. Specify the new view Cplane's origin by selecting the view on the screen.
   b. Specify two diagonal locations to indicate the view clipping boundaries. At this point, the view appears on the screen with clipping boundaries matching the drawing extents.
   c. Specify two diagonal locations to clip the view or enter a semicolon (;) to make the view remain clipped to the drawing extents.

10. If you have chosen Scale Model to selected area, specify two diagonal locations of a rectangle. Each pair of choices changes the view orientation and view scale so that the rectangular area is as large as possible without going outside the view clipping boundary. The view origin and the view scale are adjusted during the fitting.

    If Scale Model to selected area is not selected, no fitting is done. This forces the view scale to be 1.0.

11. If you have chosen BACK edge of clipping boundary or FRONT edge of clipping boundary, specify the z-front or z-back clipping plane by clicking once on the screen. Further clicks continue to be interpreted as z-front or z-back clipping planes until you enter a semicolon (;). This allows trial and error manipulation of the z front/z back clipping plane.

12. If you have chosen the SLICE edge of clipping boundary option, click once on the screen to specify both the z-front and z-back clipping planes. The front clipping is placed.005 model units in front of the specified location and the back clipping plane is placed.005 model units in back of the specified location. Further clicks continue to be interpreted as z cut locations until you enter a semicolon (;). This allows trial and error manipulation of the cutting action.

13. Choose Done from the Utilities menu.

    A new view is created on the current drawing. The view created is an accepted view, and you can use all existing view manipulation commands on this view.

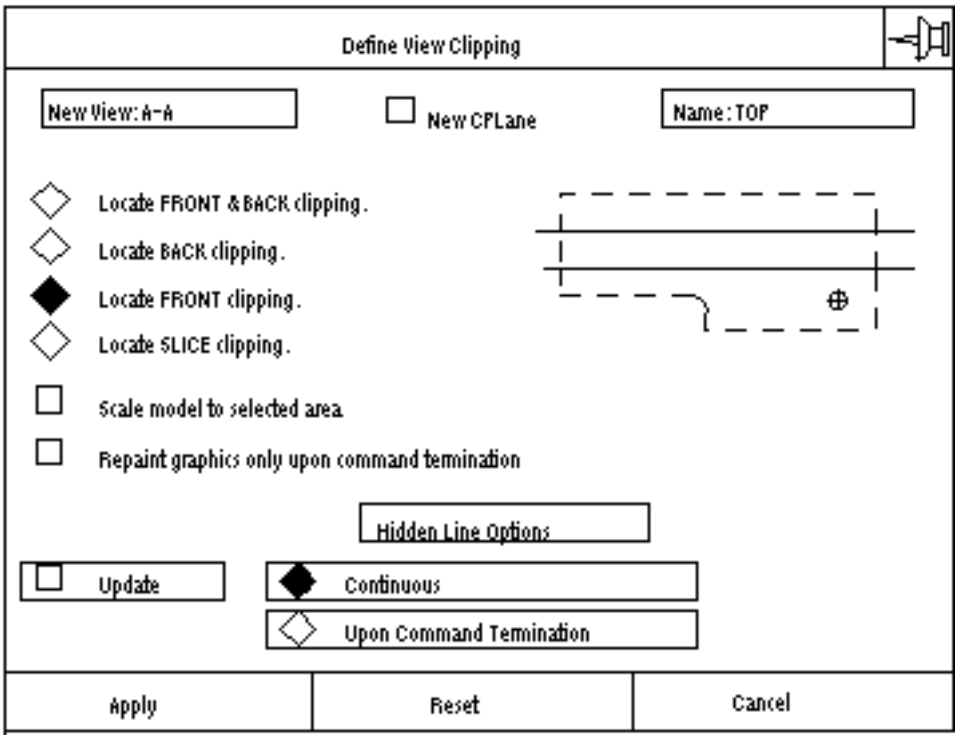
Example of Defining a View Clipping

To define a view that is clipped from an existing standard view, follow these steps:

1. Make the active Cplane TOP. (The active Cplane should be parallel to the x-depth of the cut.)
2. Choose the Define View Clipping icon from the Sectioning menu. The Define View Clipping property sheet appears.
3. Enter `A-A` as the view name in the New View field.
4. Choose TOP as the Cplane from the Name pulldown menu.
5. Choose Locate FRONT Clipping.

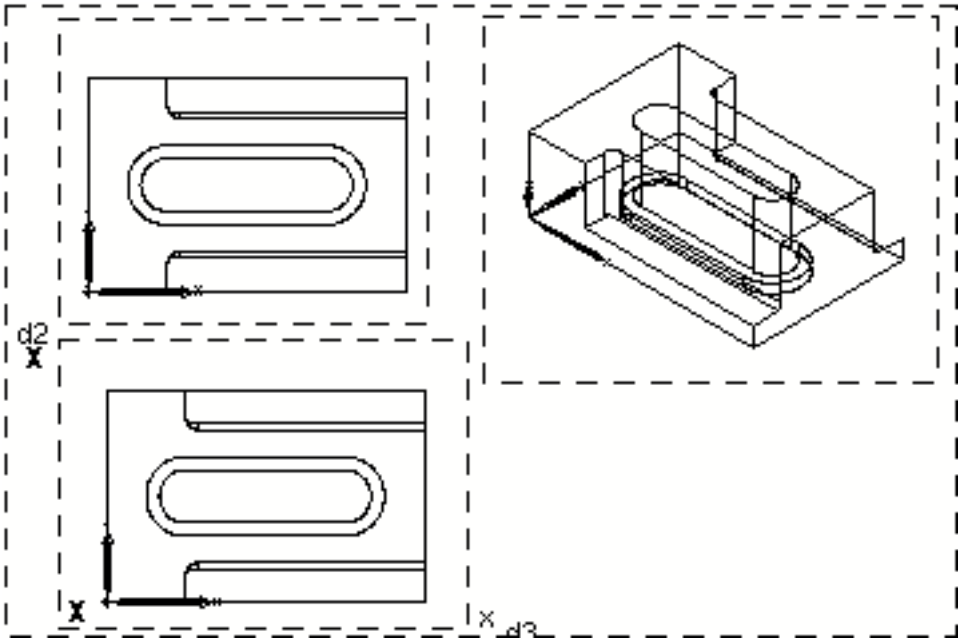
   In this example, the Scale model to selected area and Repaint graphics only upon command termination are not chosen.

Figure 2-14 Example:Define View Clipping

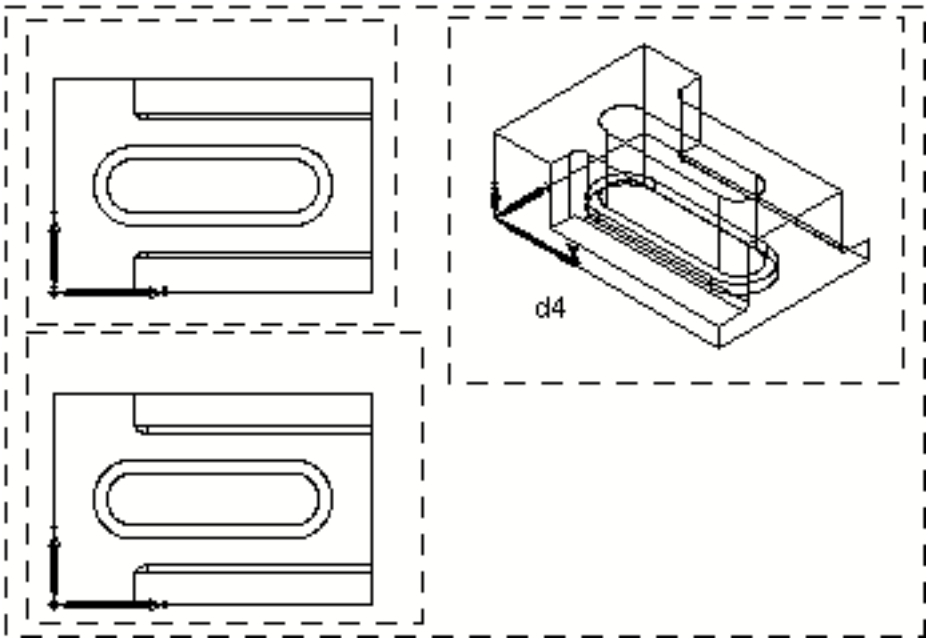


6. Click Apply.
7. Specify the view location and clipping boundaries by following these steps:
   a. Specify the new view Cplane's origin by clicking on the screen.
   b. Specify two diagonal locations to indicate the view clipping boundaries. At this point, the view appears on the screen with clipping boundaries matching the drawing extents.

c. Specify two diagonal locations to clip the view or enter a semicolon (;) to make the view remain clipped to the drawing extents.



8. Identify the front z-clipping plane by clicking on the screen.



9. Choose Done from the Utilities menu.

Revise View Clipping

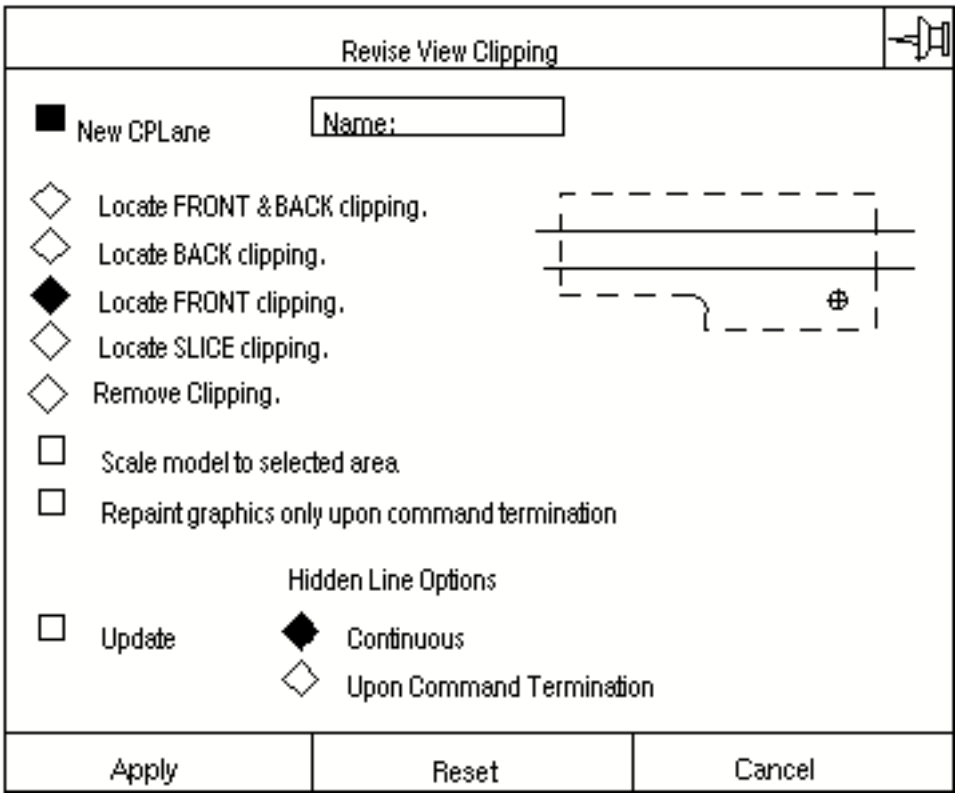
You can modify an existing view by choosing the Revise View Clipping icon from the Sectioning menu. Revise View Clipping can also be used on standard CADDS views. This command operates on permanent views only.

Please note: If changes are made to the view (ZOOM, CHANGE VIEW, and so forth.), these changes must be made permanent by using the Set View command before using the Revise Cutview command, or results will be unpredictable.

To change an existing view, follow these steps:

1. Choose the Revise View Clipping icon from the Sectioning menu. The Revise View Clipping property sheet, as shown below, appears.

Figure 2-15 Revise View Clipping Property Sheet



2. Fill in the property sheet.

   All the options that were available when defining a z-depth clipped view are available in the Revise View Clipping property sheet. However, this property sheet provides an additional option, Remove Clipping.

   For more details regarding the other options on the property sheet, see section "Define View Clipping" on page 2-17.

3. Click Apply.

4. Select the view to be revised.

5. Specify the drawing space locations as you did when using the Define View Clipping icon.

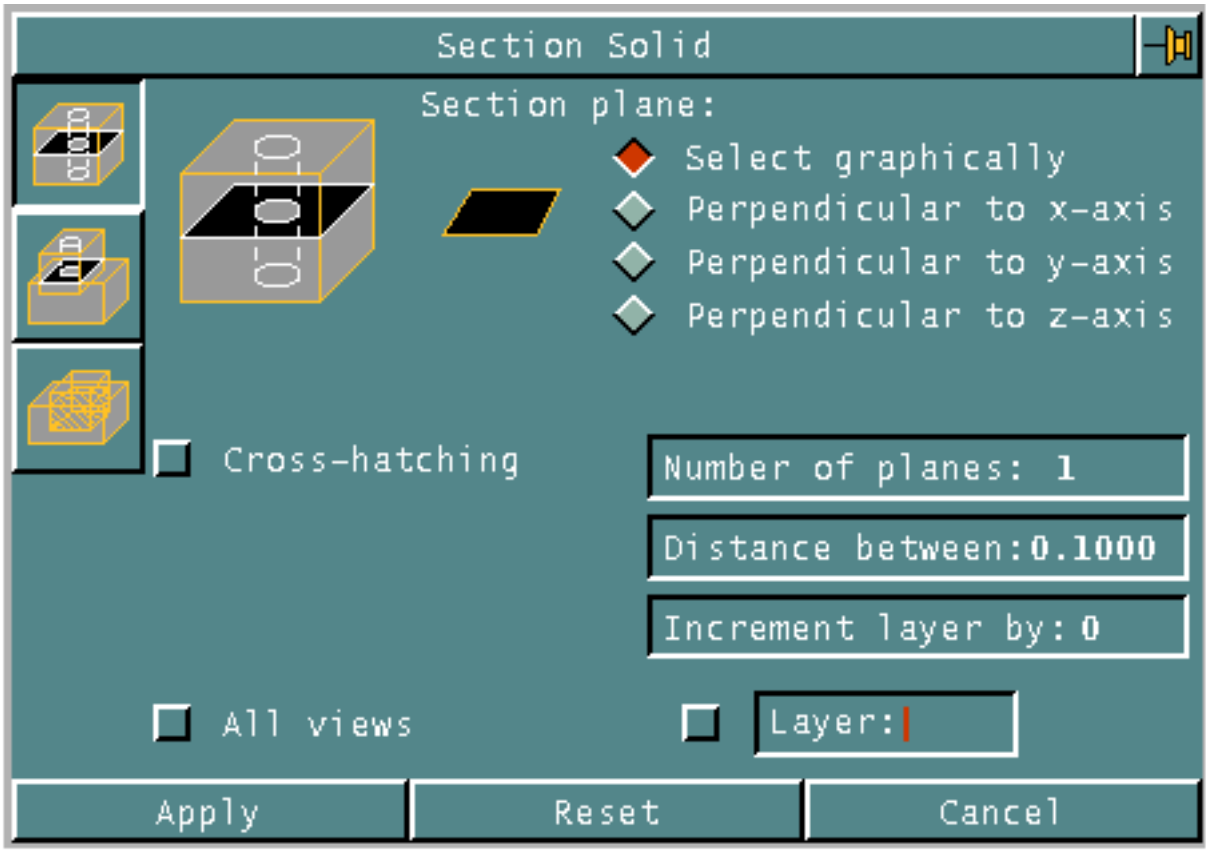
Generate Cross Sections Using Solids

You can create a cross section of a solid by using a plane or another solid to define the cross section.

To create a cross section of a solid, choose the Section Solid icon from the Sectioning menu.

The Section Solid property sheet, as shown in the figure below, appears.

Figure 2-16 Section Solid Property Sheet



Generate Cross Sections Using Planes

Choose the Plane icon to create a cross section of a solid using a plane and choose one of the following:

- Select graphically allows you to select three locations to indicate the plane. This option is selected by default.
- Perpendicular to x-axis, Perpendicular to y-axis, and Perpendicular to z-axis allow you to identify the cutting plane as perpendicular to the x-, y-, or z-axis. When you choose one of these options, you can also enter the distance from the axis for the plane by entering a value in the At distance field.

Generating Cross Sections Using a Planar Face

Choose the Generate Cross Sections using a planar face icon to create a cross section of a solid by identifying a planar face with two locations.

Generating Cross Sections Using Another Solid

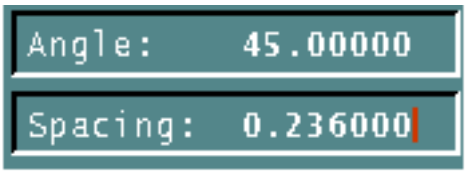
Choose the Generate Cross Sections using a solid icon to create a cross section of a solid by using another solid to define the cross section.

Other Options on the Property Sheet

The following options on the property sheet are common to the Plane, Planar Face, and Solid options on the property sheet.

Placing Crosshatching on the cross section

Choose Cross-hatching to place crosshatching on the cross section. The following fields appear on the property sheet.



1. Enter the angle for the crosshatching in the Angle field. The angle is measured from the x-axis. The default is 45 degrees.
2. Enter the distance between the crosshatching lines in the Spacing field. The default is 0.6 cm or 0.236 inch.

Using Cutting Planes

Enter the number of cutting planes in the Number of Planes field.

Enter the distance between the cutting planes in the Distance between field.

Enter a number in the Increment layer by field. This places each cross section created on a different layer and the system increments the default or the specified layer by *n* (the number entered).

Please note: These options are grayed out when generating cross sections using another solid.

Cross Sectioning Solids in All Views

Choose All Views to create the cross section of the solid in all views. If you do not use this option, the system prompts you for a view.

Specifying the Layer

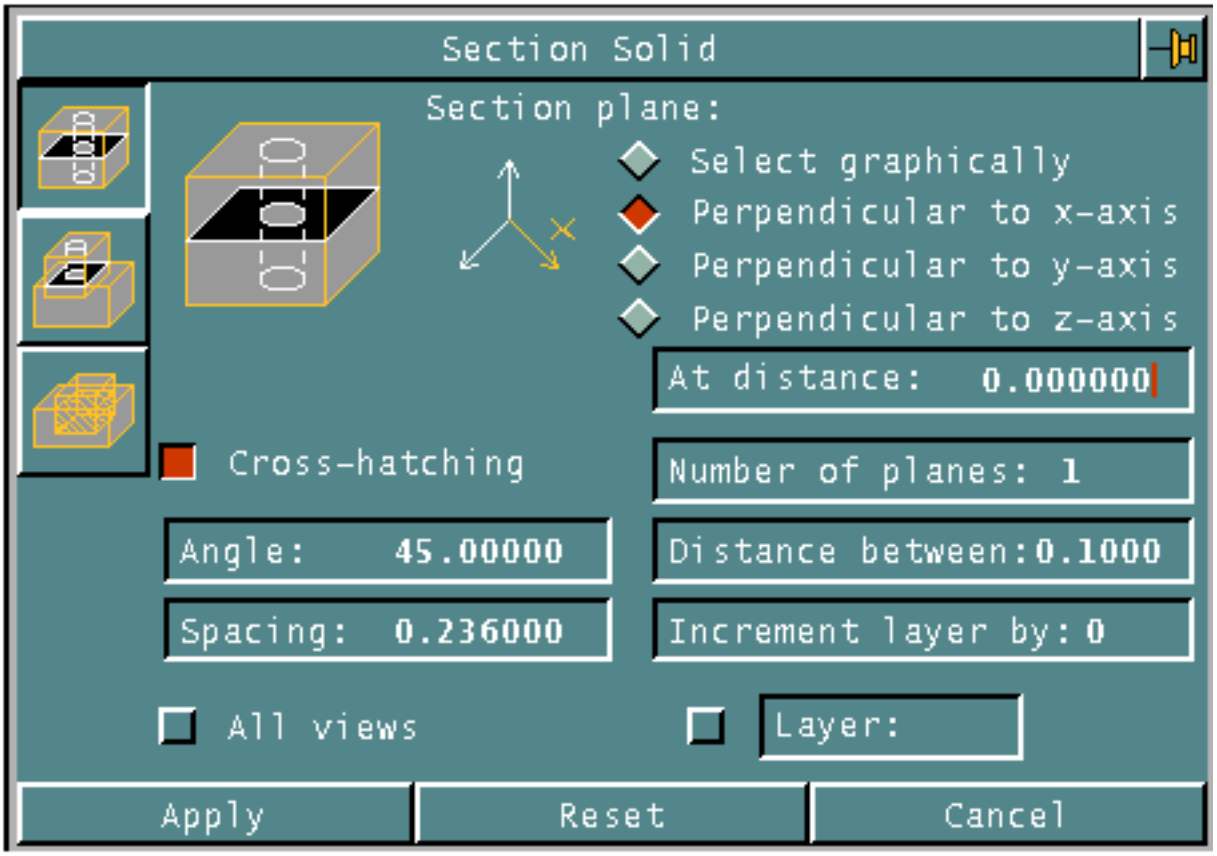
Choose Layer to place the cross section on a different layer. The default is the active layer.

Example of Generating Cross Sections Using Solids

This example uses a cutting plane perpendicular to the x-axis to create the cross section.

1. Choose the Section Solid icon from the Sectioning menu. The Section Solid property sheet appears.
2. Choose the Plane icon.

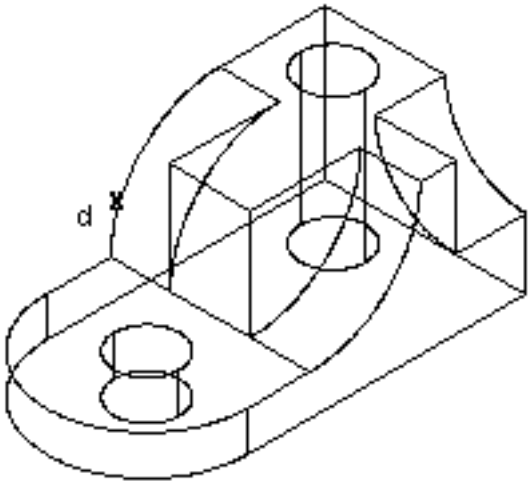
Figure 2-17 Example: Section Solid



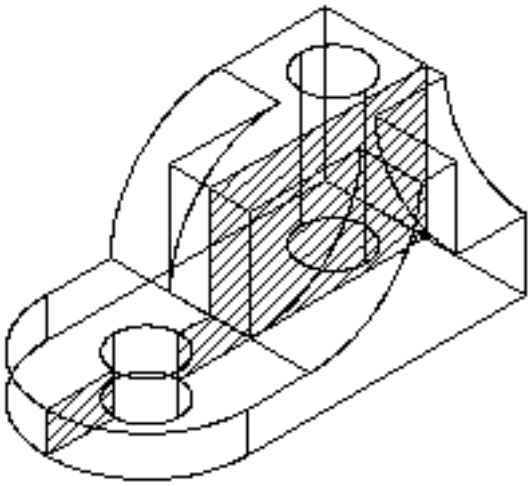
3. Choose Perpendicular to x-axis.
4. Choose Cross-hatching and enter the angle and spacing in the respective fields.

   The Section Solid property sheet must resemble the figure above.
5. Click Apply.

6. Select the solid you want to cross section.



7. Choose Done. The system creates the cross section.



The following system message appears in the text window:

```
*** Solution in progress. ***
** Cross section generated. **
Drawing mode crosshatching created.
```

Generate Cross Sections Using Strings

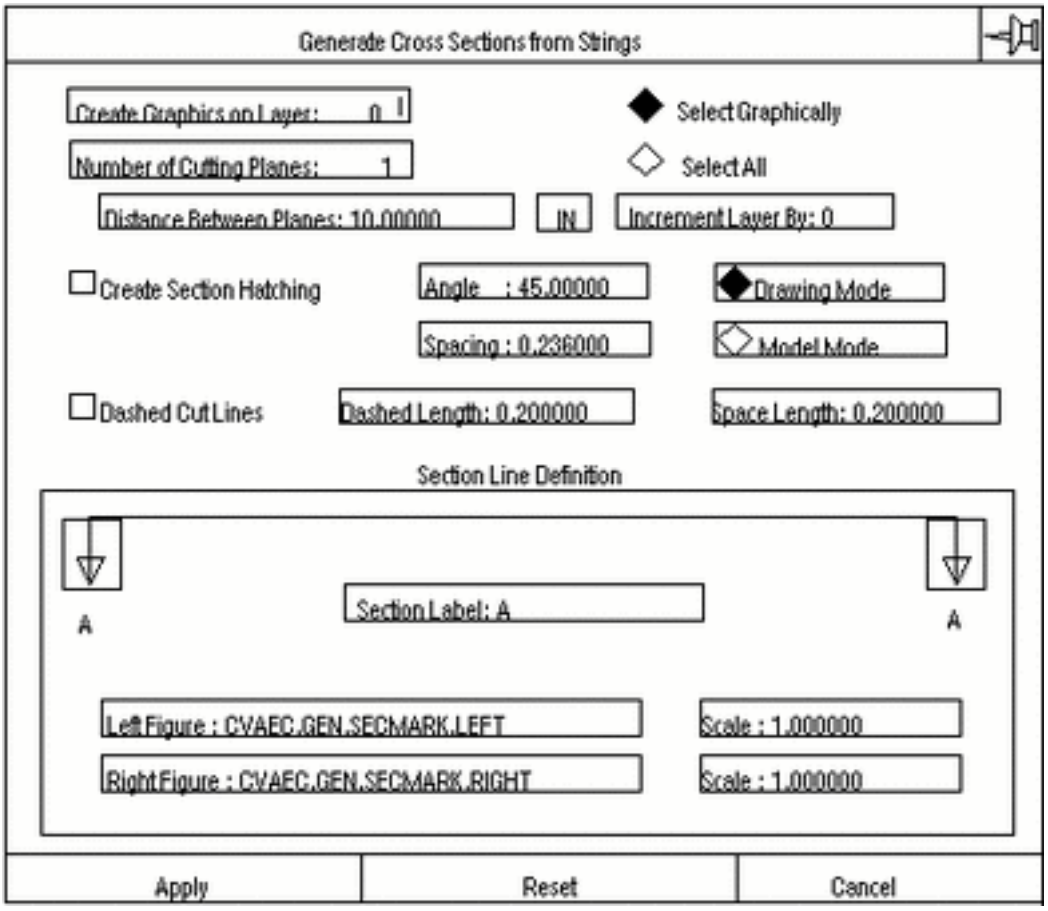
You can create cutaway sections of 3D Model graphics. Using closed and coplanar strings, you can automatically crosshatch and font the areas of the cutaway section.

The Generate Cross Sections from Strings option can be used on CADDS 4X parts (3D Model graphics generated from the PROJECT OUTLINE command) which have been converted to CADDS 5 parts.

To generate a cross section from 3D Model graphics (closed and coplanar strings), follow these steps:

1. Choose the Generate Cross Sections from Strings icon from the Sectioning menu. The Generate Cross Sections from Strings property sheet, as shown below, appears.

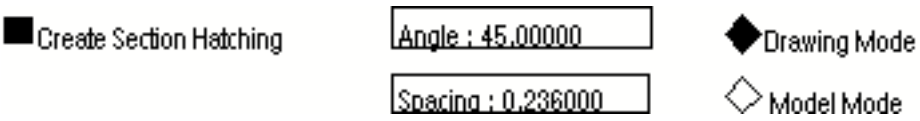
Figure 2-18 Generate Cross Sections from Strings Property Sheet



2. Enter the layer number in the Create Graphics on Layer field. The line entities representing the section are placed on this layer. The active layer is the default layer.
3. Enter the number of parallel planes used for the cutting in the Number of Cutting Planes field. If the number of planes specified is greater than one:
   a. Enter the distance between the cutting planes in the Distance Between Planes field.
   b. Enter a number in the Increment layer by field. This places each cross section created on a different layer and the system increments the default or the specified layer by *n* (the number entered).
4. Choose Select Graphically to individually select the strings you want to use. This option is selected by default.

   OR

   Choose Select All to select all the strings that lie along the defined plane.
5. Choose Create Section Hatching to crosshatch the cross section created by the cut. The following options on the property sheet are not grayed.



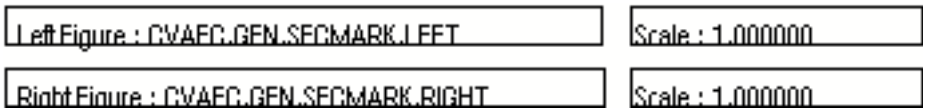
   a. Enter the angle for the crosshatching in the Angle field. The angle is measured from the x-axis. The default is 45 degrees.
   b. Enter the distance between the crosshatching lines in the Spacing field. The default is 0.6 cm or 0.236 inch.
   c. Choose Drawing Mode to create the crosshatching as a Draw mode graphic in the view you select. Drawing Mode is the default.

      (Or)

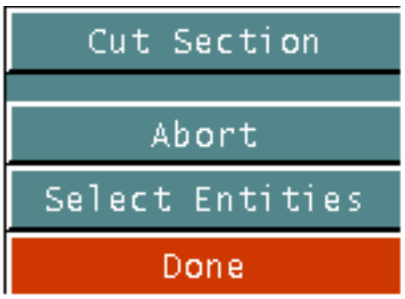
      You can also create the crosshatching as a model graphic visible in all views by clicking the Model Mode option.
6. Choose Dashed Cut lines to specify the dash length and the space length for the default font. The following options on the property sheet are not grayed.

   a. Enter the dash length in the Dashed Length field. The default is 0.200000.
   b. Enter the space length of the default section line font. The default is 0.200000.

7. Choose the Label box to select symbols for marking sections. The following options appear next to the selected box. Currently, this is the default and only symbol available. You can create or use existing Nfigure symbols from a company library.



8. Enter the text for marking sections in the Section Label field.
9. Click Apply. The following appears on the screen and the property sheet closes.



10. Select the view that you want the crosshatching to appear in.
11. Specify the two ends of the section line.
12. Choose Select Entities from the Cut Section menu.
13. Use any entity selection aid on the top bar to select the model entities that you want to section.
14. Choose Done from the menu.

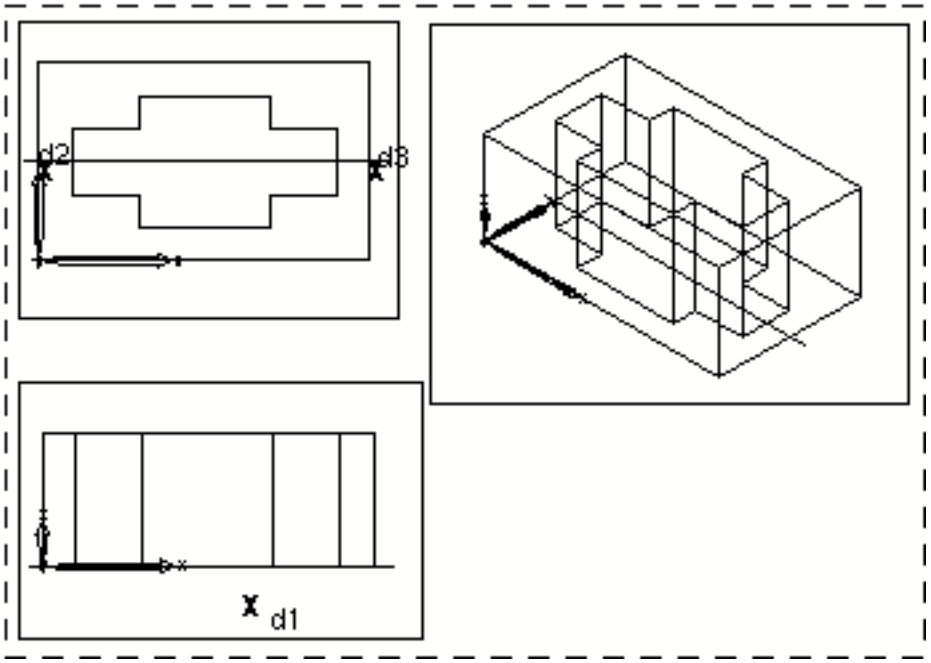
Example of Generating Cross Sections Using Strings

To generate a cross section from 3D model graphics (closed and coplanar strings), follow these steps:

1. In Model mode, choose Cplane TOP. The line selected in this view defines the cutting plane.
2. Choose the Section icon from the Drafting Task Set menu.
3. Choose the Generate Cross Sections from Strings icon from the Sectioning menu. The Generate Cross Sections from Strings property sheet appears.
4. Select layer 120 for the section entities.
5. Choose Create Section Hatching and leave the default settings for angle and spacing.
6. Click Apply. The Cut Section menu appears on the screen and the property sheet closes.

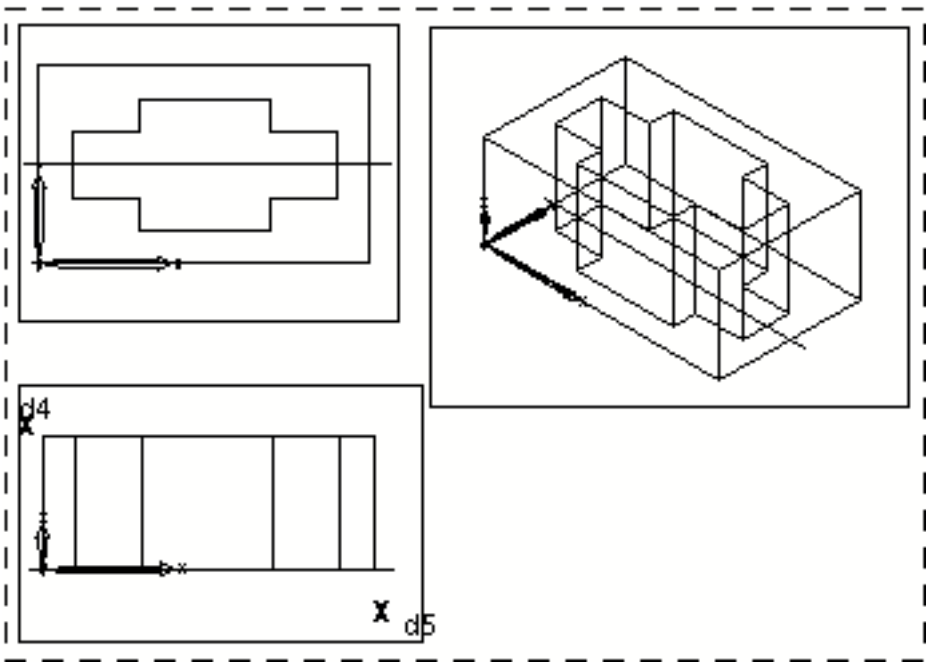
7. Select the view that you want the crosshatching to appear in.
8. Select the two ends of the section line.

In this example, the ends of a previously created line were selected by using the end constraint. The direction is important for the placement of the section label.



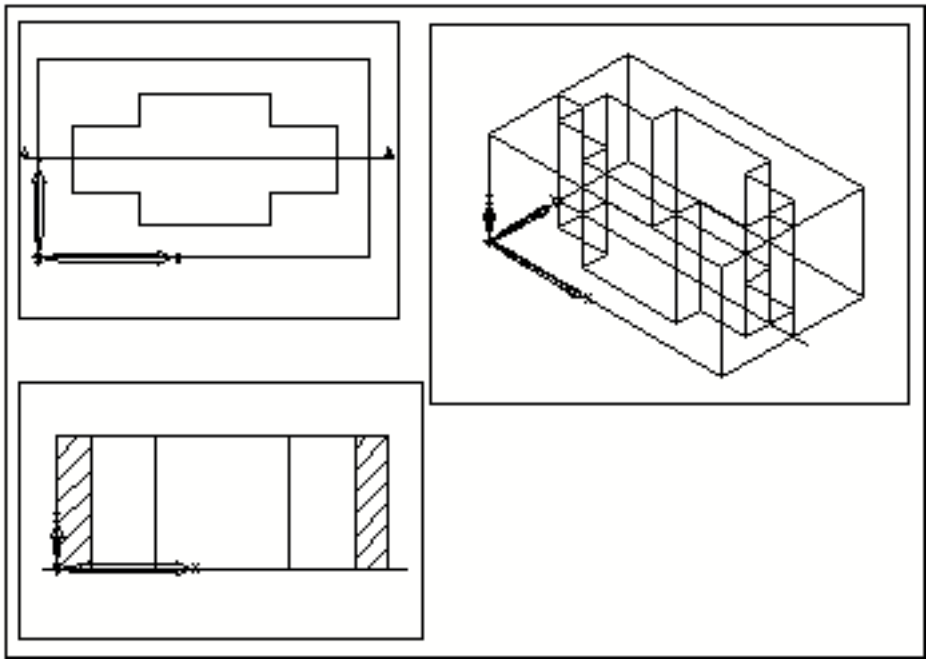
9. Choose Select Entities from the Cut Section menu.

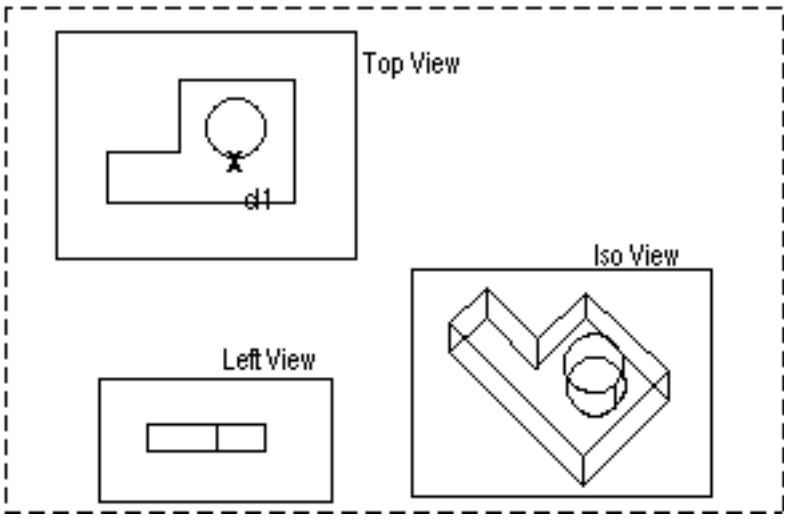
10. To select all the entities in a window, use an entity selection aid on the top bar and then select two diagonal locations to define the model entities within the window.



11. Choose Done from the menu.

Please note: To view just the cross section, echo only layer 120 in the front view.





Generate Cross Sections Using Surfaces

Using the Generate Cross Sections from Surfaces icon on the Sectioning menu, you can:

- Section entities
- Intersect entities

Section Entities

The Generate Cross Sections from Surfaces option allows you to section curves, surfaces (including planes), Tsurfaces, and solids (creating Nsplines, B-splines, strings or Cpoints):

- Parallel planes
- Any selected planar entities
- With planes defined along an spine curve.

This option also allows you to select 10,000 entities within a command.

Intersect Entities

The Generate Cross Sections from Surfaces option allows you to intersect all surface types (including planes), Tsurfaces, and solids with each other (creating Nsplines, B-splines, and strings).

You can also select multiple entities as intersecting entities and multiple entities to be intersected with. In this case, each entity from the first group is intersected with each entity from the second group.

Intersections of any two curves occur if the curves are coplanar.

All surface types intersect a curve, creating points.

Intersecting allows you to select 10,000 entities in each group within a command.

Temporary graphics appear on the visible layer as soon as intersections are found. When the command is executed, the permanent graphics can be found on the active construction layer or the layer(s) specified with modifiers in the command line.

When the system creates an entity, it reports the following information in the text window:

```
Tolerance =   0.001
  MIP      TYPE    DEG     POLY   SEGM   TOL     MAX DEVIATION
 ----      ----   ----   ------ ------ ----    ---------------
  10       NSPL     5       12      7    Y          0.0007
  11       NSPL     5        9      4    Y          0.001
```

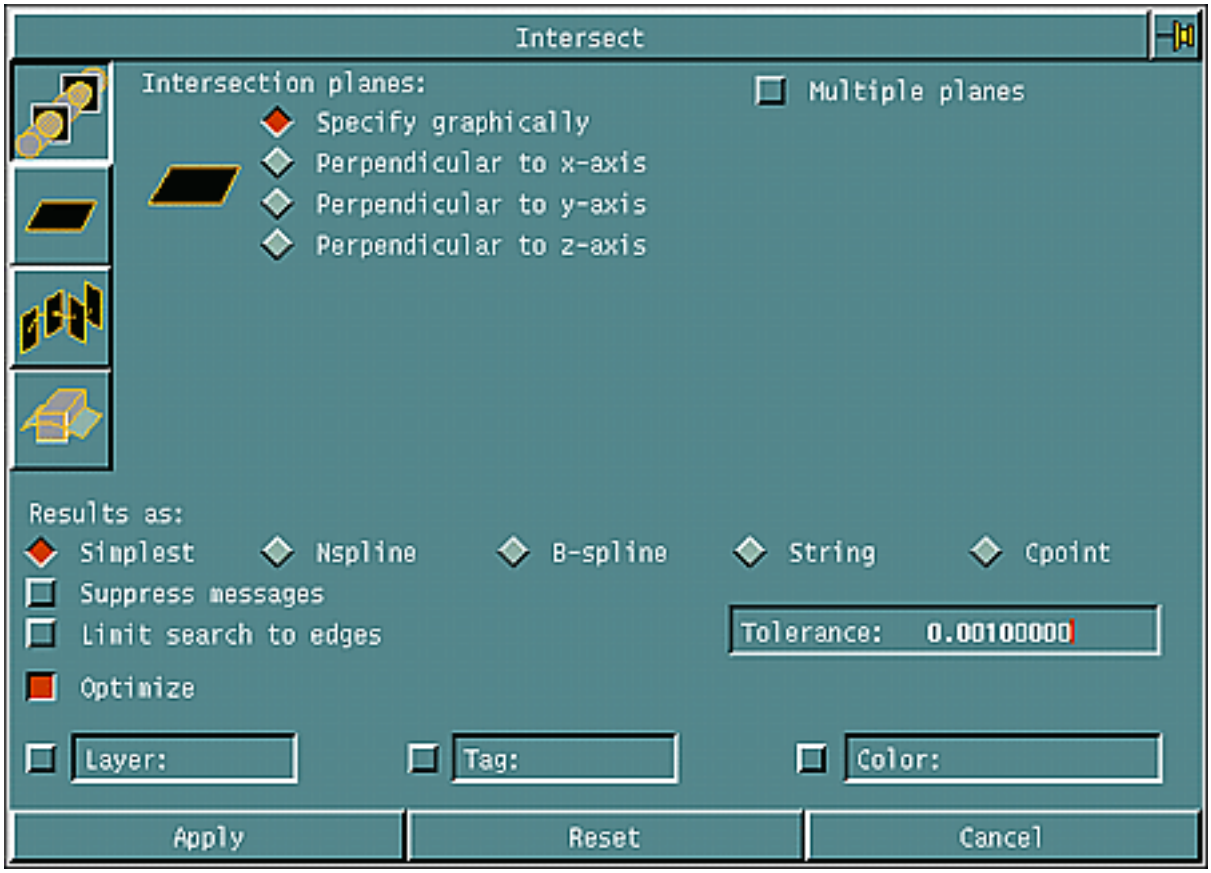
Where:

Tolerance	Is the tolerance used by the system
MIP	Is the miptr of the created entity
TYPE	Is the type of the created entity (NSPL = Nspline, LINE = line,...)
DEG	Is the degree of the created entity
POLY	Is the number of polygon points of the created entity
SEGM	Is the number of segments of the created entity
TOL	Is set to Y if the input tolerance is reached Is set to N if the input tolerance is not reached
MAX DEVIATION	Is the maximum deviation between the original curve and the smoothed curve

The Generate Cross Sections Using Surfaces Property Sheet

To section or intersect entities, select the Generate Cross Sections from Surfaces icon from the Sectioning menu. The Intersect property sheet appears.

Figure 2-19 Intersect Property Sheet



Sectioning Using Parallel Planes

Choose the Plane icon to section entities with parallel planes and choose one of the following:

- Specify graphically allows you to select three locations to indicate the plane. This option is the default.
- Perpendicular to x-axis, Perpendicular to y-axis, and Perpendicular to z-axis allow you to identify the cutting plane as perpendicular to the x-, y-, or z-axis of the active Cplane. When you select one of these options, enter the distance from the axis for the plane in the At distance field.

Sectioning Using Planar Entities

Choose the Planar Entity icon to section entities using existing planar entities (planes, surfaces and curves) in a group.

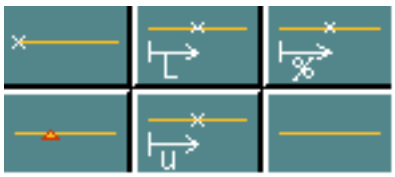
Sectioning Using Spine Curves

Choose the Spine Curve icon to section entities by creating infinite planes along a spine curve at specified points and choose one of the following:

- Specify all locations graphically to select all the plane locations on the spine curve graphically.
- Specify initial/final locations to indicate the plane locations nongraphically. Using this option, you can specify the initial and final locations of the spine curve.

  To specify the initial location of the curve, follow these steps:

1. Choose Start of Curve. The following menu appears.



2. Choose one of the following options.

Choose this option to indicate that the first location is the beginning of the spine curve.

Choose this option to indicate that the first location is at the specified distance from the beginning of the curve. Enter the distance in the field.

Choose this option to indicate that the first location is at a distance of the specified percentage of the total length (from the starting point) of the spine curve. Enter the percentage in the field.

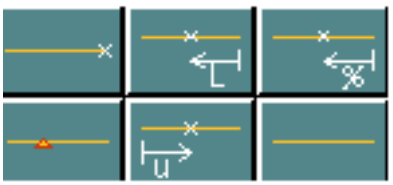
Choose this option to indicate that the first location will be selected graphically. If no location is specified, the starting point of the curve is selected.

Choose this option to indicate that the first location is at the specified parameter value along the spine curve.

Choose this option to indicate that the first location will not be specified.

To specify the final location of the curve, follow these steps:

1. Choose End of Curve. The following menu appears.



2. Select one of the following options.

Choose this option to indicate that the final location is the end of the spine curve.

Choose this option to indicate that the final location is at the specified distance from the end point of the curve. Enter the distance in the field.

Choose this option to indicate that the final location is at a distance of the specified percentage of the total length (from the end point) of the spine curve. Enter the percentage in the field.

Choose this option to indicate that the final location will be selected graphically. If no location is specified, the end point of the curve is selected.

Choose this option to indicate that the final location is at the specified parameter value from the beginning of the spine curve.

Choose this option to indicate that the final location will not be specified.

Intersecting Two Groups of Entities

Choose the Intersect Entities icon to find the intersection of two groups of entities.

Other Options on the Property Sheet

The following options on the property sheet are common to the Parallel Planes, Planar entities, Spine curves, and Intersecting options on the property sheet.

Using Multiple Planes

Choose Multiple Planes to section with multiple planes parallel to each other and at a given distance from each other. When you choose this option, follow these steps:

1. Enter the number of planes in the Number of Planes field.
   This option does not have any default value.
2. Enter the distance between the multiple sectioning planes in the Distance field.
3. Enter a number in the Increment Layer by field to define the successive layers on which to place successive cuts. The default is 0.

   For example, if construction layer 37 is active and you enter 4 in the Number of Planes field and 2 in the Increment Layer By field, then the first cut goes on layer 37, the second cut on layer 39 (37+2), the third cut on layer 41 (39+2). The value entered in the Layer field determines the layer of the first cutting plane.

Please note: If no results are obtained for certain sections, no entities are placed on the current layer specified in the Increment Layer By field. For example, if you ask for four successive cuts, and they are to be placed on successive layers, and if the third section does not take place, nothing is placed on the third layer by this process.

Smoothing the Intersection Curves

Choose Optimize to smooth the intersection curves. This option is selected by default.

The system tries to minimize the number of segments within the specified tolerance using a least square method, while increasing the degree from 3 to 6 with one span and using degree 3 and 5 for a multiple span.

If this method fails to smooth the curves within the specified tolerance, the system approximates the intersection curves by curves of degree 3 or 5. This approximation will have as its knots a subset of knots of the original curves.

If Optimize option is not chosen, no optimization is done. This could be useful when the system needs to make many intersections to speed up the sectioning of surfaces. Then the degree of the resulting entity is 3.

Limiting Search to Edges

Chose Limit search to edges to limit the search for intersection curves by starting at only those points found as intersections of the plane with the surface edges. This option is not selected by default. This option is not available when intersecting two groups of entities.

This option speeds up the execution of the command up to three or four times in some cases.

This option cannot be used when the intersection of a surface with a given plane simultaneously generates the following two kinds of curves:

- Curves having common points with the surface boundary
- Inner curves, which do not have any common points with the surface boundary

  In this case, if Limit search to edges is chosen, inner curves will not be found. However, if only inner types of curves are expected, using this option will not make any difference as all curves will be found.

Please note: Do not use this option if sectioning involves only curves, planes, or simple Srevolutions or Tcylinders (generated by a line). It will not make any difference in the results.

Displaying the Results

Choose Nspline to create the curve as an Nspline. However, if the curve is a line, arc, or a conic, it is written to the database as such.

Choose B-spline to represent the intersection of two surfaces (or sections of a surface) as a B-spline entity. One or more B-spline entities are created.

Choose String to represent the intersection of two surfaces (or sections of a surface) as a string entity. One or more B-spline entities are created.

The initial and final tangent are represented as a line at the beginning and at the end of the created strings.

Choose Cpoint to represent the intersection of two surfaces (or sections of a surface) as a Cpoint entity. One or more Bspline entities are created.

Please note: If none of the options describing the curve type are chosen (Nspline, B-Spline, String, or Cpoint) the curves are written to the database as lines, conics, or Nsplines whichever is appropriate.
If intersecting surfaces are not represented as plane, cylinder, or cone types in the database, then no analytical curves (conics) are created.

Specifying the Tolerance

Enter the tolerance value in the Tolerance field. The tolerance value determines the maximum distance of the curve of intersection from the true intersection.

Specifying the Layer Number

Specify the layer on which the intersection curves and/or points are placed in the Layer field. When incremental layering is active (for sectioning only), this is the layer of the entities created for the first section. The layer number ranges from 0 through 254. The active layer is the default layer.

Specifying the Tag name

Choose Tag and enter a tag name in the Tag field. The name must begin with an alphabet and must not exceed six characters. The characters allowed are: A-Z, 0-9 and the minus sign (-). In the case of multiple results, the tag name will appear with all the results and will be incremented.

Suppressing Messages

Choose Suppress messages to suppress the intersecting (intersection mode) or sectioning (sectioning mode) related system messages. For a sample system message, refer to page 2-44.

Specifying Colors

Enter a positive integer in the range 1-65 in the Color field to specify the color of the resulting surface.

Boolean Operations

Boolean operations join solids, subtract the volume of one solid from another, or keep the common volume between two intersecting solids. Boolean operations create new solids or change existing solids by:

- Joining two or more solids together (UNION SOLID)
- Subtracting the volume of one solid from another (SUBTRACT SOLID)
- Creating a new solid from the common volume between two intersecting solids (INTERSECT SOLID)

When joining two or more solids into one solid entity, the system creates a new solid if the two original solids intersect (common volume) and/or they have coplanar faces.
The system can union up to 30 solids at one time.

When creating new solids by subtracting volume, the system subtracts common volume(s) between the solids from the first solid selected. This operation is similar to the manufacturing methods used when machining or drilling an object.

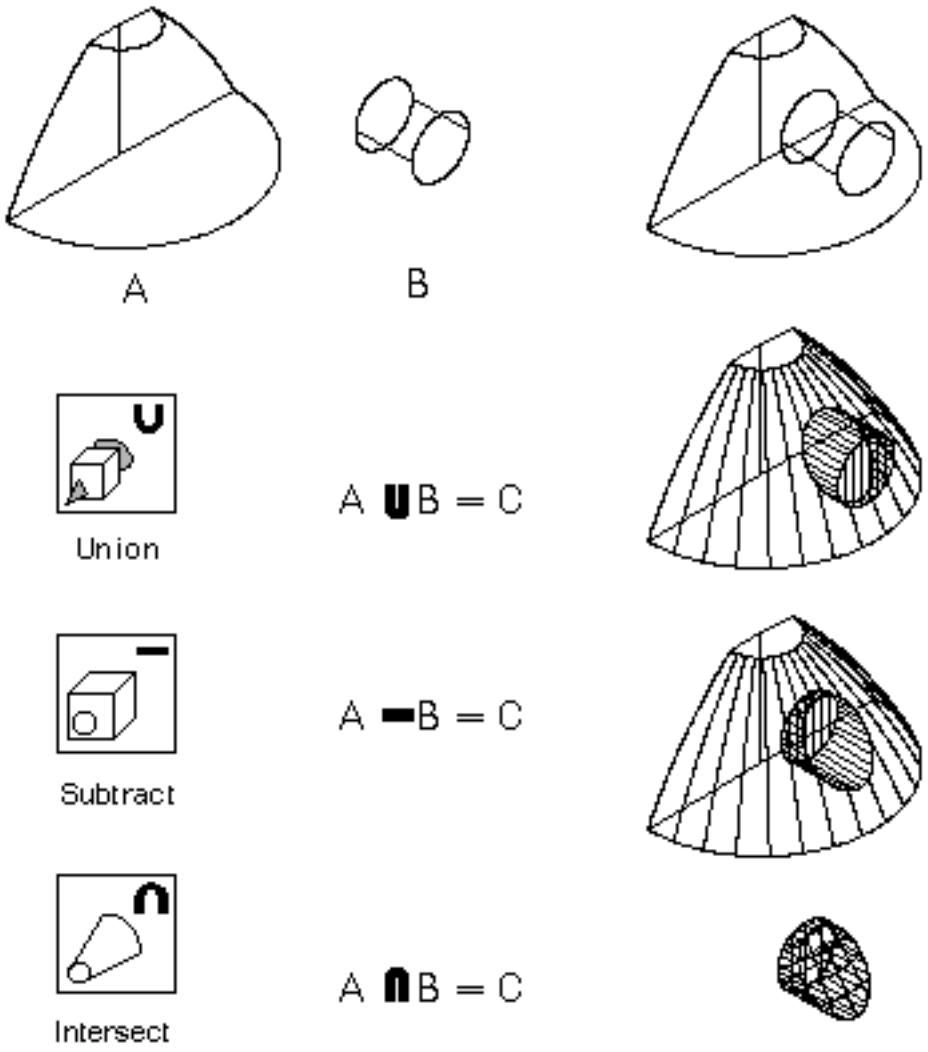
When creating new solids by intersecting solids, the system creates a new solid from the common volume between two solids.
The system can intersect up to 30 solids at one time.

Please note: The performance of Boolean operations is best with solids created in a double-precision database.

The system can perform a single Boolean operation on up to 100 solids at one time.

You can also perform several Boolean operations in sequence from the same property sheet. After the system executes the original Boolean operation, you can subtract, join, or intersect solids regardless of which Boolean operation you originally selected.

Boolean Operations and Their Results



Removing Unnecessary Topological Edges

When you are constructing a solid, you may encounter a situation where extra topological edges remain in place after a Boolean operation. By choosing the Remove Seams icon you can remove unnecessary topological edges. This replaces several adjacent faces on the same plane with a single face whose geometry is the common plane.

Limitations for Boolean Operations

- When subtracting solids, using the Subtract Solid command, to create a thin-walled vessel, there is a limitation on solid thickness. The solid thickness should be greater than (5 x Tolerance). Determine the tolerance value using either the Verify Entity command or the Dump Entity command shown as follows:

  ```
  #n#VERIFY ENTITY SOLID: MODEL ent d1   (select solid)
  ```

  When the system displays the report in the text window, look for the following information:

  ```
  Level of accuracy in model database units = xxxx
  (Value xxxx defines the TOLERANCE.)
  #n#DUMP ENTITY SOLID SREC 1248: MODEL ent d1 (select solid)
  ```

  When the system displays the report in the text window, look for the following information:

  ```
  Words (4: 5)    xxxx
  (Value xxxx defines the TOLERANCE.)
  ```

- When joining solids using the Replace modifier (Union Solid Replace) : Creating a multi-shell solid if the minimum thickness of solid walls is less than (5 x TOLERANCE) may result in an invalid solid. Determine the tolerance value using either the Verify Entity command or the Dump Entity command shown as follows:

  ```
  #n##VERIFY ENTITY SOLID: MODEL ent d1   (select solid)
  ```

  When the system displays the report in the text window, look for the following information:

  ```
  Level of accuracy in model data base units = xxxx
  (Value xxxx defines the TOLERANCE.)
  #n#DUMP ENTITY SOLID SREC 1248: MODEL ent d1 (select solid)
  ```

  When the system displays the report in the text window, look for the following information:

  ```
  Words ( 4: 5)    xxxx
  (Value xxxx defines the TOLERANCE.)
  ```

Boolean Failures

If the Boolean operation between a single pair of objects cannot be completed (and the part is in double-precision), the system makes another attempt to perform the Boolean operation, displaying a message to this effect. This can happen twice. In the instance of a multiple Boolean, the time impact on successful operations would be too great. Any multiple Boolean operation that fails at the *n*th step should be split into several separate commands.

For example, if the command Union Solid: Tag SOL1-10 returns the following message,

```
FAILED IN BOOLEAN 5
```

Execute the following commands:

UNION SOLID TAG STEP4: Tag SOL1-5

UNION SOLID TAG STEP5: Tag STEP4 SOL6

UNION SOLID TAG FINAL: Tag STEP5 SOL7-10

Explained below is an alternate solution available when using explicit solids and you encounter a Boolean failure with Union, Subtract, or Intersect Solid.

1. File the Part.
2. Use the V modifier in your command as follows:

   Union Solid V *n*

   Subtract Solid V *n*

   Intersect Solid V *n*

   The V modifier puts untrimmed intersection curves on Layer *n*+1, trimmed intersection curves on Layer *n*+2 and points of discontinuity of the intersections on Layer *n*+3. This modifier also prints intersection, edge break and vertex break tables.

   The following information is displayed on each layer:

n	Nothing new.
n+1	All intersection curves found.
n+2	Trimmed versions of these curves.
n+3	Points indicating locations where problems were detected.

   Additionally, the system displays a number of reports in the text window.
3. From the information on the layers, determine the cause of the Boolean operation failure and try changing faces, tolerances, vertices, and so forth of the geometry to get the Boolean to succeed. For example, splitting a face with which the intersector has had a problem may result in a Boolean success. You can also try translating one of the entities by a small distance so that the trimmed intersection curves form closed loops.
4. Exit the part (no file), and work on the part in its previous state, rather than work on the part with the additional entities on *n*+ layers.

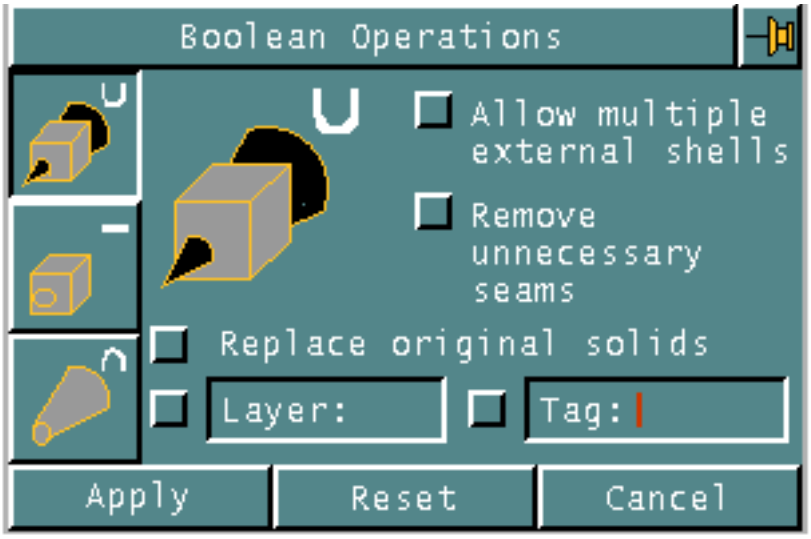
Using Boolean Operations

To create solids or change existing solids using Boolean operations, follow these steps:

1. Select the Boolean operation icon from the Sectioning menu.

   The Boolean Operations property sheet, as shown below, appears.

Figure 2-20 Boolean Operations Property Sheet



2. Chose one of the following icons:

Choose the Union icon to create a solid entity by joining two or more solids into one solid entity.

Choose the Subtract icon to create a solid entity by subtracting the volume of one solid from another.

Choose the Intersect icon to create a solid entity from the common volume between two intersecting solids.

3. Choose Remove Unnecessary Seams to replace edges or seams whose underlying geometries are identical planes (topological edges) with a single face whose underlying geometry is the common plane.

4. Choose Allow multiple external shells to join the external shells into a single solid that contains multiple external shells.

Please note: The result of the subtraction must be one or more external shells and no internal shells. Do not use the option to create solids that contain internal shells.

5. Choose Replace original solids to replace the original solids with the newly created solid after a successful Boolean or multiple Boolean operation.

   If the Boolean operation fails, this option is ignored.
   By default, the system does not replace the original solids with the newly created solids.

6. Choose Layer and enter the layer number (in the range 0-254) in the Layer field to place the new solid on a different layer.

   By default, the system places the newly created solid on the active layer.

7. Choose Tag and enter the tag name in the Tag field to assign a tag to the new solid. The tag name cannot contain more than six characters. If more than one solid is created, the system adds a number to the end of the tag name and increments it for each tag name. However, the automatic incrementation must not cause the tag name to be longer than six characters.
   There is no default tag name.

8. Click Apply.

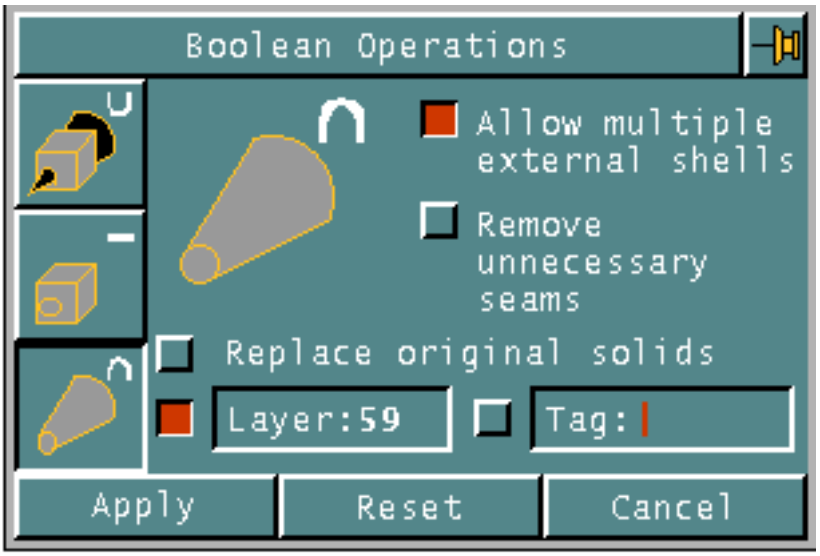
Example of Intersecting Solids

This example creates a single solid from the common volume of two solids.

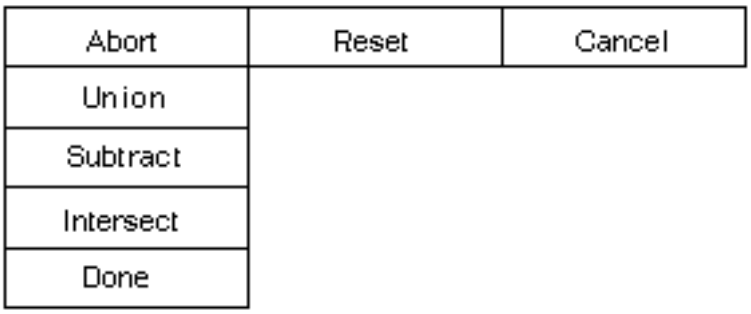
1. Choose the Boolean icon from the Sectioning menu. The Boolean Operations property sheet appears.
2. Choose the Intersect icon.
3. Choose Allow multiple external shells.
4. Choose Layer and enter the layer number as 59 in the Layer field.

   Your Boolean Operations property sheet must now resemble the following figure.

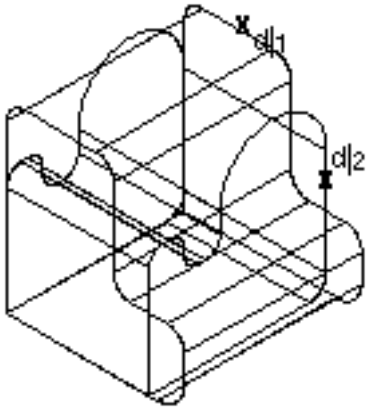
Figure 2-21 Example: Intersecting Solids



5. Click Apply. The following menu appears on the property sheet.



6. In response to the `MODEL ent` prompt in the command line, identify the solids.

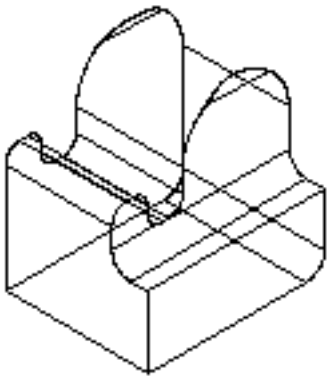


7. Choose Done from the property sheet menu.

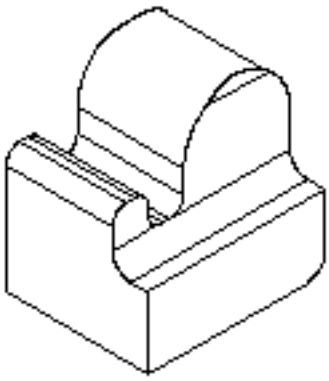
   As the system intersects the two solids to create a new solid, it displays the following messages in the text window:

```
***  Solution in progress. ***
****    Intersecting Surfaces     ****
****    Trimming 52 Intersection Curves
****    Assembling New Faces
****    Generating Database
****    Solid Complete -- Generating Graphics
** 1 SOLID(S) CREATED **
```

   The new solid created from the intersection of the two solids is shown below.



Solid created from intersecting the two solids



Solid after removing the hidden lines

Multiple Boolean Operations

You can perform several Boolean operations in sequence from the same property sheet. After the system executes the original Boolean operation, you can subtract, join, or intersect solids regardless of which Boolean operation you originally selected.

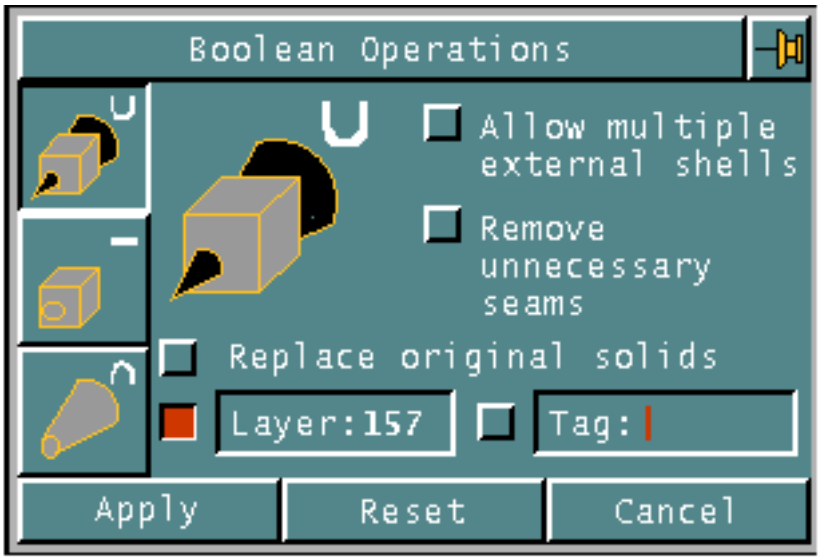
Example

This example executes multiple Boolean operations.

1. Choose the Boolean operations icon from the Sectioning menu. The Boolean Operations property sheet appears.
2. Choose the Union icon.
3. Choose Layer and enter the layer number as 157 in the Layer field.

   Your Boolean Operations property sheet must now resemble the following figure

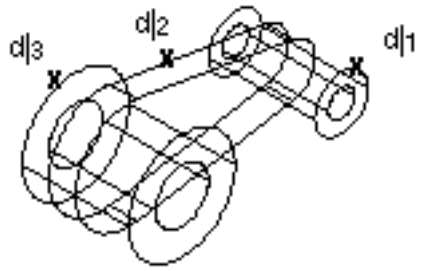
Figure 2-22 Example: Multiple Boolean Operations



4. Click Apply. The following menu appears on the property sheet.

Abort	Reset	Cancel
Union		
Subtract		
Intersect		
Done		

5. In response to `MODEL ent` prompt in the command line, identify the solids.



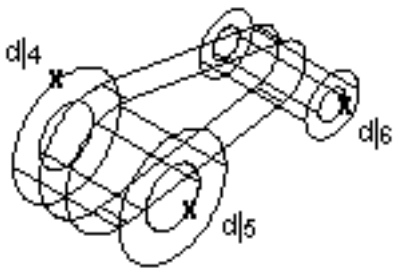
Solids to be joined together

ISO View

6. Choose the Subtract icon from the property sheet to perform the second Boolean operation.
7. When the following prompt appears, select the solids to be subtracted from each other.

```
Subtract: MODEL ent
```

Please note: The system subtracts the common volume(s) between the solids from the first solid selected.



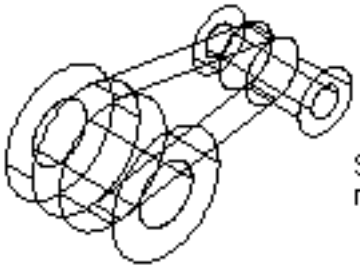
Subtracting the inner cylinders from the outer cylinders

ISO View

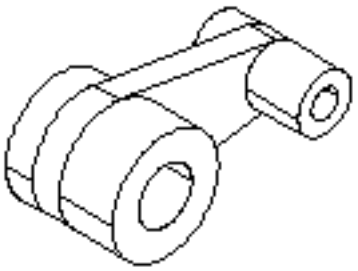
8. Choose Done from the property sheet menu.

   As the system unions the solids together and then subtracts the inner cylinders, it displays the following system messages in the text window:

```
***   Starting Operation # 1     ***
****    Intersecting Surfaces     ****
****    Trimming 8 Intersection Curves
****    Assembling New Faces
***   Starting Operation # 2     ***
****    Intersecting Surfaces     ****
****    Trimming 8 Intersection Curves
****    Assembling New Faces
***   Starting Operation # 3     ***
****    Intersecting Surfaces     ****
****    Trimming 8 Intersection Curves
****    Assembling New Faces
***   Starting Operation # 4     ***
****    Intersecting Surfaces     ****
****    Trimming 8 Intersection Curves
****    Assembling New Faces
****    Generating Database
****    Solid Complete -- Generating Graphics
** 1 SOLID(S) CREATED **
```



Solid created after performing multiple Boolean operations



Solid after removing hidden lines

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