

cam-tool V3

Text Book for Intensive Course - CAD -

Before you read this book...

In this text book, minimum contents required for operating the application are described, in order to emphasize the workflow of user's practical operation.

About this book

■ Version information

cam-tool V3 Version 3.1.4.1

cam-tool MX Version 1.1.4.1

■ Files used in this book

Use the files in the CD-ROM supplied with this book.

Refer to [File name] , [Layer number] written in each chapter.

■ About spot information



If mark is found in this text book, [spot information] is described.

■ About restrictions/precautions



If mark is found in this text book, [restrictions/precautions] are described.

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Chapter 1

Basic operation of *cam-tool V3*

How to start *cam-tool V3*

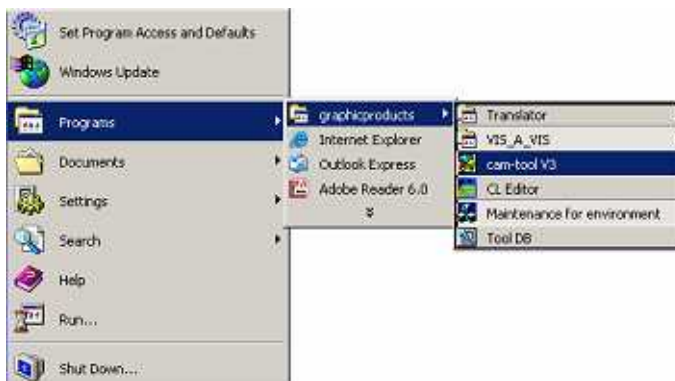
- Start *cam-tool V3* using short-cut icon



Double-click [short-cut icon of *cam-tool V3*] on the desktop.

* Short-cut icon is created automatically on the desktop when you install *cam-tool V3*.

- Start *cam-tool V3* using start menu



[Start] → [graphic products] → [cam-tool V3]

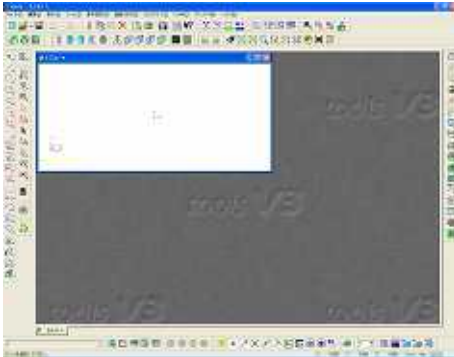
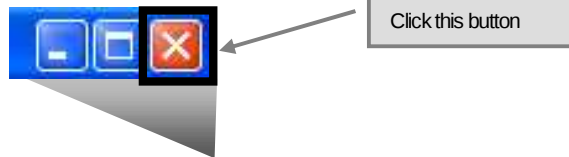


Program files of *cam-tool V3* are installed in the following folder.
Destination folder can be changed when you install *cam-tool V3*.

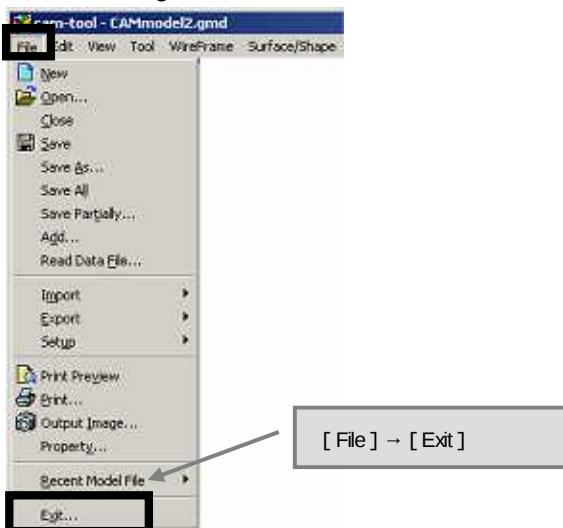
C:/Program File/graphic products inc/cam-tool V3/Bin

How to close *cam-tool V3*

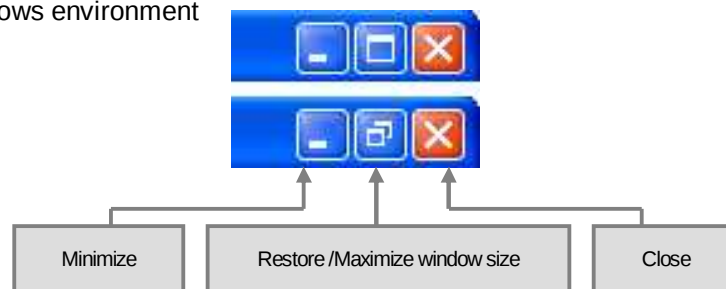
- Close using the button



- Close using the menu

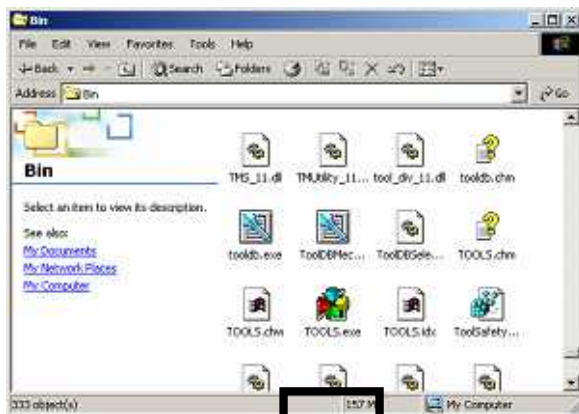


File operation in Windows environment



What if I delete short-cut icon of *cam-tool V3* accidentally?

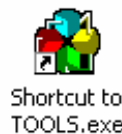
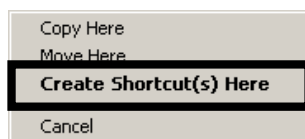
If you accidentally delete short-cut icon of *cam-tool V3* on the desktop, recreate it with the following operations. And, short-cut icon of programs other than *cam-tool V3*, such as [Tool DB] or [Translator], are not created when you install *cam-tool V3*. Start each application using the start menu. If you wish to create short-cut icon of those programs, do the same operations as the one of *cam-tool V3*.



1. Display the folder in which program is installed.

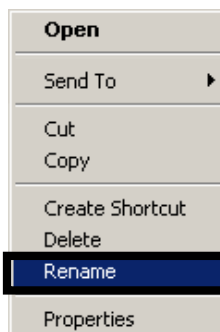
Ex.)

2. Drag the icon of [TOOLS.exe] to the desktop with the right mouse button.



3. Select [Create shortcut].

4. Short-cut icon is created.



5. Right-click on the created short-cut icon, and select [Rename].



To create short-cut icon of other programs, use the following icon.

• Tool DB



tooldb.exe

• Translator



IgesImport.exe



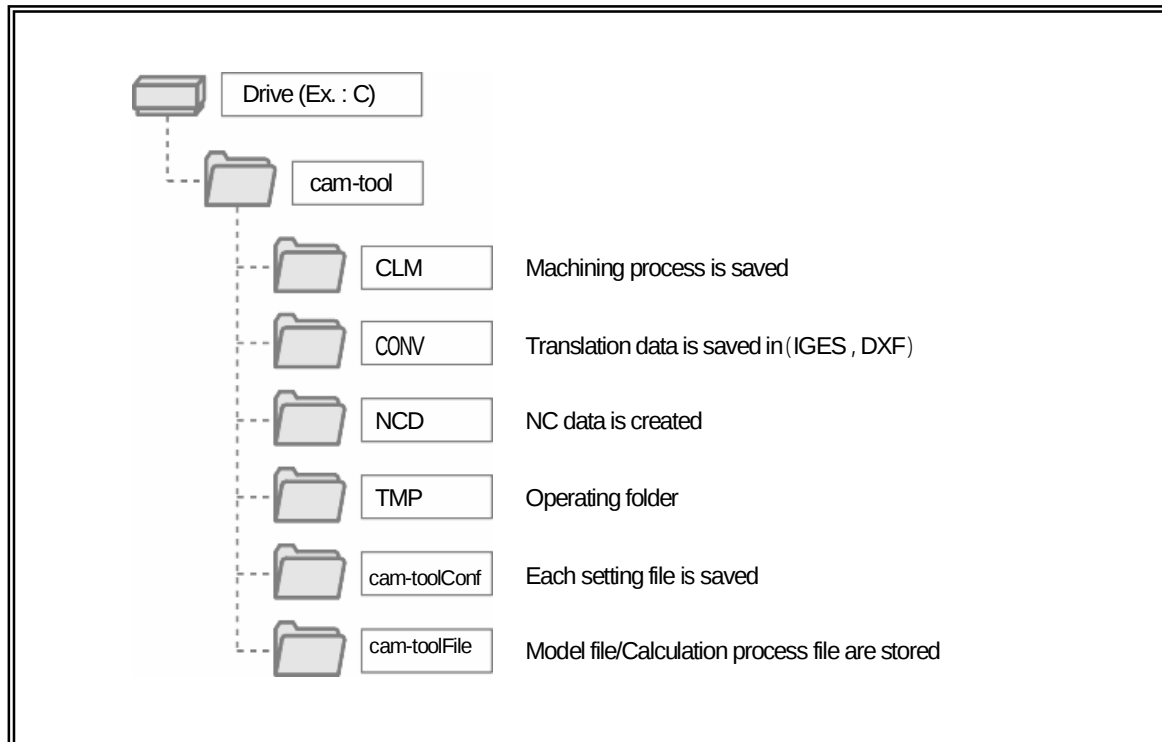
IGESEXPORTE.exe



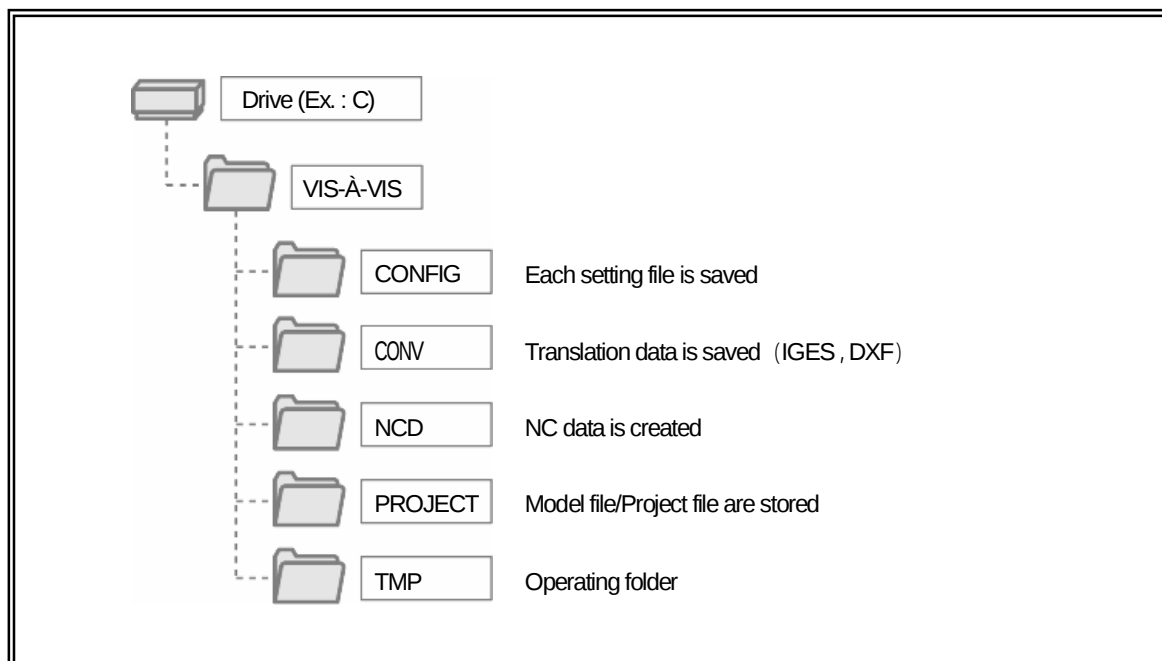
C3Import.exe

Recommended file configuration

■ CAM system: *cam-tool V3* , *cam-tool MX*



■ CAM system: *cam-tool VIS-À-VIS*

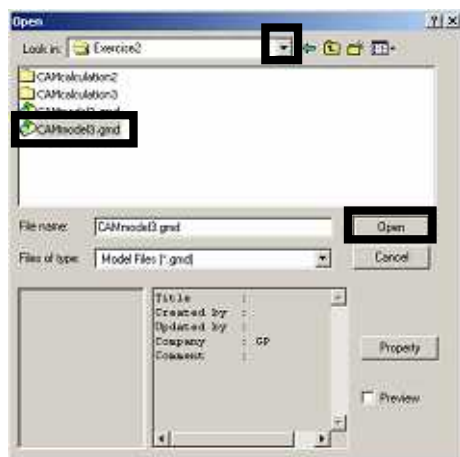


How to open a file

In **cam-tool V3**, a file that you create/edit shape is called [model file].

When you open an existing file, the model file name and extension(.gmd) is displayed on the title bar.

■ Open using icon



Convenient function



Click the triangular button above to display [Recent Model File]. When model files that you have recently used are displayed, select a model from the list.

■ Open using the menu bar



1. Select a [folder].

2. Select a [model file].

The selected file is highlighted.

3. Click the [Open] button.

* Model file can also be opened by double-clicking a model file.

• Preview

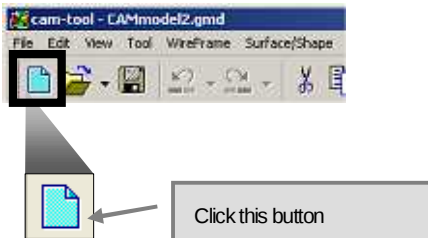
ON: Preview is displayed.

* The model file is displayed in the status that the file was saved.

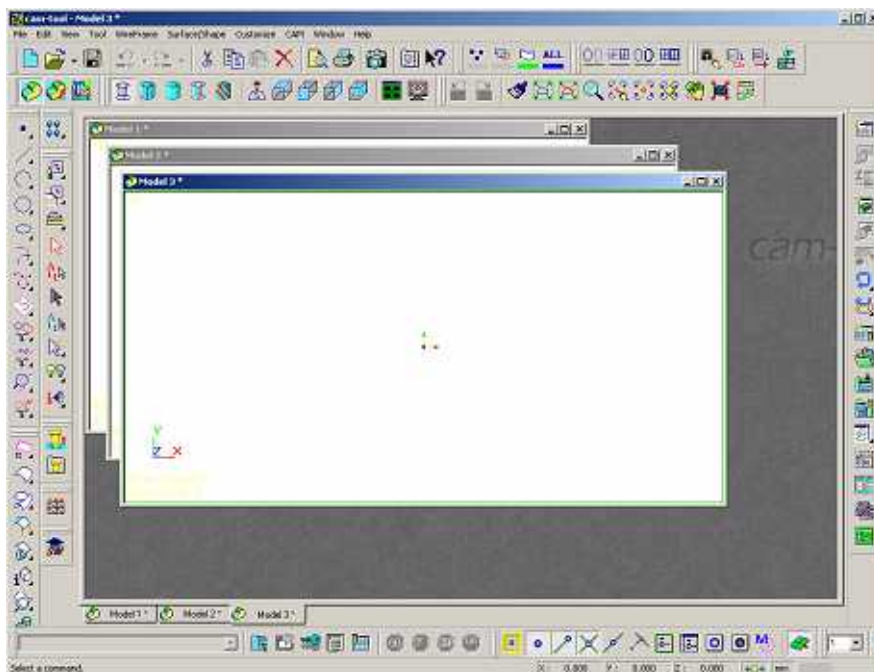
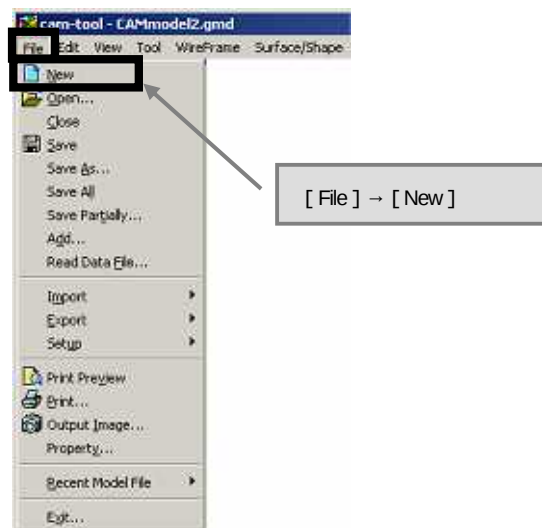
How to create a new file

When a new model file is opened, a temporary file name such as [Model 1] , [Model 2] is displayed on the title bar. Once you save a model file, the file name and extension(.gmd) is displayed on the title bar.

■ Create a new file using icon



■ Create a new file using menu bar

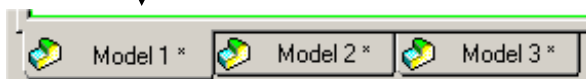
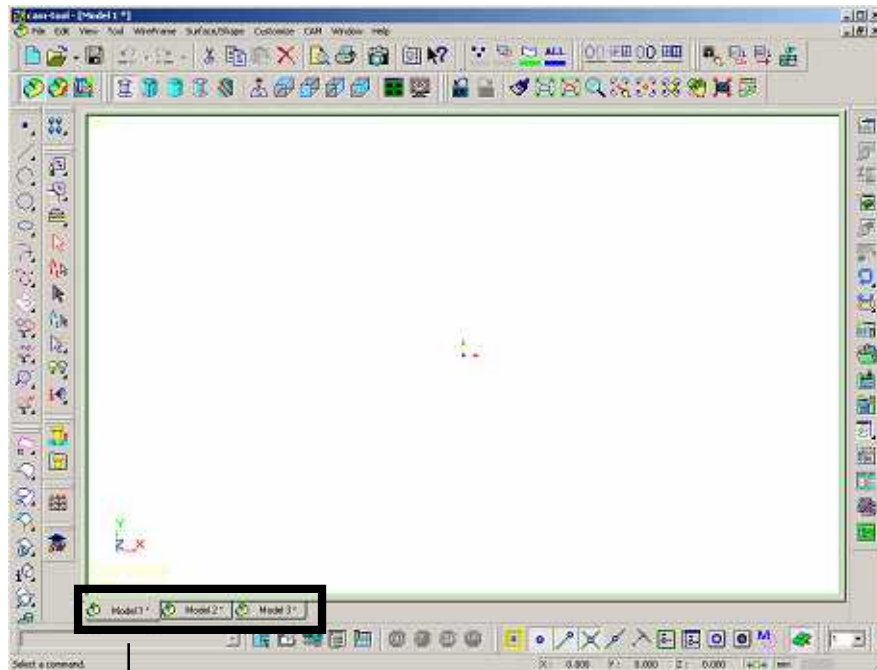
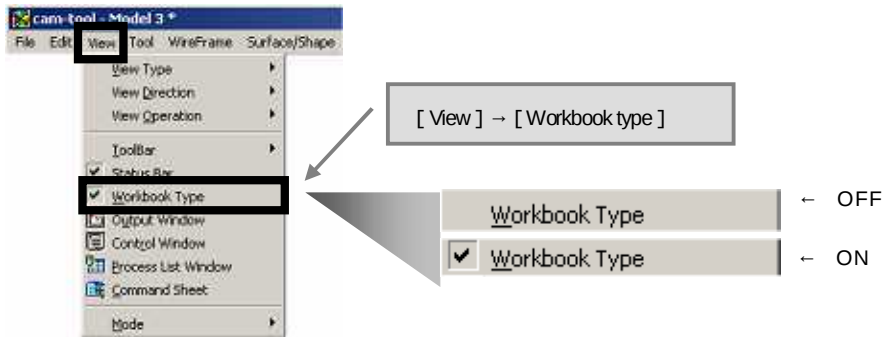


When plural model files are opened, click [Model file window] to switch active model file(model file in the status that can be edited).

How to switch model files (Workbook type)

With [Workbook type] function, active model file can be switched by clicking model file [tab].
It is useful to switch active model file such as when window size of model files are maximized.

■ Set using menu



* Switch active model file by clicking tab

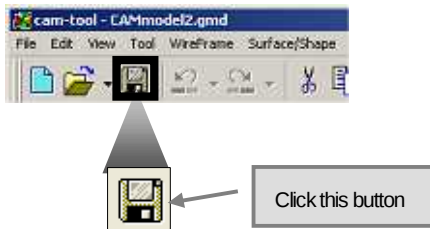
How to save a file

To save a new model file, execute [Save As] command.

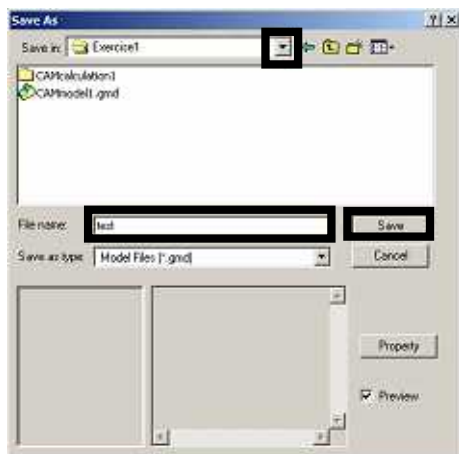
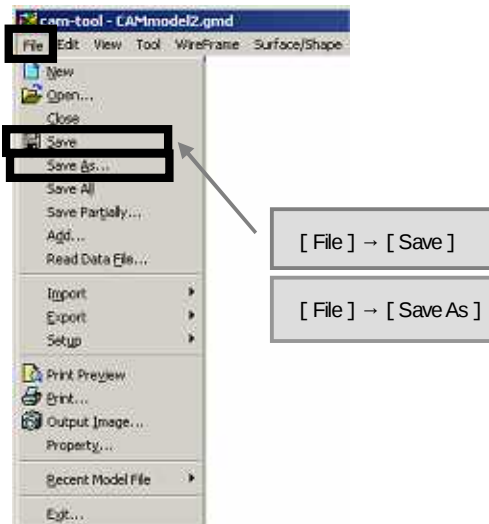
To save changed of an existing file, execute [Save] command.

Once you save a model file, [Model file name] and [Extension(.gmd)] is displayed on the title bar.

■ Save using icon



■ Save using menu bar



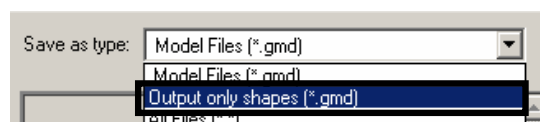
1. Select a folder.
2. Enter file name.
3. Click the [Save] button.



Type of saving a file

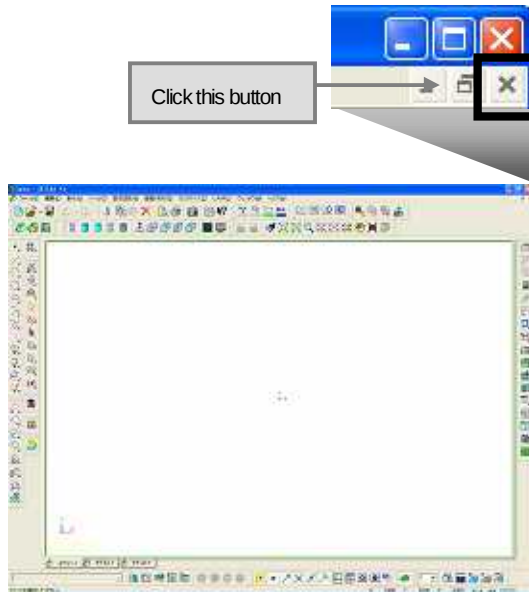
Basically, a model file that is created in **cam-tool V3** cannot be opened in the previous version of **cam-tool V3**. When you save a file, if you select [Save as type: Model Files(*.gmd)], only model file will be saved, and profile information (CAM information) will be deleted.

When you save a model file with this saving type, the model file can be opened in previous version of **cam-tool V3**.

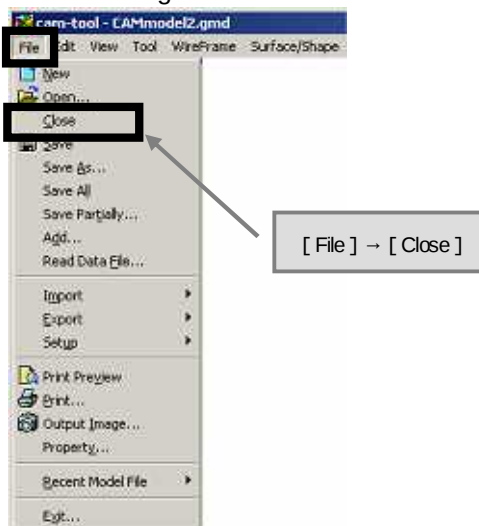


How to close a model file

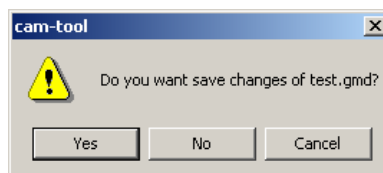
- Close using button



- Close using menu bar



Confirmation dialog



When you close a model file after editing, the dialog above is displayed to confirm whether to save the changes or not.

- Yes: [Save As] dialog is displayed.
- No: A model file is closed without saving the changes.

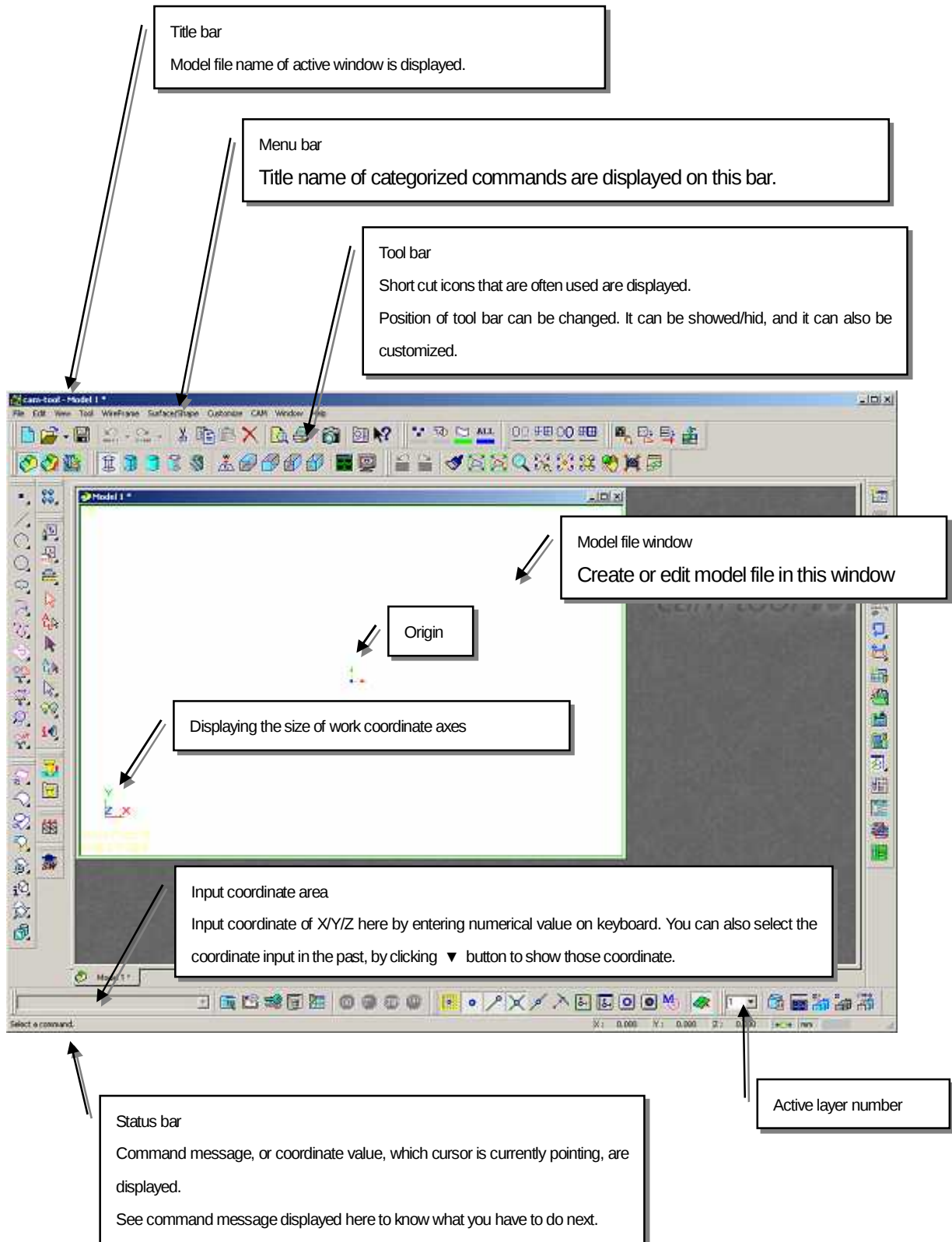


Chapter 2

Basic knowledge of *cam-tool V3*

cam-tool V3 - Screen configuration and description of each part

When you start the application, the following screen is displayed.

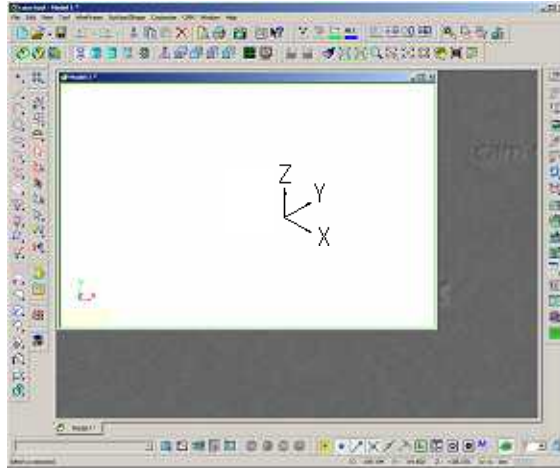


What is origin / coordinate system of a model ?

■ Origin

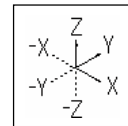
The position of $X=0, Y=0, Z=0$ is called [Origin].

Coordinate axes(X/Y/Z direction) are displayed at the position of origin of a model file.



[+] direction: The side of [X/Y/Z axis] is showed.

[-] direction: The other side of [X/Y/Z axis] is showed.



■ World coordinate system

[Origin] or [X/Y/Z axis direction] that is fixed by the system, is called world coordinate system.

World coordinate system is also fixed by the system, and its position or angle cannot be changed.

■ Work coordinate system

[Origin] or [X/Y/Z axis direction] that can be changed by user, is called as work coordinate system. When [work coordinate system] is not set, [Work coordinate system] would be equal to [World coordinate system].(For how to change, refer to [Chapter 6]).



Work plane

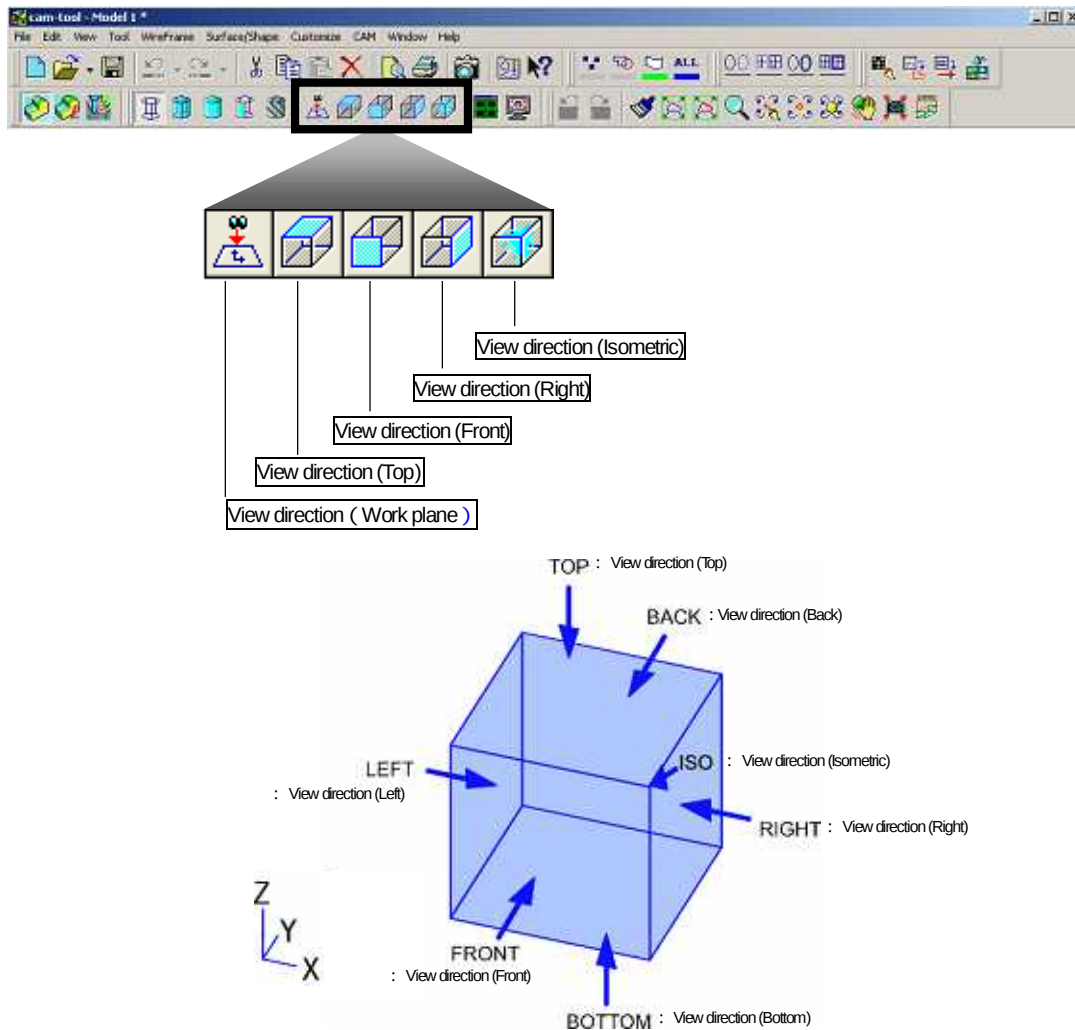
A plane that is composed of [X axis] and [Y axis] of work coordinate system, is called [work plane].

When work coordinate system is not set, [XY plane] of [World coordinate system] is defined work plane.

How to change view direction of a model

Direction of displaying model shape is called [View direction].

Execute the [View direction] command to change the direction of displaying a model shape.



Only icons of [View direction] that are often used, are displayed on the tool bar by each icon. If you wish to use other icons that are not displayed on the tool bar, select the icon with the following step.

- Select from the menu

[Menu] >> [View] >> [View direction] >> Select a view direction

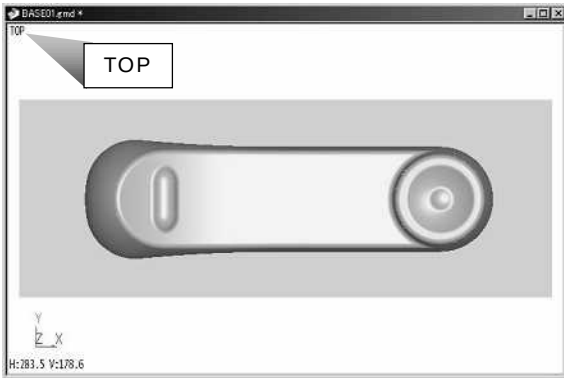
- Display tool bar

[Menu] >> [Tool] >> [Set tool bar] >> Check [View Type/Direction/Operation] ON

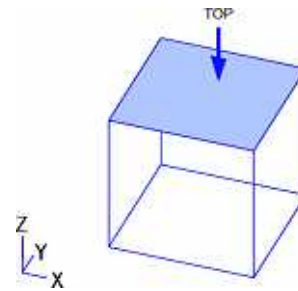
<<Exercise>> Open the model file [CAD model1.gmd] and confirm the operation above.

CD-ROM /Intensive Course/CADmodel1.gmd

■ View direction (TOP) = Top view



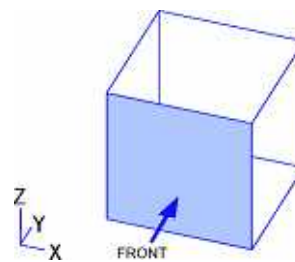
You see [XY plane] from [Z+ axis direction]



■ View direction (FRONT) = FRONT view



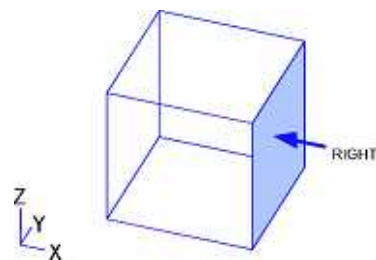
You see [ZX plane] from [Y- axis direction]



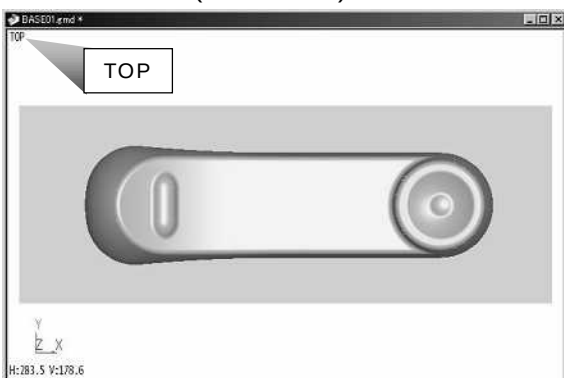
■ View direction (RIGHT) = RIGHT view



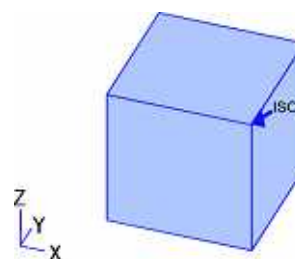
You see [YZ plane] from [X+ axis direction]



■ View direction (ISOMETRIC) = ISOMETRIC view



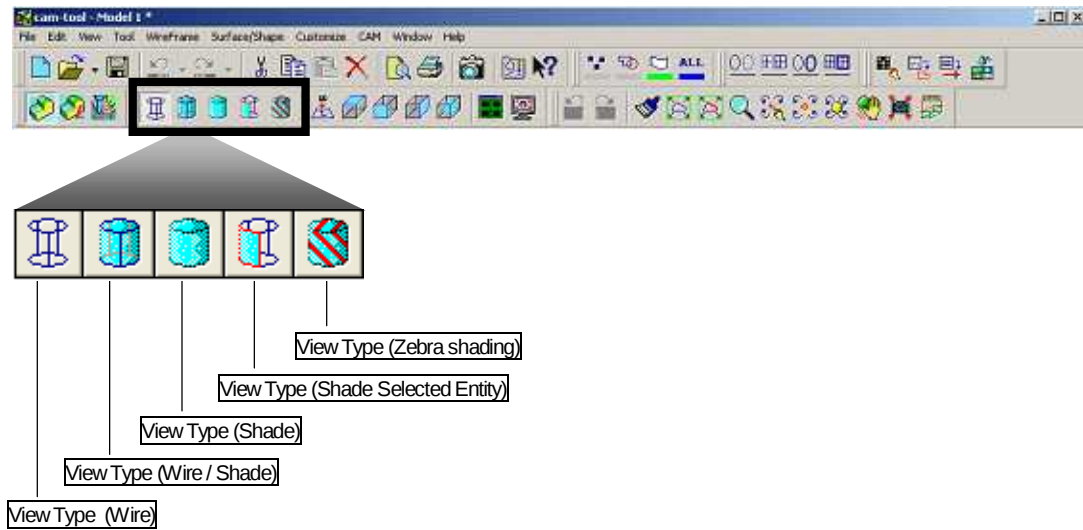
You see [XY plane] from [X+ axis direction], [Y+ axis direction], and [Z+ axis direction].



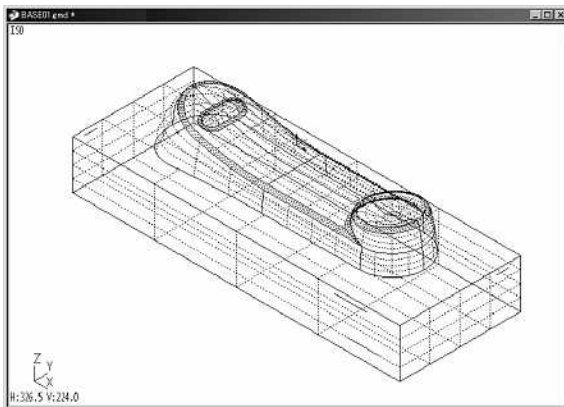
How to change view type of model

Type of displaying model shape is called [View type].

Select view type to confirm the modeling shape.



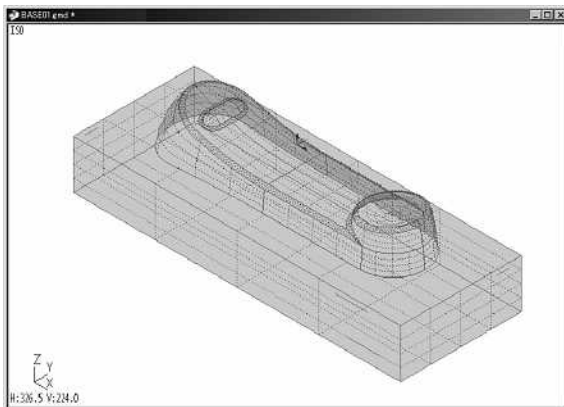
■ View type (Wire)



All wire frames in the active view are displayed.
Surfaces are displayed by [Mesh].

* Modeling is available in this view.

■ View type (Wire / Shade)



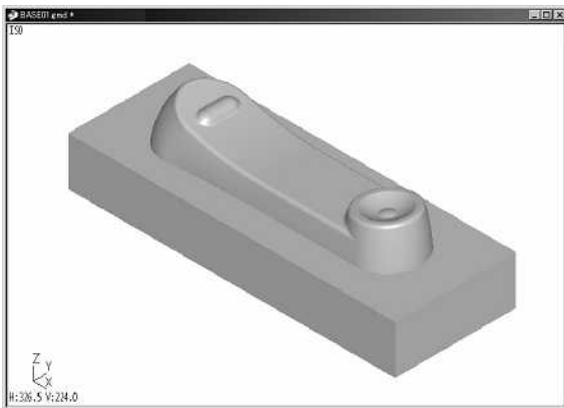
[Shade] >> [Translucent] / [Back face color off]

* Modeling is available in this view.



Setting for [Shading] command can be edited.

■ View type (Shade)

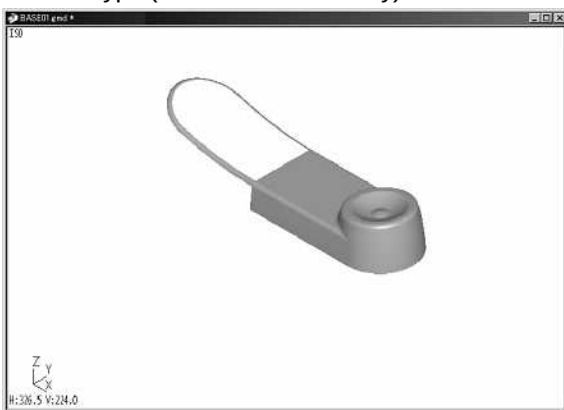


Surfaces are displayed by [Shading].

Wire frame is not displayed.

* You can confirm the model in this view. (Modeling is not available in this view.)

■ View type (Shade Selected Entity)

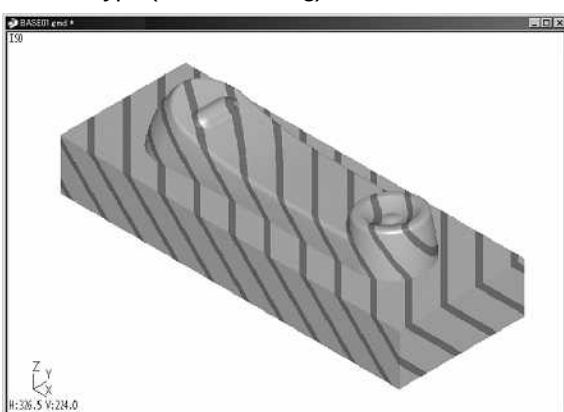


Selected surfaces are displayed by [Shading].

Wire frame is not displayed.

* You can confirm the model in this view. (Modeling is not available in this view.)

■ View type (Zebra Shading)



Surfaces are displayed by [Shading of stripe pattern].

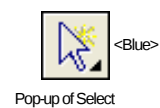
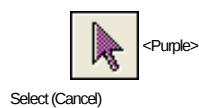
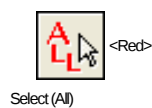
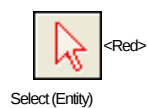
Wire frame is not displayed.

* You can confirm the model in this view. (Modeling is not available in this view.)



[Select] is the command to put an entity into the selected status.(displayed in red)

If you put an entity into the selected status in advance, number of operations in the next command can be reduced.



How to zoom in/out, rotate, move the screen

Learn the [View operations(screen operations)], such as zooming in/out the screen here.



View operation (UNDO)

Restore the view to the previous one.



View operation (REDO)

Restore the view to the previous one restored by [Undo] command.



Redraw

Clear up the screen when drawings are patchy.



Zoom extent

Zooms up all entities so as to fill the active view.



Zoom selected

Zooms up enlarge the selected entities so as to fill the active view.



Zoom

- Drag with the left mouse button: Specify area to zoom.
(Zoom in: down to up, Zoom out: up to down)
- Drag with the right mouse button: Zoom in by the specified magnification.
- Ctrl key + click with the left mouse button: Zoom out by the specified magnification.
- Shift key + click with the left mouse button: Zoom in by exact size.



Convenient mouse operation

Zoom by specifying area: Drag with the right mouse button pressing [Ctrl key]

([Ctrl key] can be changed in [Environment setting]. (Zoom in: down to up, Zoom out: up to down)



Rotate view

Rotate view by dragging with the left mouse button. The center point of rotation is the gravity center of boundary of the shape. *Boundary: Cuboid shape that contains the right size shape.



Convenient mouse operation

[View rotation] can be operated by dragging with the right mouse button. [View rotation] can be controlled by the function of [Lock Vertical Drag]/[Lock Horizontal Drag] displayed by right-click on the screen.



Set center of Rotation

Specify the center of [View rotation].

* When it is specified, [Rotate on Selected Pt] is automatically set ON.



Rotate on Selected Pt

Change the center of [View rotation].

OFF: The gravity center of the shape is set as center of rotation.

ON: The coordinate specified by [Set center of Rotation] is set as the center of rotation.



Pan

Move the shape on the screen by dragging with left mouse button.



Convenient mouse operation

Moves the shape on the screen by dragging with the right mouse button pressing [Shift key].
([Shift key] can be changed in [Environment setting].)



Reset

- Restore the view to the initial state based on the [World coordinate].
- [Shift key] + Reset: Restore the view to the initial state based on the [World coordinate].
- [Ctrl key] + Reset: Restore the divided view to the initial state.



Show ruler

Measure the distance between the two points on the view plane, and display the result.



Convenient mouse operation [Consecutive zoom]

- If you use the mouse with wheel,

Zoom consecutively by rolling the wheel. Zoom in/out at the center of the screen.

- If you use the mouse with two buttons/three buttons,

Zoom by dragging with the right mouse button pressing [Ctrl key] + [Shift key].

([Ctrl key] + [Shift key] can be changed in [Environment setting]. (Zoom in: down to up, Zoom out: up to down))

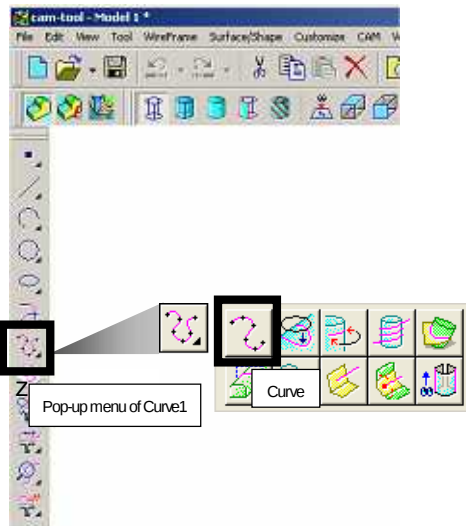


Once [Zoom] / [Rotate on Selected Pt] / [Rotate view] / [Pan] command is executed, it will be set ON consecutively. If you wish to cancel the command, click [Cancel] button, or click the command icon again to set OFF. * [Rotate on Selected Pt] command cannot be ended by clicking the [Cancel] button.

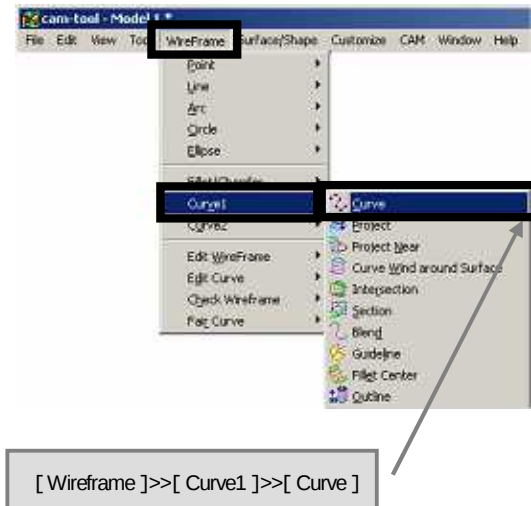
How to select / execute / cancel command

There are a few methods to select a command.

- Select a command by clicking command icon



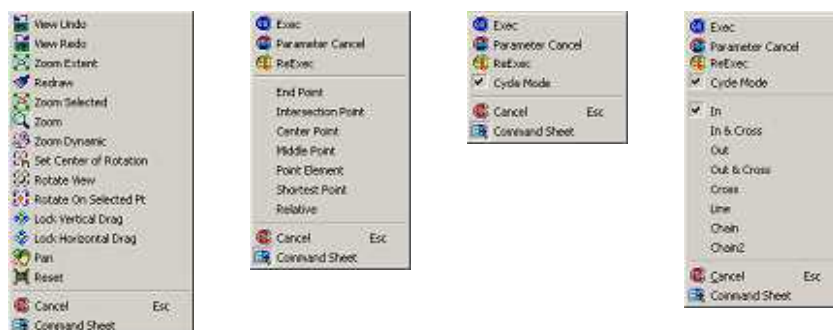
- Select a command by using the menu



- Select a command from the menu displayed by right-click on the screen.

The followings are the short-cut functions. Right-click on the screen, the menu of commands, which are available to execute at the time, are displayed. And you can select a command in the menu.

Selectable commands in the menu vary depending on the selected command, or the state of command operation.



- Select a command from the menu displayed by center-click on the screen.

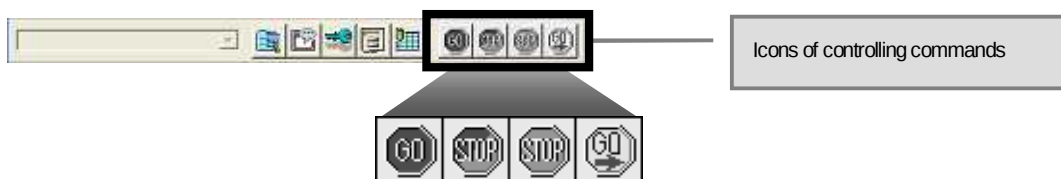
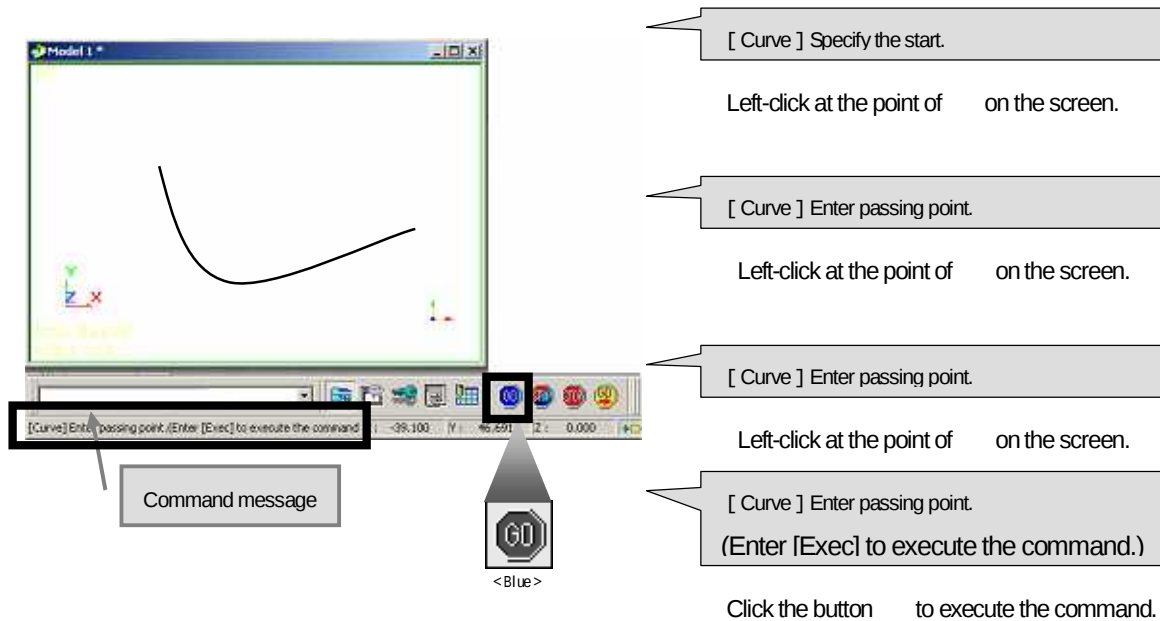
While no command is being executed, click the center mouse button.

Commands that were executed in the past are displayed. And, select one in the command history.



■ Execute/cancel a command

When you select a command, command message is displayed on the status bar. (On the left of the screen)
Follow the command message for your operation.



Execute

Execute the command, or determine the specifying during the command operation.
This function can also be done by pressing the center mouse button.



<Blue/Red>

Parameter cancel

While a command is being executed, cancel the past operations one by one.



<Red>

Cancel

Cancel the command.

Command can also be canceled by pressing the [Esc] key.



<Yellow>

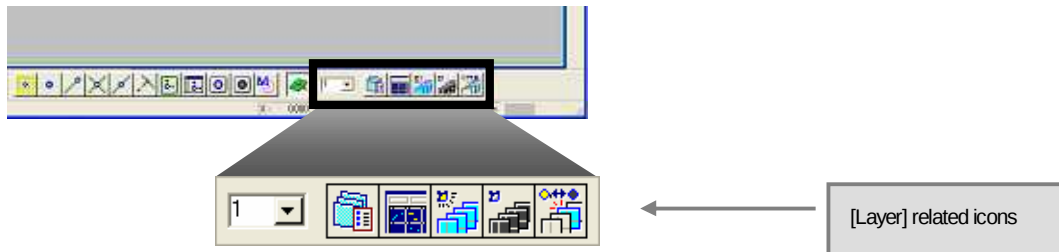
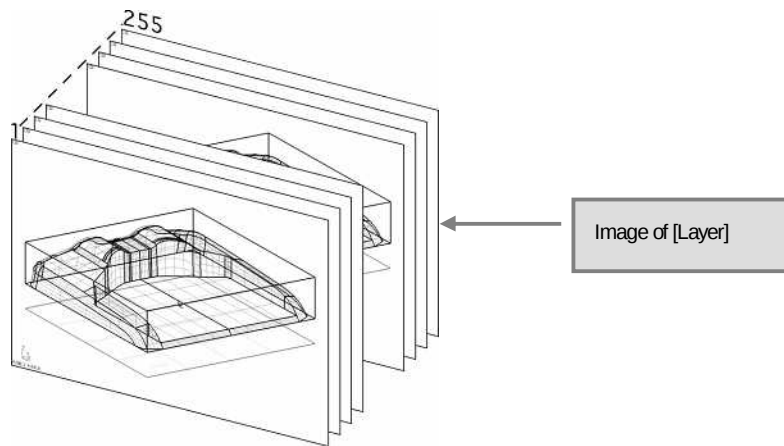
Re-execute

Cancel all of past operations of the command, and re-execute the command. Use this function when you wish to operate the command again.

How to use layer

Layer is like a transparent drawing paper.

In a model file, you can use 255 layers. Using layer makes your modeling operation more efficient.



Activelayer

Activelayer number

Layer that you are currently using for modeling operation is called [Active layer].
To change the active layer, enter a number and press [Enter] key.
Active layer is displayed on the screen, even if it is set hidden.



Show all layers

Show all layers



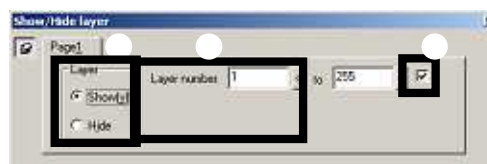
Hide all layers

Hide all layers. Only active layer is displayed on the screen.



Show/Hide layer

Show/Hide the specified number of layer.



Select Show/Hide
Enter layer number
Specify single layer number>>ON
Press the center mouse button to execute.

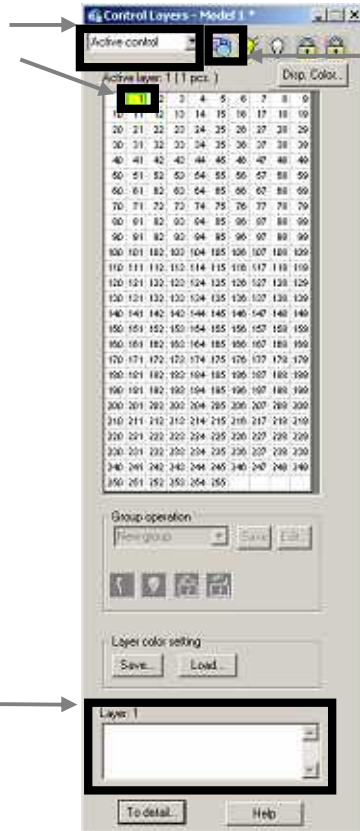


Control layer

The status of all layers can be checked/edited in this dialog.

Controlling layers, such as [Change active layer] / [Show/Hide layer] / [Input comment] is available in the dialog.

Change active layer
Set setting/hiding layer



Apply the settings in the dialog to the screen

Input comment



Browse layer

The status of all layers can be checked/edited visually in this window.

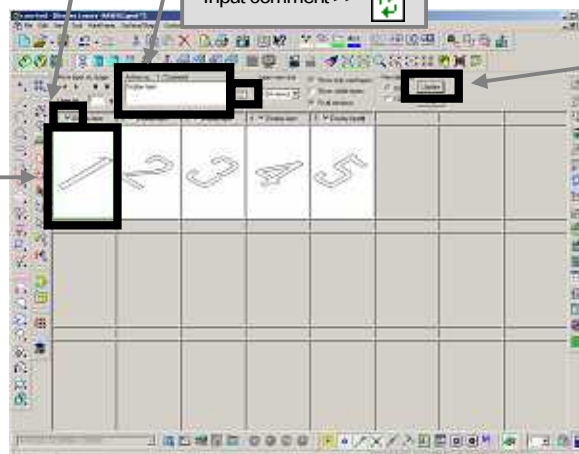
Controlling layers, such as [Change active layer] / [Show/Hide layer] / [Input comment] is available in the dialog.

Change active layer

Show/Hide layer

Input comment >>

After you set, click [Update] button to apply the setting



Chapter 3

Create Wire frame

Open the model file below.

CD-ROM/ Intensive Course/CADmodel2.gmd

- If [Layer number] is written in the description, change the active layer number.
- Follow the [command message] displayed at the lower left of the screen, for your operation.

Ex.) Create a [Single] line by inputting [absolute coordinate value]. [Layer1] ← layer number

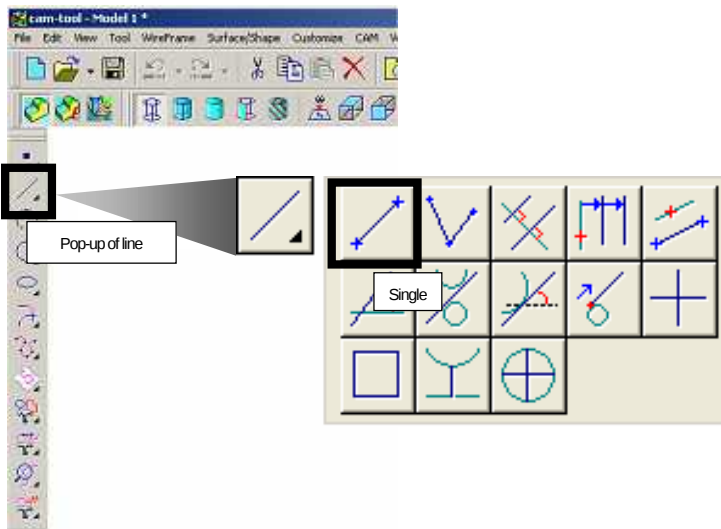


[Curve] Specify the start.
Left-click at the coordinate of start.

[Curve] Enter passing point.
Left-click at the coordinate of end.

Command message

Create a line that connects the specified two points. (If you wish to create lines consecutively, use [Line Segments] command.)



■ Create wire frame by inputting coordinate value

Specify coordinate value by inputting [X/Y/Z] value with comma.

Input coordinate value by either [Absolute coordinate] or [Relative coordinate].

Absolute coordinate: Input coordinate value based on the current work origin (0,0,0).

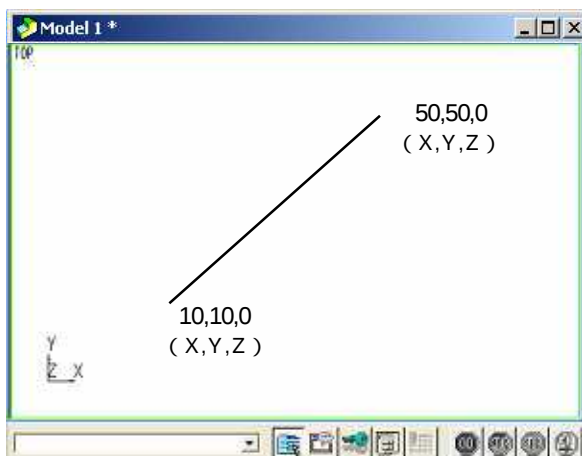
Relative coordinate: Input [/slash] first, and input incremental value based on the previous value.



Convenient function - [,(comma)]

- Press the [Space] key one time. (comma) is input.
- Press the [(Dot)] key twice. [,(comma)] is input.

Create a [Single] line by inputting [absolute coordinate value]. [Layer1]



Select the [Single] command.

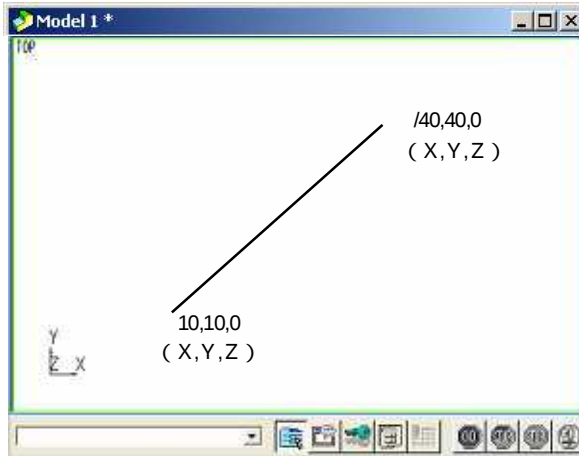
[Single] Specify the start.

Input coordinate value. 10,10,0 <Enter>

[Single] Specify the end.

Input coordinate value. 50,50,0 <Enter>

Create a [Single] line by inputting [relative coordinate value]. [Layer2]



Select the [Single] command.

[Single] Specify the start.

Input coordinate value. 10, 10, 0 <Enter>

[Single] Specify the end.

Input coordinate value. /40, 40, 0<Enter>

* incremental value: [X=40], [Y=40]



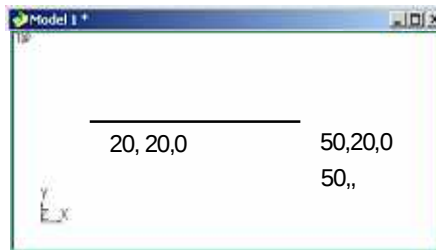
Convenient function

- F12 key

Work coordinate origin (X=0,Y=0,Z=0) can be specified by pressing [F12] key.

- Omit inputting coordinate value [Layer3]

If you omit inputting coordinate value, the previously input value is substituted.

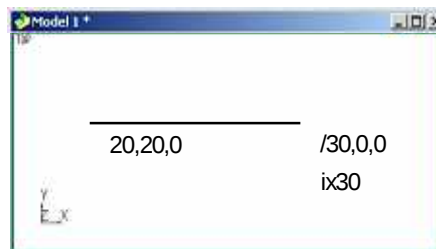


<Normal> Start 20, 20, 0
End 50, 20, 0

<Omit> Start 20, 20, 0
End 50,,

- Simplify inputting coordinate value [Layer4]

This is the simplified function to input either of [X]/[Y]/[Z] relative value.



<Normal> Start 20, 20, 0
End /30, 0, 0

<Simplify> Start 20, 20, 0
End ix30

* Simplified inputting way
ix10 = /10, 0, 0
iy10 = /0, 10, 0
iz10 = /0, 0, 10

- Change relative point

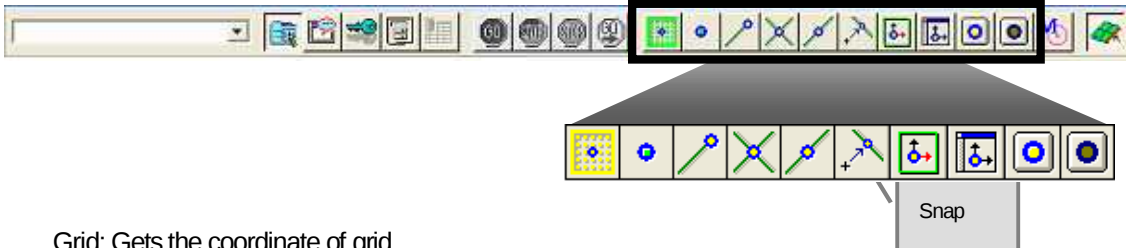
When you input relative coordinate, the [previously input coordinate] that is [referred coordinate] can be changed to [any coordinate]. You can input relative coordinate based on the point, which you specify with the [Change relative point] function.

■ Creating wire frame using the particular point

<Snap>

This is the function to specify the [particular point (End point/Intersection point...)] of entity. To set ON/OFF, click on each icon.

When you put your cursor near each singular point, [Snap name] is displayed. Click the coordinate.



Grid: Gets the coordinate of grid.

Point: Gets the coordinate of point entity.

End: Gets the coordinate of end point of an entity. End point of circle cannot be specified.

Intersection point: Gets the coordinate of intersection of the specified entities. Intersection of surface contour cannot be specified.

Middle point: Gets the midpoint of an entity. The middle point of circle(arc) cannot be specified.

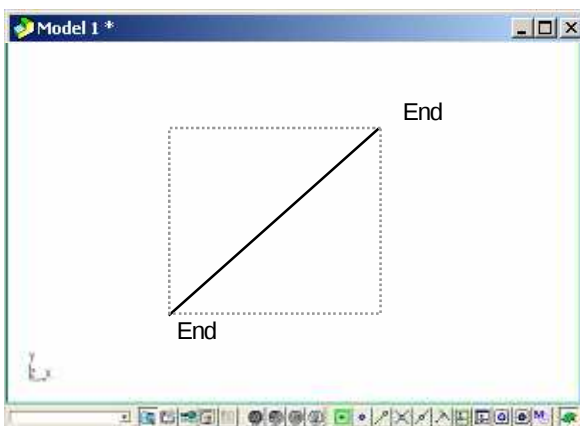
On: Gets the coordinate on an entity.

Work: Gets the coordinate of work origin.(0,0,0).

World: Gets the coordinate of world origin.(0,0,0).

Snap All ON/ Snap All OFF

Create a [Single] line by specifying end points of rectangle entity. [Layer 5]



Select the [Single] command.

Set [Snap(End)] ON.

[Single] Specify the start.

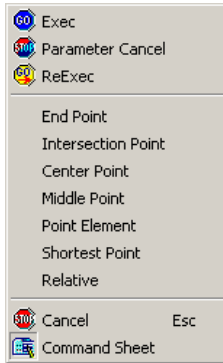
Click the end point on the screen.

[Single] Specify the end.

Click the end point on the screen.

<Input by Get-data>

This is the function to specify the singular point of an entity by compulsion. Select a [Singular point] from the menu displayed by right-click, and specify an entity. Basically, this function is valid one time after you select a [Singular point].



- End Point:: Gets the coordinate of an end point nearer to the picked point of a specified entity. (End point of circle: 0 degree, 90 degree, 180 degree, 270 degree)
- Intersection Point: Gets the coordinates of and intersection of two specified entities. If plural [Intersection point] exist, the nearest intersection point is specified.
- Center point:: Gets the coordinate of center point of a specified entity/surfaces.
- Middle point:: Gets the coordinate of middle point of the two specified points.
- Shortest point:: Gets the coordinates of a point on entity to make the shortest distance between the entity and the specified coordinates.



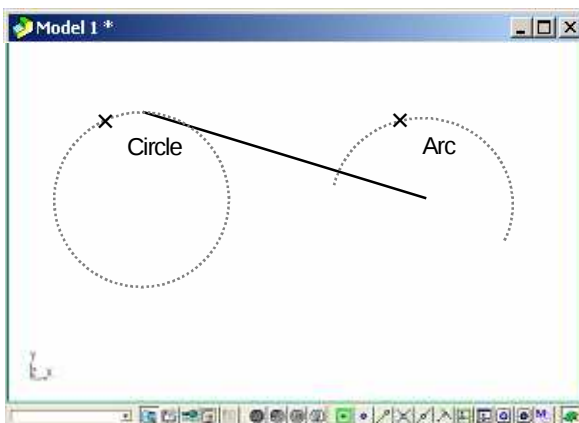
- Change relative point: When you input relative coordinate, the [previously input coordinate] that is [referred coordinate] can be changed to [any coordinate].

Modal function of [Input by Get-data]



Basically, specifying singular point using [Input Get-data] function is valid only one time. However, it can be used consecutively with [Modal] function. *[Modal] is not available for [Change relative point].

Create [Single] line by specifying the end point of the circle and the center point of the arc. [Layer 6]



Select the [Single] command.

[Single] Specify the start.

Display the menu by right-click.

Select end point.

Click the circle on the screen.

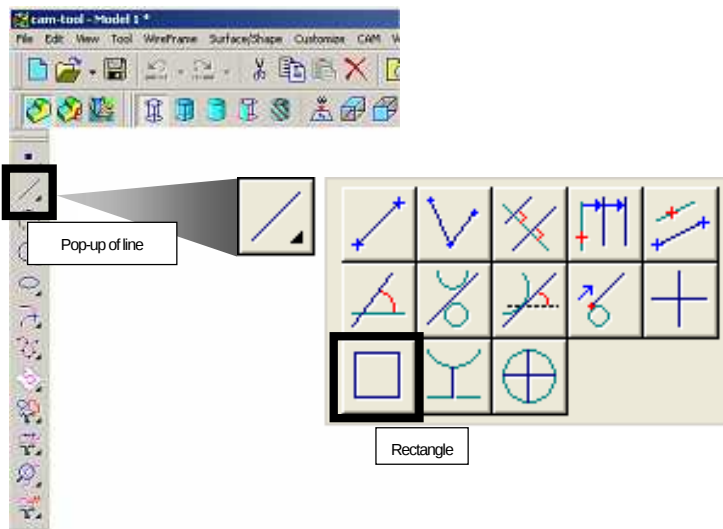
[Single] Specify the end.

Display the menu by right-click.

Select end point.

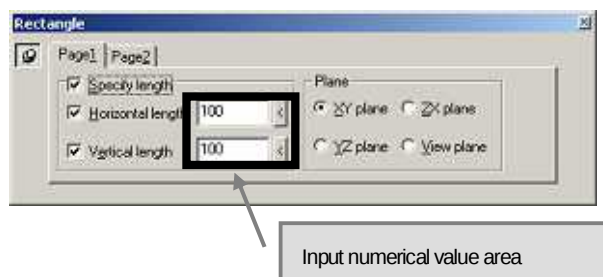
Click the circle on the screen.

Create a rectangle [Layer7]



Command sheet

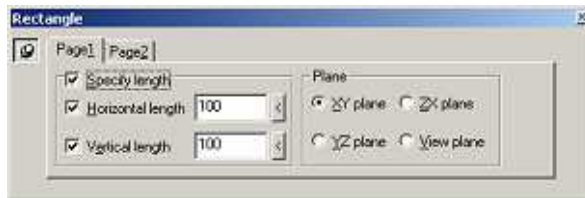
When you execute the command that has parameters to be set, its command sheet is displayed. Command sheet may have plural pages depending on the number of parameters. Click tab to switch page.



Input numerical value in the area above. * Input one-byte numerical value.
[Four-function] is available as below.

+	Addition	Ex) 2+1 >> 3
-	Subtraction	Ex) 5-2 >> 3
*	Multiplication	Ex) 3*3 >> 9
/	Division	Ex) 2-1/2 >> 1.5

< Command sheet >



[Page (1)]

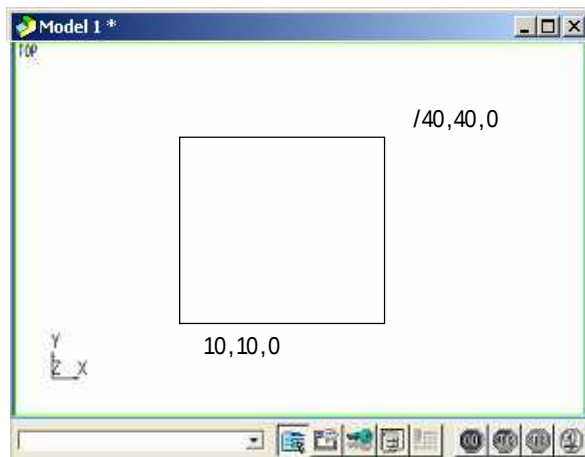
- Specify length: Input value for vertical/horizontal length.
- Plane: Specify the plane to create a rectangle.



[Page (2)]

- Radius of fillet: Input arc radius.
- Length of chamfer: Input dimension of chamfer.

Create a [Rectangle] by inputting coordinate value.



Select the [Rectangle] command.

Specify parameters in the command sheet.

Specify length: [OFF]

Plane: Specify: XY Plane

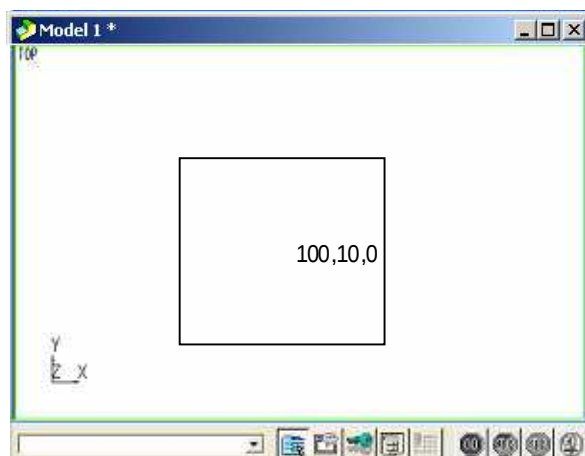
[Rectangle] Specify the start.

Input coordinate for the start point. 10,10,0 <Enter>

[Rectangle] Specify the end.

Input coordinate for the end point. /40,40,0 <Enter>

Create a [Rectangle] by specifying value in the command sheet.



Select the [Rectangle] command.

Specify parameters in the command sheet.

Specify length: [ON]

Horizontal: 40

Vertical: 40

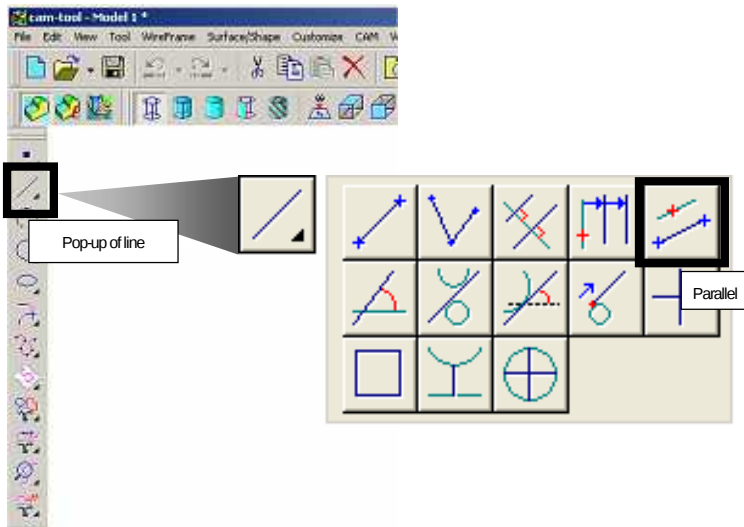
Plane: Specify: XY Plane

[Rectangle] Enter the center point.

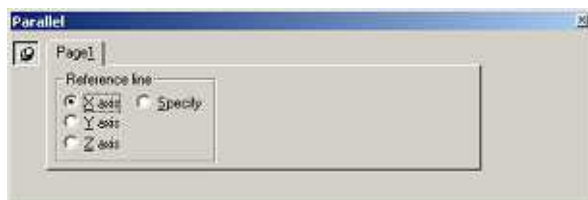
Input coordinate for the center point. 100,10,0 <Enter>

How to create line parallel to X/Y/Z axis <Parallel>

Create a line that is parallel to the specified X/Y/Z axis. [Layer 8]

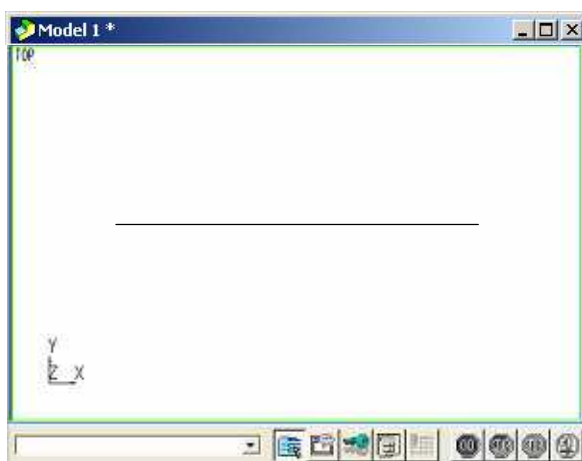


< Command sheet >



- Reference line
 - X-axis: Create a line parallel to X-axis.
 - Y-axis: Create a line parallel to Y-axis.
 - Z-axis: Create a line parallel to Z-axis.
 - Specify: Create a line parallel to the specified line.

< Operation >



Select the [Parallel] command.

Specify parameters in the command sheet.

Ex.) Reference line : X-axis

[Parallel] Enter the start point.

Input coordinate value or click a certain position on the screen.

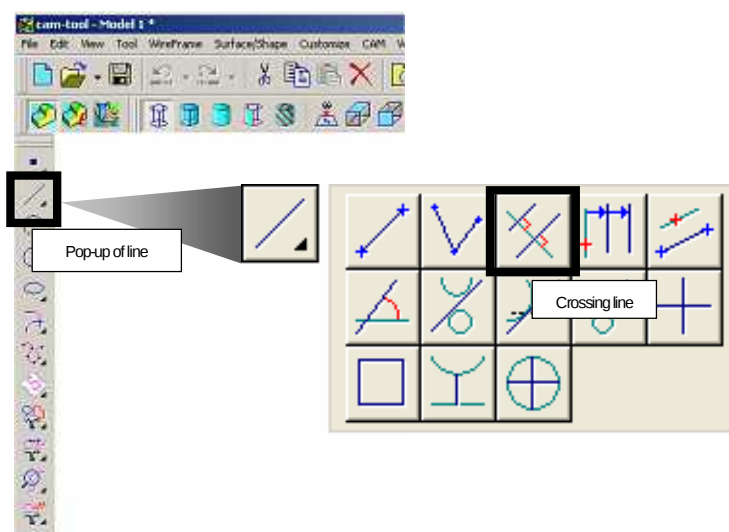
Enter the end point.

Input coordinate value or click a certain position on the screen.

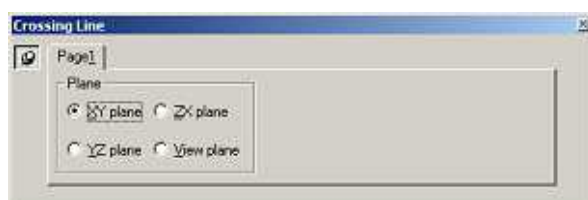
A line parallel to the reference line(X-axis) is created.

How to create line crosses at right angles to reference line <Crossing line>

Create a line that crosses at right angles to reference line. [Layer 9]

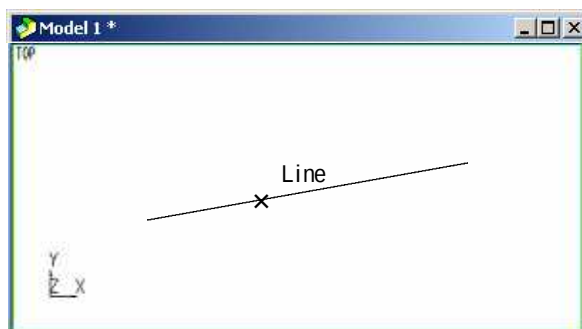


< Command sheet >



- Plane: Specify the plane to create a line.

< Operation >



Select the [Crossing line] command.
Specify parameters in the command sheet.
Ex.) Plane: XY plane

[Crossing line] Pick a reference line.

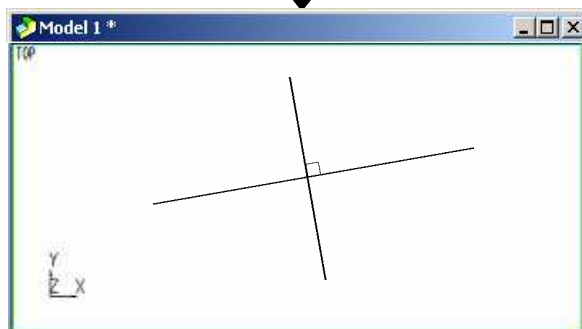
Click the reference line. (Line)

[Crossing line] Enter the start point direction.

Click the coordinate of the start point.

[Crossing line] Enter the end point direction.

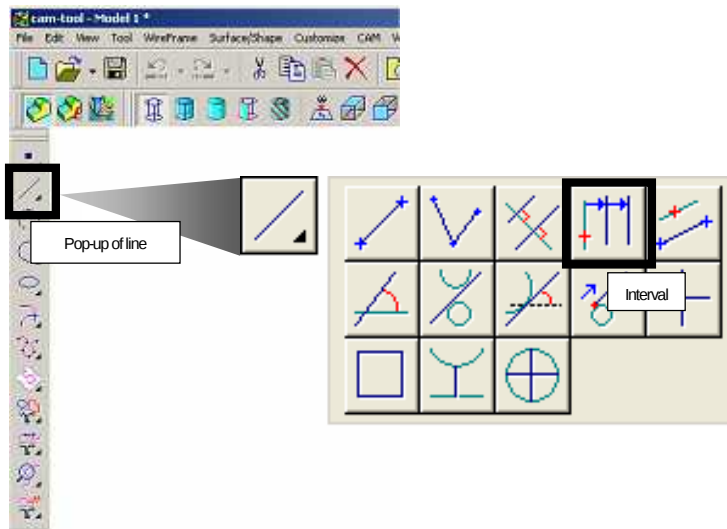
Click the coordinate of the end point.



A line that crosses the specified reference line is created.

How to create line, which is parallel to, and the specified distance far from reference line < Interval >

Create a parallel line, which is parallel to, and the specified distance far from reference line. [Layer 10]

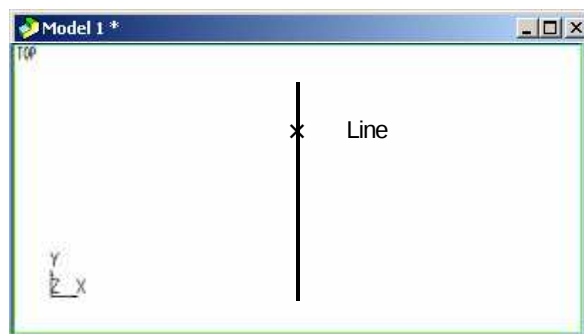


< Command sheet >



- Distance: Input value for the distance from reference line
- Repeat: Specify number of creating lines.
- Both sides: Create lines at both sides of reference line.
- Plane: Specify the plane to create a line.

< Operation >



Select the [Interval] command.

Specify parameters in the command sheet.

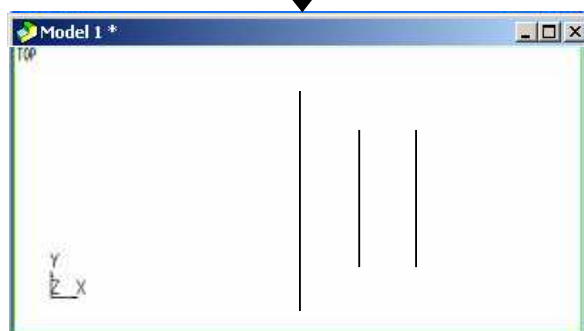
Ex.) Distance: 10

Repeat: 2

Plane: XY plane

[Interval] Pick a reference line.

Click the reference line. (Line)



[Interval] Enter the start point direction.

Click the coordinate of the start point.

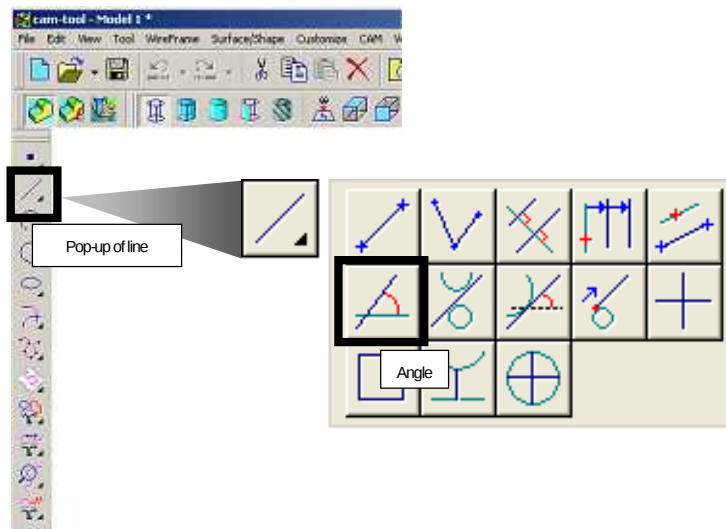
[Interval] Enter the end point direction.

Click the coordinate of the end point.

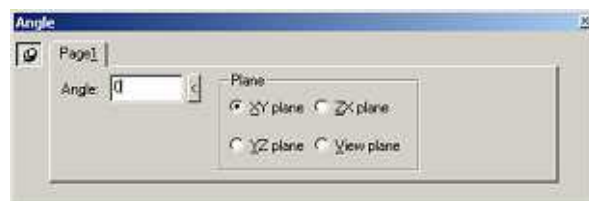
How to create line that has the specified angle?

<Angle>

Create a line that has the specified angle. As a reference line, choose [X-axis on work plane] or [Reference line].
[Layer 11]



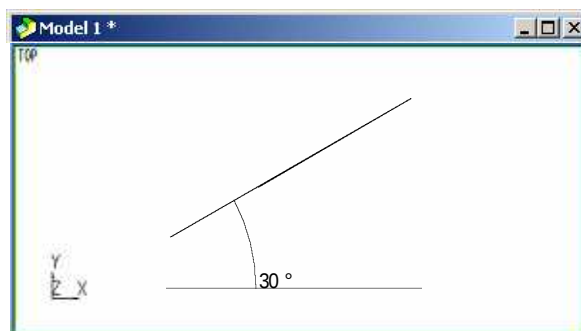
< Command sheet >



- Angle: Input value for angle
- Plane: Specify the plane to create a line.

< Operation >

<<Specify X-axis on work plane as reference line>>



Select the [Angle] command.

Specify parameters in the command sheet. Ex.)

Angle:30

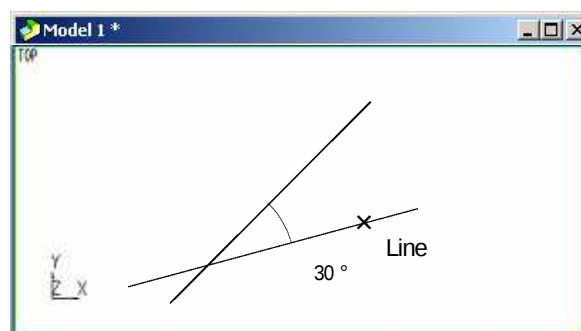
[Angle] Pick a reference line or enter the start point.

Click the coordinate of the start point.

[Angle] Enter the end point direction.

Click the coordinate of the end point.

<<Create a line crosses reference line at the specified angle>>



Select the [Angle] command.

Specify parameters in the command sheet. Ex.)

Angle:30

[Angle] Pick a reference line or enter the start point.

Click the reference line. (Line)

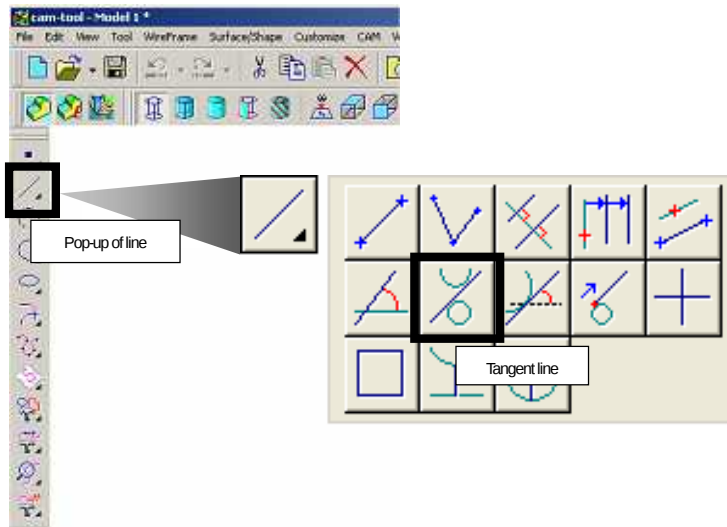
[Angle] Enter the start .

Click the coordinate of the start point.

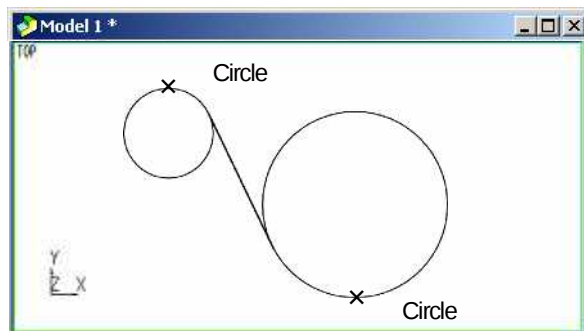
[Angle] Enter the end point direction.

Click the coordinate of the end point.

Create a line, which is tangent to the specified two entities. [Layer 12]



< Operation >



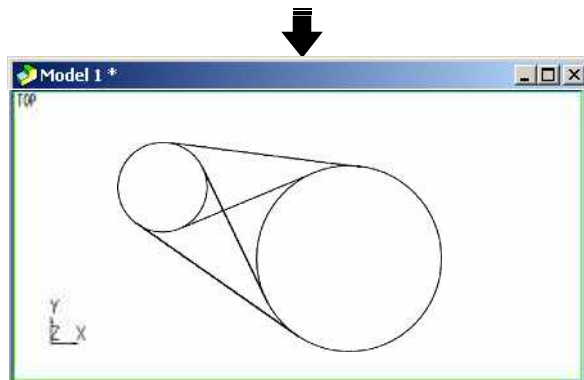
Select the [Tangent line] command.

[Tangent line] Pick entities for the first group.

Click the first entity. (Circle)

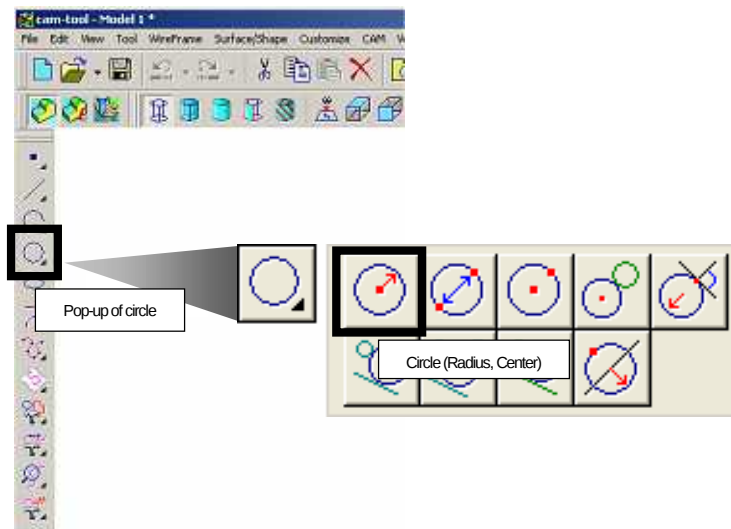
[Tangent line] Pick the second entity or enter the coordinates.

Click the second entity. (Circle)

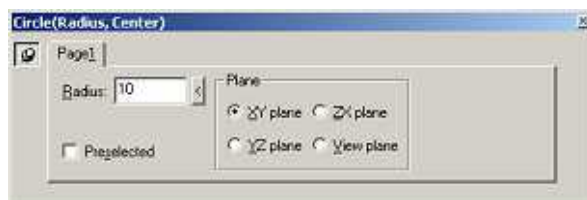


* Tangent line is created differently depending on the specified position.

Create a circle by specifying radius and coordinate of center point. [Layer13]



< Command sheet >

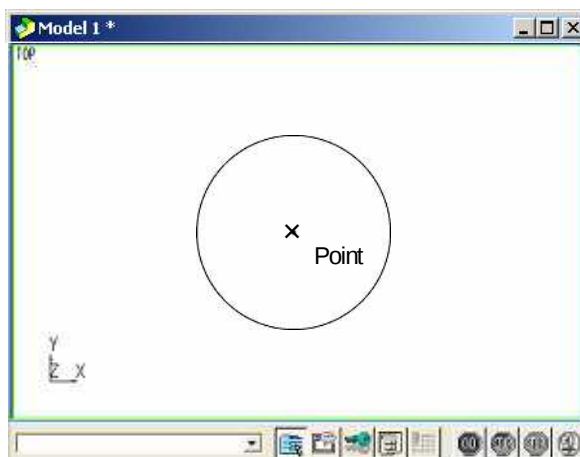


- Radius: Input value for radius of circle.
- Plane: Specify the plane to create a circle.
- Pre-selected

Exist on the plane, multiple circles of the same radius

are created at the selected center point simultaneously.

< Operation >



Select the [Circle (Radius, Center)] command.

Specify parameters in the command sheet.

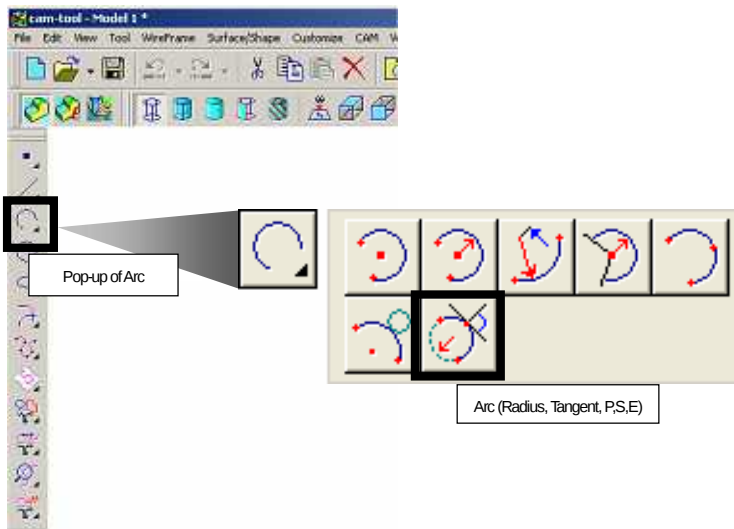
Ex.) Radius: 50

[Circle (Radius, Center)] Specify a center point.

Input value for the center point, or click a certain position on the screen. (Point)

Create an arc of the specified radius and angle, that is tangent to the specified point. Arc is created on work plane.

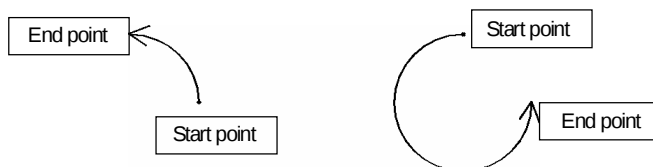
[Layer14]



■ Start point/End point of arc

Arc is basically created in the [Counterclockwise direction].

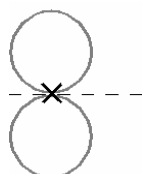
Arc is created differently depending on the start/end position.



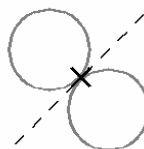
■ What is tangent angle?

It is the angle at which arc(circle) is tangent to a point. Arc(Circle) which has the same [Tangent point]/[Tangent angle] can exist in the two directions. The direction which arc(circle) is created, is determined by the position of the start point.

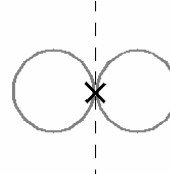
Tangent angle: 0



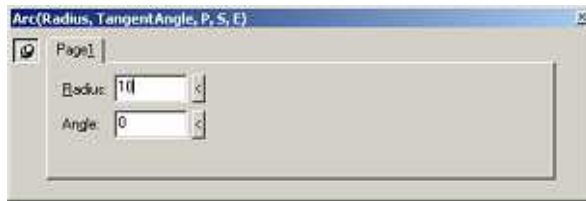
Tangent angle: 45



Tangent angle: 90

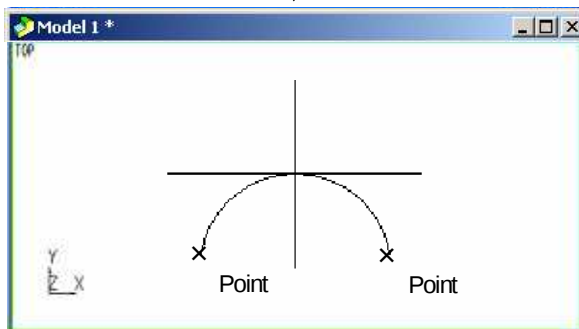
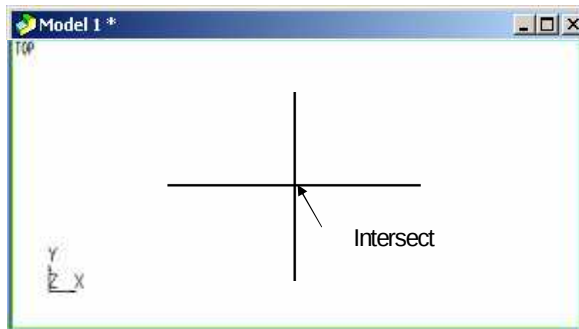


< Command sheet >



- Radius: Input value for radius.
- Angle: Input value for tangent angle.

< Operation >



Select the [Arc (Radius, Tangent line, P,S,E)] command.

Specify parameters in the command sheet.

Ex.) Radius: 30

Angle: 0

[Arc(Radius, ...) Enter the tangent point.

Input coordinate for the tangent point, or click a certain position on the screen.

[Arc(Radius, ...) Enter the start point direction.

Click the coordinate for the start point. (Point)

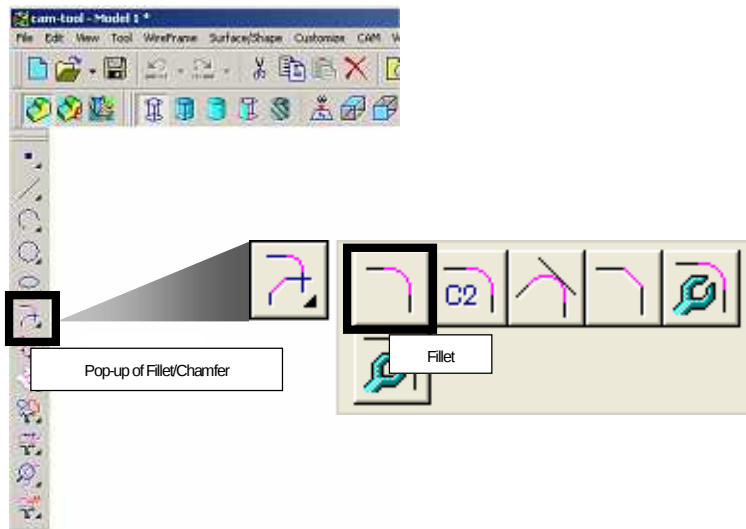
[Arc(Radius, ...) Enter the end point direction.

Click the coordinate for the end point. (Point)

* Specify start / end point in the counterclockwise direction.

How to create fillet < Fillet >

Create a fillet tangent to the two entities. [Layer15]

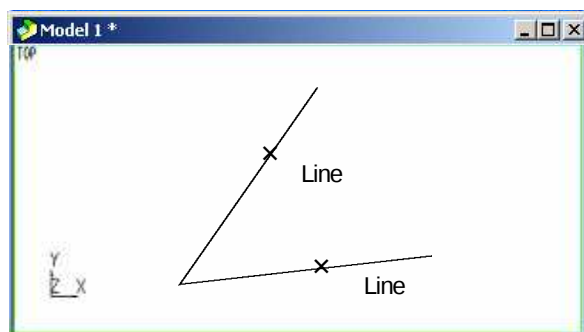


< Command sheet >



- Trim
 - Both: Trim both entities.
 - First: Trim the first entity.
 - Second: Trim the second entity.
 - No trim: No trim.
- Trim circle
 - Select whether trim or not trim circle.

< Operation >



Select the [Fillet] command.
Specify parameters in the command sheet.

Ex.) Radius: 10

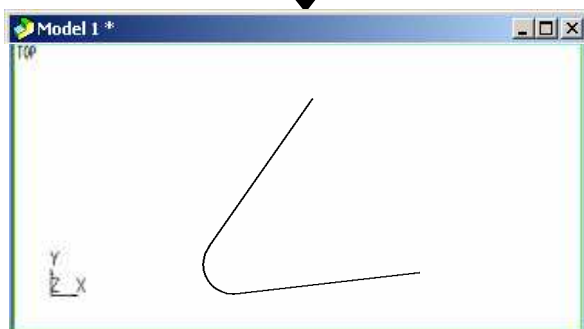
Trim: Both

[Fillet] Pick the first tangent entity or enter the coordinates.

Click the first entity. (Line)

[Fillet] Pick the second tangent entity or enter the coordinates.

Click the second entity. (Line)



Chapter 4

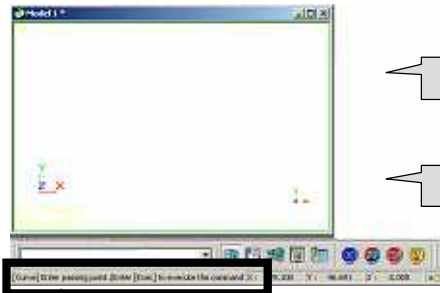
Edit Wire frame

Open the model file below. (Same model file as in [Chapter3])

CD-ROM/ Intensive Course/CADmodel2.gmd

- If [Layer number] is written in the description, change the active layer number.
- Follow the [command message] displayed at the lower left of the screen, for your operation.

Ex.) Create a [Single] line by inputting [absolute coordinate value]. [Layer1] ← Layer number



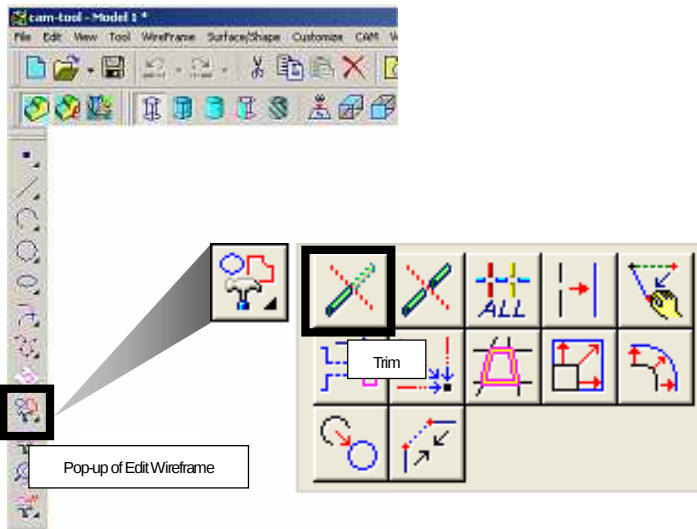
[Curve] Specify the start.
Left-click at the coordinate of start.

[Curve]Enter passing point.
Left-click at the coordinate of end.

Command message

How to cut/lengthen entity <Trim>

Cut/lengthen an entity with other entity or coordinate.



How to cut entity at the intersection of two entities

How to lengthen entity to the intersection of the entity and the other entity

How to cut entity using the other two entities

How to cut center part of entity using the other two entities

How to cut circle

How to cut /lengthen entity, when intersection does not exist

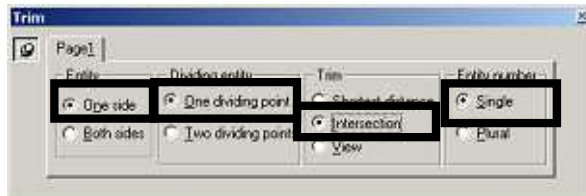
How to cut /lengthen entity when intersection does not exist on the same plane

How to cut /lengthen plural entities simultaneously

How to trim entity at the intersection of two entities (Dividing entity: One dividing point)

Trim (Cut) entity at [Intersection] of the entity and the reference entity. [Layer20]

<Operation>



Select the [Trim] command.

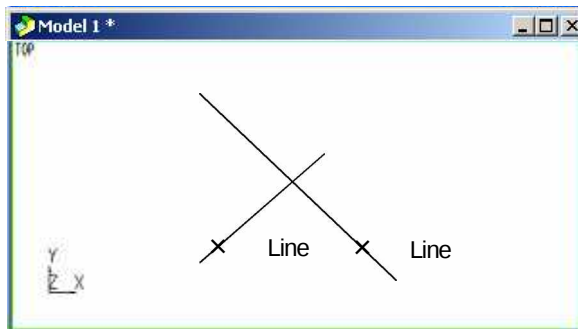
Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

Entity number: Single

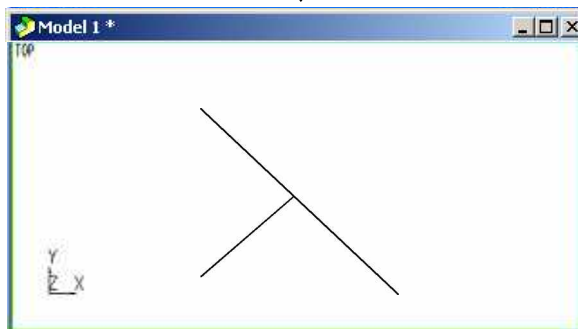


[Trim] Pick an entity to be trimmed.

Click the entity to be left. (Line)

[Trim] Pick a reference entity.

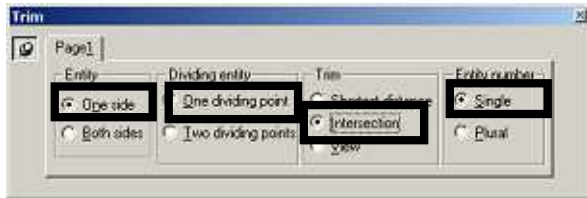
Click the reference entity. (Line)



How to lengthen entity to the intersection of the entity and the other entity (Dividing entity: One dividing point)

Trim (Lengthen) entity to [Intersection] of the entity and reference entity. [Layer21]

<Operation>



Select the [Trim] command.

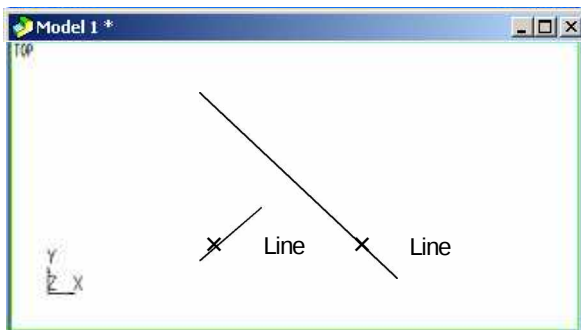
Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

Entity number: Single

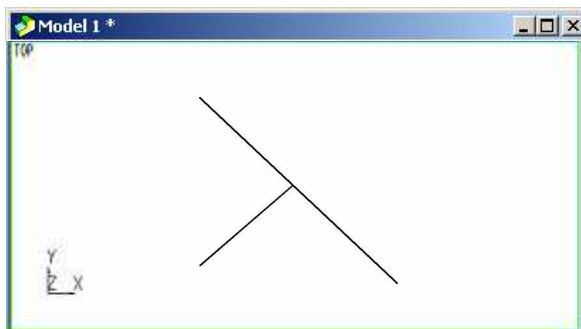


[Trim] Pick an entity to be trimmed.

Click the entity to be trimmed. (Line)

[Trim] Pick a reference entity.

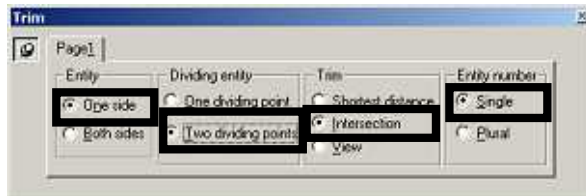
Click the reference entity. (Line)



How to cut entity using the other two entities (Dividing entity: Two dividing points)

Trim (Cut) entity at [Intersection] of the entity and the reference entity. [Layer21]

<Operation>



Select the [Trim] command.

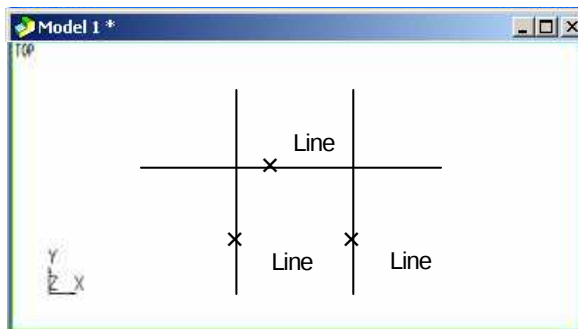
Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

Entity number: Single

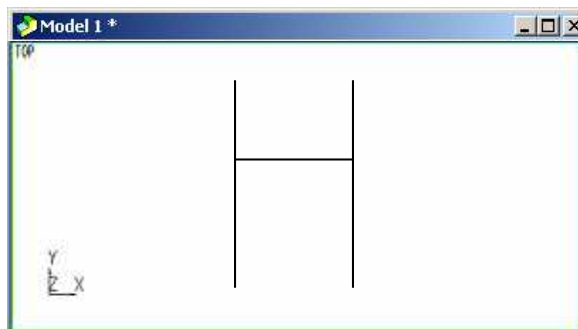


[Trim] Pick an entity to be trimmed.

Click the entity to be left. (Line)

[Trim] Pick the first reference entity.

Click the first reference entity. (Line)



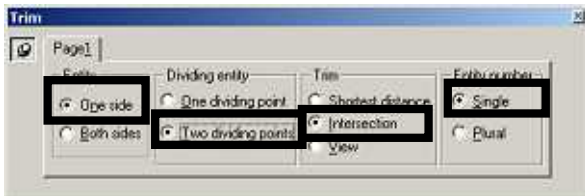
[Trim] Pick the second reference entity.

Click the second reference entity. (Line)

How to cut center part of entity using the other two entities (Dividing entity: Two dividing points)

Cut center part of an entity using the other two entities. [Layer23]

<Operation>



Select the [Trim] command.

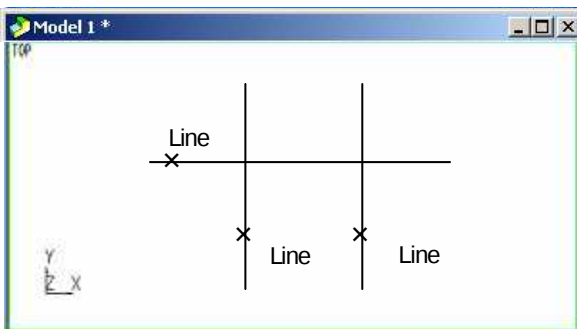
Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

Entity number: Single

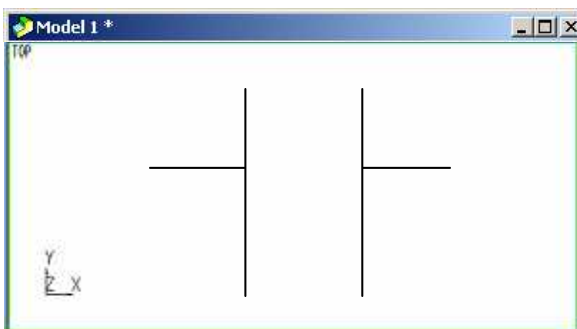


[Trim] Pick an entity to be trimmed.

Click the entity to be left. (Line)

[Trim] Pick the first reference entity.

Click the first reference entity. (Line)



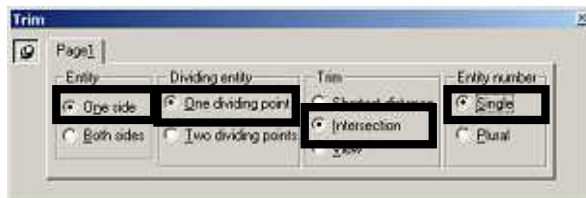
[Trim] Pick the second reference entity.

Click the second reference entity. (Line)

How to cut circle (Dividing entity: Two dividing points)

Circle cannot be cut at a single position. For [Dividing entity], [Two dividing point] is automatically set. [Layer 24]

<Operation>



Select the [Trim] command.

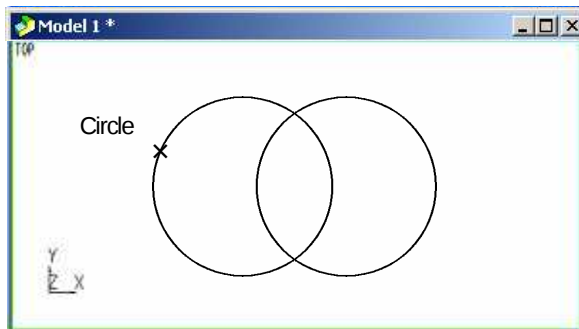
Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

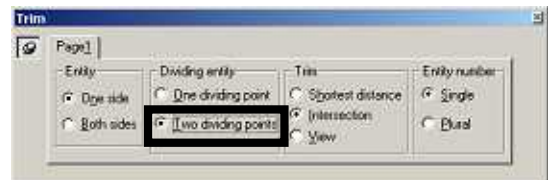
Entity number: Single



[Trim] Pick an entity to be trimmed.

Click the entity to be left. (Circle)

* [Two dividing points] is automatically set.

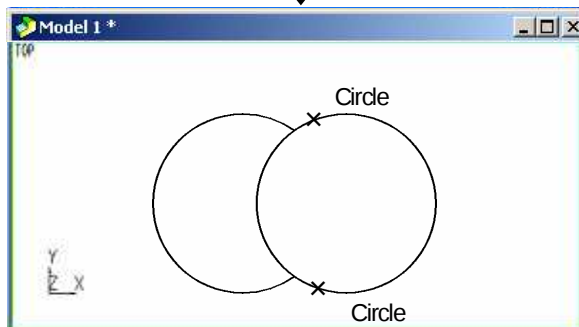


[Trim] Pick the first reference entity.

Click the first reference entity. (Circle)

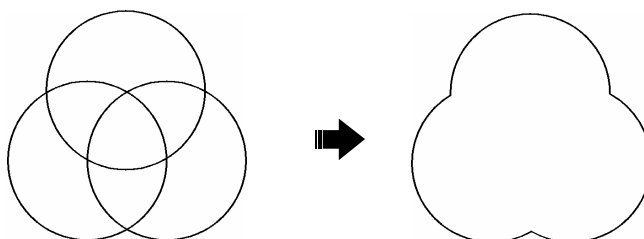
[Trim] Pick the second reference entity.

Click the second reference entity. (Circle)



<<Exercise>>

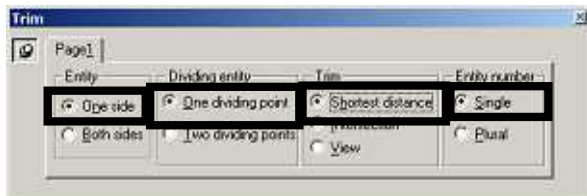
Trim the left shape so as to be the right shape.



How to cut /lengthen entity, when intersection does not exist (Trim: Shortest distance)

Cut/lengthen an entity at the shortest position from the specified coordinate. Entity can be cut/lengthened at/to proper position. [Layer 25]

<Operation>



Select the [Trim] command.

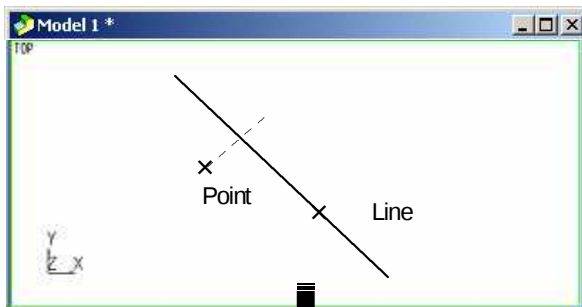
Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Shortest distance

Entity number: Single



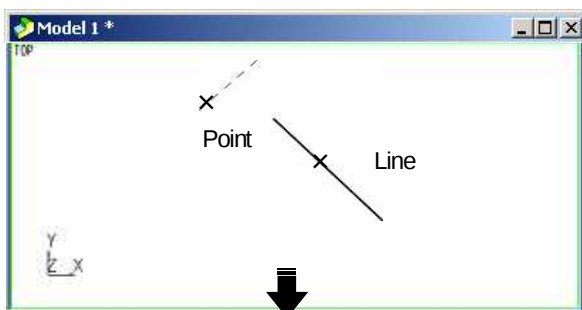
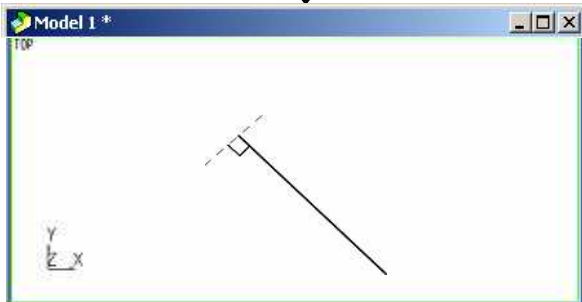
[Trim] Pick an entity to be trimmed.

Click the entity to be left. (Line)

[Trim] Pick a reference entity or enter the coordinate.

Click the coordinate. (Point)

* Entity is cut at the shortest position from the specified coordinate.



Select the [Trim] command.

Specify parameters in the command sheet.(Same as above)

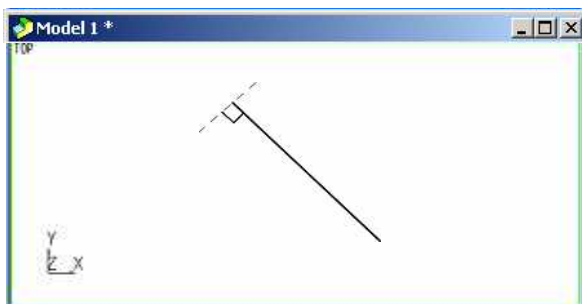
[Trim] Pick an entity to be trimmed.

Click the entity to be left. (Line)

[Trim] Pick a reference entity or enter the coordinate.

Click the coordinate. (Point)

* Entity is lengthened to the shortest position from the specified coordinate.

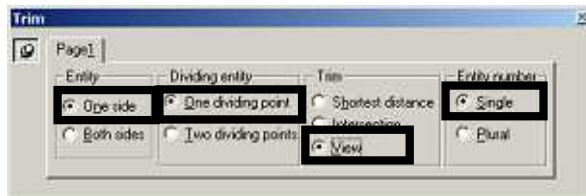


How to cut /lengthen entity when intersection does not exist on the same plane (Trim: View)

Cut/lengthen entity at/to [intersection in the view direction] [Layer 26]

* See the entity of [Layer 26] in the [View direction (ISO)], and confirm that entities does not exist on the same plane.

<Operation>



Select the [Trim] command.

Specify parameters in the command sheet.

Entity: One side

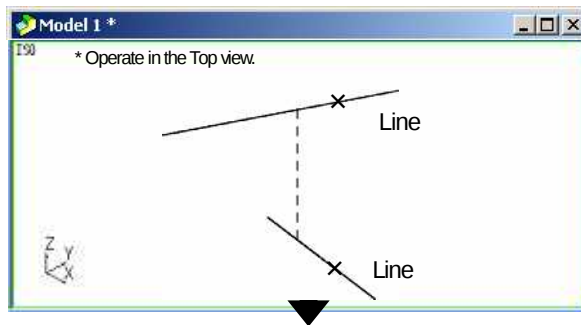
Dividing entity: One dividing point

Trim: View

Entity number: Single

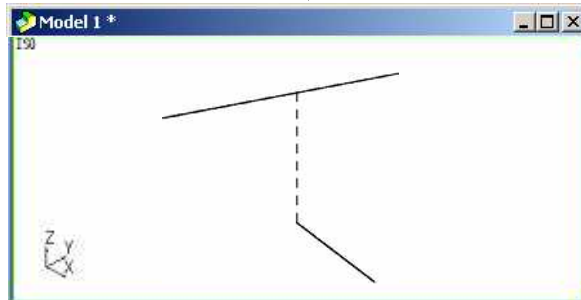
Set the view direction to execute [Trim] command.

View direction: Top



[Trim] Pick an entity to be trimmed.

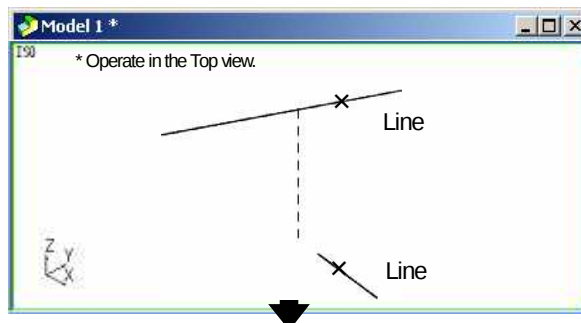
Click the entity to be left. (Line)



[Trim] Pick a reference entity or enter the coordinate.

Click the reference entity. (Line)

*Entity is cut at intersection in the set view direction (at the point which looks intersection visually).



Select the [Trim] command.

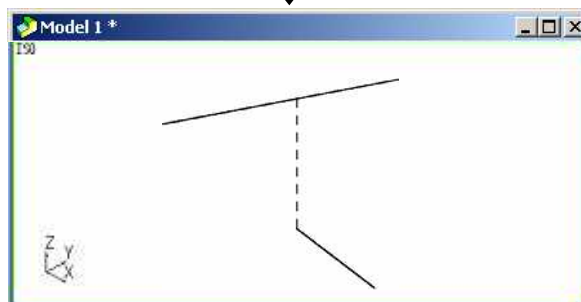
Specify parameters in the command sheet.(Same as above)

Set the view direction to execute [Trim] command.

View direction: Top

[Trim] Pick an entity to be trimmed.

Click the entity to be left. (Line)



[Trim] Pick a reference entity or enter the coordinate.

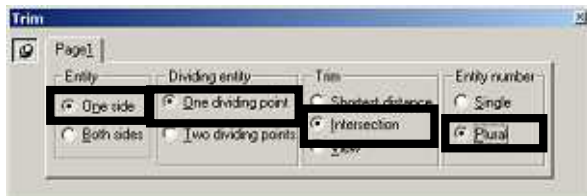
Click the reference entity. (Line)

*Entity is lengthened to intersection in the set view direction (at the point which looks intersection visually).

How to cut /lengthen plural entities simultaneously (Entity number: Plural)

[Plural entities] can be selected as trim entity. [Layer 27]

<Operation>



Select the [Trim] command.

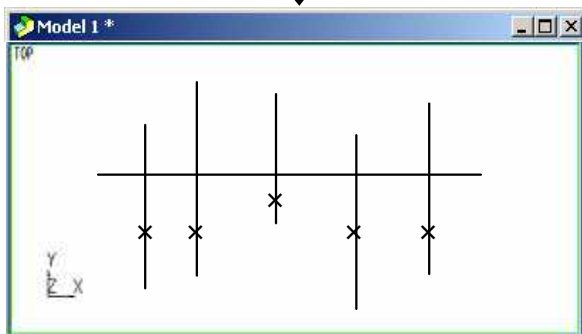
Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

Entity number: Plural

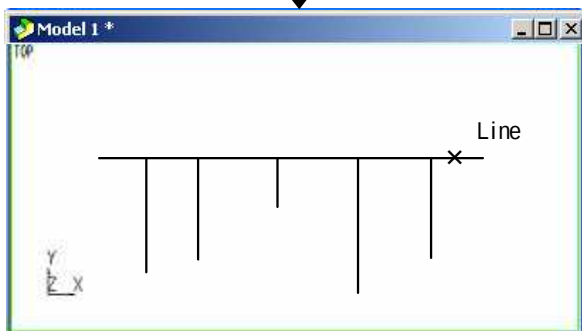


[Trim] Pick an entity to be trimmed.

- Click the entities to be left. (Line)

Press the center mouse button to determine the specifying.

- * When [Plural] is set in the command sheet, press the center mouse button to finish selecting entities.

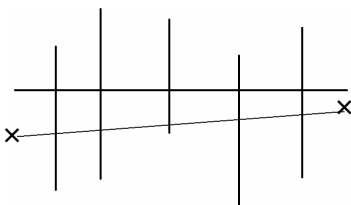


[Trim] Pick a reference entity.

Click the reference entity. (Line)



When [Entity number: Plural] is set, entities can be selected using [Specify by area: Line].



[Trim] Pick an entity to be trimmed.

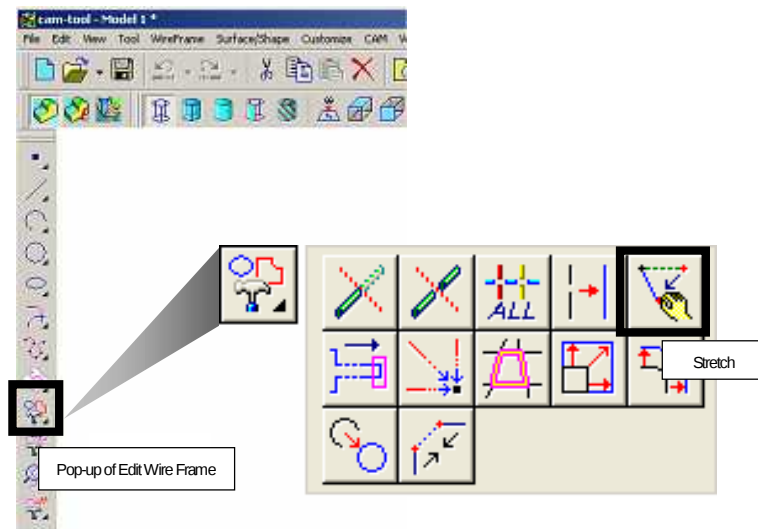
- Specify entities to be left by [Specify by area].



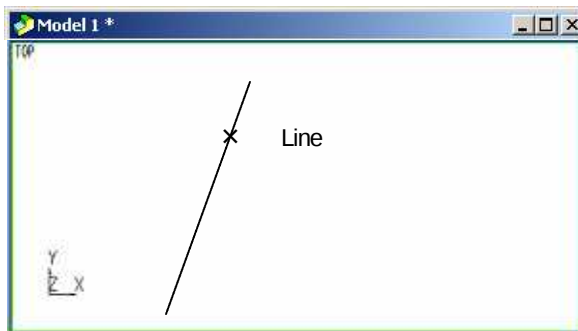
Settings in the command sheet will be kept until you close the application. To prevent mistakes from occurring in the next operation, restore the default settings before you end the command.

How to stretch entity < Stretch >

Move the [end point] of the specified entity to the specified coordinate.



<Operation>



Select the [Stretch] command.

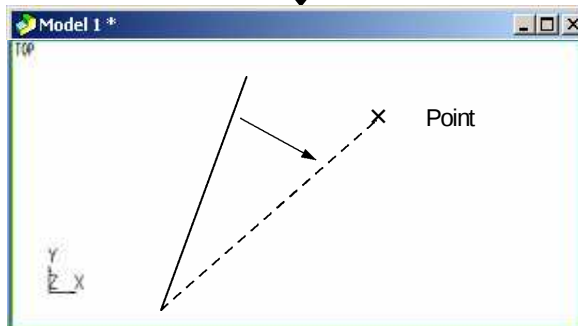
[Stretch] Pick an entity.

Click the entity. (Line)

* End point closer to the clicked point is moved.

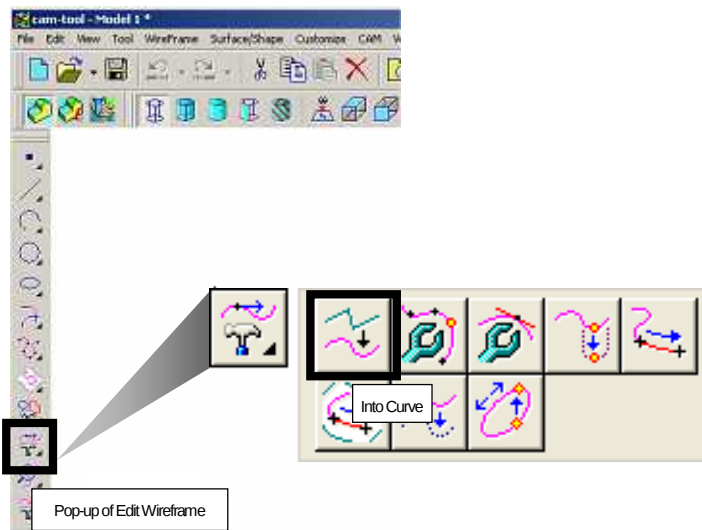
[Stretch] Specify coordinates.

Click the coordinate. (Point)



How to convert entities into a single entity < Into Curve >

Convert wire frames such as [Circle], [Arc], [Line] into [Curve]. If you put a group of entities to be converted in the selected status in advance, those entities can be converted into a [single curve].

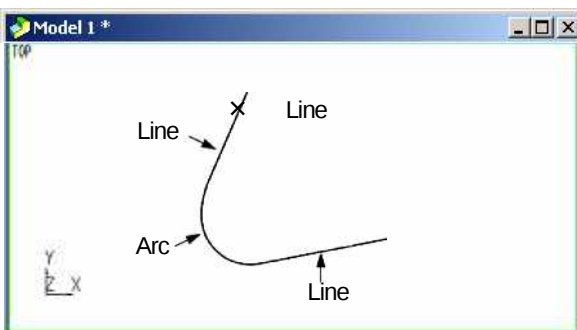


<Command sheet>



- Closed contour: Select open curve or closed curve
- Individual
ON: Convert each entity into curve
OFF: Convert the selected entities into curve.

<Operation>



Execute the [Select(Entity)] command.

[Select(Entity)] Pick an entity.

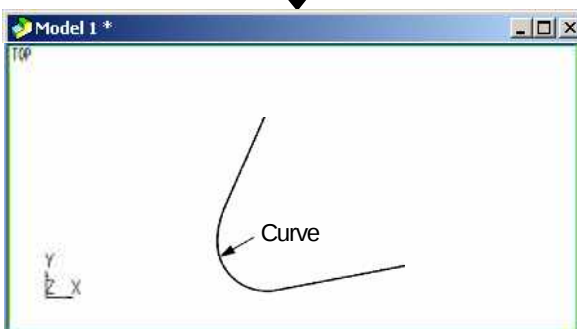
Right-click [Chain] >> Click the entity. (Line)

Select the [Into Curve] command.

You do not need to set parameters.

[Into Curve] (The entity is already selected) Pick an entity.

Press the center button to execute the command.

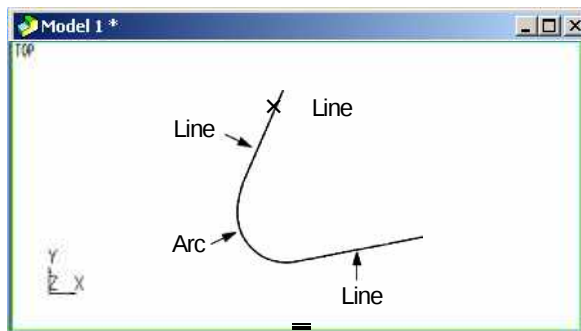


* [Plural entities] such as Line, Arc, Line, have been converted into a [single curve].

How to remove micro section in a curve

When a micro section exists in a curve, temporary mark is displayed at the position. If you execute CAM calculation with the data that includes a micro section, invalid data may be output. Therefore, execute [Re-form] command to remove the micro section before CAM calculation.

<Operation>

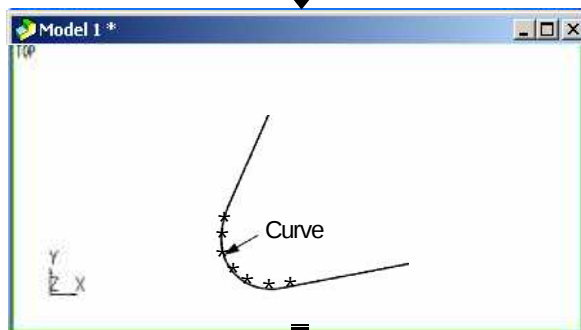


Execute the [Select(Entity)] command.

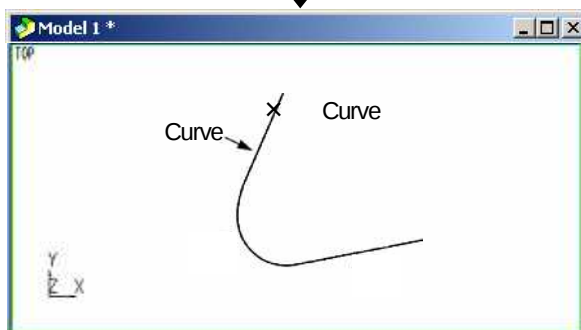
Right-click [Chain] >> Click the entity. (Line)

Select the [Into Curve] command.

Press the center button to execute the command.



Temporary mark is displayed at the position where a micro section exists.



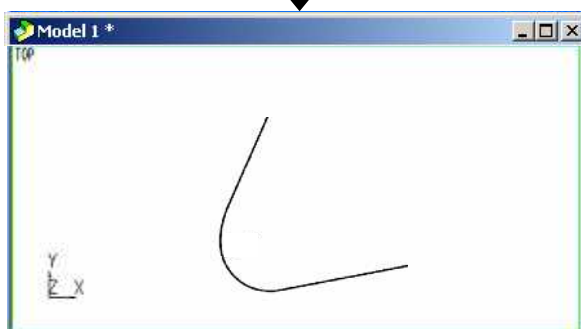
[Menu bar] >> [WireFrame] >> [Fair Curve] >>

[Re-Form]

Execute the [Re-form] command.

Click the entity. (Curve)

Press the center button to execute the command.



The micro section has been removed.

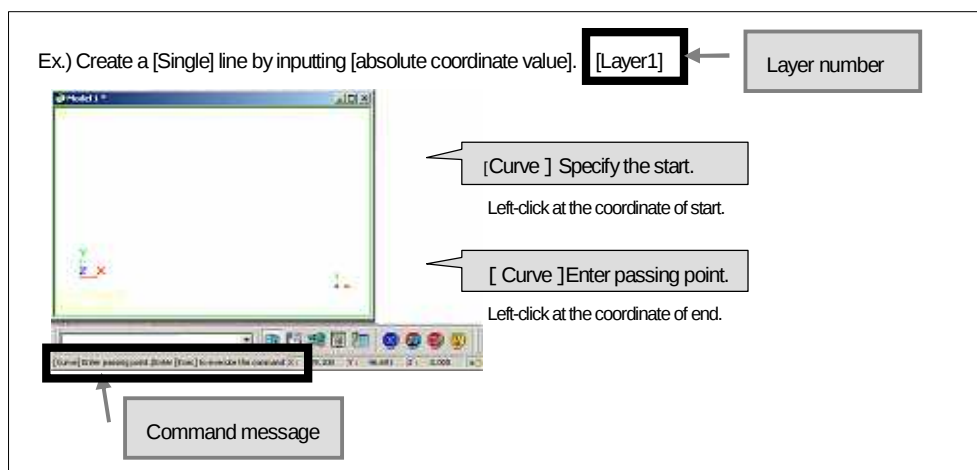
Chapter 5

Move/Copy

Open the model file below. (Same model file as in [Chapter3 - Chapter4])

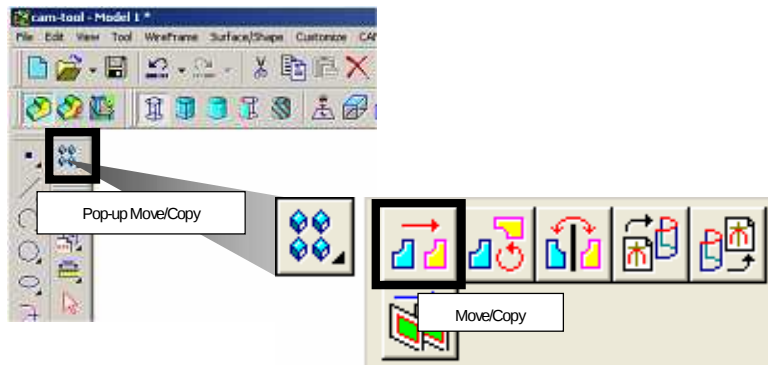
CD-ROM/ Intensive Course/CADmodel2.gmd

- If [Layer number] is written in the description, change the active layer number.
- Follow the [command message] displayed at the lower left of the screen, for your operation.

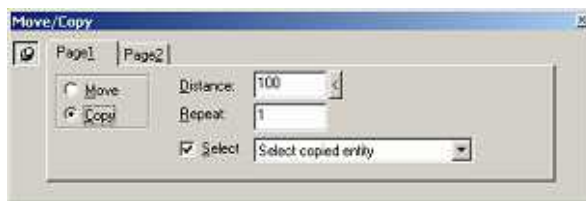


How to move/copy entity <Move/Copy>

Move/copy an entity to the specified position.



<Command sheet>

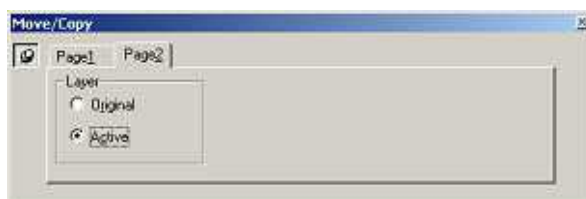


Move: Move an entity in parallel to the specified line.

Copy: Copy an entity in parallel to the specified line.

Distance: Input value for moving distance.

Repeat: Input number for moving.



Original:

Copy an entity in the layer, which the entity to be copied exists.

How to move/copy an entity in either of [X]/[Y]/[Z] direction

How to move/copy an entity by specifying two points

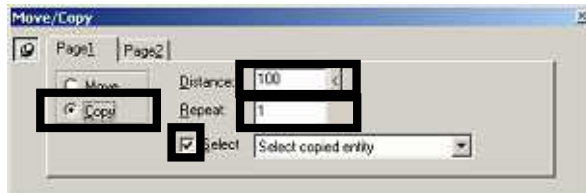
How to move/copy an entity by specifying moving distance

How to move/copy an entity by specifying a line as moving direction, or specifying line length as moving distance

How to move/copy an entity in either of [X]/[Y]/[Z] direction. (Use guide axis)

Ex.) Copy the entity in the Z-axis direction of work coordinate by [100mm]. [Layer30]

<Operation>



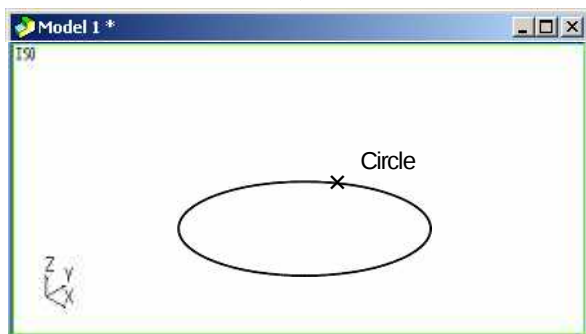
Select the [Move/Copy] command.
Specify parameters in the command sheet.

Move/Copy: Copy

Distance: 100

Repeat: 1

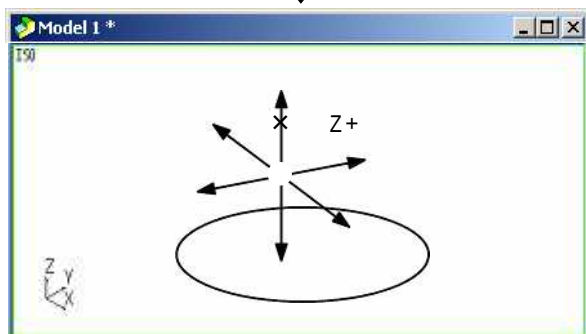
Select: OFF



[Move/Copy] Pick an entity.

Click the entity to be copied. (Circle)

Press the center mouse button to determine the specifying.



[Move/Copy] Pick a line to indicate the moving direction, or enter the origin.

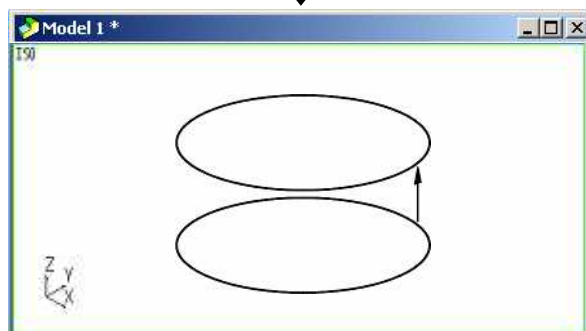
Click the moving origin (proper position).

Guide axes in the [X/Y/Z] direction are displayed.

[Move/Copy] Pick a direction entity to indicate the moving direction, or enter the end point.

Click the direction entity. (Z +)

Ex.) Z + axis direction

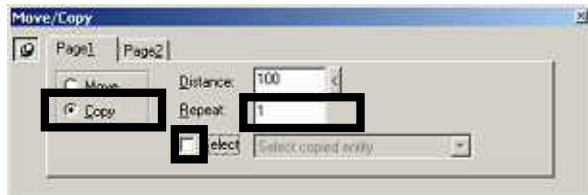


Press the center mouse button to execute the command.

How to move/copy an entity by specifying two points (Specify moving origin/moving end point)

Ex.) Copy the entity from the end point of the crossing line to the specified end point. [Lyer31]

<Operation>

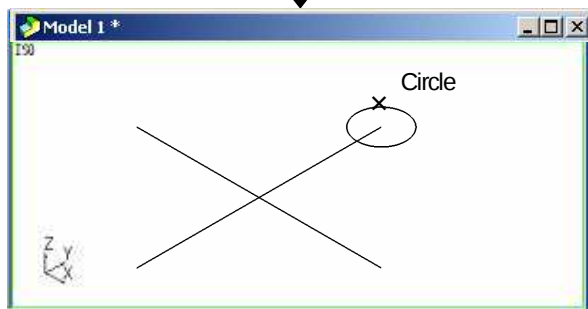


Select the [Move/Copy] command.
Specify parameters in the command sheet.

Move/Copy: Copy

Repeat: 1

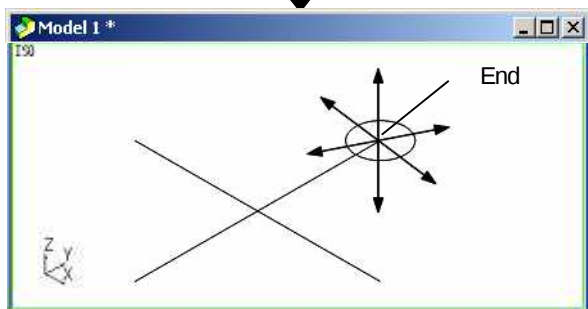
Select: OFF



[Move/Copy] Pick an entity.

Click the entity to be copied. (Circle)

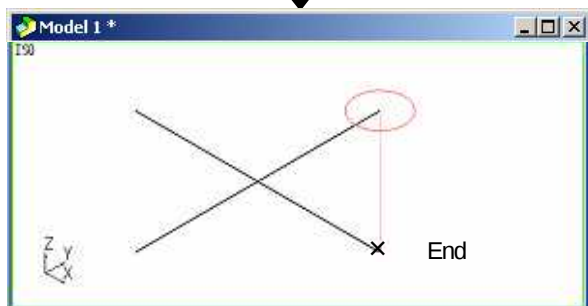
Press the center mouse button to determine the specifying.



[Move/Copy] Pick a line to indicate the moving direction, or enter the origin.

Click the moving origin (End).

Guide axes in the [X/Y/Z] direction are displayed.



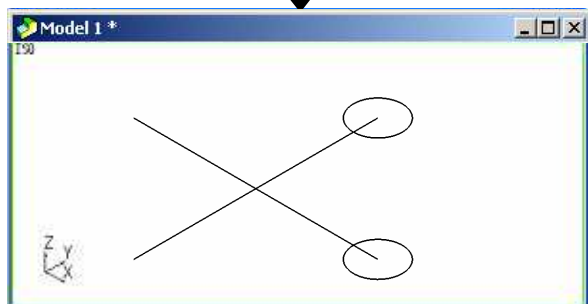
[Move/Copy] Pick a direction entity to indicate the moving direction, or enter the end point.

Click the moving end point.(End)

* Moving distance is input in the command sheet.



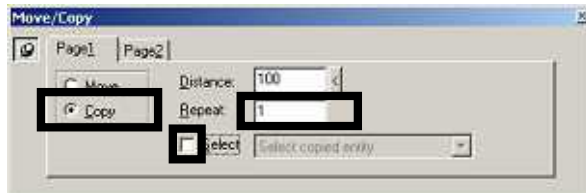
Press the center mouse button to execute the command.



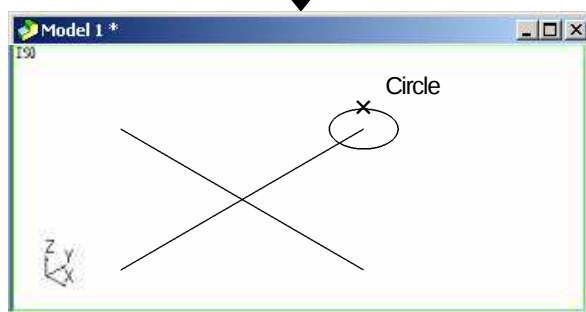
How to move/copy an entity by specifying moving distance(Specify moving origin/moving end point)

Ex.) Copy the entity in the X-axis direction of work coordinate by [50mm], in the Y-axis direction by [50mm]
[Layer32]

<Operation>

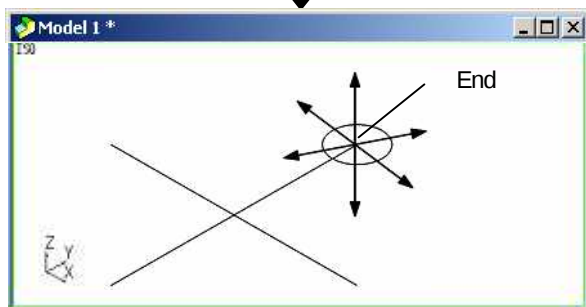


Select the [Move/Copy] command.
Specify parameters in the command sheet.
Move/Copy: Copy
Repeat: 1
Select: OFF



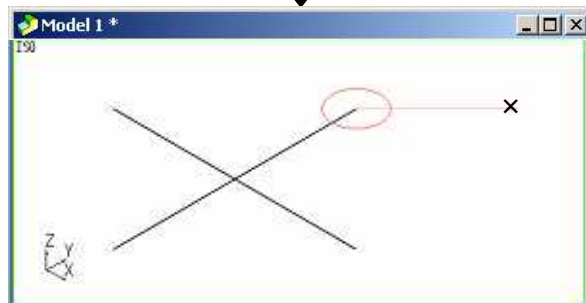
[Move/Copy] Pick an entity.

Click the entity to be copied. (Circle)
Press the center mouse button to determine the specifying.



[Move/Copy] Pick a line to indicate the moving direction, or enter the origin.

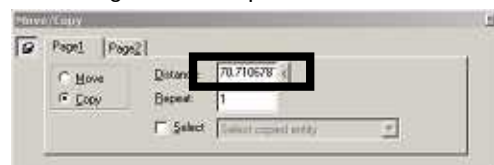
Click the moving origin (End).
Guide axes in the [X/Y/Z] direction are displayed.



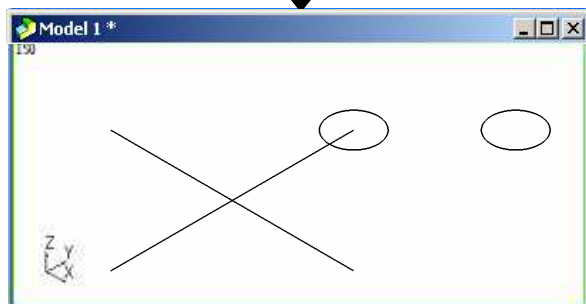
[Move/Copy] Pick a direction entity to indicate the moving direction, or enter the end point.

Input coordinate value for moving end point.
例) /50 , 50 , 0 <Enter>

* Moving distance is input in the command sheet.



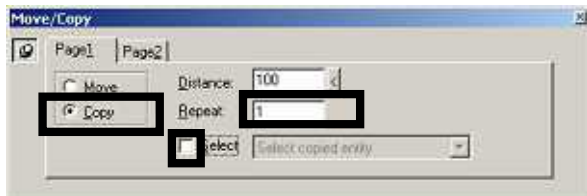
Press the center mouse button to execute the command.



How to move/copy an entity by specifying a line as moving direction/specifying line length as moving distance (Getting entity information of line)

Ex.) Copy the entity using the [parallel line to Y-axis] of [150mm]. [Layer33]

<Operation>

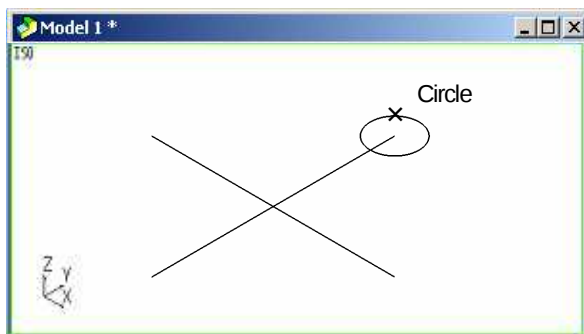


Select the [Move/Copy] command.
Specify parameters in the command sheet.

Move/Copy: Copy

Repeat: 1

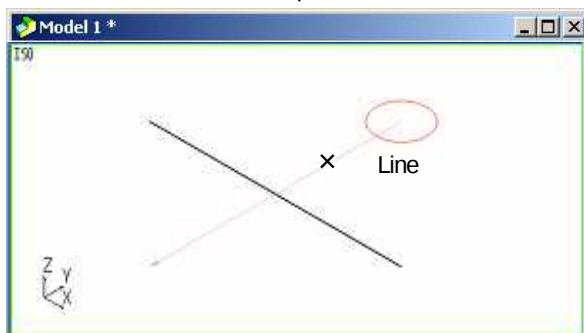
Select: OFF



[Move/Copy] Pick an entity.

Click the entity to be copied. (Circle)

Press the center mouse button to determine the specifying.

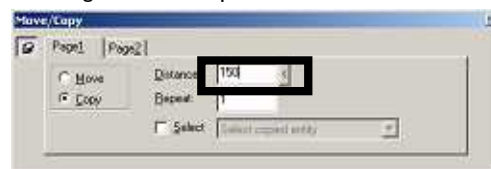


[Move/Copy] Pick a line to indicate the moving direction, or enter the origin.

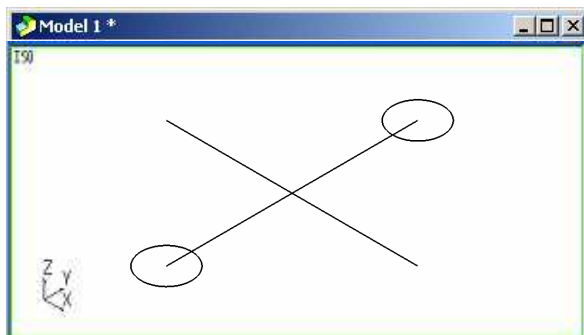
Click the line to indicate the moving direction. (Line)

- End point closer to the picked position is determined as moving origin.
- End point far from the picked position is determined as moving end point.

* Moving distance is input in the command sheet.



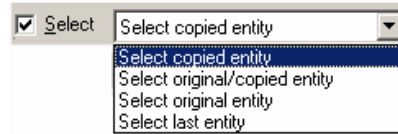
Press the center mouse button to execute the command.





[Select] function

Set the selecting status of entity when the command has been executed.



Select ON: Entity is in the selected status when the command has been executed.

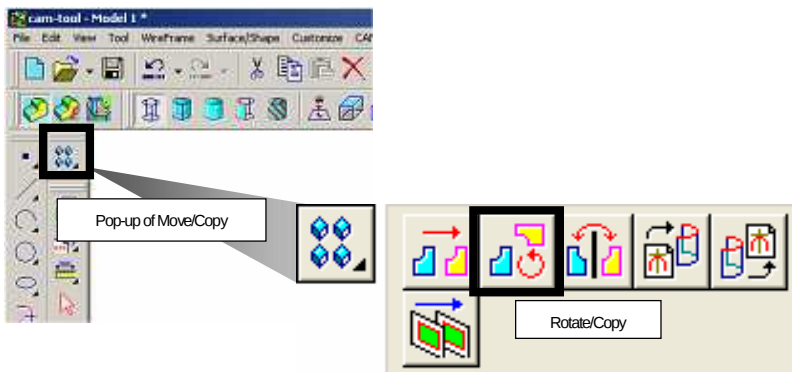
Select ONFF: Entity is not in the selected status when the command has been executed.

Selecting type (For copying)

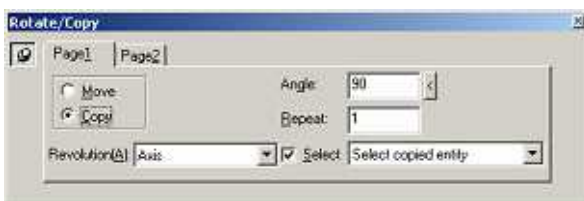
- Select copied entity: All copied entities are in the selected state.
- Select original/copied entity: Both original and copied entities are in the selected state.
- Select original entity: Only original entities are in the selected state.
- Select last entity: Only the last entity is in the selected state when copying is repeated.

How to rotate and move/copy entity <Rotate/Copy>

Rotate an entity and move/copy it.



<Command sheet>



Move: Rotate and move an entity.

Copy: Rotate and copy an entity.

Angle: Input value for rotating.

Repeat: Input value for repeating number.

Revolution: Select the method of specifying rotating axis.

- Axis: Specify axis and angle for rotating entity.
- Center and two points: Specify angle for rotating entity with coordinates of three points.

How to move/copy by specifying either of [X]/[Y]/[Z] axis as rotating axis

How to move/copy by specifying line as rotating axis

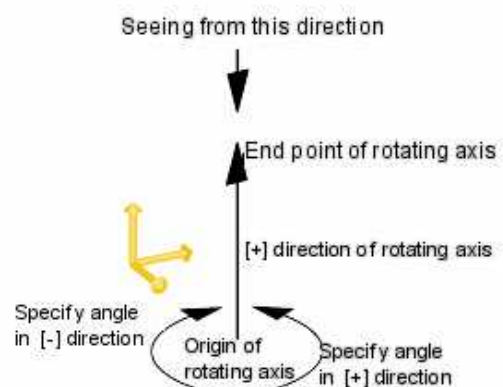
How to move/copy by specifying coordinates of three points



Rotating axis

Specify [rotating angle] seeing from the end point of rotating axis

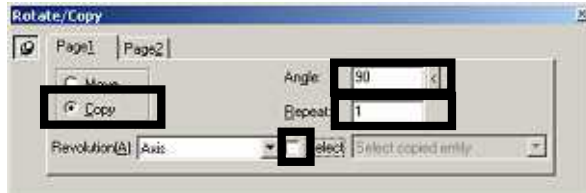
- Counterclockwise: Specify angle in the [+] direction
- Clockwise: Specify angle in the [-] direction



How to move/copy entity by specifying either of [X]/[Y]/[Z] axis as rotating axis (Using guide axis)

Ex.) Rotate an entity and copy it by specifying Z-axis or work coordinate as rotating axis by [90 degree]. [Layer34]

<Operation>



Select the [Rotate/Copy] command.

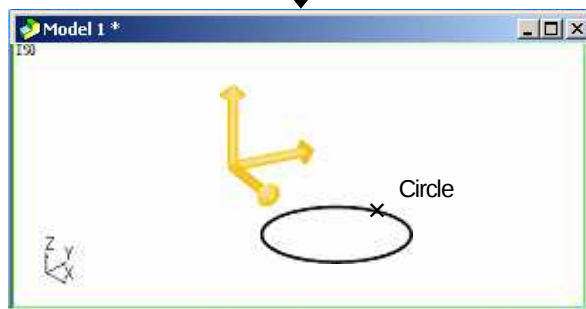
Specify parameters in the command sheet.

Move/Copy: Copy

Angle: 90

Repeat: 1

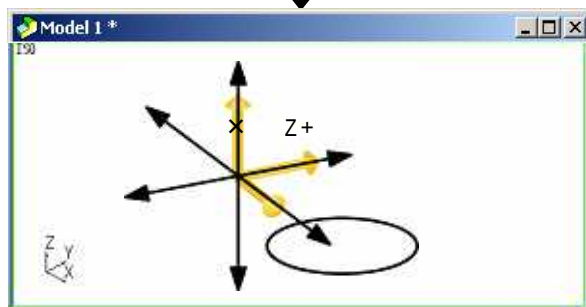
Select: OFF



[Rotate /Copy] Pick an entity.

Click the entity to be copied. (Circle)

Press the center mouse button to determine the specifying.

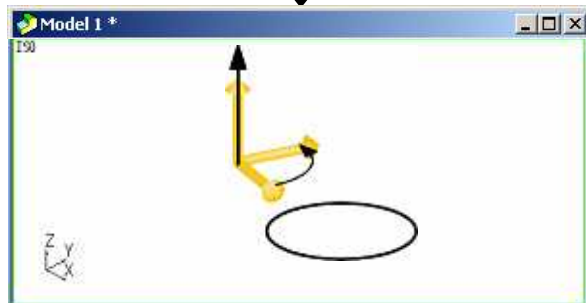


[Rotate /Copy] For axis of revolution, pick a straight line or enter the start of vector.

Input coordinate value for rotating origin.

Ex.) 0, 0, 0 <Enter>

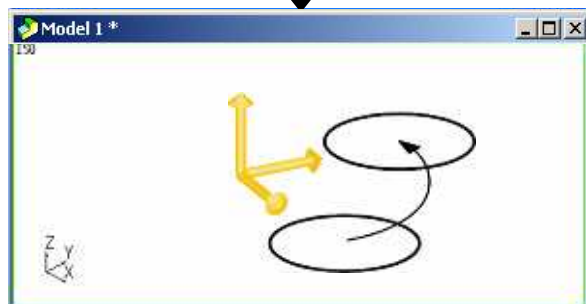
Guide axes in the [X/Y/Z] direction are displayed.



[Rotate /Copy] For axis of revolution, pick a straight line or enter the end of vector.

Click the direction entity. (Z+)

Ex.) Z + direction



Press the center mouse button to execute the command.

How to move/copy entity by specifying line as rotating axis (Using line)

Ex.) Rotate an entity and copy it by specifying line in Y-axis direction of work coordinate as rotating axis by [90 degree]. [Layer35]

<Operation>



Select the [Move/Copy] command.

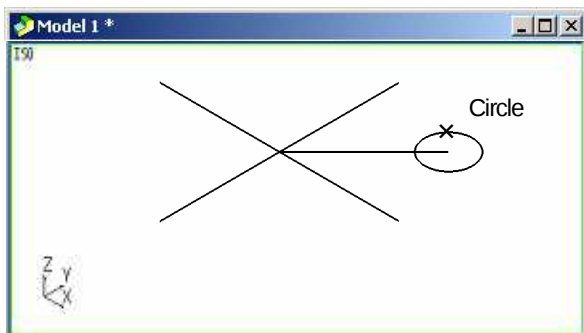
Specify parameters in the command sheet.

Move/Copy: Copy

Angle: 90

Repeat: 1

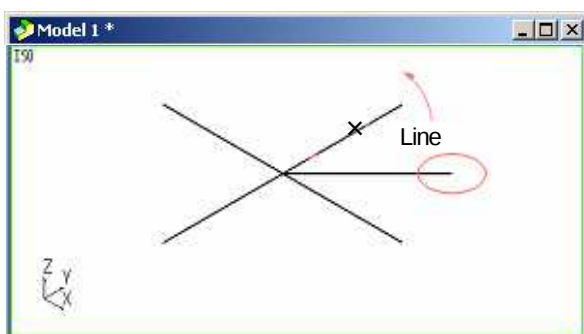
Select: OFF



[Rotate /Copy] Pick an entity.

Click the entity to be copied. (Circle)

Press the center mouse button to determine the specifying.

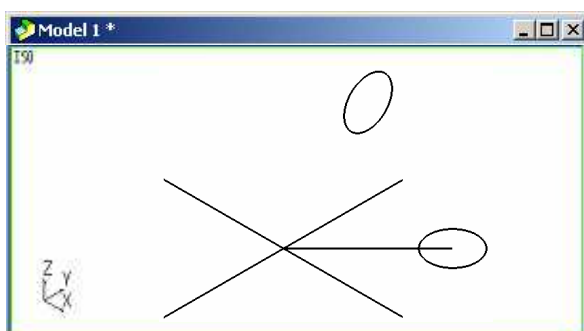


[Rotate /Copy] For axis of revolution, pick a straight line or enter the start of

Click the line as rotating origin.

- End point closer to the picked position is determined as origin of axis.

- End point closer to the picked position is determined as end point of axis.

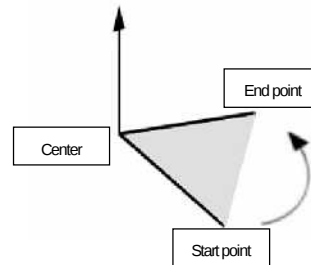


Press the center mouse button to execute the command.



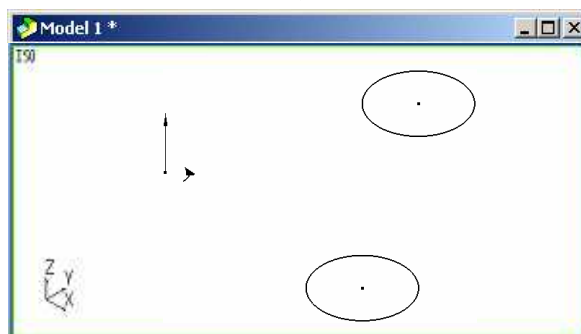
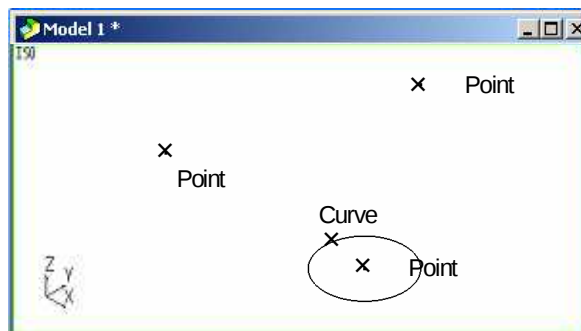
What is [center and two points] ? [Layer 36]

Determine [rotating axis]/[rotating angle] by specifying the coordinates of three points



- Determine the plane by specifying the coordinate of three points such as [origin], [start point of rotation], [end point of rotation]
- Set normal direction of [plane that is created with three points], which runs through the origin as [rotating axis].
- Set angle between [line created with origin and start point of rotation], and [line created with origin and end point of rotation]

<Operation>



Select the [Move/Copy] command.

Specify parameters in the command sheet.

Move/Copy: Copy

Axis: Center and 2 points

Repeat: 1, Select: OFF

[Rotate /Copy] Pick an entity.

Click the entity to be copied. (Curve)

Press the center mouse button to determine the specifying.

[Rotate /Copy] Enter the center of rotation.

Specify the coordinate of center of rotation. (Point)

[Rotate /Copy] Enter the start point of rotation.

Specify the coordinate of start point of rotation. (Point)

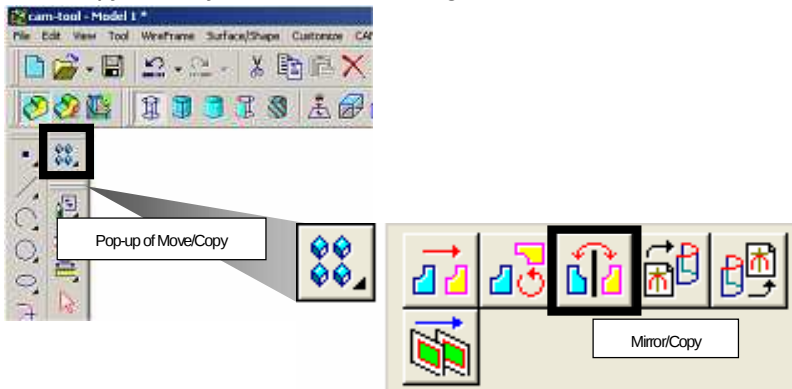
[Rotate /Copy] Enter the end point of rotation.

Specify the coordinate of end point of rotation. (Point)

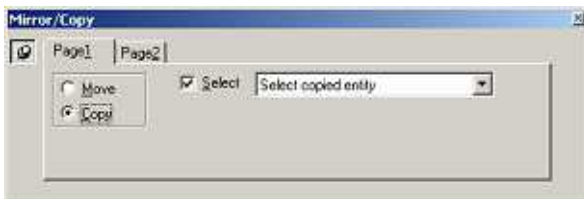
Press the center button to execute the command.

How to move/copy entity to the other side with mirror <Mirror/Copy>

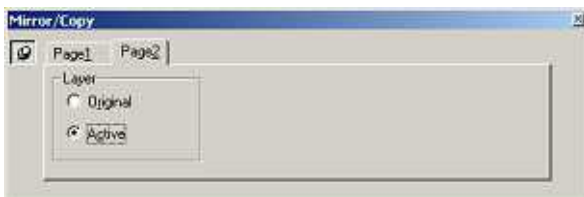
Move/copy an entity to the other side using mirror.



<Command sheet>



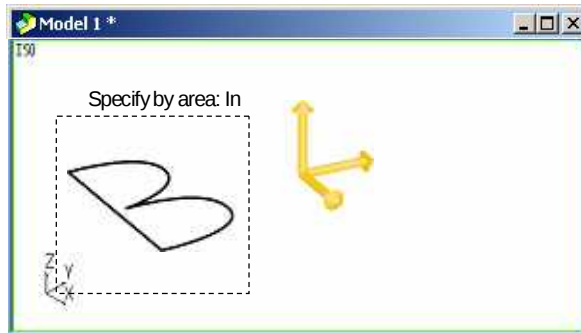
Move: Move the specified entity to the other side using mirror.



How to move/copy an entity to the other side with either of [XY plane]/[ZX plane]/[YZ plane] (Using guide plane)

Ex.) Copy the entity to the other side using [ZX plane] of work coordinate.

<Operation>



Select the [Mirror/Copy] command.
Specify parameters in the command sheet.

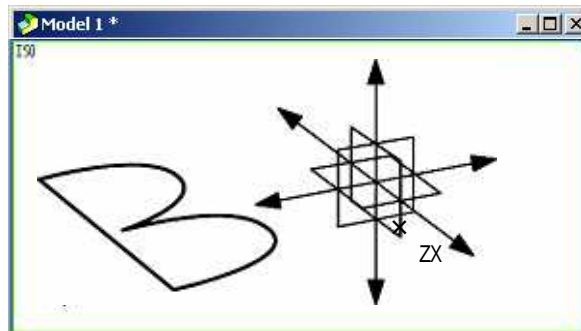
Move/Copy: Copy

Select: OFF

[Mirror /Copy] Pick an entity.

Click the entity to be copied. (Circle)

Press the center mouse button to determine the specifying.



[Mirror /Copy] Pick a plane to make the inversion..., or pick a line, or enter the origin.

Input coordinate value for origin of inverting plane.

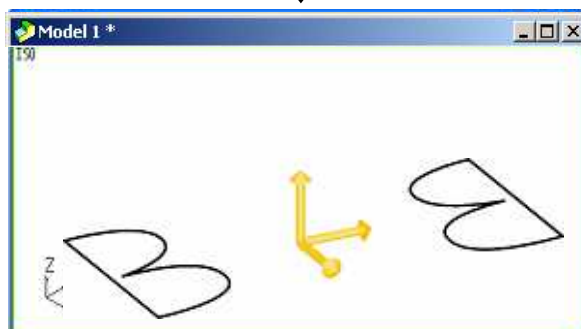
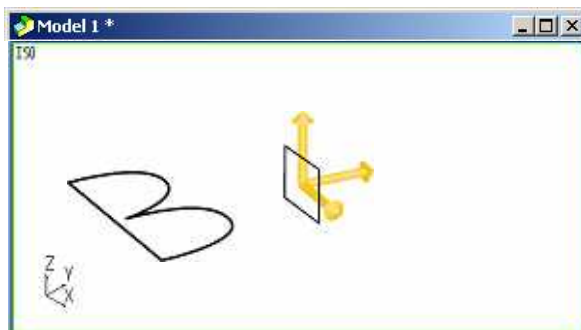
Ex.) <Enter>

Guide plane of [XY]/[ZX]/[YZ] direction are displayed.

Mirror /Copy] Pick a entity to make the inversion plane..., or enter the second coordinates.

Click the direction entity.

Ex.) ZX plane



Press the center mouse button to execute the command.

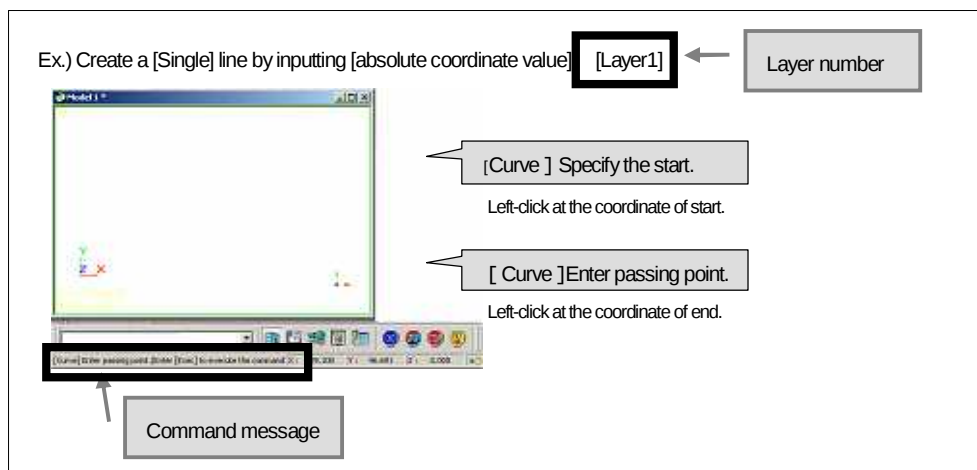
Chapter 6

Change origin/axis direction

Open the model file below. (Same model file as in [Chapter3 - Chapter5])

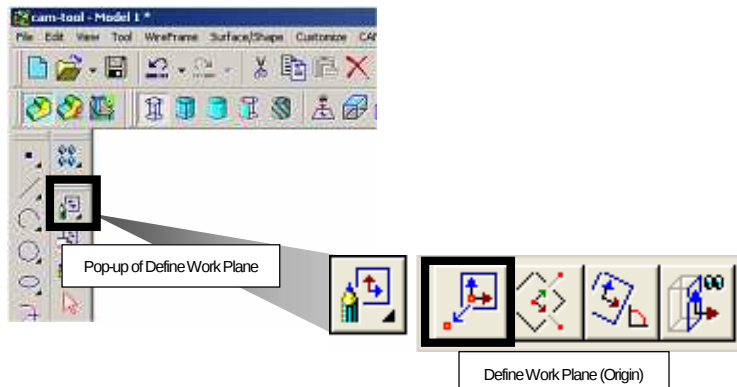
CD-ROM/Intensive Course/CADmodel2.gmd

- If [Layer number] is written in the description, change the active layer number.
- Follow the [command message] displayed at the lower left of the screen, for your operation.



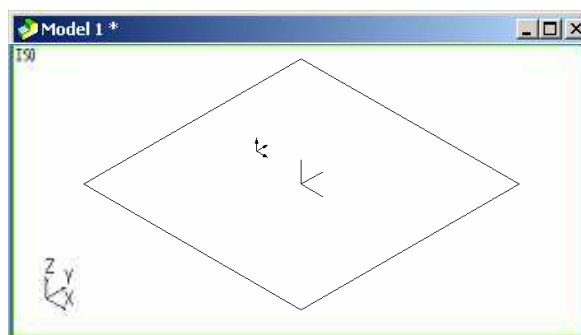
How to change coordinate of origin <Define Work Plane (Origin)>

Change the origin of work plane. The direction of [X-axis]/[Y-axis]/[Z-axis] is not changed. [Layer40]



Ex.) Change the origin of work coordinate to [X=50], [Y=50], [Z=50].

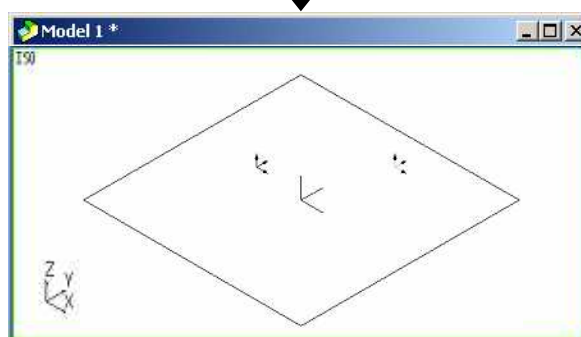
<Operation>



Select the [Define Work Plane(Origin)] command.

Work plane and coordinate axis is displayed by rubber band.

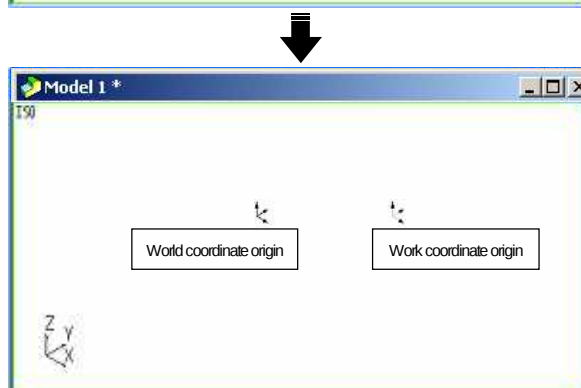
* If rubber band is not displayed clearly, zoom out the screen using [Zoom] command.



Input value for origin, or click the coordinate.

Ex.) Input the coordinate value.

50 , 50 , 0 <Enter>



The origin is changed.

World coordinate origin = Coordinate axis in blue

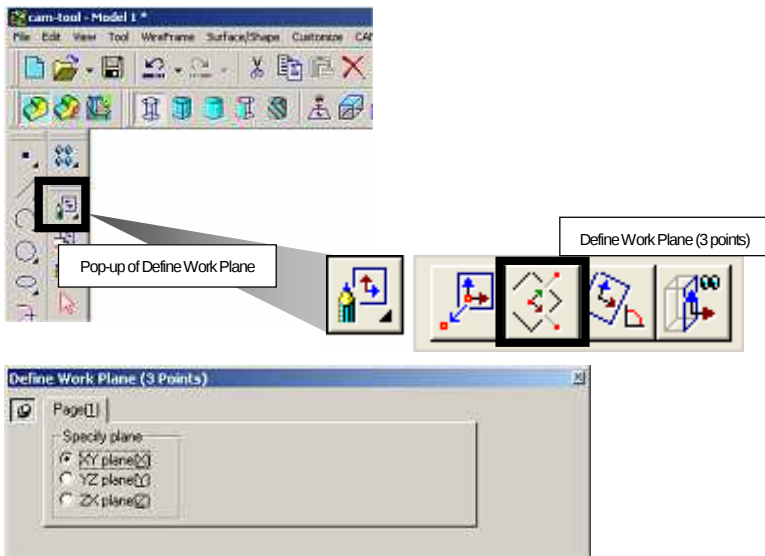
Work coordinate origin = Coordinate axis in Yellow

Click [Cancel] to end the command.

How to change X/Y/Z axis direction

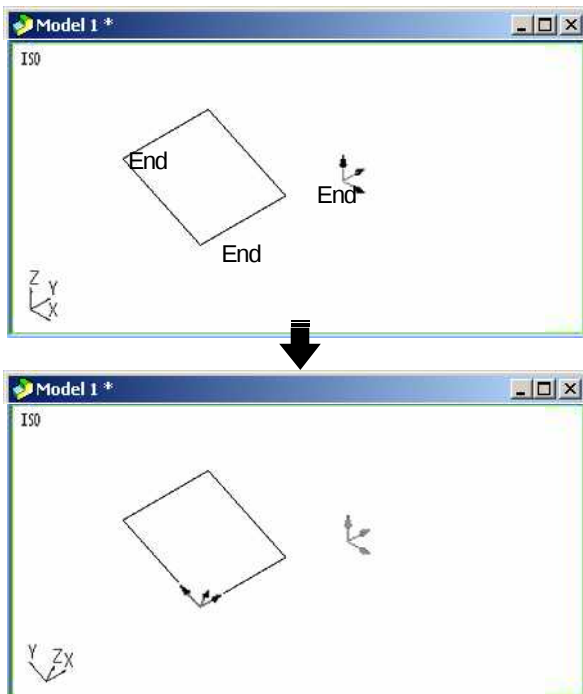
<Define Work Plane (3 points)>

Define work plane by specifying three points (Origin, X-axis direction, Y-axis direction).



Ex.) Change origin/axis-direction of work coordinate using [entity on the screen].

<Operation>



Select the [Define Work Plane(3 points)] command.

[Define Work Plane (3 points)] Enter origin, pick an entity..., or pick a straight line for X-axis.

Click the coordinate for origin. (End/Intersect)

[Define Work Plane (3 points)] Enter coordinates for..., or straight line for X-axis.

Click the coordinate for X-axis direction. (End/Intersect)

[Define Work Plane (3 points)] Enter coordinates for..., or straight line for Y-axis.

Click the coordinate for Y-axis direction. (End/Intersect)

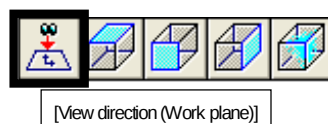
The origin/coordinate axis is changed.

Click [Cancel] to end the command.



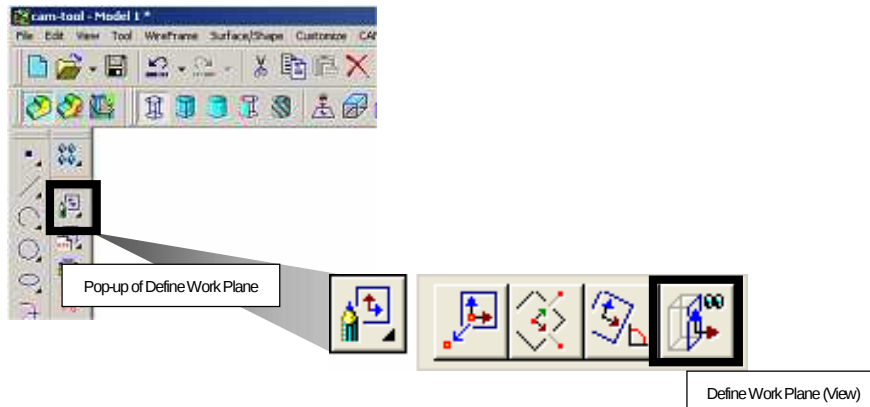
When you change axis direction, click the [View direction (Work plane)] to displayed work plane.

And view direction of work coordinate system is displayed by clicking [View direction] button pressing [Shift key].



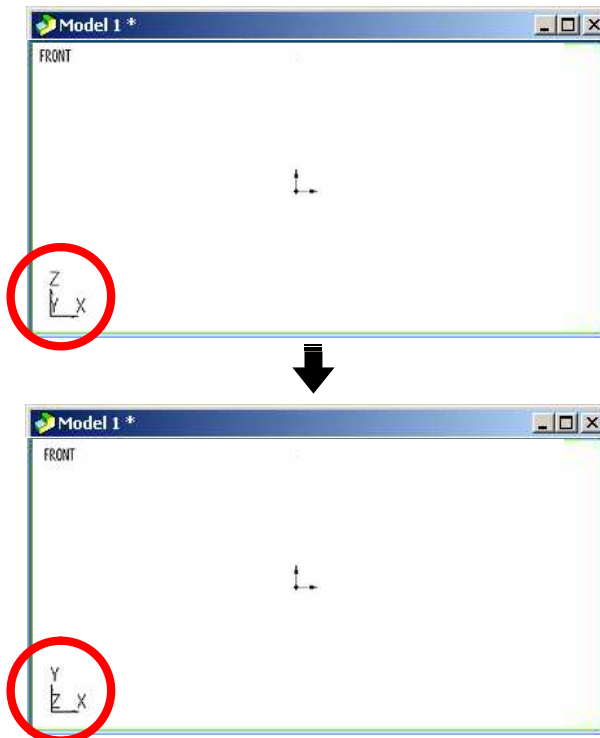
How set the current view direction as XY plane <Define Work Plane (View)>

Set the current active view direction as [Work plane (XY plane)]. [Layer42]



Ex.) Set the plane seeing from [View direction (FRONT)] as [XY plane].

<Operation>



Select the [View direction(FRONT)] command.

Select the [Define Work Plane(View)] command.

At the point when the command is selected, the command is executed.

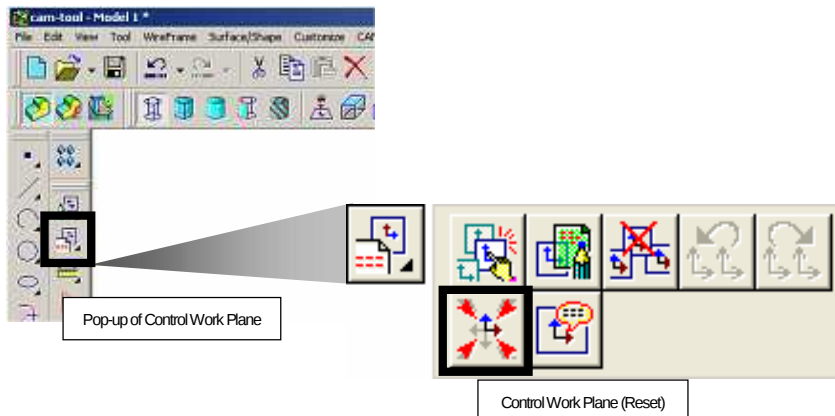
Axis-direction is changed.

Plane seeing from [View direction (FRONT)] (ZX plane) is defined as work plane. (XY plane)

How to restore the changed origin/axis direction

<Control Work Plane (Reset)>

Restore the defined [Work plane] to the initial setting. [Layer42] * In the initial state, [Work coordinate system] = [World coordinate system]



<Operation>



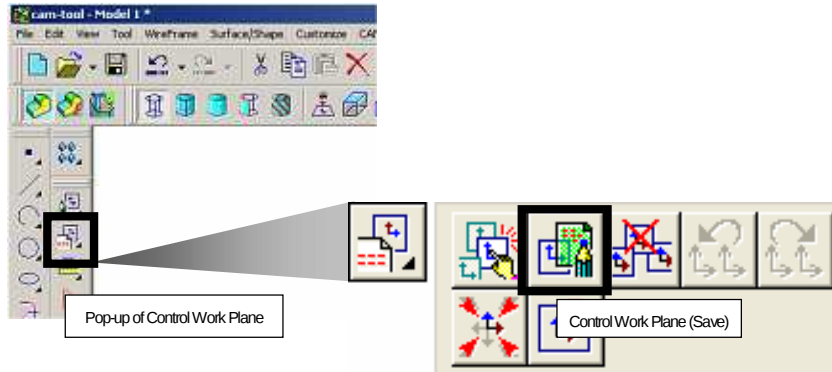
Select the [Control Work Plane(Reset)] command.

The defined origin/axis-direction is restored to the initial setting.

How to save the changed coordinate system

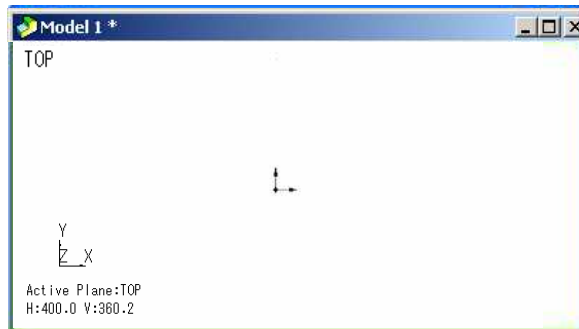
<Control Work Plane (Save)>

Register the current active [Work plane].



Ex.) Register [Work plane] that its origin/axis-direction has been changed.

<Operation>



Select the [Control Work Plane(Origin)] command.

[Define Work Plane (Origin)] Enter origin.

Input value for origin, or click the coordinate.

Ex.) 50 , 50 , 0 <Enter>

Select the [View direction(FRONT)] command.

Select the [Define Work Plane(View)] command.

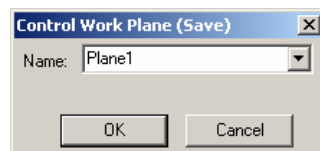
Select the [Control Work Plane(Save)] command.

[Control Work Plane (Save)] dialog is displayed.

Input plane name.

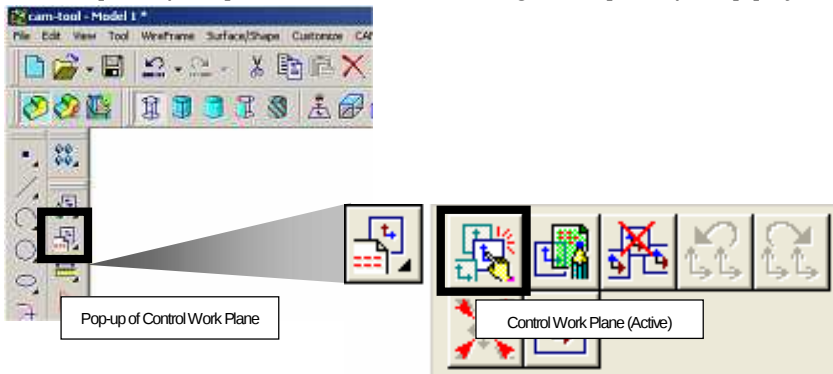
Ex.) Plane1

Click the [OK] button.



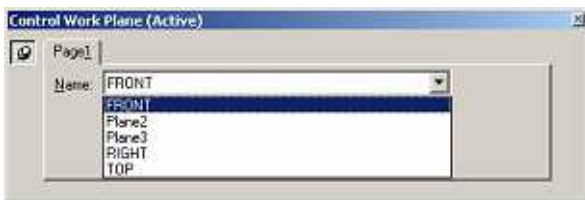
How to use the saved coordinate system <Control Work Plane (Active)>

Select a [Work plane] to be activated from the registered [Work plane]. [Layer42]



Ex.) Use the registered [Work plane2]

<Operation: Select from the list>



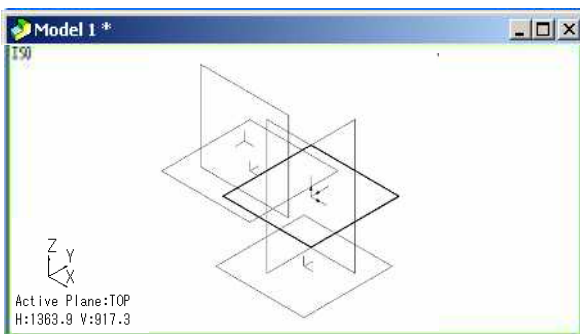
Select the [Control Work Plane(Active)] command.

The registered [Work plane] are displayed.

Select one from the list.

The selected [Work plane] is activated.

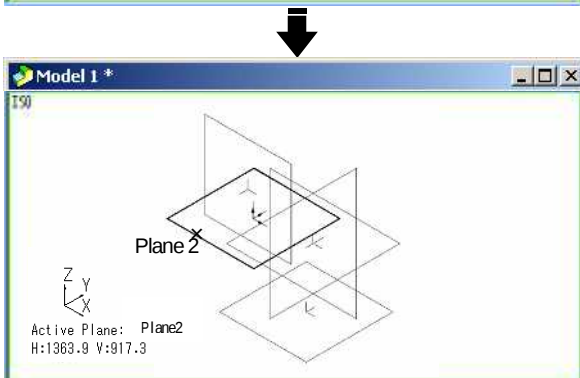
<Operation: Select on the screen>



Select the [Control Work Plane(Active)] command.

The registered [Work plane] are displayed by rubber band.

* If rubber band is not displayed clearly, zoom out the screen using [Zoom] command.



[Control Work Plane (Active)] Choose a work plane.

Specify the work plane.

* Work plane name is displayed in the pre-selected state.

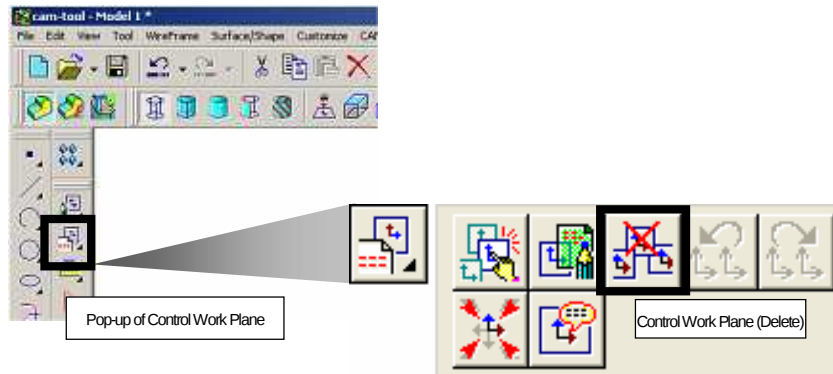
The selected [Work plane] is activated.

Click [Cancel] to end the command.

How to delete the saved coordinate system

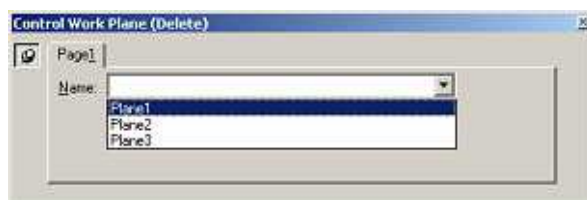
<Control Work Plane (Delete)>

Delete the registered [Work plane]. [Layer42]



Ex.) Delete the registered [Work plane1]

<Operation: Select from the list>



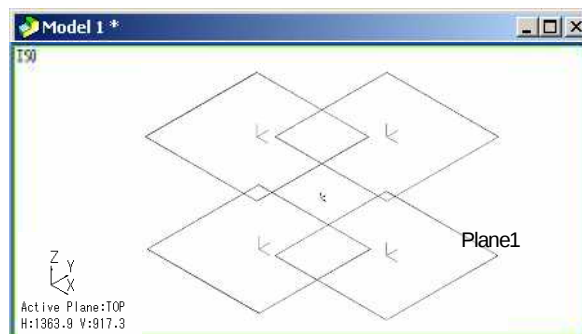
Select the [Control Work Plane(Delete)] command.

The registered [Work plane] are displayed.

Select one from the list.

The selected [Work plane] is deleted.

<Operation: Select on the screen>



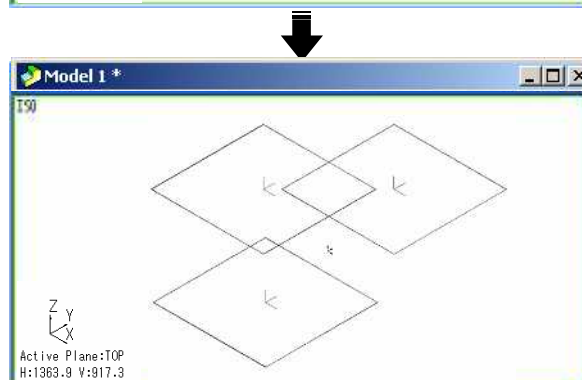
Select the [Control Work Plane(Delete)] command.

The registered [Work plane] are displayed by rubber band.

* If rubber band is not displayed clearly, zoom out the screen using [Zoom] command.

[Control Work Plane (Delete)] Choose a work plane to delete.

Specify the [Work plane] to be deleted.



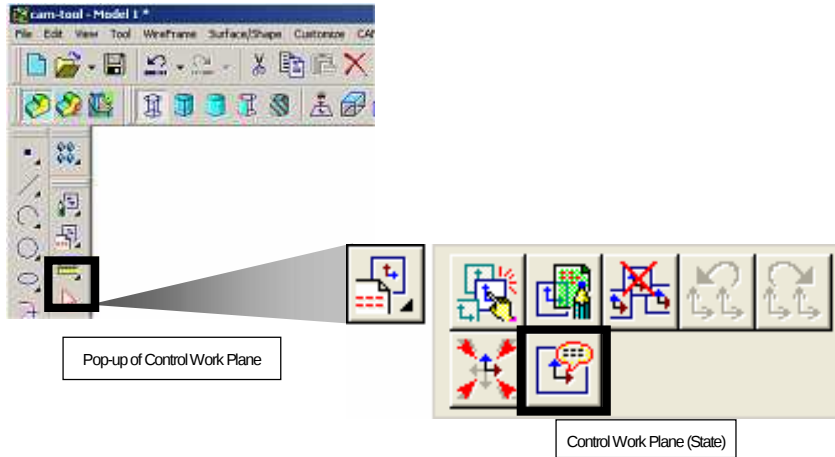
The selected [Work plane] is deleted.

Click [Cancel] to end the command.

How to confirm the information of current coordinate system

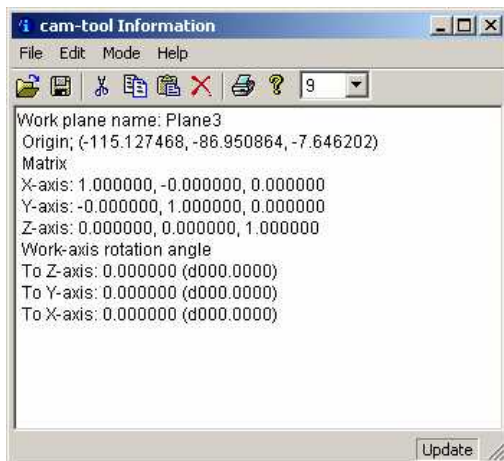
<Control Work Plane (State)>

Information of the current active [Work plane] is displayed on the [Information window]. [Layer42]



Ex.) Confirm the information of [Work coordinate system] that is currently being used.

<Operation>



Specify the [Work plane] to be confirmed.

Select the [Control Work Plane(State)] command.

Information of [Work plane] is displayed on the information window.

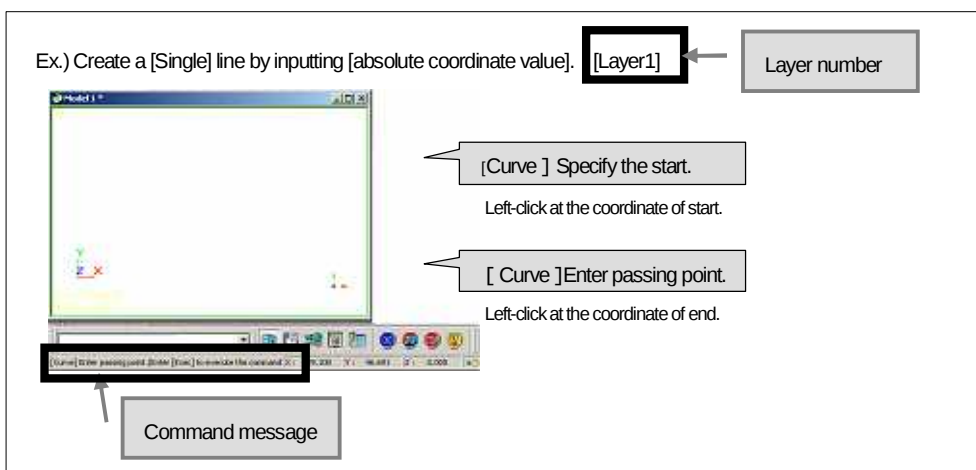
Chapter 7

Create surface

Open the model file below. (Same model file as in [Chapter3 - Chapter6])

CD-ROM/Intensive Course/CADmodel2.gmd

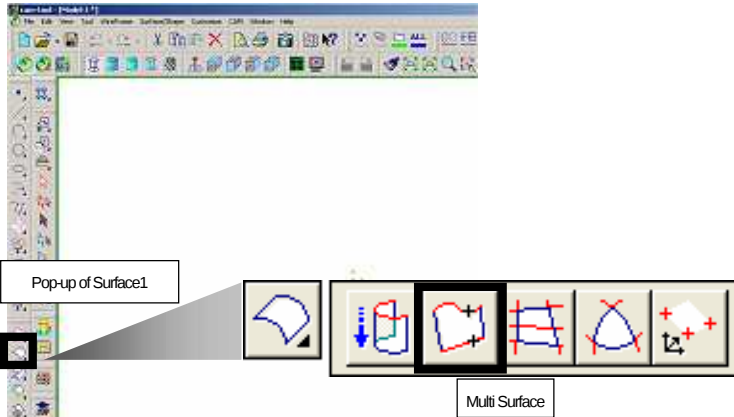
- If [Layer number] is written in the description, change the active layer number.
- Follow the [command message] displayed at the lower left of the screen, for your operation.



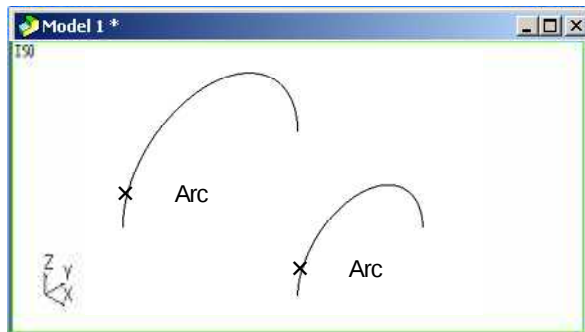
How to create surface by interpolating space between two entities

<Multi Surface>

Create a surface by interpolating a space between two entities with [Multi Surface]. [Layer50]



<Operation>



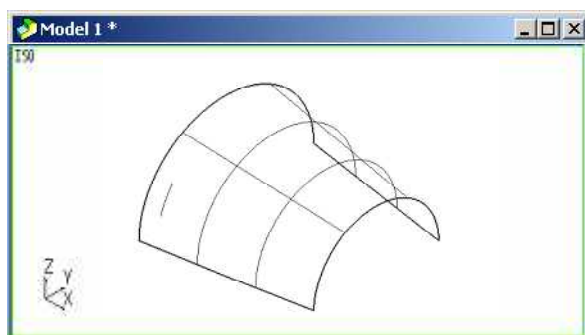
Select the [Multi surface] command.

[Multi surface] Pick entities.

Click the contour entity. (Arc)

[Multi surface] Pick entities. (Enter [Exec] to execute.)

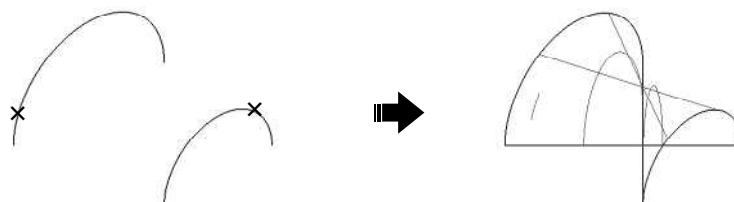
Click the contour entity. (Arc)



Press the center button to execute the command.



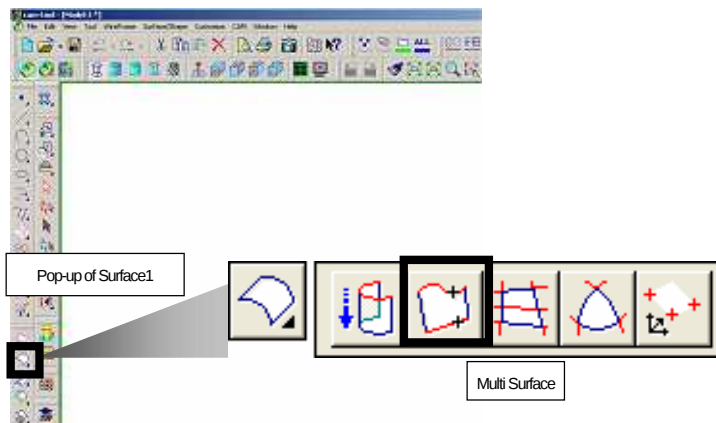
A surface is created by interpolating space between the picked point and the end point closer to the picked position. Therefore, pick the same side of entities.



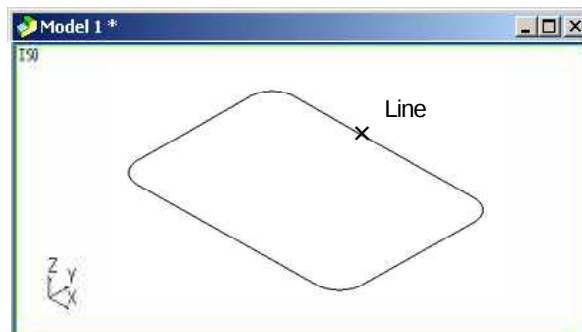
How to create trim surface with closed contour <Multi Surface>

Create a trim surface with closed contour with [Multi Surface].

Closed contour is need to be in the selected status before the [Multi Surface] command is executed.



<Operation>

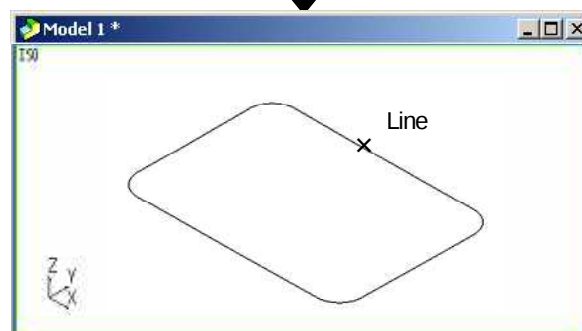


Execute the [Select (Entity)] command.

[Select (Entity)] Pick an entity.

Right-click >> Pop-up menu/Chain

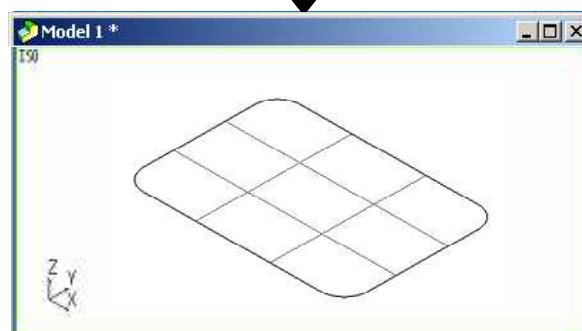
Specify the entity. (Line)



Select the [Multi surface] command.

[Multi surface] Pick entities *Trim surface is created by specifying selected closed contour.

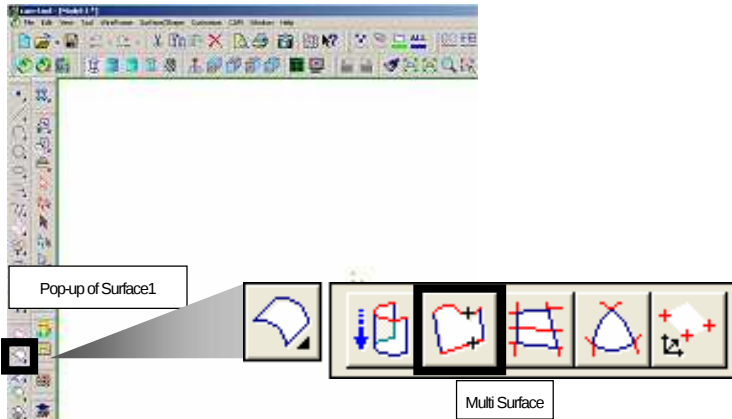
Specify the closed contour in the selected status.(Line)



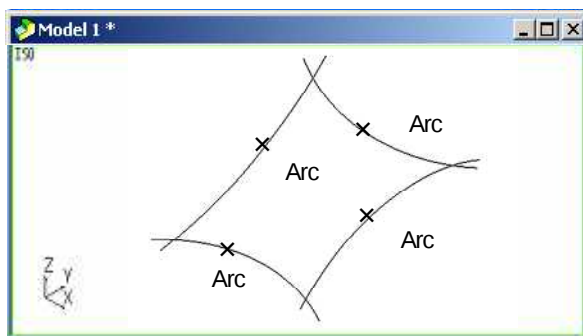
Press the center button to execute the command.

How to create surface to interpolate closed contours <Multi Surface>

Create a surface to interpolate closed contours of three to six sides, with [Multi Surface]. [Layer52]



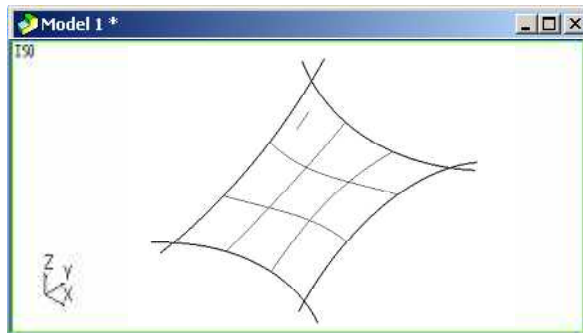
<Operation>



Select the [Multi surface] command.

[Select (Entity)] Pick an entity.

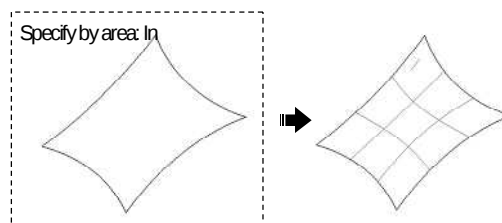
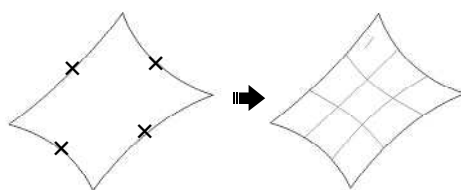
- Specify the entities. (Arc)



Press the center button to execute the command.



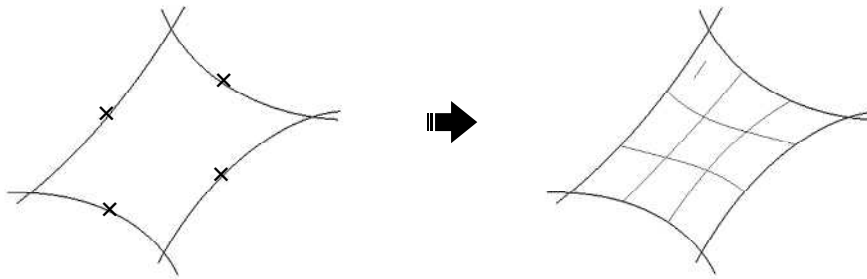
If the contour entity has been trimmed at each intersection, multi surface can be created by specifying entities with [Specify by Area: In] function instead of specifying each contour one by one.



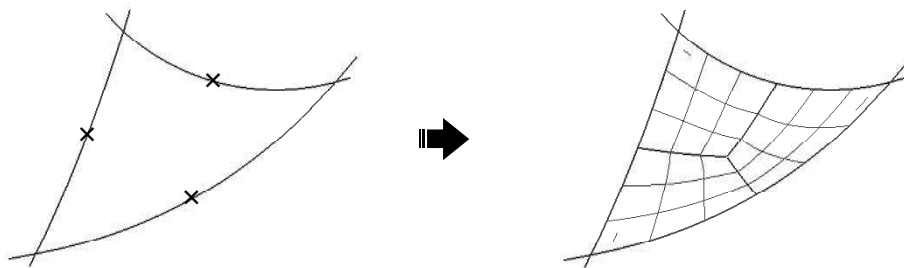


A surface is created differently depending on the number of side of contours. Only when there are four sides, multi surface is created with a single surface. And when there are three, five or six sides, multi surface is created with each number of surfaces.

<For four sides> Multi surface is created with a single surface. [Layer53]



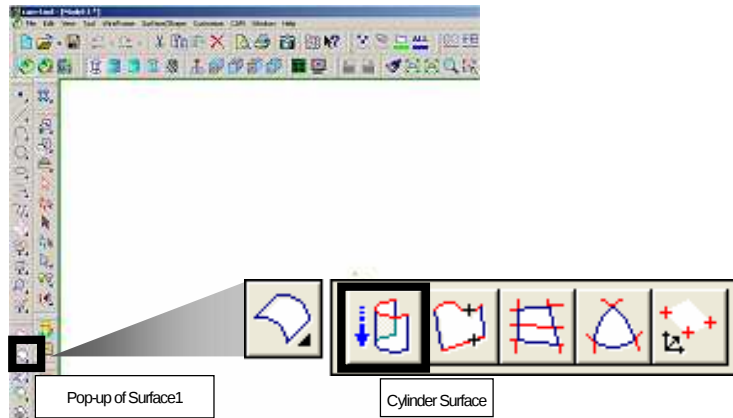
<For three sides> Multi surface is created with three surfaces. [Layer53]



How to create surface by copying contour in parallel <Cylinder Surface>

Create a surface by copying entity in the specified direction, and interpolating the space between the entities with [Cylinder Surface].

(The method of specifying direction/distance for moving is the same as the operation of [Move/Copy] command.)



<Command sheet>



How to create a cylinder surface in either of [X]/[Y]/[Z] direction

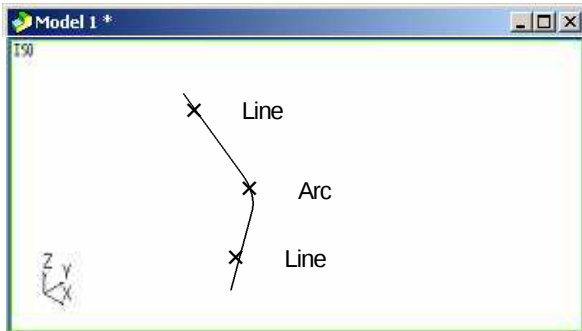
How to create a cylinder surface by specifying two points

How to create a cylinder surface by specifying line as [direction/length] for copying

How to create a cylinder surface in either of [X]/[Y]/[Z] direction (Use guide axis)

Ex.) Create a cylinder surface by copying the entity in the [Y-axis direction] of work coordinate by [100mm], and interpolating the space between the entities. [Layer54]

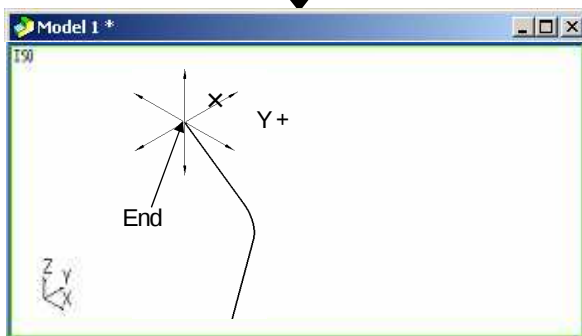
<Operation>



Select the [Cylinder Surface] command.
Specify parameters in the command sheet.
Length: 100
Start position: 0
Deselect: ON
Trim: OFF

[Cylinder Surface] Pick an entity.

- Specify the entities.(Line, Arc, Line)
Press the center button to determine the specifying.



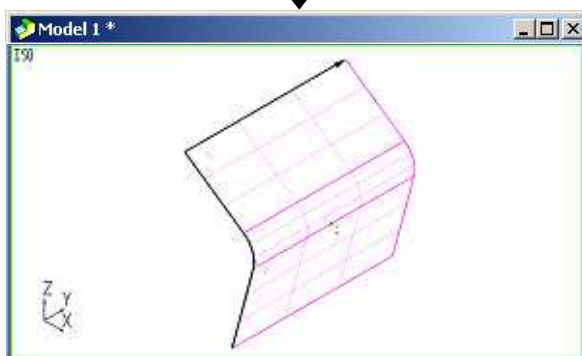
[Cylinder Surface] Pick a straight line or enter the start of vector.

Specify the point. (End)
Guide lines in the [X/Y/Z] direction are displayed.

[Cylinder Surface] Pick a guide axis or enter the end of vector.

Specify the direction entity. (Y+)
Ex.) Y + direction

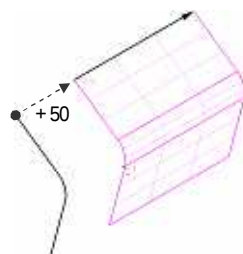
Press the center button to execute the command.



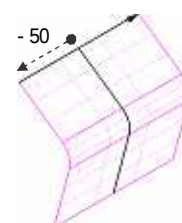
What is the start point?

To create cylinder surface, copy the contour from the position specified by [Start position] in the command sheet, in the specified direction.

Distance:	100	<
Start position:	50	<



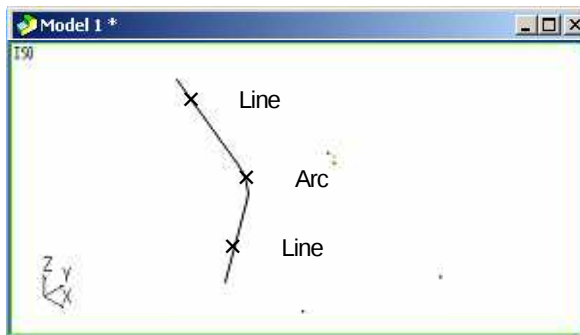
Distance:	100	<
Start position:	-50	<



How to create a cylinder surface by specifying two points (Specify two points)

Ex.) Create a cylinder surface by copying entities in the direction specified by two points. [Layer55]

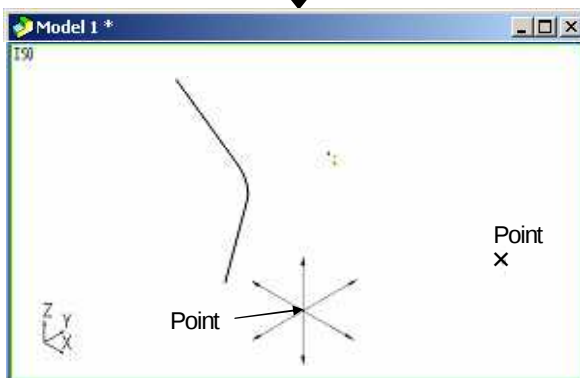
<Operation>



Select the [Cylinder Surface] command.
Specify parameters in the command sheet.
Length: Any parameter is available
Start position: 0
Deselect: ON
Trim: OFF

[Cylinder Surface] Pick an entity.

- Specify the entities.(Line, Arc, Line)
Press the center button to determine the specifying.



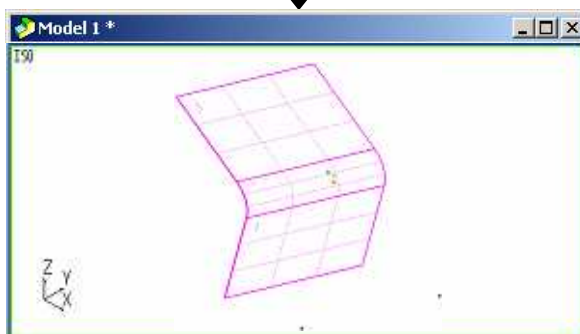
[Cylinder Surface] Pick a straight line or enter the start of vector.

Specify the start point. (Point)
Guide lines in the [X/Y/Z] direction are displayed.

[Cylinder Surface] Pick a guide axis or enter the end of vector.

Specify the end point. (Point)

* The distance between the two points is input in the command sheet.

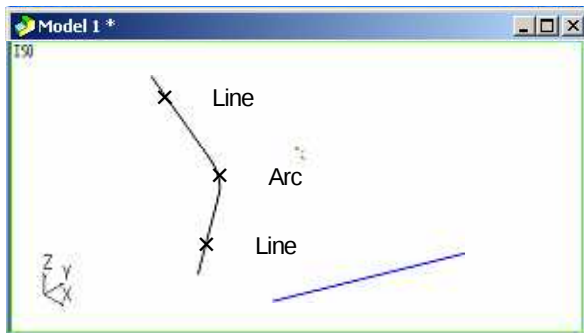


Press the center button to execute the command.

How to create by specifying a line as the [direction/length] for copying (Getting information of line)

Ex.) Create a surface by specifying a line of [80mm] as the [direction/length] of copying entities [Layer56]

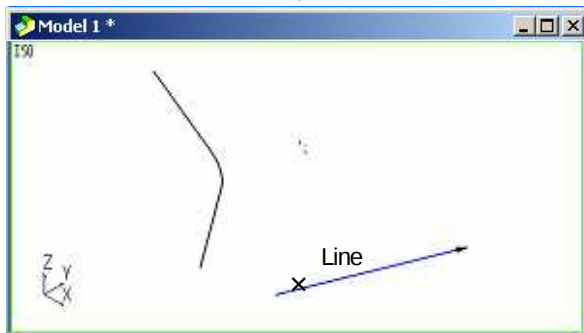
<Operation>



Select the [Cylinder Surface] command.
Specify parameters in the command sheet.
Length: Any parameter is available
Start position: 0
Deselect: ON
Trim: OFF

[Cylinder Surface] Pick an entity.

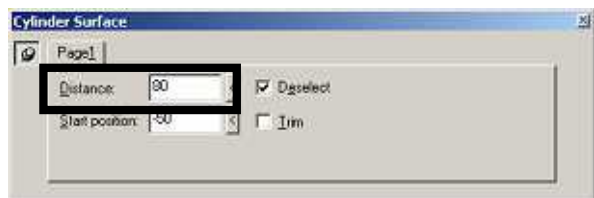
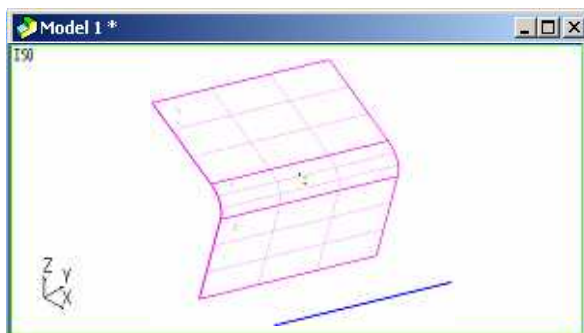
- Specify the entities.(Line, Arc, Line)
- Press the center button to determine the specifying.



[Cylinder Surface] Pick a straight line or enter the start of vector.

- Specify the line. (Line)
- The end point closer to the specified position is determined as the start point.
 - The end point far from the specified position is determined as the endpoint.

* The length of the line is input for the [Length] in the command sheet.



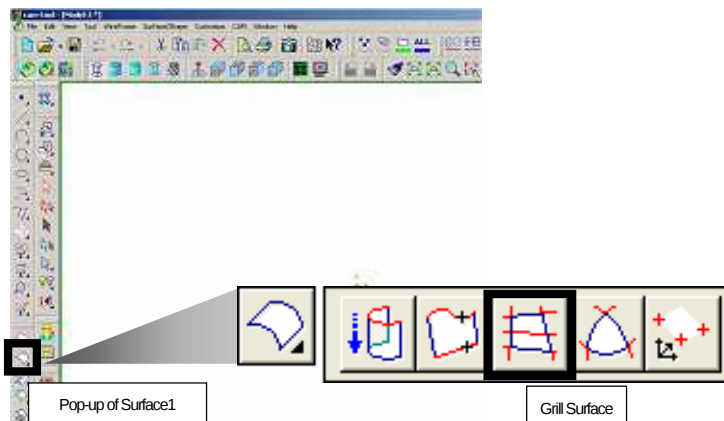
Press the center button to execute the command.

How to create surface by specifying section for closed contour <Grill Surface>

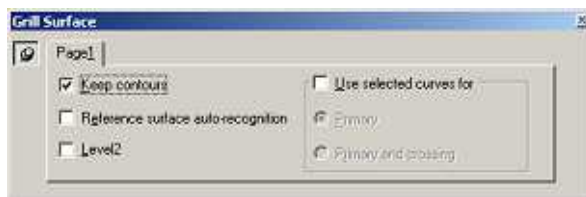
Create a surface by keeping grid-like contours that is consisted of four sides.

Specify a group of outer contours as [Primary curve], and a group of contours cross the primary contours, as [Crossing curve].

All [Primary curves] are need to be crossed at intersections of the curves and [Crossing curves].



<Command sheet>



How to create a grill surface by keeping grid-like contours

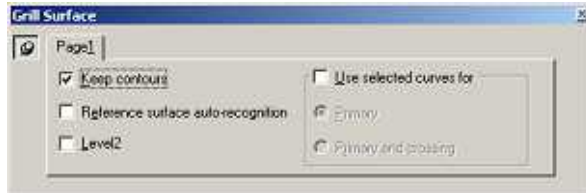
How to create a surface that is connected smoothly with adjacent surface

How to create a surface that is partly connected smoothly with adjacent surface

How to create a grill surface by keeping grid-like contours

Ex.) Create a grill surface by keeping grid-like contours of [Primary curve] / [Crossing curve]. [Layer57]

<Operation>

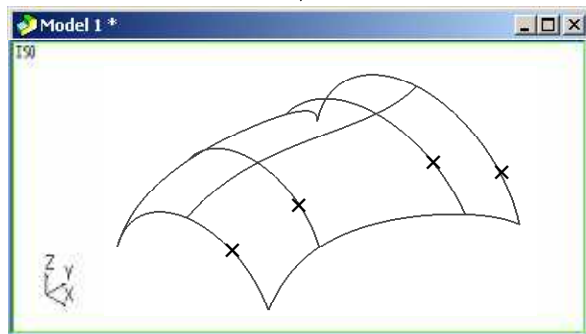


Select the [Grill Surface] command.

Specify parameters in the command sheet.

Keep contours: ON

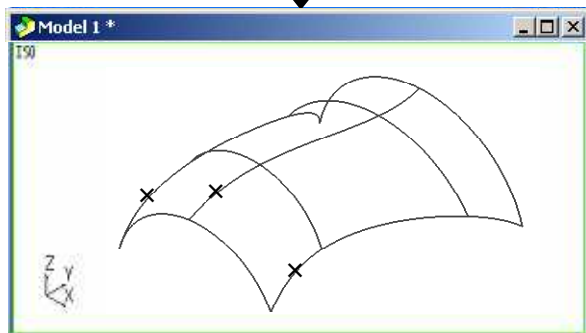
Reference surface auto-recognition: OFF



[Grill Surface] Pick a primary curve.

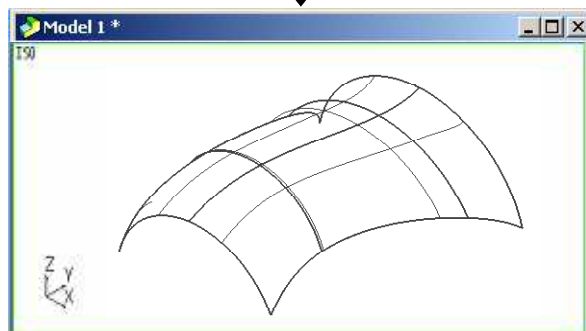
- Specify the entities for primary curves. (Curve)

Press the center button to determine the specifying.



[Grill Surface] Pick crossing curves.

- Specify the entities for crossing curves. (Curve)



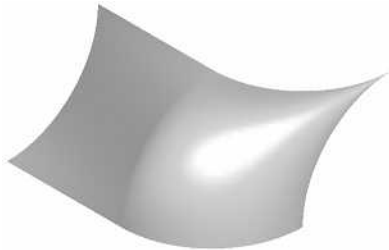
Press the center button to execute the command.

* If an error occurred because the primary curve and the crossing curve are not connected smoothly, the distance is temporarily displayed at the position. (Within 1mm)

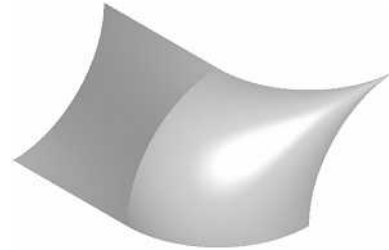
How to create a surface that is connected smoothly with adjacent surface (Reference surface auto-recognition: ON)

Ex.) When there are surfaces that are adjacent to [Primary curve] and [Crossing curve], create a surface which has the same tangent direction with all of adjacent surfaces. [Layer58]

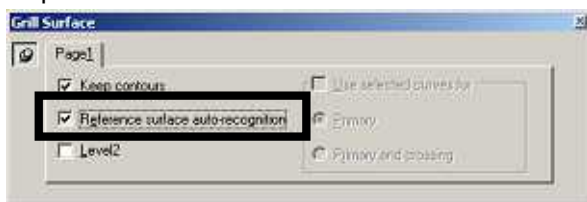
Reference surface auto-recognition: ON



Reference surface auto-recognition: OFF



<Operation>

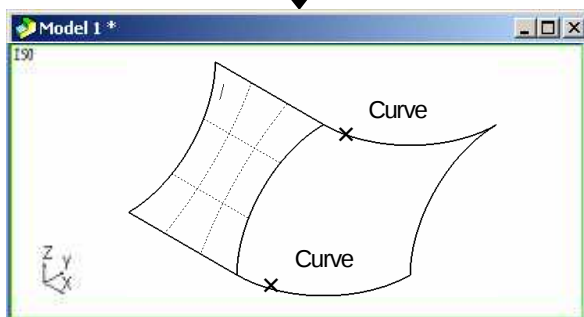


Select the [Grill Surface] command.

Specify parameters in the command sheet.

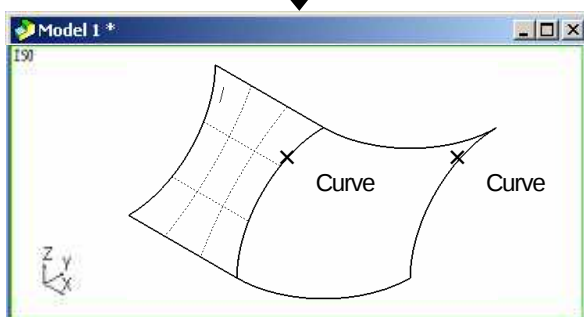
Keep contours: ON

Reference surface auto-recognition: ON



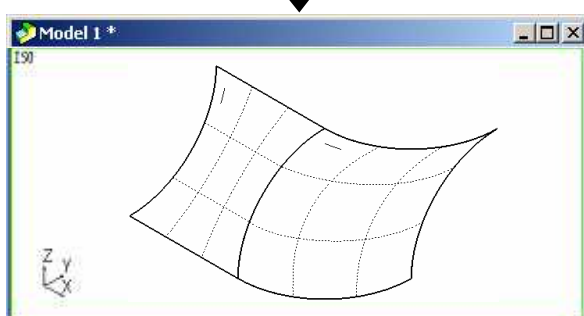
[Grill Surface] Pick a primary curve.

- Specify the entities for primary curves. (Curve)
- Press the center button to determine the specifying.



[Grill Surface] Pick crossing curves.

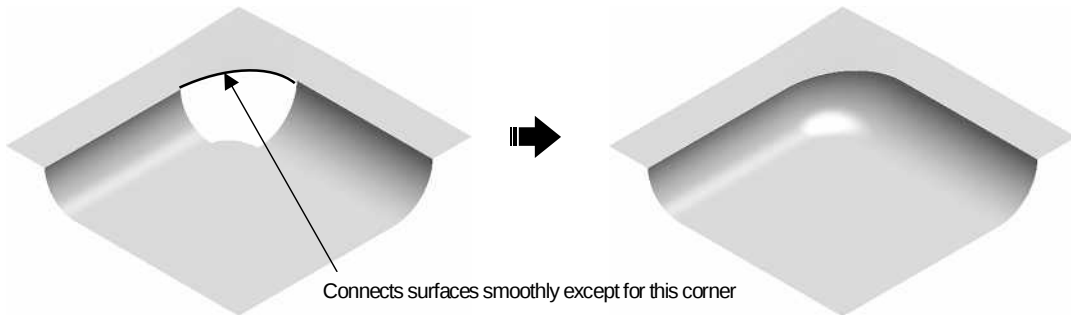
- Specify the entities for crossing curves. (Curve)
- Press the center button to determine the specifying.



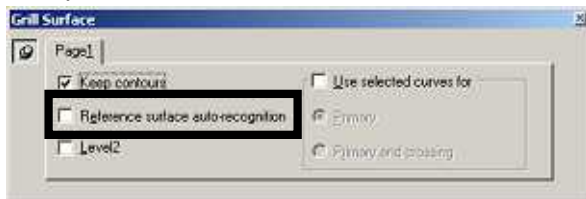
Press the center button to execute the command.

How to create a surface that is partly connected smoothly with adjacent surface
(Reference surface auto-recognition: OFF)

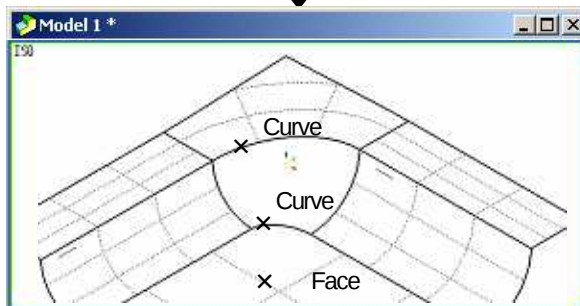
Ex.) To create a grill surface, for three sides, connects adjacent surfaces smoothly so that surfaces have the same tangent direction. And for one side, connects adjacent surfaces not smoothly. [Layer59]



<Operation>



Select the [Grill Surface] command.
Specify parameters in the command sheet.
Keep contours: ON
Reference surface auto-recognition: OFF



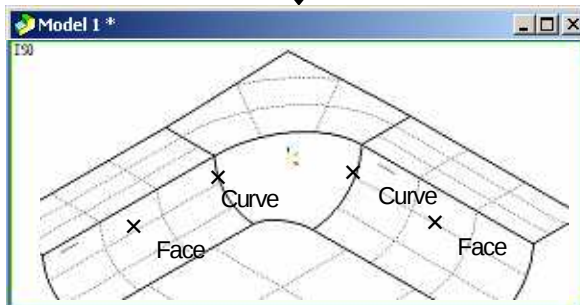
[Grill Surface] Pick a primary curve.

Specify the entity for primary curve. (Curve)

Specify the entity for primary curve. (Curve)

Pick the surface for tangent direction. (Face)

Press the center button to determine the specifying.



[Grill Surface] Pick crossing curves.

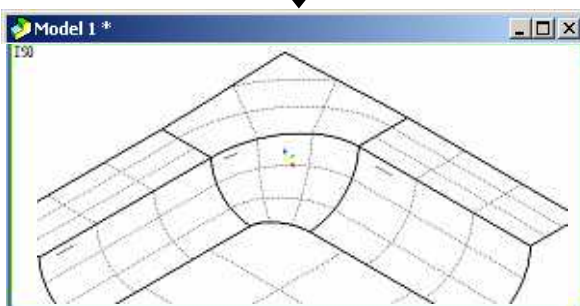
Specify the entity for crossing curve. (Curve)

Pick the surface for tangent direction. (Face)

Specify the entity for crossing curve. (Curve)

Pick the surface for tangent direction. (Face)

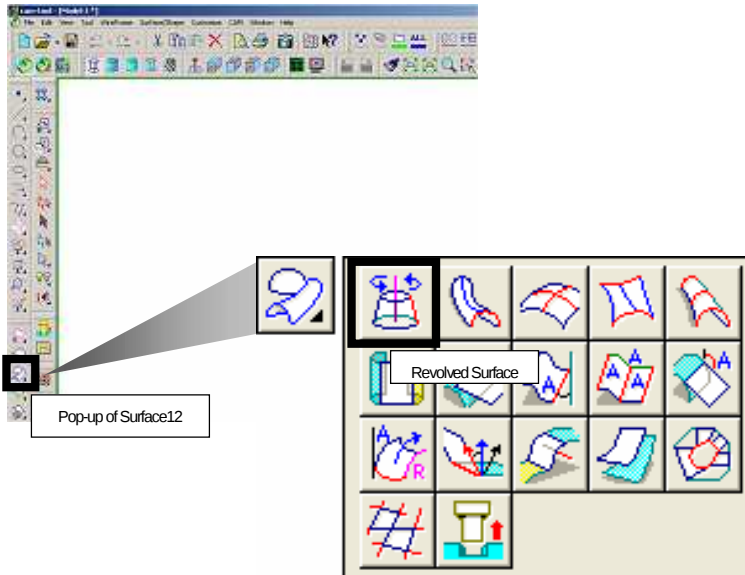
Press the center button to execute the command.



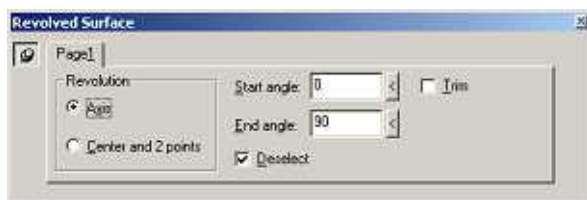
How to create surface by rotating contour <Revolved Surface>

Create a surface by rotating the specified entity with [Revolved Surface].

(The method of specifying rotating axis is the same as the operation of [Move/Copy] command.)



<Command sheet>



How to create a revolved surface by specifying either of [X]/[Y]/[Z] direction as rotating axis

How to create a revolved surface by specifying a line as rotating axis

How to create a revolved surface by specifying three points as rotating axis

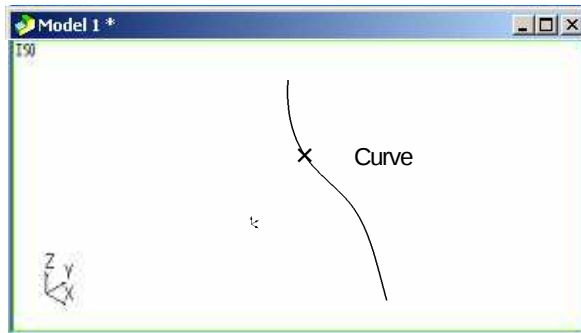


It is recommended not to create a revolved surface by rotating the entity by 360 degree. If you use the data that includes the surface created in that way, for CAM calculation, an error may be occurred, or invalid [CL data] may be output.

How to create a revolved surface by specifying either of [X]/[Y]/[Z] direction as rotating axis (Use guide axis)

Ex.) Specify [Z axis] of work coordinate, and rotate the entity by [90 degree]. [Layer60]

<Operation>



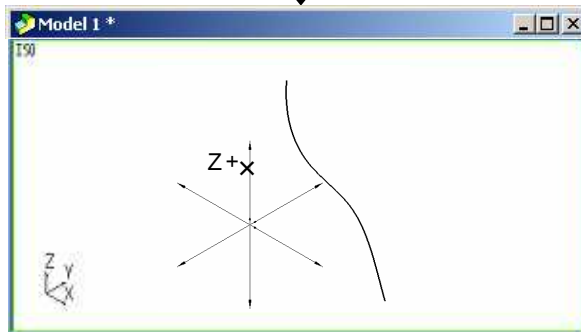
Select the [Revolved Surface] command.
Specify parameters in the command sheet.

Revolution: Axis
Angle: 0
End angle: 90
Deselect: ON
Trim: OFF

[Revolved Surface] Pick an entity.

Specify the entity. (Curve)

Press the center button to determine the specifying.



[Revolved Surface] For axis of revolution, pick a straight line or enter the start of vector.

Specify the point. >> [F12] (Work origin)

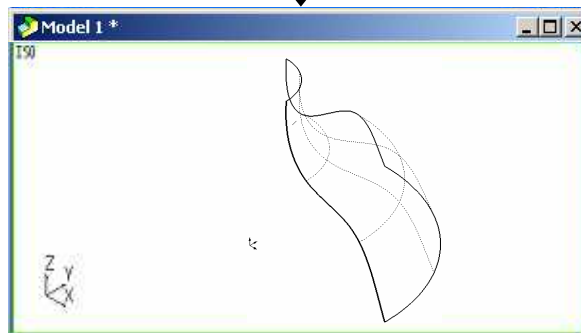
Guide axes of [X]/[Y]/[Z] are displayed.

[Revolved Surface] For axis of revolution, pick a guide axis or enter the end of vector.

Specify the direction entity. (Z+)

Ex.) Z + direction

Press the center button to execute the command.



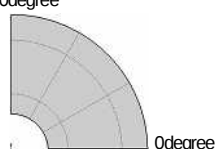
What is start/end angle?

Set the specified entity as at being [0 degree], and create a surface from [start angle] to [end angle].

Start angle: 0
End angle: 90

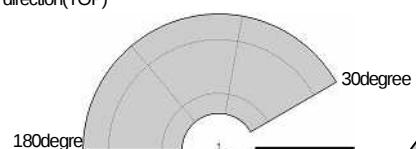
View direction(TOP)

90degree



Start angle: 30
End angle: 180

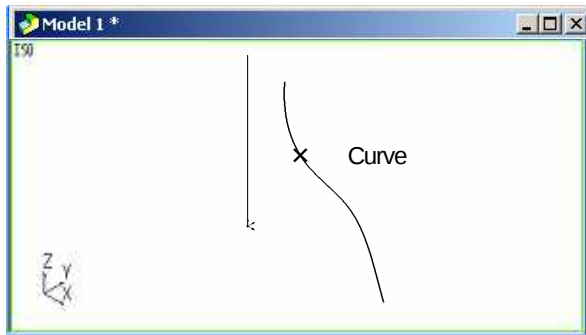
View direction(TOP)



How to create a revolved surface by specifying a line as rotating axis (Use line)

Ex.) Specify the [line in the Z-axis direction] as the rotating axis, and rotate the entity by [90 degree]. [Layer61]

<Operation>



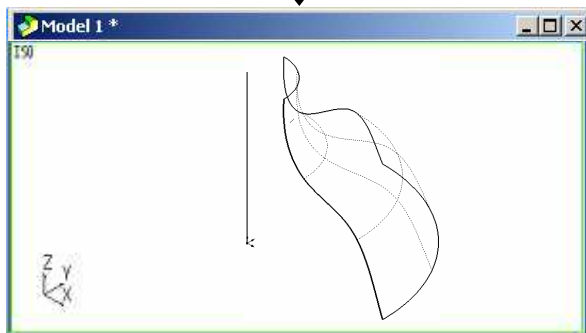
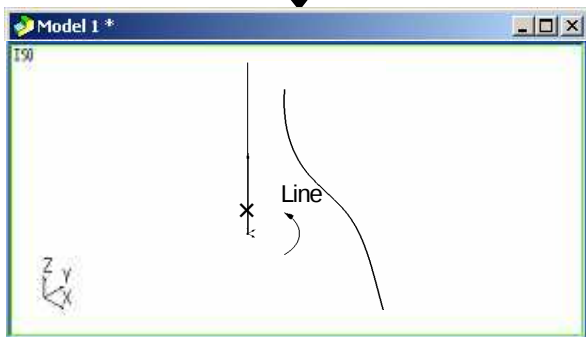
Select the [Revolved Surface] command.
Specify parameters in the command sheet.
Revolution: Axis
Angle: 0
End angle: 90
Deselect: ON
Trim: OFF

[Revolved Surface] Pick an entity.

Specify the entity. (Curve)
Press the center button to determine the specifying.

[Revolved Surface] For axis of revolution, pick a straight line or enter the start of vector.

Specify the line. (Line)



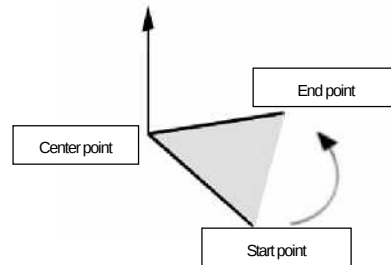
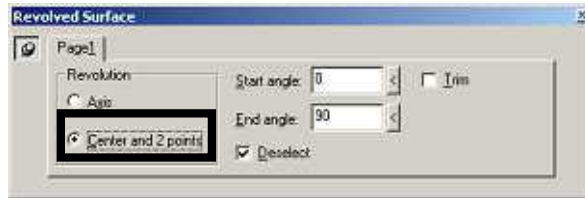
Press the center button to execute the command.

How to create a revolved surface by specifying three points as rotating axis
(Determine rotating axis/angle by three points.)



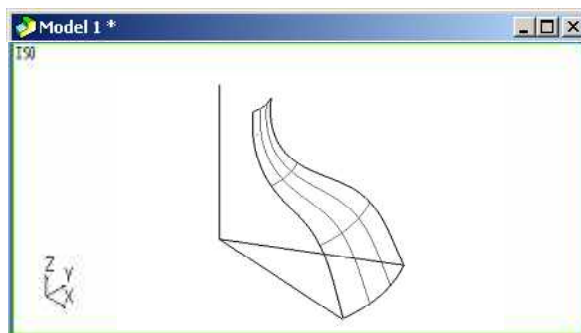
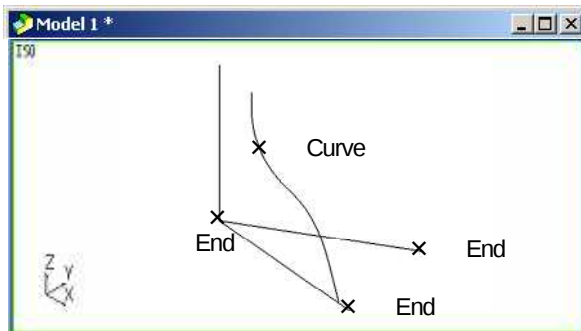
What is [Center and 2 points] ? [Layer62]

Determine rotating axis/angle by specifying three points.



- Determine the plane by specifying the coordinate of three points such as [origin], [start point of rotation], [end point of rotation]
- Set normal direction of [plane that is created with three points], which runs through the origin as [rotating axis].
- Set angle between [line created with origin and start point of rotation], and [line created with origin and end point of rotation] as [rotating angle].

<Operation>



Select the [Revolved Surface] command.

Specify parameters in the command sheet.

Revolution: Center and 2 points

Deselect: ON

Trim: OFF

[Revolved Surface] Pick an entity.

Pick the entity. (Curve)

Press the center mouse button to determine the specifying.

[Revolved Surface] Enter coordinates of the center of revolution.

Specify the coordinate of the center. (End/Intersect)

[Revolved Surface] Enter coordinates of the start of revolution.

Specify the coordinate of the start. (End)

[Revolved Surface] Enter coordinates of the end of revolution.

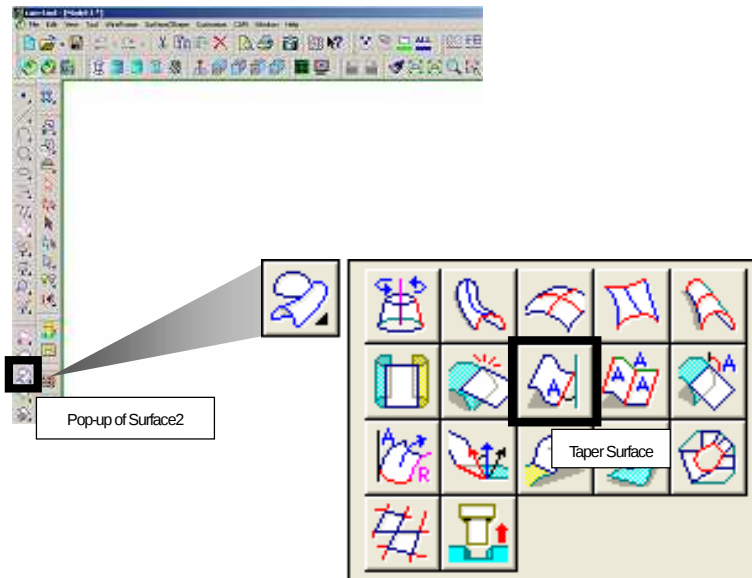
Specify the coordinate of the end. (End)

Press the center button to execute the command.

How to create surface by specifying angle <Taper Surface>

Create a surface that has the angle based on the [Z-axis of work coordinate] with [Taper surface].

If you wish to create a taper surface from plural contours, you need to put those contours in the selected status before executing [Taper Surface] command.



<Command sheet>



How to create taper surfaces separately from plural contours

How to create a single taper surface from plural contours

If space is generated between adjacent surfaces...,

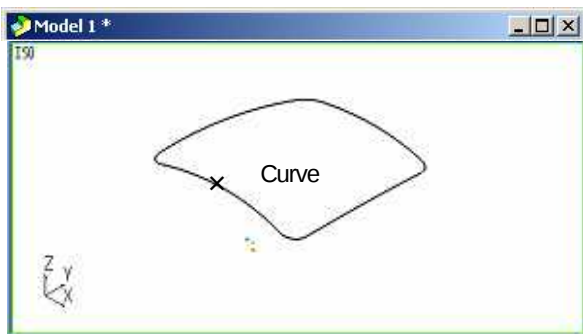
How to create taper surfaces and cut the surfaces at the certain position

How to create taper surfaces separately from plural contours (Each contour: ON)

Taper surfaces can be created from each contour separately. If you wish to create taper surfaces that have the same angle, put the all contours in the selected status in advance. Taper surfaces can be created from all contours simultaneously.

It is valid for the contour that can be selected by [Select-Chain] function. [Layer63]

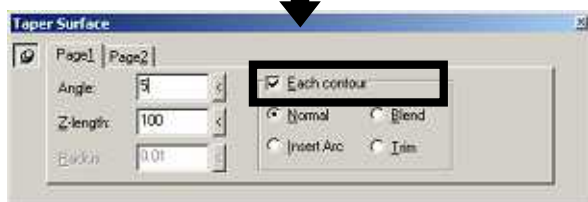
<Operation>



Execute the [Select(Entity)] command.

[Select (Entity)] Pick an entity.

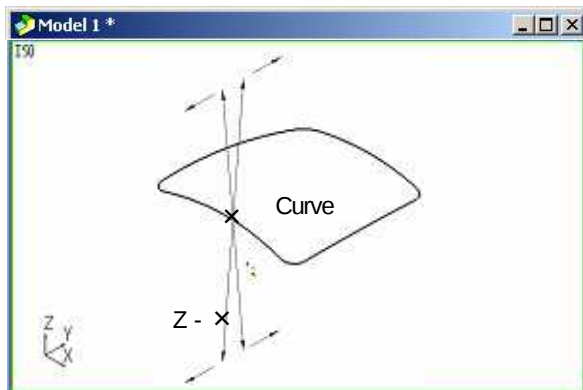
Right-click >> Pop-up menu/Chain
Specify the entity. (Curve)



Select the [Taper Surface] command.

Specify parameters in the command sheet.

Angle: 5
Z-length: 100
Each contour: ON [Normal]

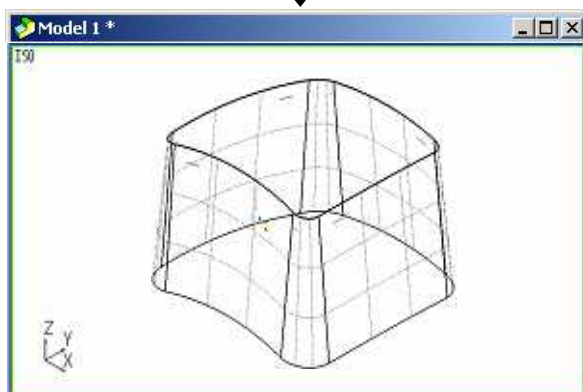


[Taper Surface] Pick the guideline.

Specify the selected entity. (Curve)

[Taper Surface] Pick taper direction.

Specify the direction. (Z-)
Ex.) Outer Z - direction



Press the center button to execute the command.

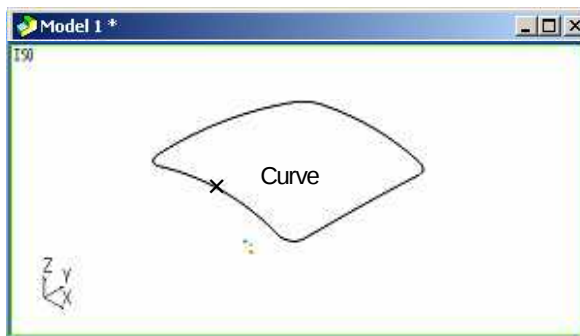
* Taper surfaces are created from each contour.

How to create a single taper surface from plural contours (Each contour: OFF)

If you wish to create a taper surface from plural contours, you need to put those contours in the selected status before executing [Taper Surface] command.

It is valid for the contour that can be selected by [Select-Chain] function. [Layer64]

<Operation>

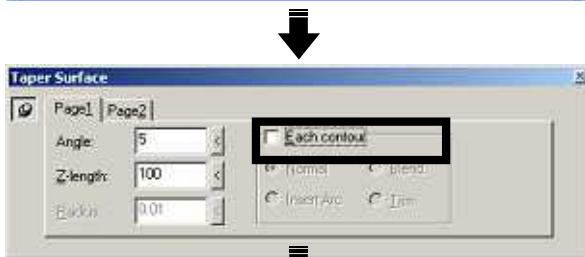


Execute the [Select(Entity)] command.

[Select (Entity)] Pick an entity.

Right-click >> Pop-up menu/Chain

Specify the entity. (Curve)



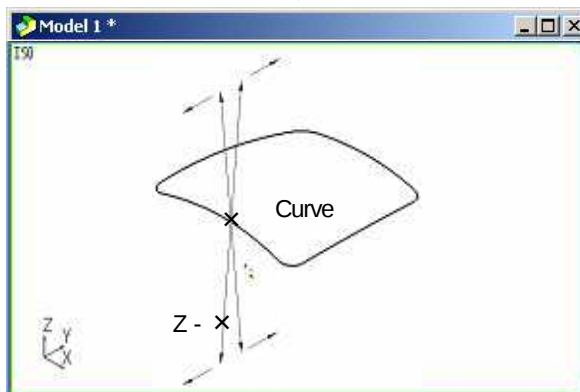
Select the [Taper Surface] command.

Specify parameters in the command sheet.

Angle: 5

Z-length: 100

Each contour: OFF



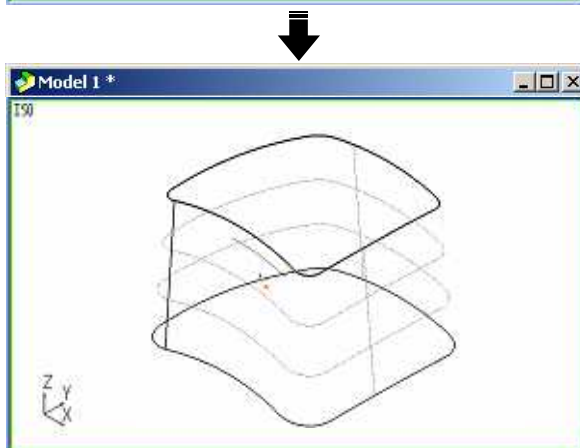
[Taper Surface] Pick the guideline.

Specify the selected entity. (Curve)

[Taper Surface] Pick taper direction.

Specify the direction. (Z-)

Ex.) Outer Z - direction



Press the center button to execute the command.

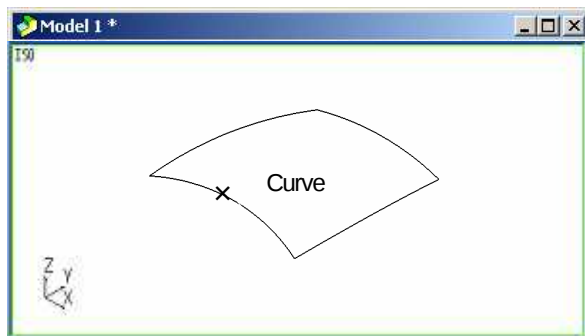
* A single taper surface is created.

If space is generated between adjacent surfaces...,
(Each contour: ON >> Specify the method of interpolating space)

If adjacent contours do not have the same tangent direction, space is generated between taper surfaces.
In that case, put the contours in the selected status in advance. Taper surfaces are created from each contour, and space can be interpolated with the specified function.

It is valid for the contour that can be selected by [Select-Chain] function. [Layer65]

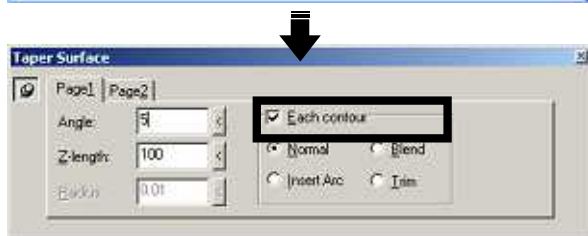
<Operation>



Execute the [Select(Entity)] command.

[Select (Entity)] Pick an entity.

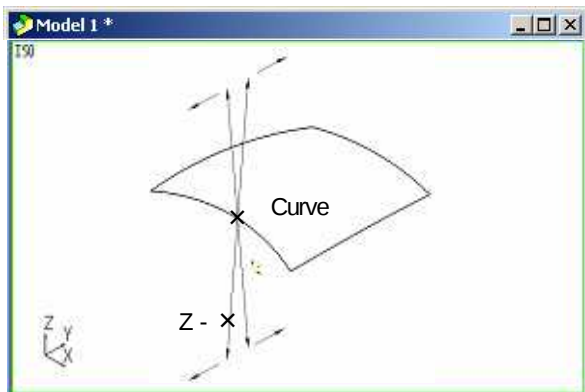
Right-click >> Pop-up menu/Chain
Specify the entity. (Curve)



Select the [Taper Surface] command.

Specify parameters in the command sheet.

Angle: 5
Z-length: 100
Each contour: ON [Normal]

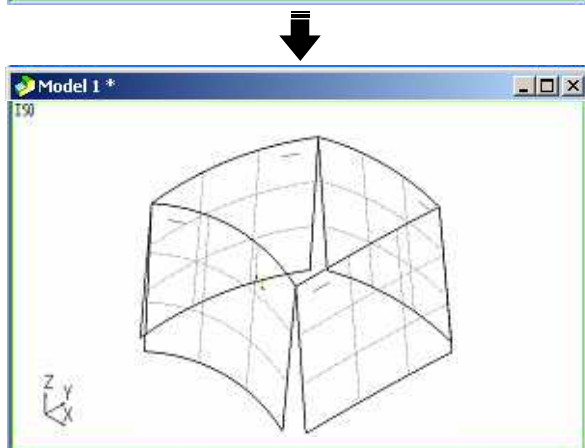


[Taper Surface] Pick the guideline.

Specify the selected entity. (Curve)

[Taper Surface] Pick taper direction.

Specify the direction. (Z-)
Ex.) Outer Z - direction



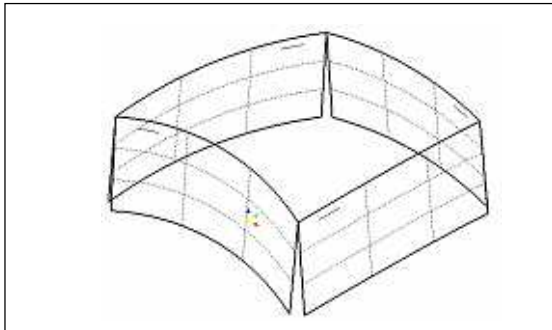
Press the center button to execute the command.

* Space is generated between the taper surfaces.

■ Method of interpolating space [Layer66]

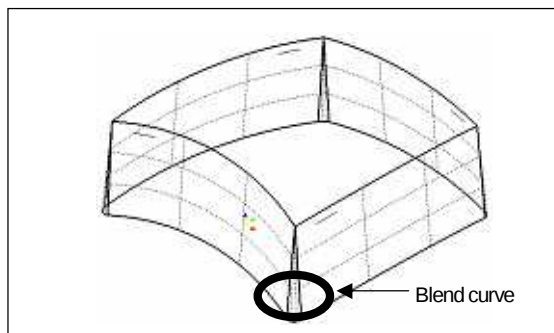


<Normal>



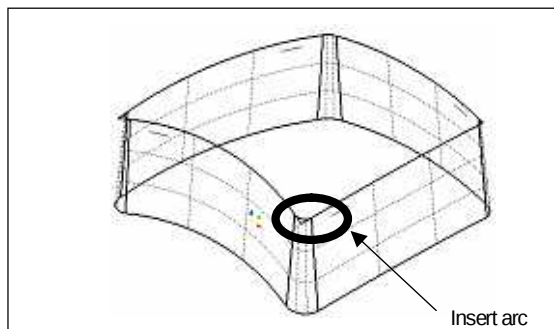
Create taper surfaces separately.

<Blend>



If adjacent contours do not have the same tangent direction, contours are connected with a blend curve that has the same tangent direction, (Refer to page107), and a compensate surface is created between the surfaces.

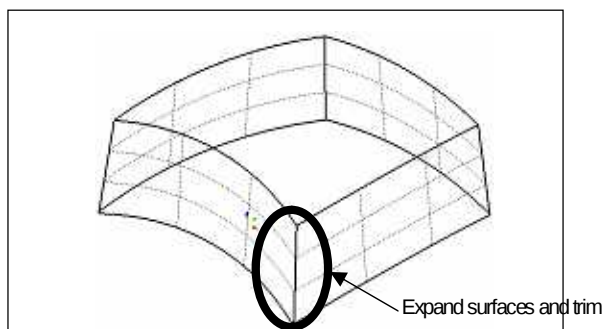
<Insert Arc>



If adjacent contours do not have the same tangent direction, a curve that is approximately the same as the specified arc is inserted into the corner(in the tangent direction), and a compensate surface is created between the surfaces.

Input [radius value] for arc in the command sheet.

<Trim>

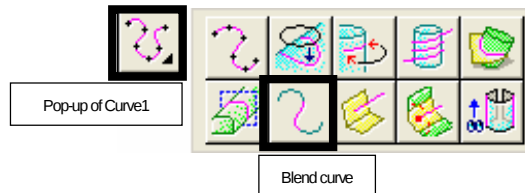


If adjacent contours do not have the same tangent direction, expand surfaces, and trim them at the line of intersection of the surfaces.

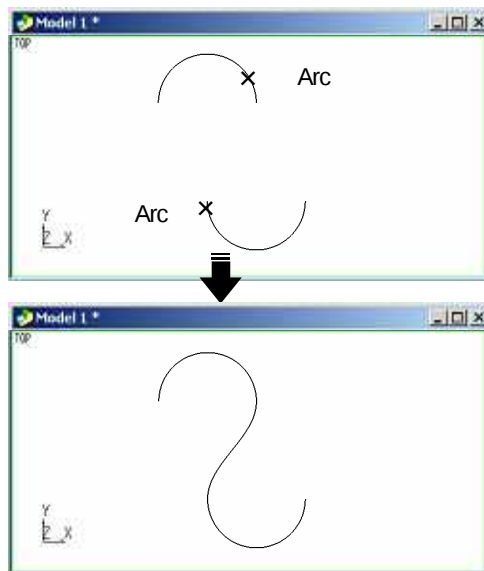


What is [Blend curve]? [Layer67]

Create a curve that connects space between the two entities smoothly.



<Operation>



Select the [Blend Curve] command.

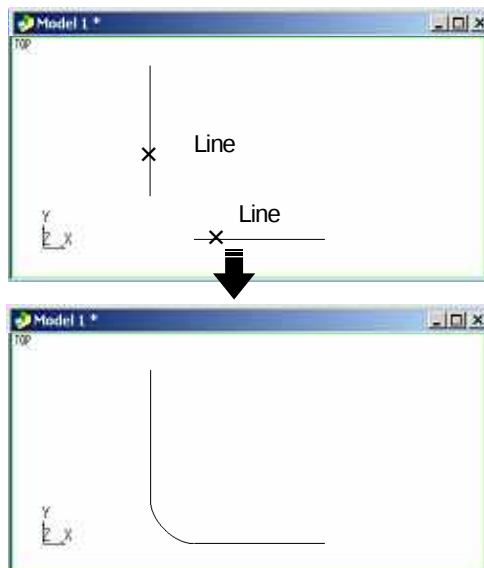
[Blend Curve] Pick entities for the first group.

Specify the first entity. (Arc)

[Blend Curve] Pick the second entity or specify passing point.

Specify the second entity. (Arc)

<Operation>



Select the [Blend] command.

[Blend Curve] Pick entities for the first group.

Specify the first entity. (Line)

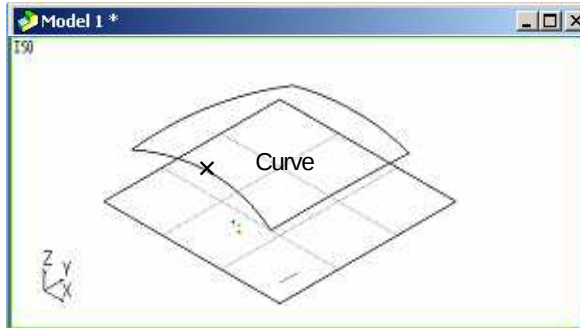
[Blend Curve] Pick the second entity or specify passing point.

Specify the second entity. (Line)

How to create taper surfaces and cut the surfaces at the certain position (Trim: ON)

Ex.) Create taper surfaces and trim the surfaces with the specified surface. [Layer68]

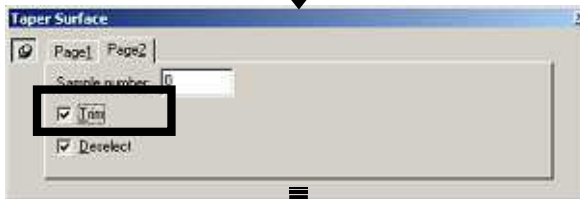
<Operation>



Execute the [Select(Entity)] command.

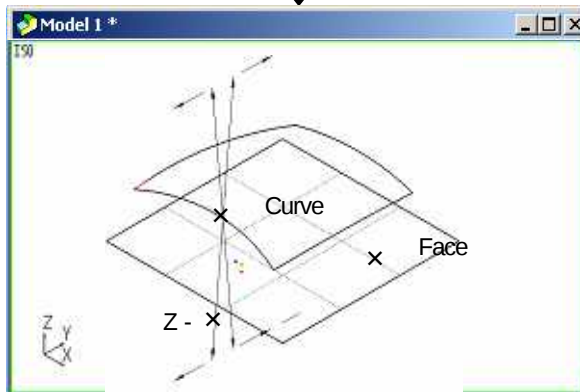
[Select (Entity)] Pick an entity.

Right-click >> Pop-up menu/Chain
Specify the entity. (Curve)



Select the [Taper Surface] command.
Specify parameters in the command sheet.
(Page1) Angle: 5, Z-length: 100
Each contour: ON : [Trim]

(Page2) Trim: ON



[Taper Surface] Pick the guideline.

Specify the selected entity. (Curve)

[Taper Surface] Pick taper direction.

Specify the direction. (Z-)
Ex.) Outer Z-direction

[Taper Surface] Pick a trimming surface.

Specify the surface. (Face)

Press the center button to execute the command.

* Although [100mm] is specified for [Z-length], the surfaces are trimmed with the surface specified in the operation].

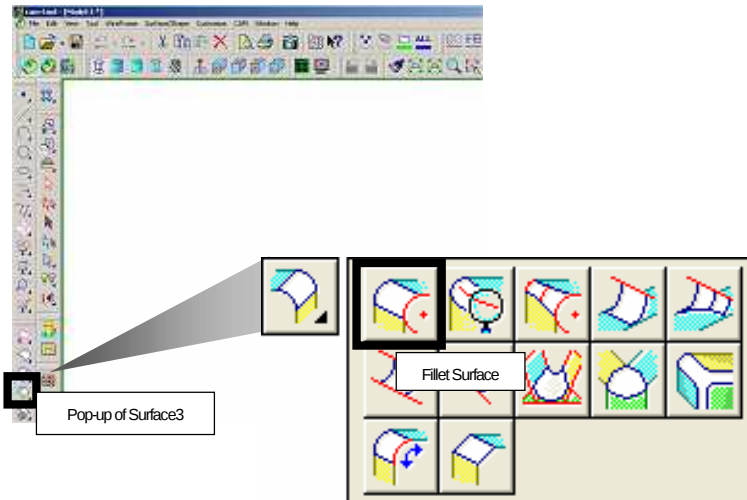


This [Trim] function is included in [Cylinder Surface], [Revolved Surface] as well.

How to create fillet surface between the surfaces <Fillet Surface>

Create a surface of the specified arc between the two groups of surfaces with [Fillet Surface].

Place the specified arc so as to be adjacent to the first group of surfaces and the second group of surfaces, and create a surface to interpolate the arc. The arc is placed on the [normal plane] of crossing line of the first group of surfaces and the second group of surfaces.



How to create a fillet surface between the specified surfaces

How to create fillet surfaces between surfaces simultaneously

How to create a fillet surface that compensates the border of surfaces based on arc value

How to confirm direction of creating a fillet surface using arrow

How to extend a surface to create a fillet surface

How to extend a surface along surface to create a fillet surface

If you wish not to trim the specified surface from the created surfaces...



Precautions for creating fillet surface

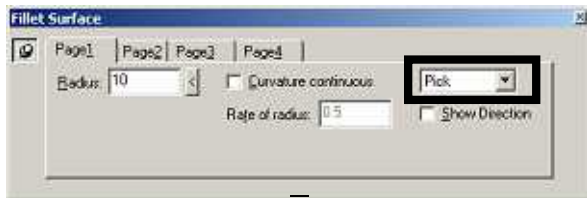
- When two cavity fillets/core fillets are overlapped each other, create a surface in descending order.
- If surfaces are created in descending order, surfaces are unspread at corner.
- Surfaces of the [same radius] cannot be created at corner, with [Fillet Surface].

>> Use [Corner Fillet Surface]

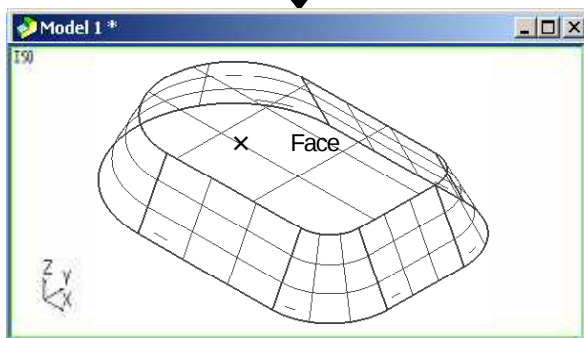
How to create a fillet surface between the specified surfaces (Specify surface: Pick)

Specify surfaces for both the first group of surfaces and the second group of surfaces. [Layer70]

<Operation>

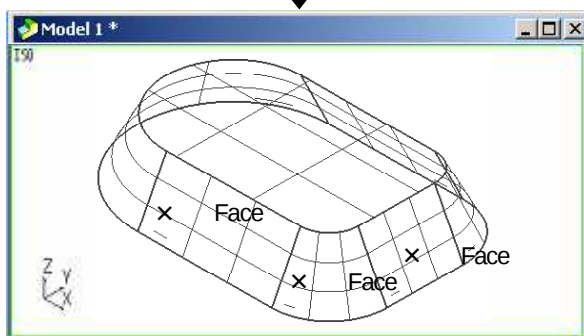


Select the [Fillet Surface] command.
Specify parameters in the command sheet.
Radius:10
Curvature continuous: OFF
Specify surface: Pick



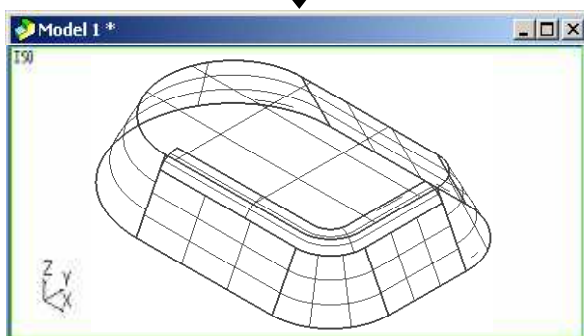
[Fillet Surface] Pick a surface for the first group.

Specify the surface (Face)
Press the center button to determine the specifying.



[Fillet Surface] Pick the second surface.

- Specify the surfaces (Face)
Press the center button to determine the specifying.



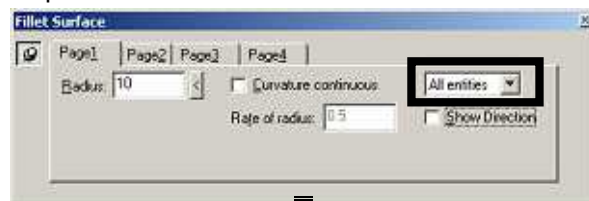
Press the center button to execute the command.

How to create fillet surfaces between surfaces simultaneously (Specify surface: All entities)

All surfaces, which are connected smoothly with the specified surface are automatically selected, and fillet surfaces are created between the surfaces simultaneously.

The [First group of surfaces] and the [Second group of surfaces] need to be connected smoothly. [Layer71]
(If those surfaces are not connected smoothly, a fillet surface will end at the border of the surfaces.)

<Operation>



Select the [Fillet Surface] command.

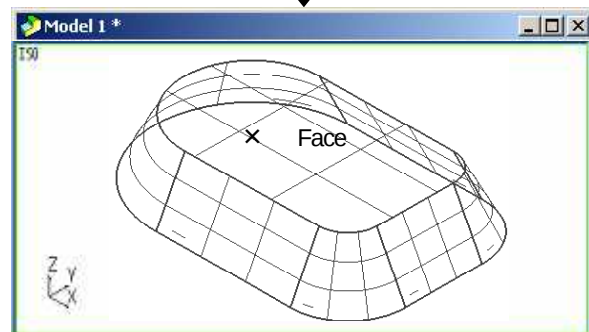
Specify parameters in the command sheet.

Radius:10

Curvature continuous: OFF

Specify surface: All entities

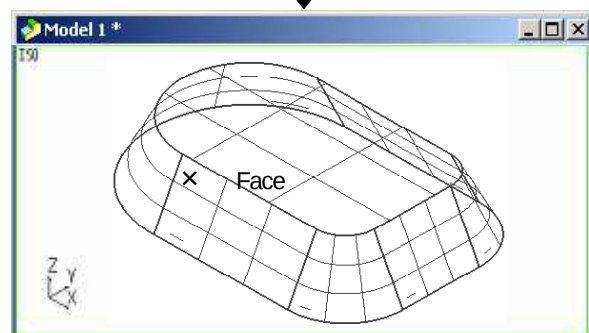
Show direction: OFF



[Fillet Surface] Pick a surface for the first group.

Specify the surface (Face)

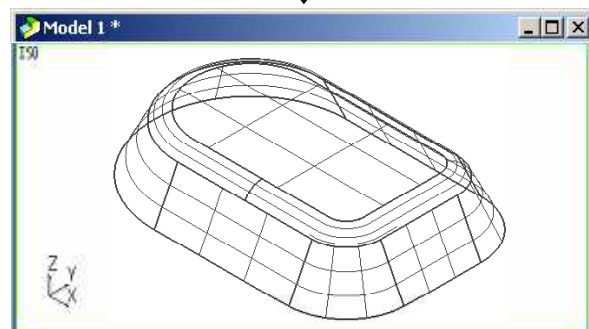
* When [All entities] is selected, you do not need to press the center mouse button after specifying the first group of surfaces.



[Fillet Surface] Pick the second surface.

Specify the surface (Face)

* When [All entities] is selected, you do not need to press the center mouse button after specifying the second group of surfaces.

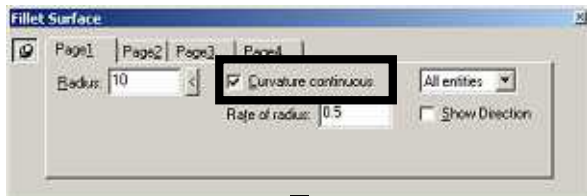


Press the center mouse button to execute the command.

How to create a fillet surface that compensates the border of surfaces based on arc value
(Curvature continuous: ON)

Create a [Fillet surface] keeping the curvature in the normal direction of the surface, with [Curvature continuous] function. Specify the [range] for keeping arc radius. [Layer72]

<Operation>



Select the [Fillet Surface] command.

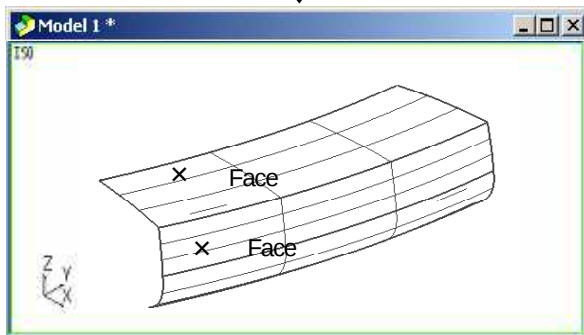
Specify parameters in the command sheet.

Radius:10

Curvature continuous: ON, Rate of radius: 0.5

Specify surface: All entities

Show direction: OFF



[Fillet Surface] Pick a surface for the first group.

Specify the surface (Face)

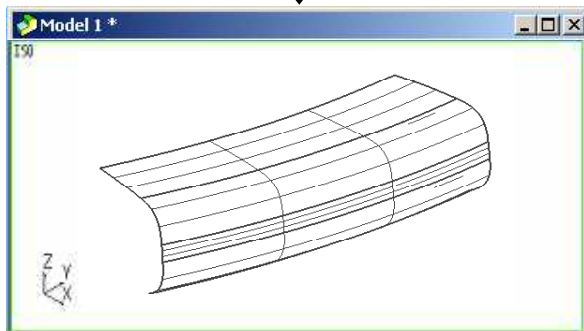
* When [All entities] is selected, you do not need to press the center mouse button after specifying the first group of surfaces.

[Fillet Surface] Pick the second surface.

Specify the surface (Face)

* When [All entities] is selected, you do not need to press the center mouse button after specifying the first group of surfaces.

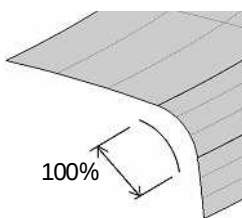
Press the center button to execute the command.



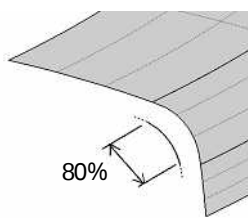
Keep the radius of a fillet surface only within the specified range.

In the area out of the range, a fillet surface is created keeping the curvature in the normal line direction of the surface.

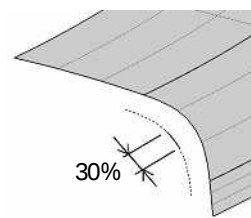
Curvature continuous: OFF



Rate of radius: 0.8



Rate of radius: 0.3

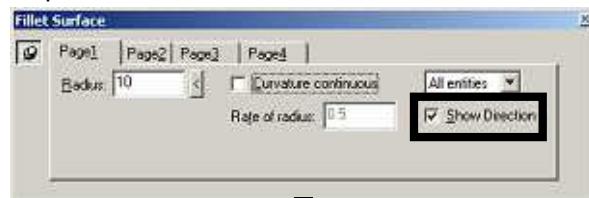


* Since it is not a fillet surface, the radius of the surface cannot be displayed by [Fillet surface Radius Information].

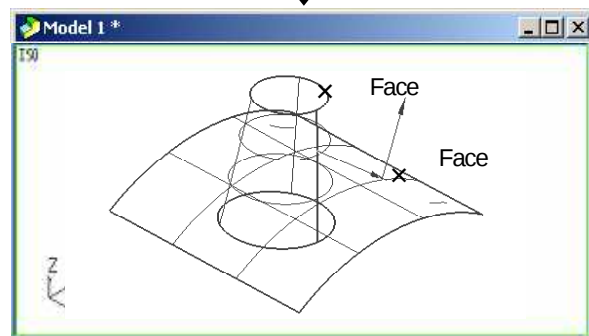
How to confirm direction of creating a fillet surface using arrow (Show direction: ON)

When [Show direction] is ON, an arrow of [indicating the direction of fillet center] is displayed.
The direction of fillet center can be selected by changing the direction of arrow. [Layer73]

<Operation>



Select the [Fillet Surface] command.
Specify parameters in the command sheet.
Radius:10
Curvature continuous: OFF
Specify surface: All entities
Show direction: ON

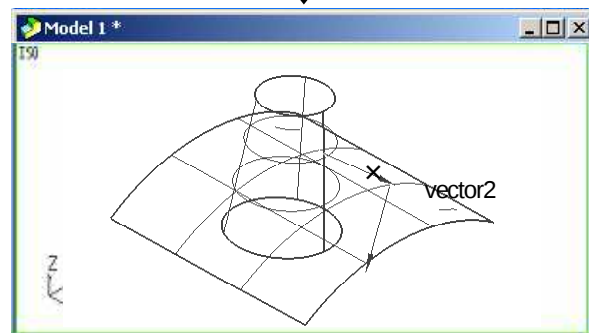


[Fillet Surface] Pick a surface for the first group.

Specify the surface (Face)

[Fillet Surface] Pick the second surface.

Specify the surface (Face)

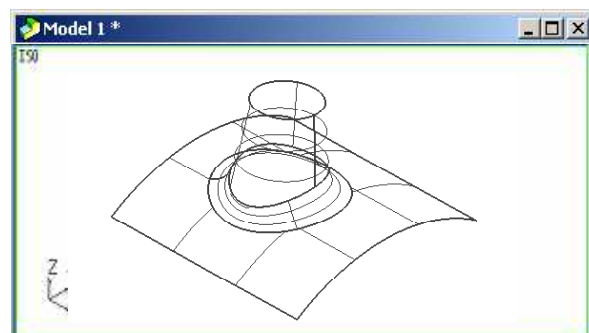


[Fillet Surface] Choose either direction for the fillet-arc center.

Arrow of indicating direction is displayed.

* Click the arrow, and the direction will be changed. If you do not change the direction, you do not need to click the arrow.

*The direction of the arrow is set for the fillet center. Set the arrow to the upper direction this time.



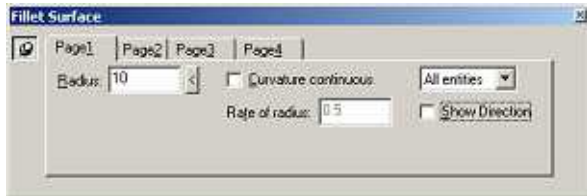
Press the center mouse button to execute the command.

How to extend a surface to create a fillet surface (Extent fillet: ON)

Extend a surface by the specified [start length], [end length] to create a fillet surface.

Use this function when the specified surface is not long enough to create a complete fillet surface. [Layer74]

<Operation>



Select the [Fillet Surface] command.

Specify parameters in the command sheet.

(Page1) Radius:10, Curvature continuous: OFF

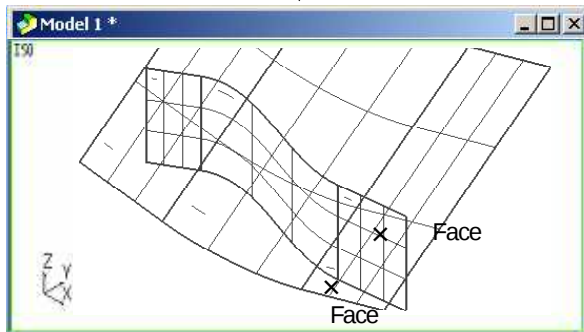
Specify surface: All entities, Show direction : OFF



(Page2) Extend fillet: ON

Start:: 10, End: 15

Extent surface: OFF



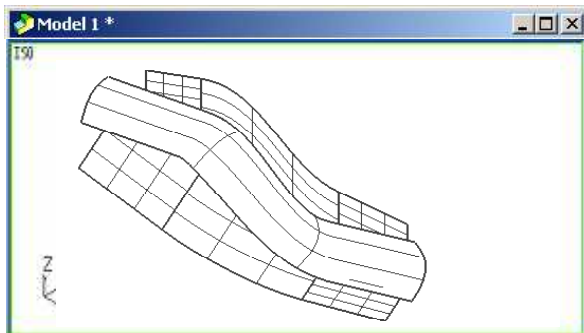
[Fillet Surface] Pick a surface for the first group.

Specify the surface (Face)

* The surface closer to the specified position of the first surface, is extended by the [start length], and the other side surface is extended by the [end length].

[Fillet Surface] Pick the second surface.

Specify the surface (Face)



Press the center button to execute the command.



Restrictions for [Extent fillet] function

Since the created fillet surface is extended, the extended part may not be connected to the surface.
In that case, the specified surface cannot be trimmed.

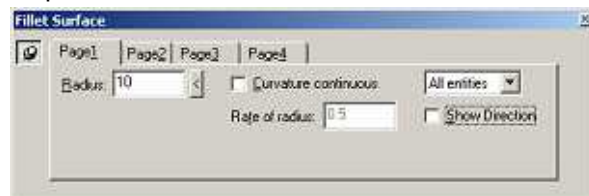
How to extend a surface along surface to create a fillet surface (Extent fillet: ON)

Extend the specified surface internally to create a fillet surface, with the [Extent fillet] function.

Input value for [distance] to extend the original surface. And, if the specified surface is a trimmed surface, the surface is extended to the length of the original surface by [distance: 0]. Only a single surface can be specified for both the [first group of surfaces], the [second group of surfaces]. (Plural surfaces cannot be specified) [Layer75]

* Use this function when you wish to extend a surface along surface, or you cannot trim a surface with the [Extend fillet] function.

<Operation>



Select the [Fillet Surface] command.

Specify parameters in the command sheet.

(Page1) Radius:10, Curvature continuous: OFF

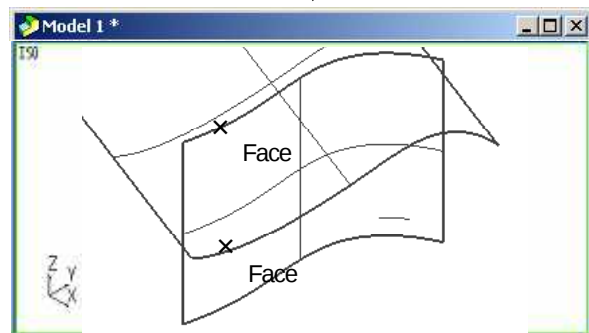
Specify surface: All entities, Show direction: OFF



(Page2) Extent fillet: OFF

Extend surface: ON

Distance: 20



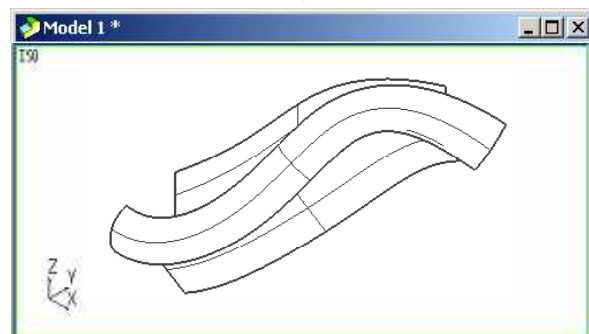
[Fillet Surface] Pick a surface for the first group.

Specify the surface (Face)

[Fillet Surface] Pick the second surface.

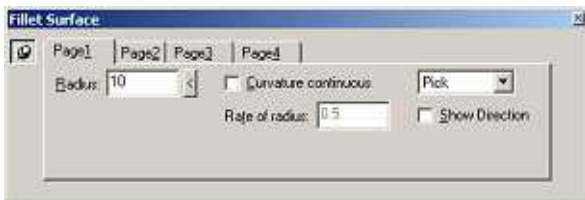
Specify the surface (Face)

Press the center button to execute the command.

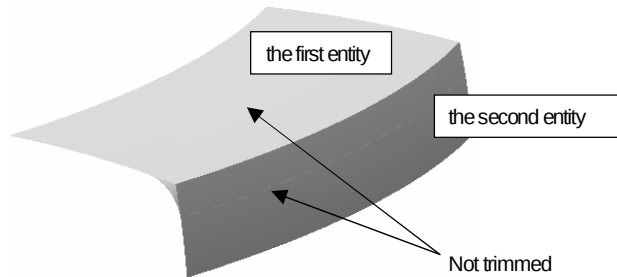


If you wish not to trim the specified surface from the created surfaces..., (Trim)

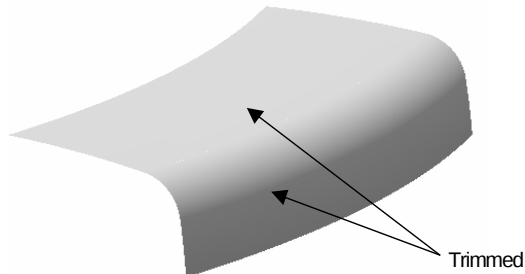
Specify whether trim or not trim the specified surface with the [Trim] function. [Layer76]



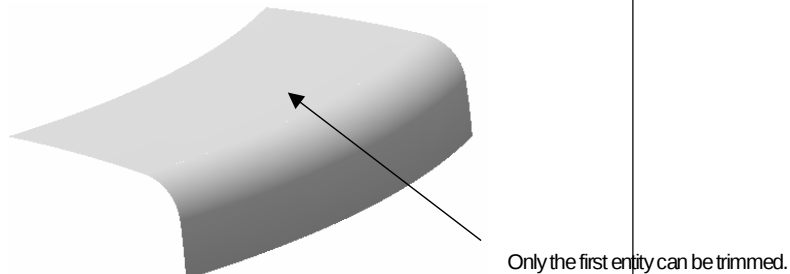
• Trim: OFF



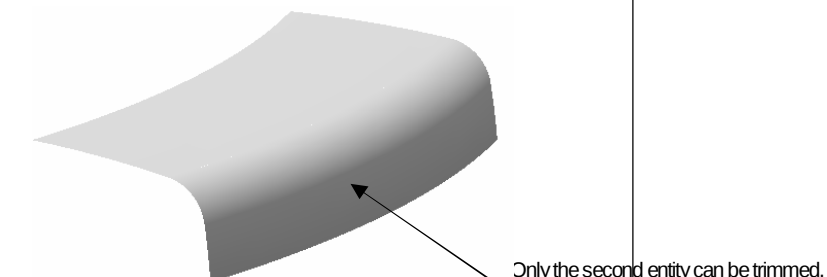
Trim: ON
First: ON
Second: ON



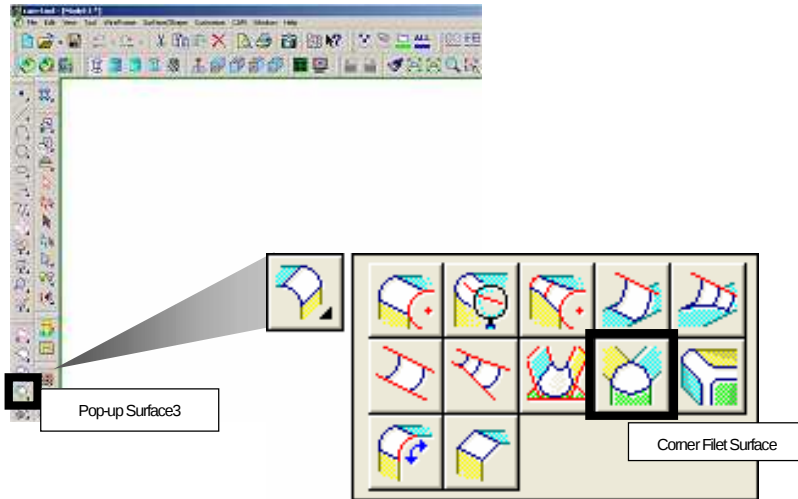
Trim: ON
First: ON
Second: OFF



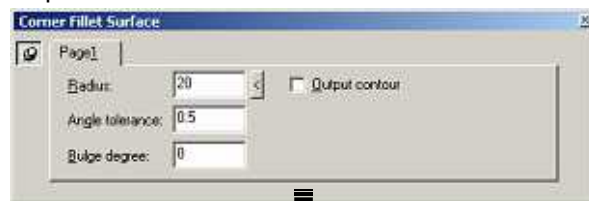
Trim: ON
First: OFF
Second: ON



Create a fillet surface at the corner which has three surfaces. [Layer77]



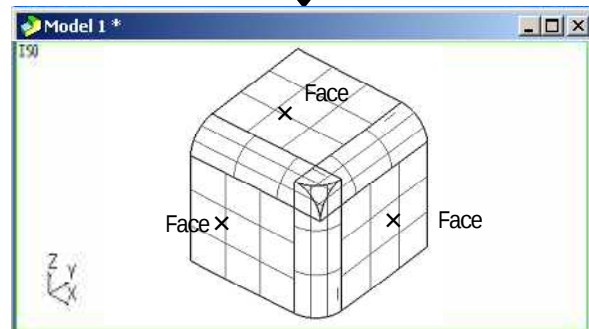
<Operation>



Select the [Corner Fillet Surface] command.

Specify parameters in the command sheet.

Radius:20



[Corner Fillet Surface] Pick the first surface.

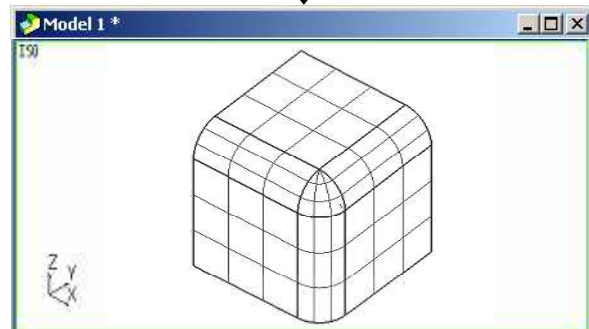
Specify the first surface (Face)

[Corner Fillet Surface] Pick the second surface.

Specify the second surface (Face)

[Corner Fillet Surface] Pick the third surface.

Specify the third surface (Face)



Mesh direction of surface

Mesh direction of [Corner fillet surface] varies depending on the specifying order of surface.
Mesh is unspread in the direction of the third surface (the surface specified last).

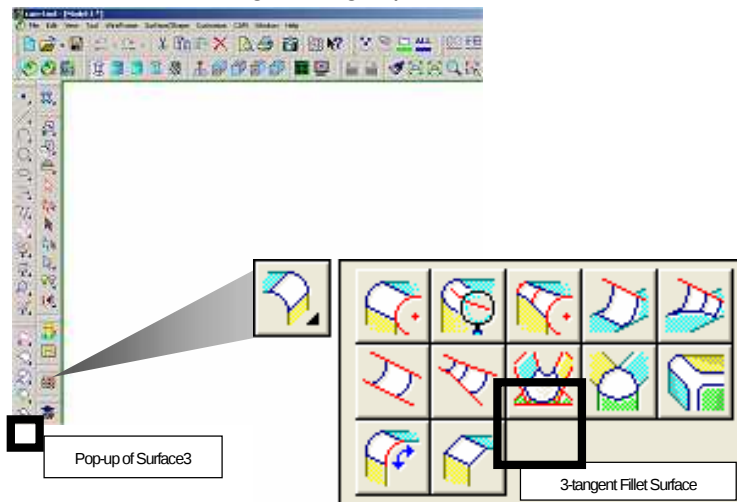
How to create fillet surface tangent to three entities

<3-Tangent Fillet Surface>

Create a fillet surface tangent to three entities with [3-Tangent Fillet Surface]. (Radius is not specified.)

For combination of three surfaces, select one from the following patterns.

- Surface tangent to three groups of surfaces
- Surface tangent to two groups of surfaces, and runs through a group of surfaces
- Surface tangent to one group of surfaces, and runs through two groups of surfaces
- Surface runs through three groups of surfaces



When you use line of intersection of surfaces as guide line...,

When you use X-axis as guide line..., (Guide line: X-axis)

When you use Y-axis as guide line...,

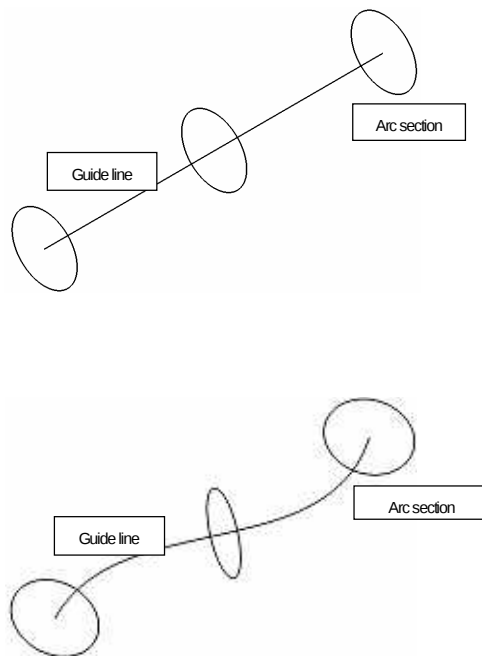
When you specify guide line...,



What is guide line?

Arc section is placed in the normal line direction of [Guide line]. A surface is created to interpolate those arc sections.

Fillet surface is created differently depending on the method of specifying guide line.



Guide line: Specify entity for [Guide line].

X-axis: Specify [X-axis] for [Guide line].

Y-axis: Specify [Y-axis] for [Guide line].

Z-axis: Specify [Z-axis] for [Guide line].

None: Two or more surfaces are specified

>> Line of intersection of the first surface and the second surface is set as guide line.

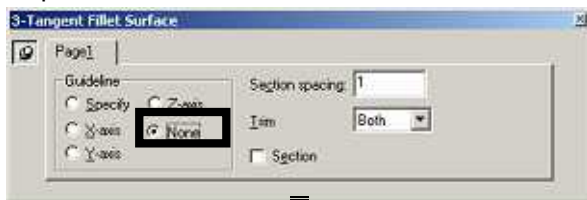
Only one surface is specified

>> The curve specified first is set as guide line.

When you use line of intersection of surfaces as guide line..., (Guide line: None)

Ex.) When two or more surfaces are specified with [Guide line: None] is set, line of intersection of the first group of surfaces and the second group of surfaces is determined as guide line. This function is available only when arc section can be placed. This time, specify [surface] / [surface] / [surface] for three entities here. [Layer78]

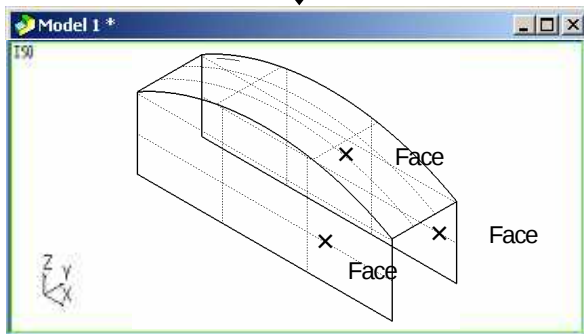
<Operation>



Select the [3-Tangent Fillet Surface] command.

Specify parameters in the command sheet.

Guide line: None



[3-Tangent Fillet Surface] Pick a surface or curve for the first group.

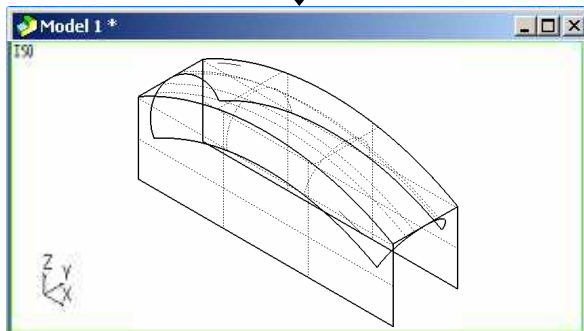
Specify the first group. (Face)

Press the center button to determine the specifying.

[3-Tangent Fillet Surface] Pick a surface or curve for the second group.

Specify the second group. (Face)

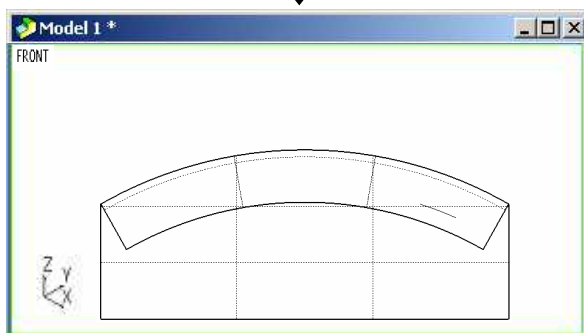
Press the center button to determine the specifying.



[3-Tangent Fillet Surface] Pick a surface or curve for the third group.

Specify the third group. (Face)

Press the center button to determine the specifying.



* Arc section is placed in the normal plane direction of [line of intersection].

When you use X-axis as guide line..., (Guide line: X-axis)

Ex.) Specify [X-axis of work] as guide line, and place arc section in the normal plane direction of X-axis. This time, specify [surface] / [surface] / [surface] for three entities. [Layer79]

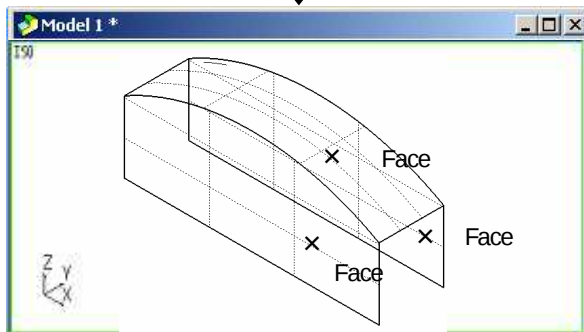
<Operation>



Select the [Fillet Surface] command.

Specify parameters in the command sheet.

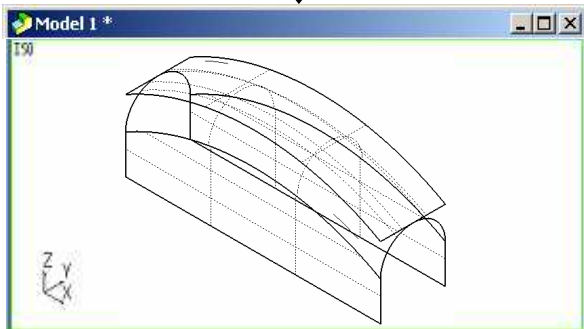
Guide line: X-axis



[3-Tangent Fillet Surface] Pick a surface or curve for the first group.

Specify the first group. (Face)

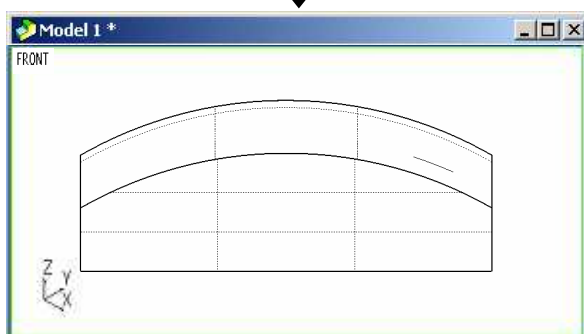
Press the center button to determine the specifying.



[3-Tangent Fillet Surface] Pick a surface or curve for the second group.

Specify the second group. (Face)

Press the center button to determine the specifying.



[3-Tangent Fillet Surface] Pick a surface or curve for the third group.

Specify the third group. (Face)

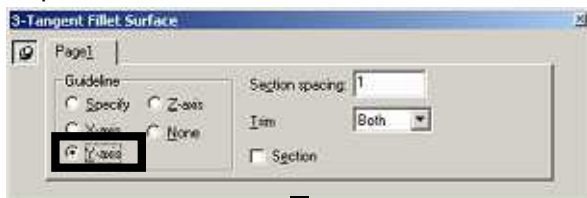
Press the center button to determine the specifying.

* Arc section is placed in the normal plane direction of [X-axis].

When you use Y-axis as guide line..., (Guide line: Y-axis)

Ex.) Specify [Y-axis of work] as guide line, and place arc section in the normal plane direction of Y-axis. This time, specify [surface] / [curve] / [surface] for three entities. [Layer80]

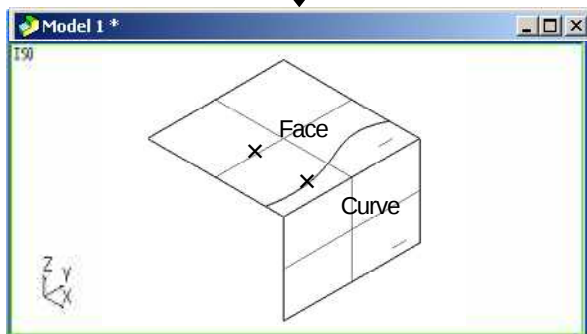
<Operation>



Select the [Fillet Surface] command.

Specify parameters in the command sheet.

Guide line: Y-axis



[3-Tangent Fillet Surface] Pick a surface or curve for the first group.

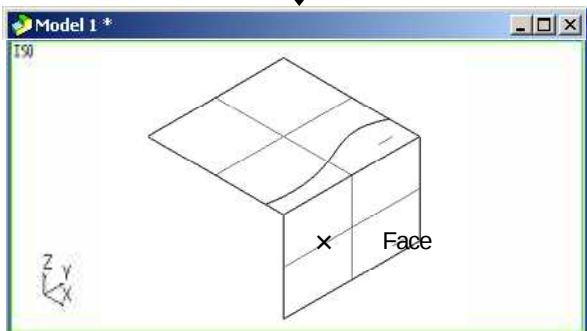
Specify the first group. (Face)

Press the center button to determine the specifying.

[3-Tangent Fillet Surface] Pick a surface or curve for the second group.

Specify the second group. (Face)

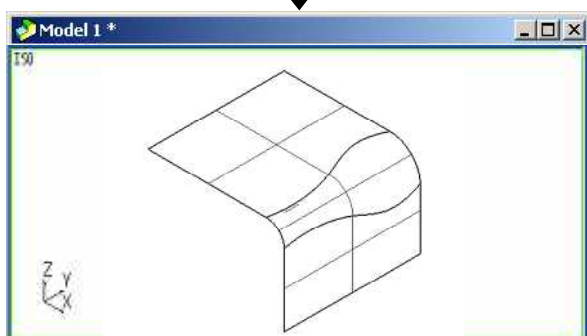
Press the center button to determine the specifying.



[3-Tangent Fillet Surface] Pick a surface or curve for the third group.

Specify the third group. (Face)

Press the center button to execute the command.



When you specify guide line..., (Guide line: Specify)

Ex.) Specify guide line, and place arc section in the normal plane direction of guide line. This time, specify [surface] / [curve] / [surface] for three entities. [Layer81]

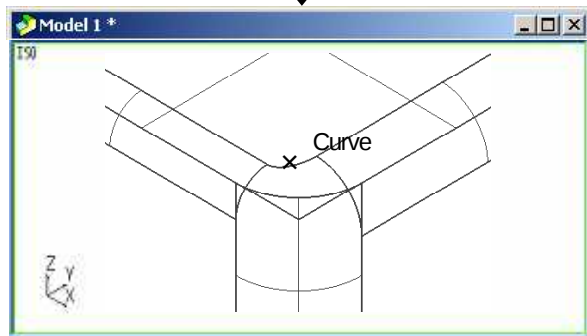
<Operation>



Select the [Fillet Surface] command.

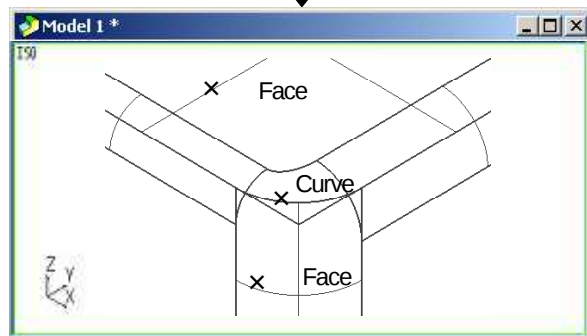
Specify parameters in the command sheet.

Guide line :Specify



[3-Tangent Fillet Surface] Pick the guide line.

Specify the curve for guide line.(Curve)



[3-Tangent Fillet Surface] Pick a surface or curve for the first group.

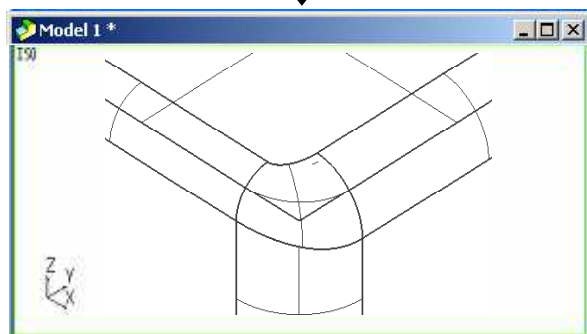
Specify the first group. (Face)

Press the center button to determine the specifying.

[3-Tangent Fillet Surface] Pick a surface or curve for the second group.

Specify the second curve. (Curve)

Press the center button to determine the specifying.



[3-Tangent Fillet Surface] Pick a surface or curve for the third group.

Specify the third group. (Face)

Press the center button to execute the command.

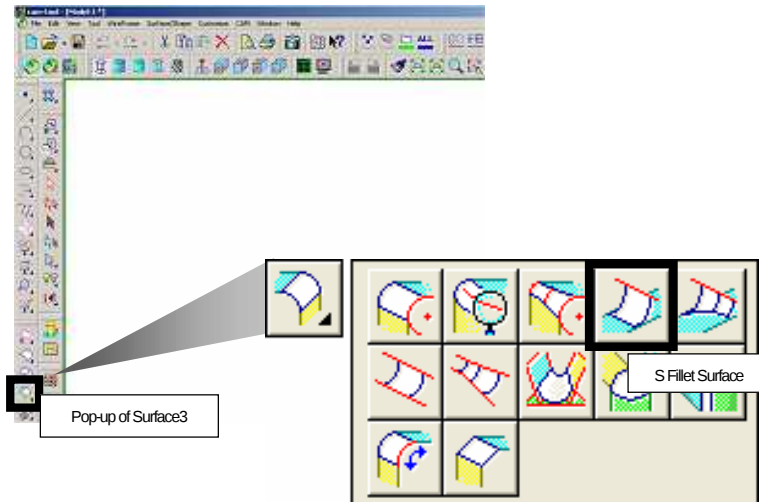


This guide line has been created with [Blend curve] command. (Refer to page107)

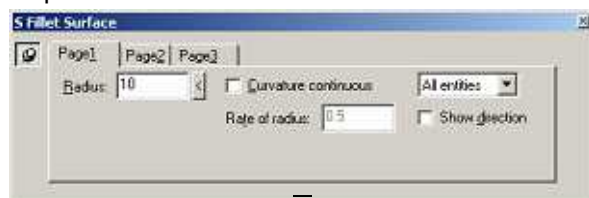
How to create fillet surface between surface and curve

<S Fillet Surface>

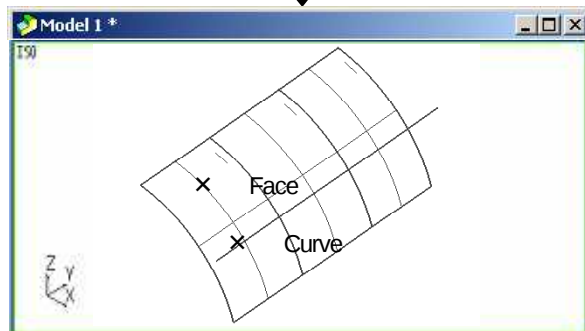
Create [Fillet surface] that has a single radius between curve and a group of surfaces that are connected smoothly.,
[Layer82]



<Operation>



Select the [S Fillet Surface] command.
Specify parameters in the command sheet.
Radius:10
Specify surface: All entities

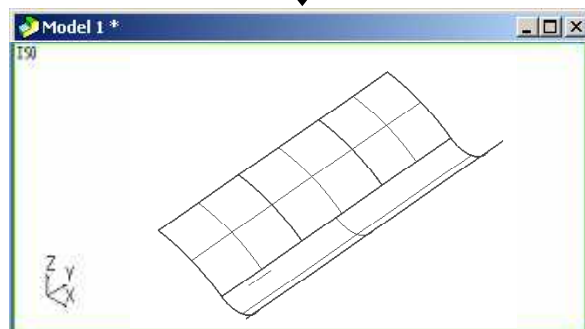


[S Fillet Surface] Pick the guide line.

Specify the curve. (Curve)

[S Fillet Surface] Pick a surface.

Specify the surface. (Face)



Press the center button to execute the command.

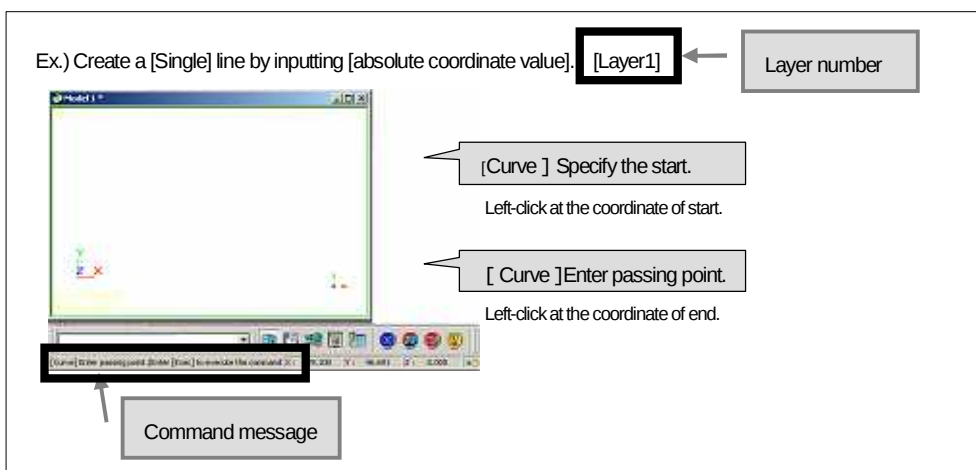
Chapter 8

Edit surface

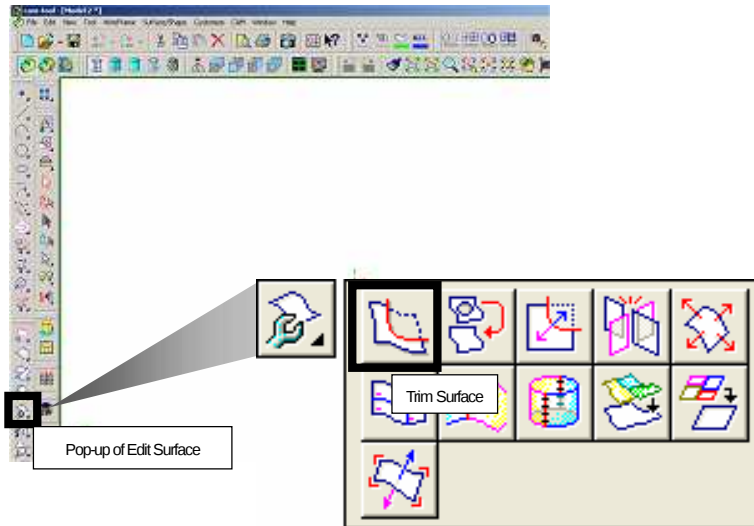
Open the model file below. (Same model file as in [Chapter3 - Chapter7])

CD-ROM/Intensive Course/CADmodel2.gmd

- If [Layer number] is written in the description, change the active layer number.
- Follow the [command message] displayed at the lower left of the screen, for your operation.



Cut (Trim) the specified surface with entity (border).



How to trim surface with wire frame

How to trim surface with other surface

How to trim surfaces with a simple operation

How to trim other surface with surface contour (Method1)

How to trim other surface with surface contour (Method2)

How to specify plural areas as to be left

How to trim surface with small surface

How to trim both surfaces

How to trim surface with unadjacent surface or wire frame

How to trim surface with wire frame

Trim surface with contour. [Layer90]

<Operation>

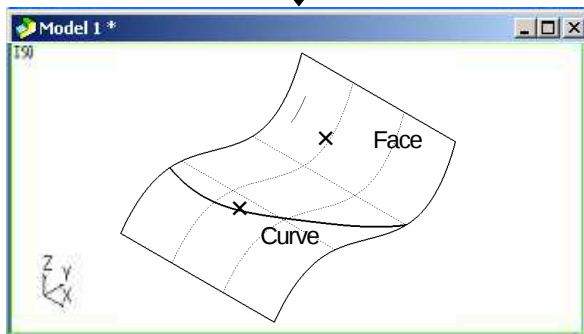


Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON (Other parameters: All OFF)

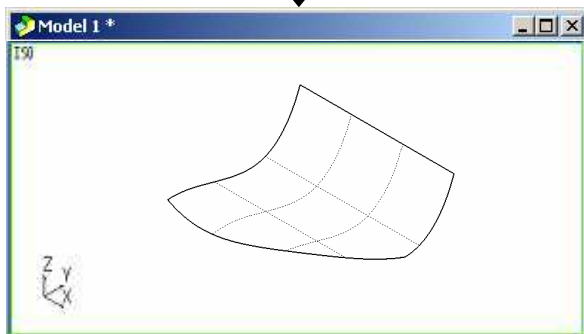
Direction of Project: Not project



[Trim Surface] Pick a surface.

Specify the surface. (Face)

Press the center button to determine the specifying.



[Trim Surface] Pick a trimming entity.

Specify the contour. (Curve)

Press the center button to execute the command.



Basically, wire frame is need to be on the surface. If wire frame does not exist on the surface, use [Direction of Project] function. (Refer to page 138)

How to trim surface with other surface

Trim surface with the [Line of intersection] of the surface and the surface to be referred. [Layer91]

<Operation>

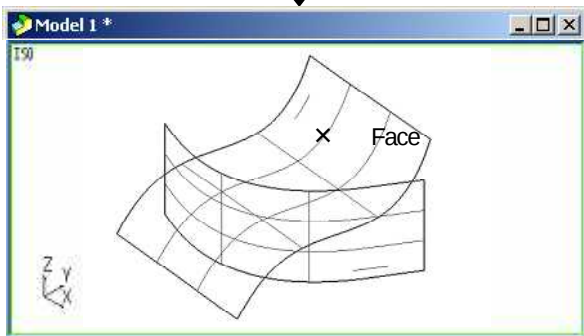


Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON (Other parameters: All OFF)

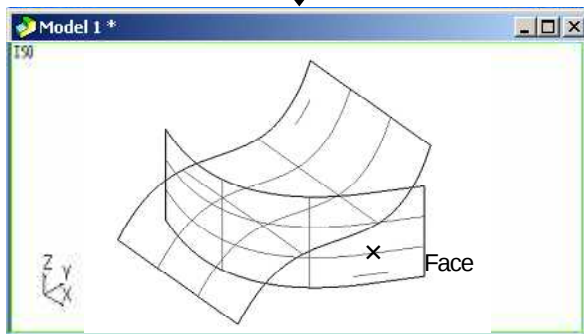
Direction of Project: Not project



[Trim Surface] Pick a surface.

Specify the surface to be left. (Face)

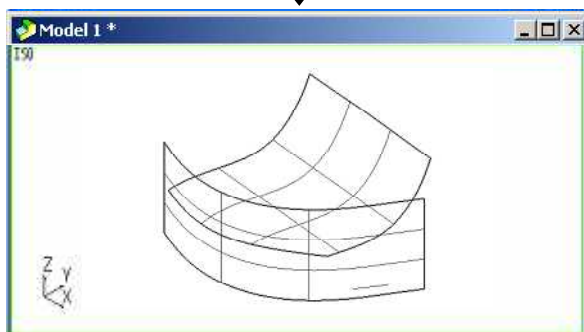
Press the center button to determine the specifying.



[Trim Surface] Pick a trimming entity.

Specify the surface to be referred. (Face)

Press the center button to execute the command.

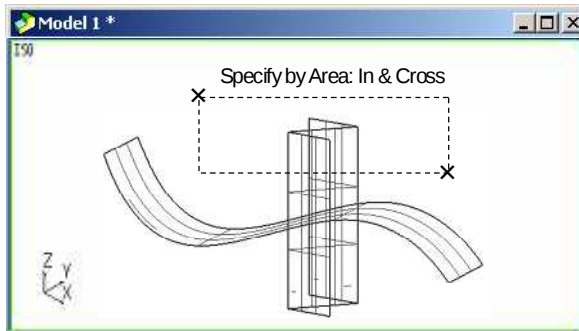


Basically, surface to be referred is need to be larger than surface to be trimmed. If surface to be referred is smaller, use the [Trim extended] function. (Refer to page 136)

How to trim surfaces with a simple operation (Surface contour: ON)

Put the surface to be referred into the selected status, and execute the [Trim Surface] command. Operation of specifying surface to be referred can be omit.[Layer92]

<Operation>



Execute the [Select (Entity)] command.

[Select (Entity)] Pick an entity.

Right-click >> Pop-up menu /In & Cross

Specify the area includes entities to be referred.

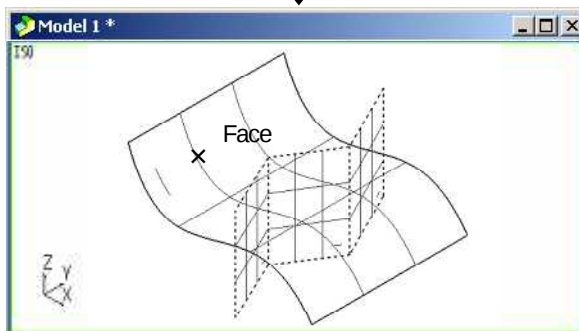


Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON (Other parameters: All OFF)

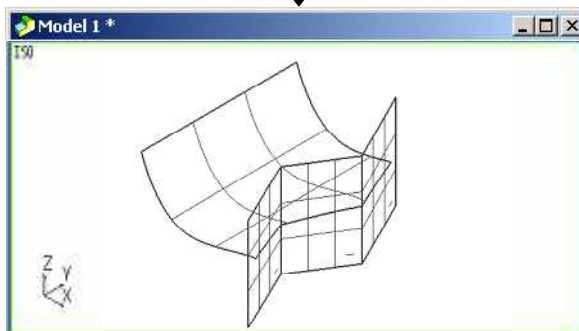
Direction of Project: Not project



[Trim Surface] Pick a surface.

Specify the surface to be left. (Face)

Press the center button to determine the specifying.



[Trim Surface] Pick a trimming entity.

Specify the surface to be referred.

(Already selected) >> No need to specify here.

Press the center button to execute the command.



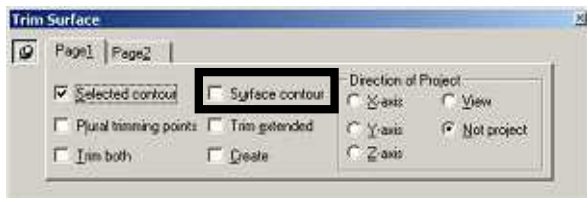
[Selected contour] function is available only when the selected entities exist.

Even if no selected entity exist, [Selected contour] can be set ON in the command sheet.

How to trim other surface with surface contour (Method1) (Surface contour: OFF)

When the surface to be trimmed and the surface to be referred are not crossed, [Line of intersection] is not determined. In this case, trim surface with surface contour. [Layer93]

<Operation>



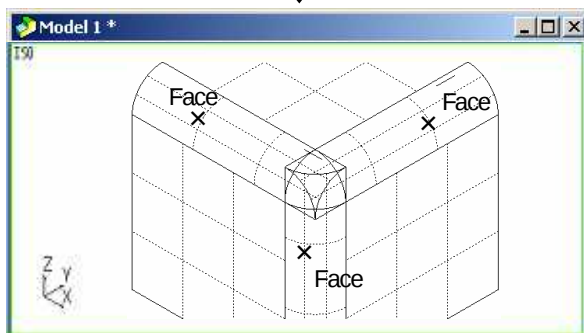
Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON

Surface contour: OFF (Other parameters: All OFF)

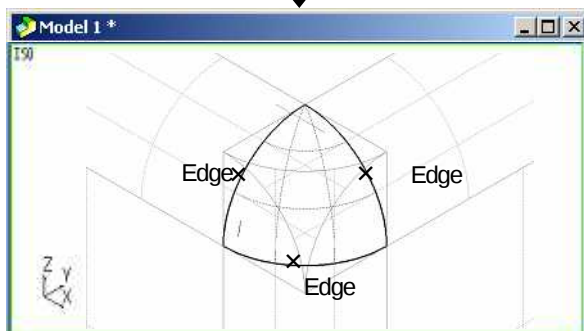
Direction of Project: Not project



[Trim Surface] Pick a surface.

- Specify the surfaces to be left. (Face)

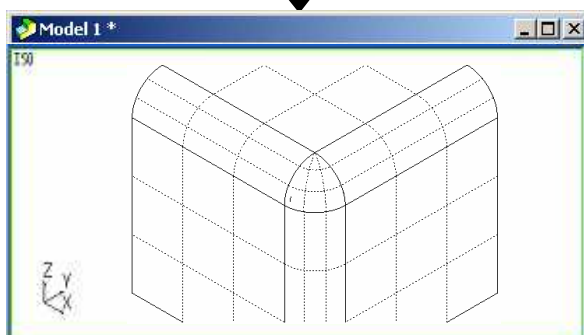
Press the center button to determine the specifying.



[Trim Surface] Pick a trimming entity.

- Specify the surface contours. (Face)

Press the center button to execute the command.



How to trim other surface with surface contour (Method2) (Selected contour: ON)

When the surface to be trimmed and the surface to be referred are not crossed, trim surface with surface contour.
[Layer94]

<Operation>



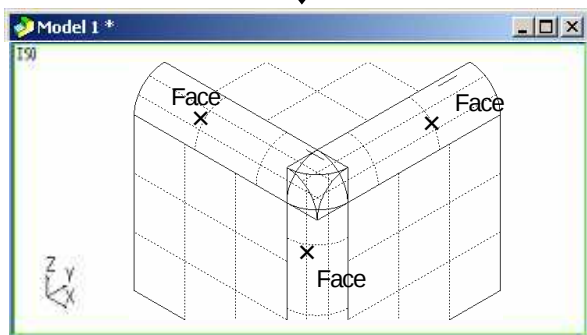
Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON

Surface contour: ON (Other parameters: All OFF)

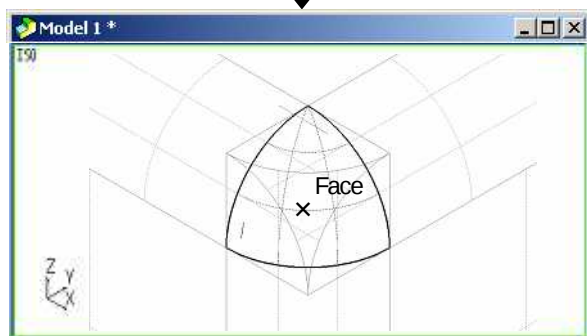
Direction of Project: Not project



[Trim Surface] Pick a surface.

- Specify the surfaces to be left. (Face)

Press the center button to determine the specifying.

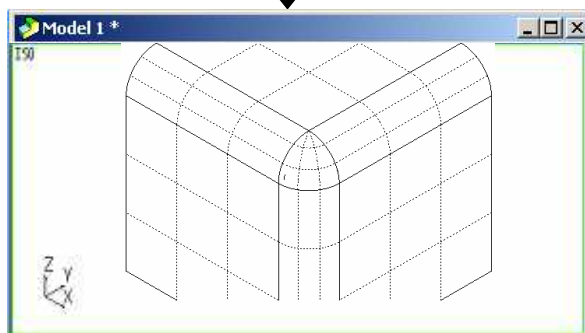


[Trim Surface] Pick a trimming entity.

Specify the surface (Mesh) to be referred. (Face)

* You can also specify a contour instead of mesh, as the surface to be referred.

Press the center button to execute the command.



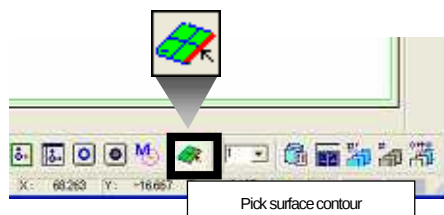


[Pick surface contour] button

Set the button ON, and surface contour can be selected. Set the button OFF, and surface contour cannot be selected.

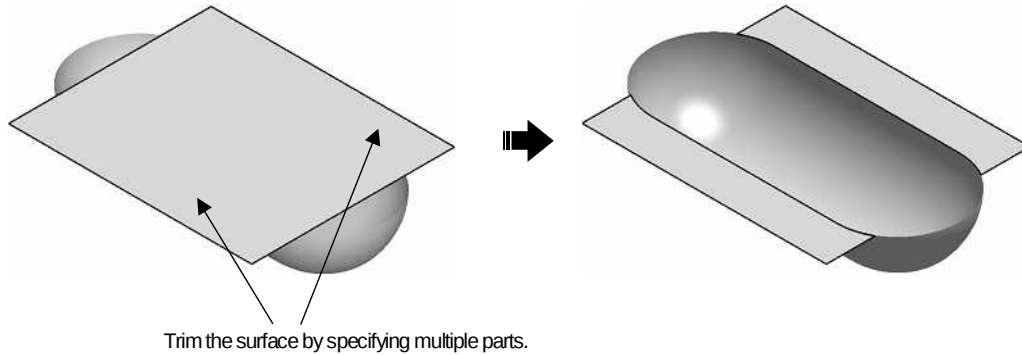
Basically, set it ON, and only when you do not wish to select surface contour, set it OFF.

When [Pick surface contour] button is set OFF, contour of [surface] cannot be specified in the operation of [How to trim other surface with surface contour (Method1)].

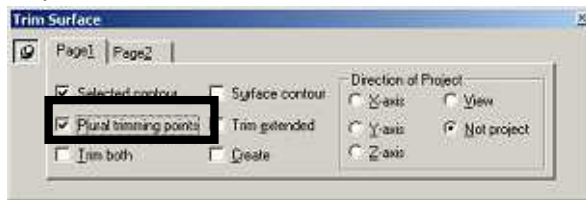


How to specify plural areas as to be left (Plural trimming point: ON)

Trim surface by specifying multiple parts of the surface to be left. Multiple surfaces cannot be specified as the surface to be trimmed. [Layer95]



<Operation>



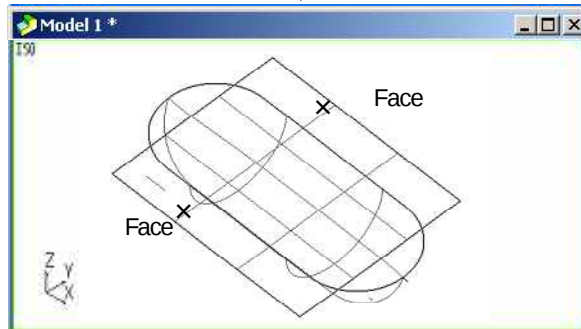
Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON

Plural trimming points: ON (Other parameters: All OFF)

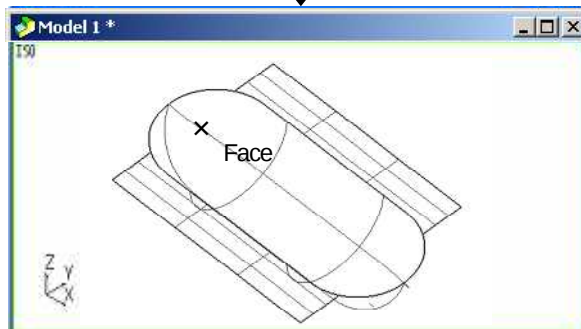
Direction of Project: Not project



[Trim Surface] Pick a surface.

- Specify the surfaces to be left. (Face)

Press the center button to determine the specifying.



[Trim Surface] Pick a trimming entity.

Specify the surface (Mesh) to be referred. (Face)

Press the center button to execute the command.

How to trim surface with small surface (Trim extended: ON)

When the surface to be referred is shorter than the surface to be trimmed, [line of intersection] is lengthened on the surface and the surface is trimmed. [Layer96]

<Operation>



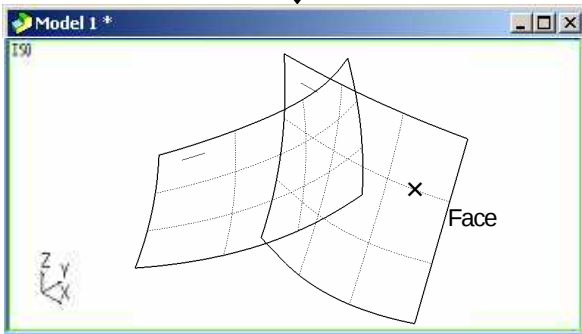
Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON

Trim extended: ON (Other parameters: All OFF)

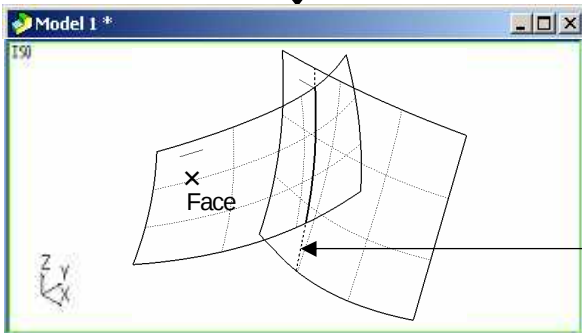
Direction of Project: Not project



[Trim Surface] Pick a surface.

Specify the surfaces to be left. (Face)

Press the center button to determine the specifying.

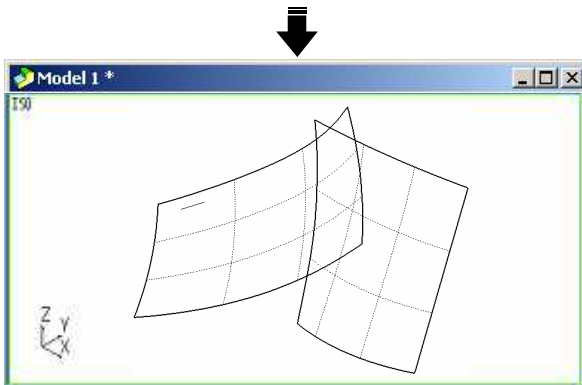


[Trim Surface] Pick a trimming entity.

Specify the surface to be referred. (Face)

Press the center button to execute the command.

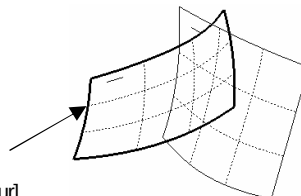
* [Line of intersection] is lengthened on the surface and the surface is trimmed.



When [surface contour] is set ON in the command sheet,

[Trim extended] function may be invalid.

Surface may be trimmed with [Surface contour].



How to trim both surfaces (Trim both: ON)

Trim both the surface to be trimmed and the surface to be referred, with the line of intersection of both surfaces, each other.

Specify [Layer97]

<Operation>



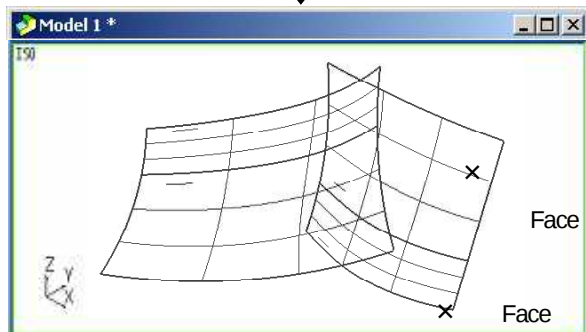
Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON

Trim both: ON (Other parameters: All OFF)

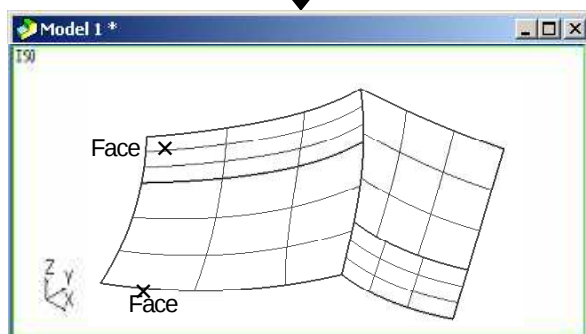
Direction of Project: Not project



[Trim Surface] Pick a surface.

- Specify the surfaces to be trimmed. (side to be left)
(Face)

Press the center button to determine the specifying.



[Trim Surface] Pick a trimming entity..

- Specify the surfaces to be referred. (side to be left)
(Face)

Press the center button to execute the command.

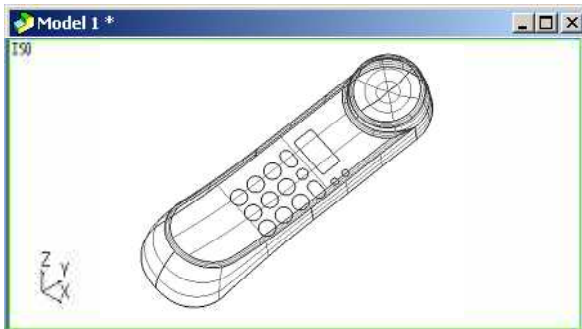
How to trim surface with unadjacent surface or wire frame (Direction of project)

When you trim a surface with the [surface to be referred] or [wire frame], which are not adjacent to the surface to be trimmed, project the [surface to be referred] or [wire frame] in the specified direction for trimming. [Layer98]

* If there are many [surface to be referred] or [wire frame], put those entities in the selected status in advance, and operations of [Trim surface] can be reduced.

Ex.) Project wire frame (button shapes) on the work plane to the top surface of the phone, and execute [Trim].

<Operation>



Execute the [Select (Except Surface)] command.

* Entities of button shape are selected.

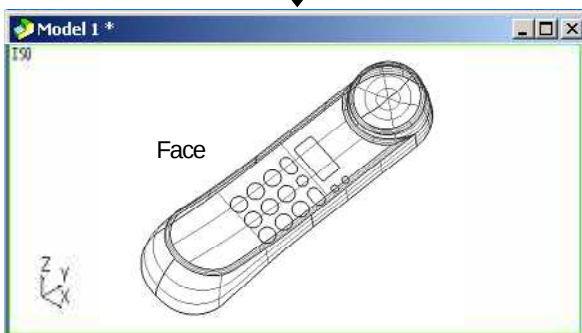


Select the [Trim Surface] command.

Specify parameters in the command sheet.

Selected contour: ON (Other parameters: All OFF)

Direction of Project: Z-axis

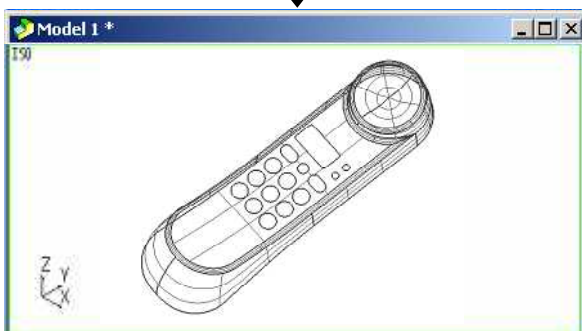


[Trim Surface] Pick a surface.

Specify the surface to be trimmed. (side to be left)
(Face)

* Set [View direction (TOP)] and click the position to be left.

Press the center button to determine the specifying.



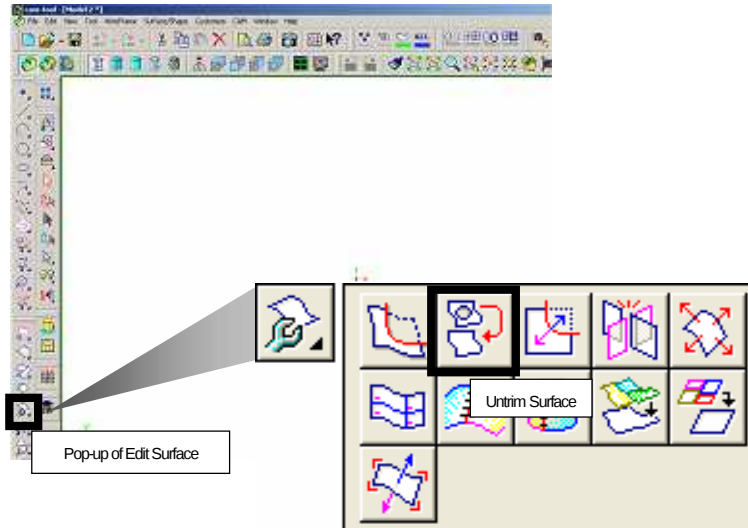
[Trim Surface] Pick a trimming entity.

Specify the surfaces to be referred.

(Already selected) >> No need to select here

Press the center button to execute the command.

Restore the trimmed surface with the [Untrim Surface].



How to cancel the trimmed the surface to restore original surface

How to untrim

How to untrim inner contour and create entity at the position

Create another surface at the trimmed part of surface

How to cancel the trimmed of the surface to restore original surface (All)

Basically, cancel the all of trimmed parts of the surface to restore the original surface [Layer100]

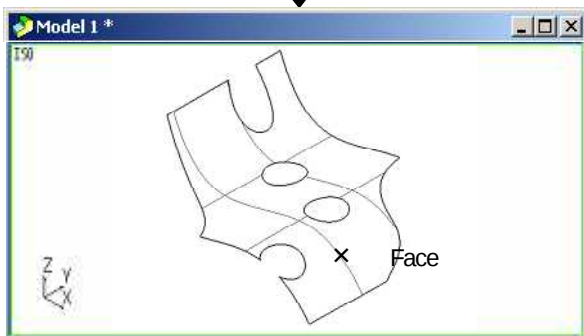
<Operation>



Select the [Untrim Surface] command.

Specify parameters in the command sheet.

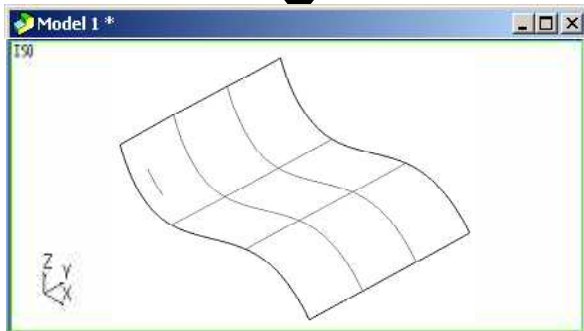
Part to untrim: All



[Untrim Surface] Pick a trimmed surface.

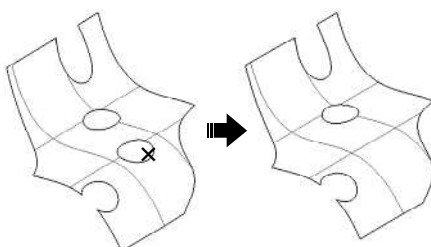
Specify the surface to be untrimmed. (Mesh). (Face)

Press the center button to execute the command.

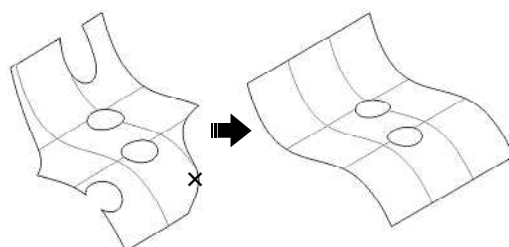


If surface contour is specified, only [specified contours] can be untrimmed.

<If inner contour is specified..., >



<If outer contour is specified..., >



How to untrim surface partially (Specified part)

When plural parts are trimmed in a surface, use this function to [untrim surface partially]. [Layer101]

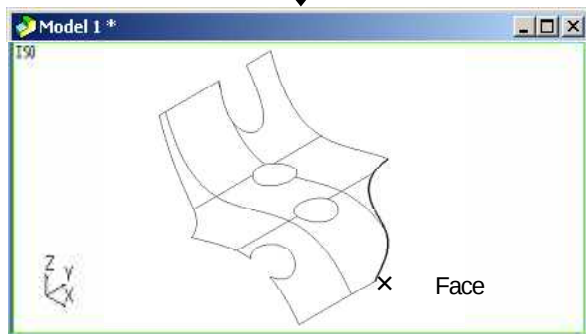
<Operation>



Select the [Untrim Surface] command.

Specify parameters in the command sheet.

Part to untrim: Specified part

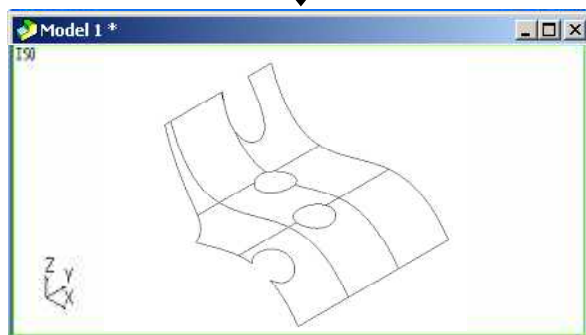


[Untrim Surface] Pick a trimmed boundary to detach.

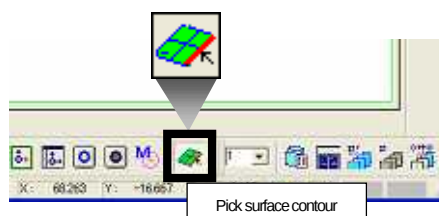
Specify the surface contour to be untrimmed. (Face)

* Mesh cannot be specified with the [Specified part] function.

Press the center button to execute the command.



- Complex surface such as plural surface contours are overlapped, may turn original surface.
- When [Pick surface contour] is OFF, [contour] of surface cannot be specified.



How to untrim inner contour and create entity at the position (Part to untrim: Inside)

Untrim the trimmed part (inner contour) of the trimmed surface, and create curves at the position where the contours existed. [Layer102]

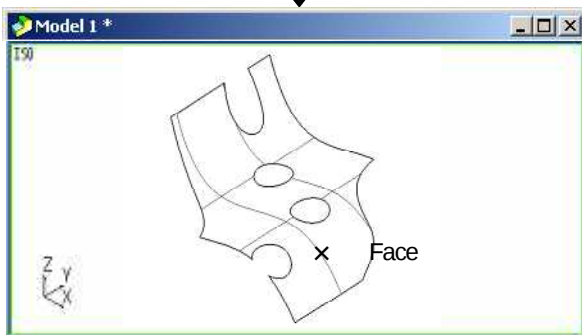
<Operation>



Select the [Untrim Surface] command.

Specify parameters in the command sheet.

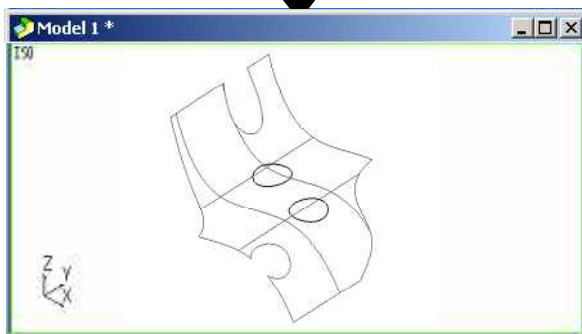
Part to untrim: Inside



[Untrim Surface] Pick a trimmed surface.

Specify the surface to be untrimmed. (Mesh). (Face)

Press the center button to execute the command.

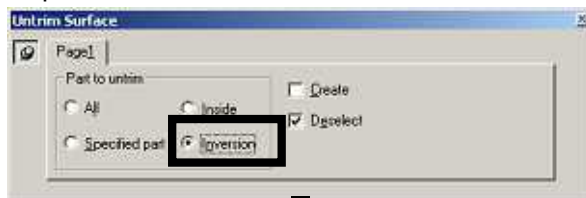


Curve entities are created at the position where trimmed contours existed.

Create another surface at the trimmed part of surface (Part to untrim: Inversion)

When inner contours exist on the trimmed surface, untrim the outer contour, and create another surfaces at the inner contours. [Layer103]

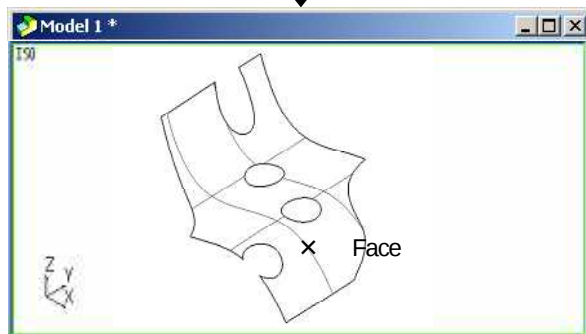
<Operation>



Select the [Untrim Surface] command.

Specify parameters in the command sheet.

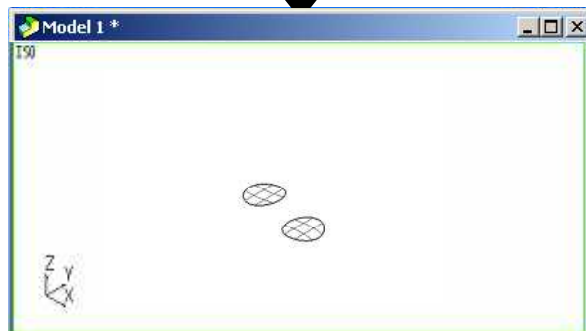
Part to untrim: Inversion



[Untrim Surface] Pick a trimmed surface.

Specify the surface to be untrimmed. (Mesh). (Face)

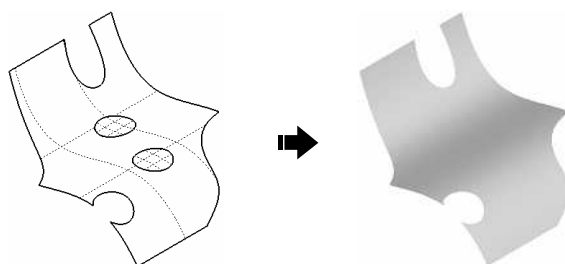
Press the center button to execute the command.



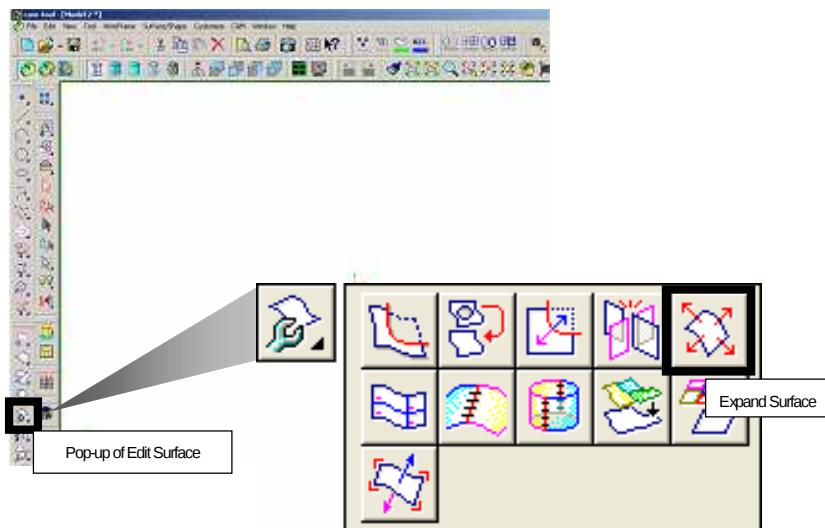
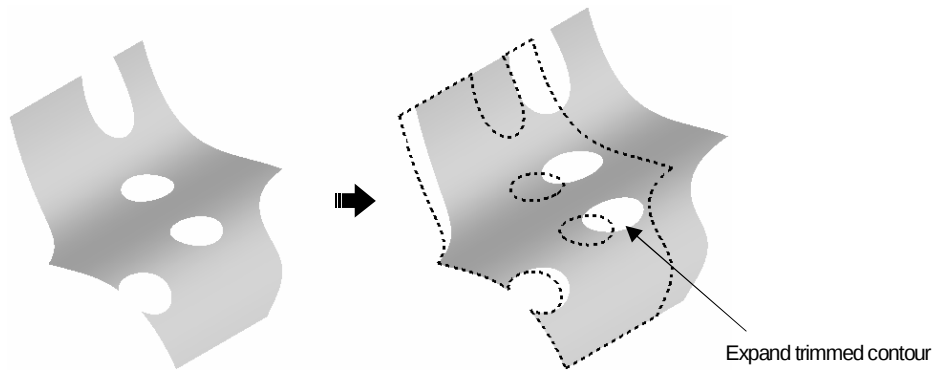
Another surfaces are created at the inner contours.



If [Inversion] is used with [Create] ON, the inner trimmed part is untrimmed with another surface.



Expand or contract surface based on the original surface. If you expand trimmed surface, trimmed contour may also be expanded. To prevent mistakes from occurring, specify parameters in the command sheet properly, or refer to examples in this text book.



How to expand surface linearly

How to expand surface along surface smoothly

How to expand/contract surface by specifying parameter

How to expand trimmed surface

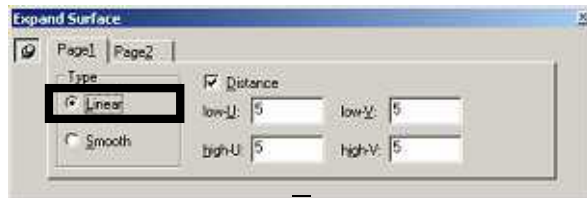
How to expand trimmed surface partially

How to expand surface linearly (Liner)

Expand surface linearly.

For expanding surface, specify one direction from [U+], [U-], [V+], [V-] direction displayed on the surface.
[Layer104]

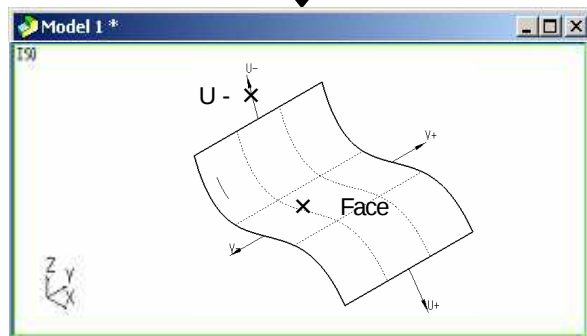
<Operation>



Select the [Expand Surface] command.

Specify parameters in the command sheet.

Type: Linear



[Expand Surface] Pick a surface.

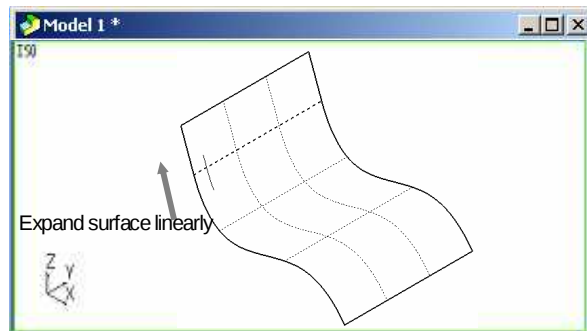
Specify the surface to be expanded. (Face)

[Expand Surface] Pick expanding /contracting direction.

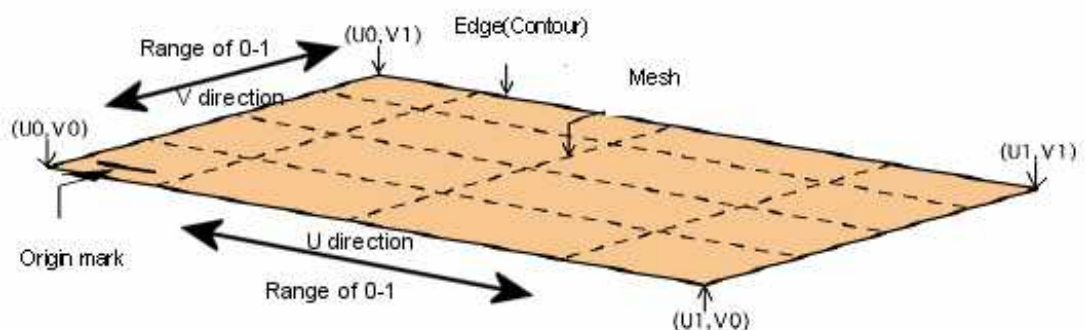
Specify the direction. (Multiple directions can be selected.)

Input value for the specified direction in the command sheet. Ex.) U- : 30

Press the center button to execute the command.



Surface is expanded in the specified direction selected from [U+], [U-], [V+], [V-] direction.
If the direction is not specified, surface is expanded in all directions.

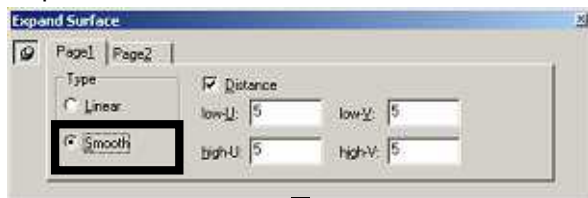


How to expand surface along surface smoothly (Smooth)

Expand surface along surface smoothly.

For expanding surface, specify one direction from [U+], [U-], [V+], [V-] direction displayed on the surface.
[Layer104]

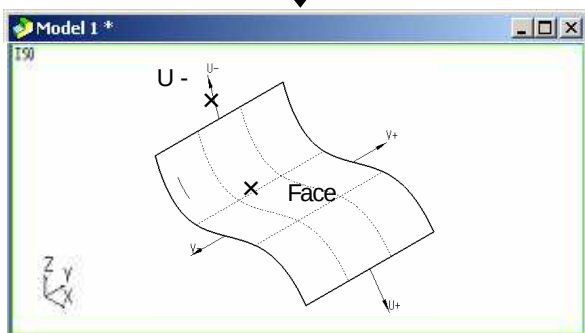
<Operation>



Select the [Expand Surface] command.

Specify parameters in the command sheet.

Type: Smooth



[Expand Surface] Pick a surface.

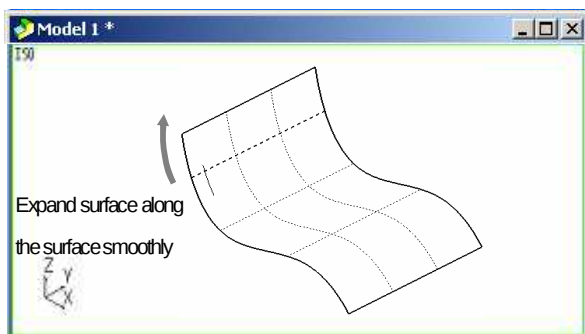
Specify the surface to be expanded. (Face)

[Expand Surface] Pick expanding /contracting direction.

Specify the direction. (Multiple directions can be selected.)

Input value for the specified direction in the command sheet.

Ex.) U- : 30



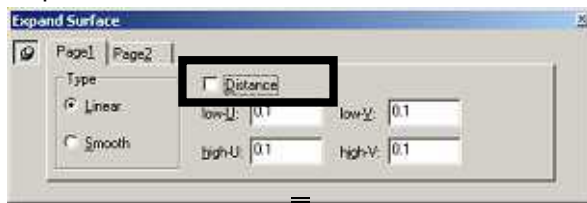
Press the center button to execute the command.

How to expand/contract surface by specifying parameter (Distance: ON/OFF)

Specify unit for executing [Expand Surface]. [Layer105]

- Distance: ON Specify length by [mm].
- Distance: OFF Specify length by parameter.

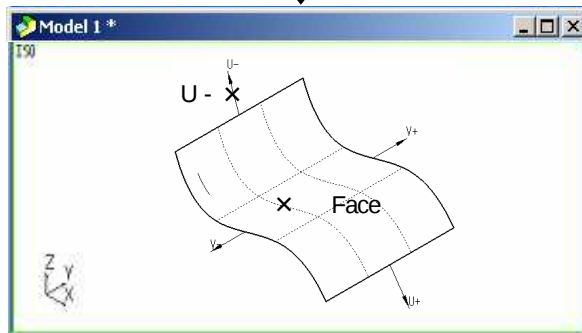
<Operation>



Select the [Expand Surface] command.

Specify parameters in the command sheet.

Distance: OFF



[Expand Surface] Pick a surface.

Specify the surface to be expanded. (Face)

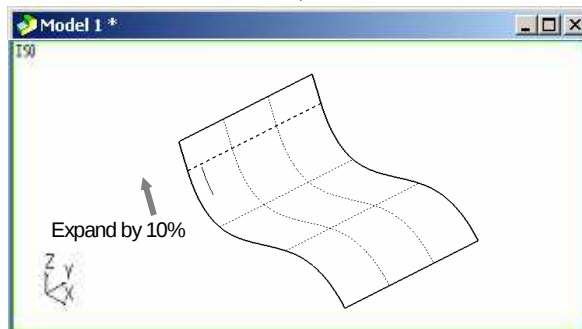
[Expand Surface] Pick expanding /contracting direction.

Specify the direction. (Multiple directions can be selected.)

Input value for the specified direction in the command sheet.

Ex.) U- : 0.1 (Expand original surface by 10%)

Press the center button to execute the command.



When you specify length by parameter with the [Distance] is set OFF, [minus(-)] value can be input.

It is available for contracting surface as well.

* When [Distance] is set ON, [minus(-)] value cannot be input.

How to expand trimmed surface (Untrim to extend: ON)

When you expand trimmed surface, expand the surface without untrimming, or untrim the surface and expand it.

* When [Untrim to extend] is set ON, [Keep trim contour] is automatically set OFF. [Layer106]

- Untrim to extend: ON Untrim the trimmed surface and expand it.
- Untrim to extend: OFF Expand the trimmed surface without untrimming.

<Operation>

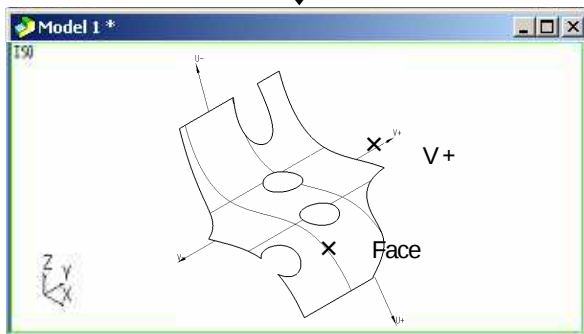


Select the [Expand Surface] command.

Specify parameters in the command sheet.

Untrim to extend :ON

(* Keep trim contour: OFF)



[Expand Surface] Pick a surface.

Specify the surface to be expanded. (Face)

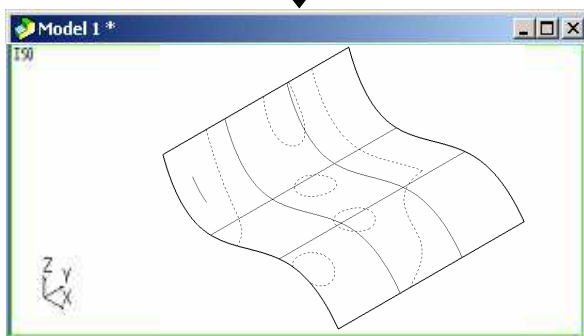
[Expand Surface] Pick expanding /contracting direction.

Specify the direction. (Multiple directions can be selected.)

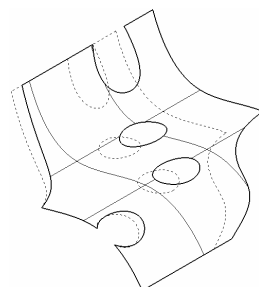
Input value for the specified direction in the command sheet.

Ex.) V+ : 20

Press the center button to execute the command.



When [Untrim to extend] / [Keep trim contour] is OFF in the command sheet, trimmed contour is also expanded.



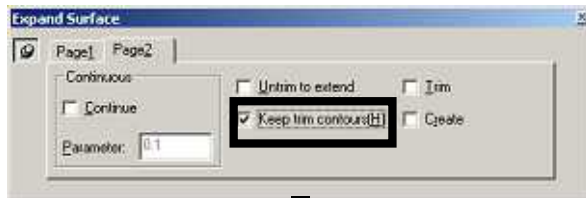
How to expand trimmed surface partially (Keep trim contour: ON)

When [Expand Surface] is executed, basically, original surface is expanded. However, part of surface can be expanded keeping trimmed contour depending on the settings in the command sheet.

* When [Keep trim contours set ON], [Untrim to extend] is automatically set OFF. [Layer107]

- Keep trim contour: ON Expand part of surface keeping trimmed contour.
- Keep trim contour: OFF Trimmed contour is also expanded.

<Operation>

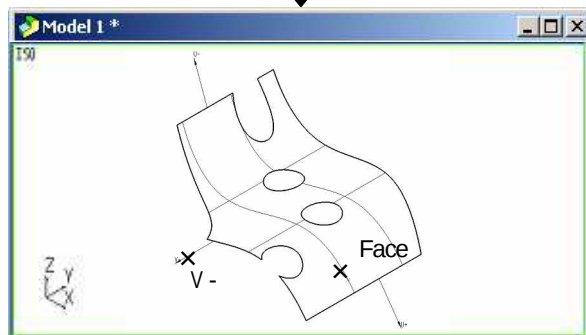


Select the [Expand Surface] command.

Specify parameters in the command sheet.

Keep trim contour: ON

(Untrim to extend: OFF)

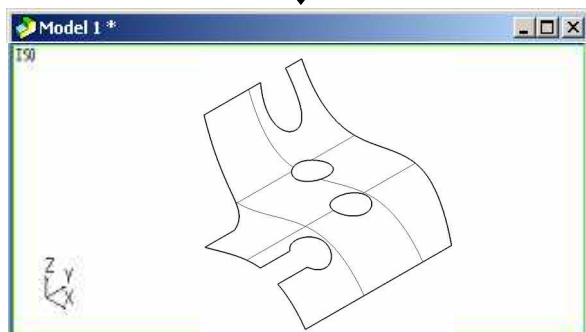


[Expand Surface] Pick a surface.

Specify the surface to be expanded. (Face)

[Expand Surface] Pick expanding /contracting direction.

Specify the direction. (Multiple directions can be selected.)



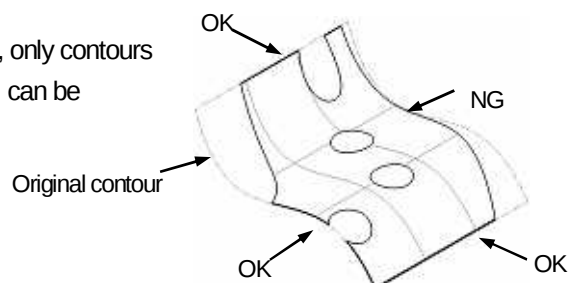
Input value for the specified direction in the command sheet.

Ex.) V- : 20

Press the center button to execute the command.



When [Keep trim contour] is set ON, only contours which original surface contour is left, can be expanded.





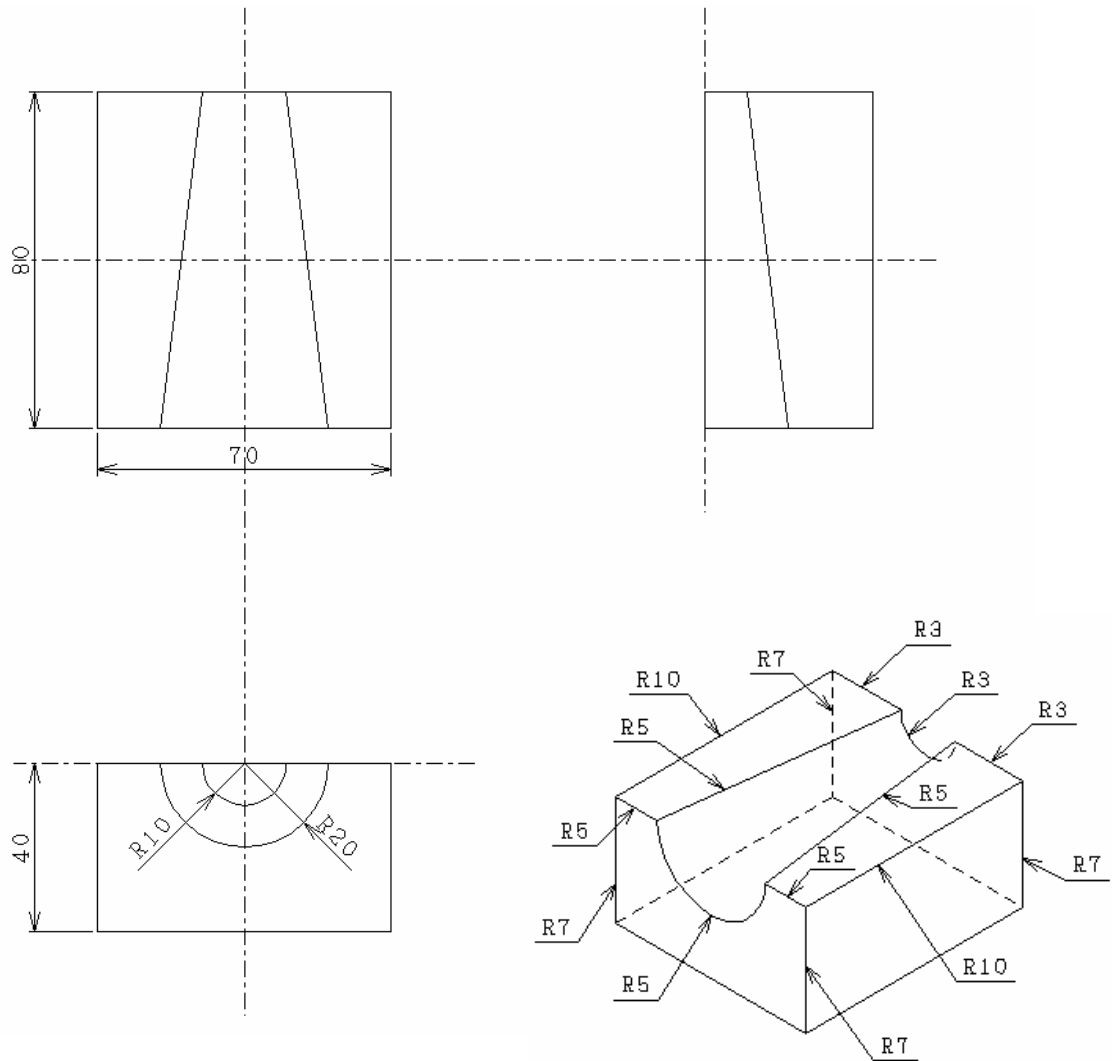
Chapter 9

Exercise

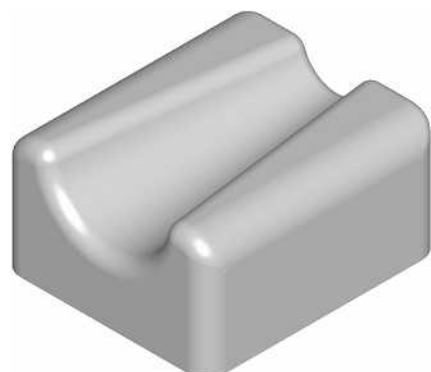
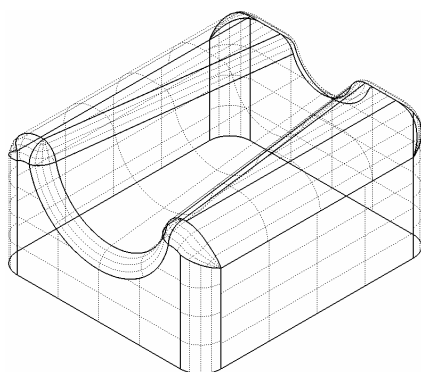
- Create /Edit shape -

Exercise1

Create the following shape.



<Completed shape>

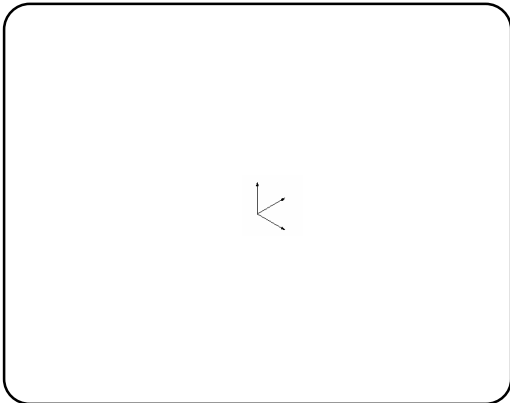


1. Open a new model file.



Select the [Open(New)] command.

2. Create a rectangle.



Select the [Rectangle] command.

Specify parameters in the command sheet.

Specify length: ON

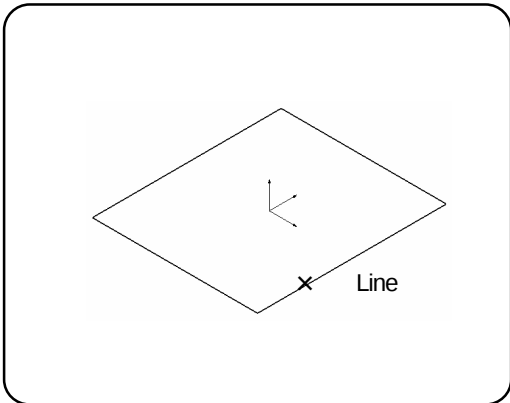
Horizontal: 70

Vertical: 80

Plane: XY plane

Specify the center point. >> [F12]

3 . Put the created rectangle into the selected state.



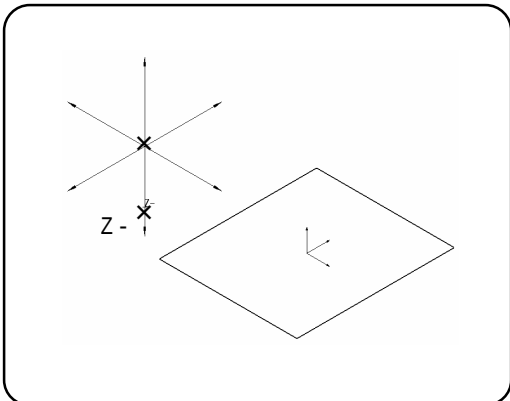
Execute the [Select(Entity)] command.

Right-click >> pop-up menu >> Chain

Specify the entity to be selected. (Line)

* If you put an entity into the selected state in advance, you can omit the operation of selecting the entity in the next command.

4. Copy the selected rectangle in the Z- direction.



Select the [Move/Copy] command.

Specify parameters in the command sheet.

Copy, Distance: 40, Select: OFF

Specify the entity. (Already selected >> No need to select here).

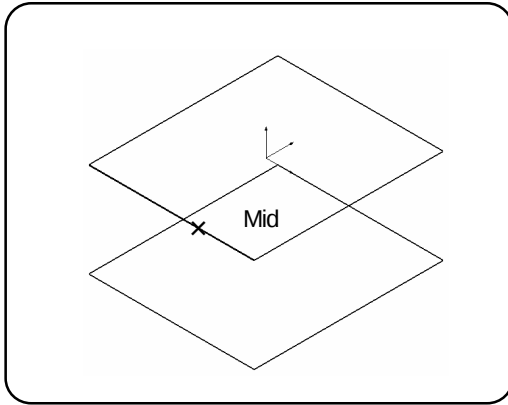
Press the center button to execute the command.

Click a certain position on the screen.

Select the entity (Z- axis) from the displayed guide axes.

Press the center button to execute the command.

5. Create a circle of R20 on the ZX plane.



Select the [Circle (Radius, Center)] command.

Specify parameters in the command sheet.

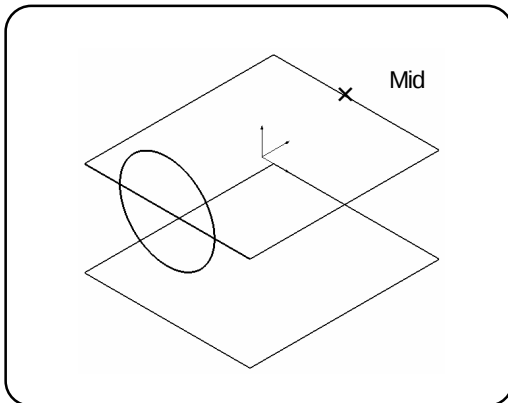
Plane: ZX Plane

Radius: 20

[Snap: Middle]: ON

Specify the center point. >> (Mid)

6. Create a circle of R10.



Select the [Circle (Radius, Center)] command.

Specify parameters in the command sheet.

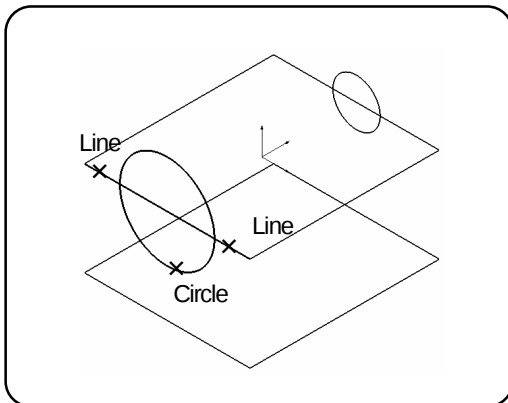
Plane: ZX Plane

Radius: 10

Specify the center point. >> (Mid)

[Snap: Middle]: OFF

7 . Trim the circle of R20 to make it to be arc.



Select the [Trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

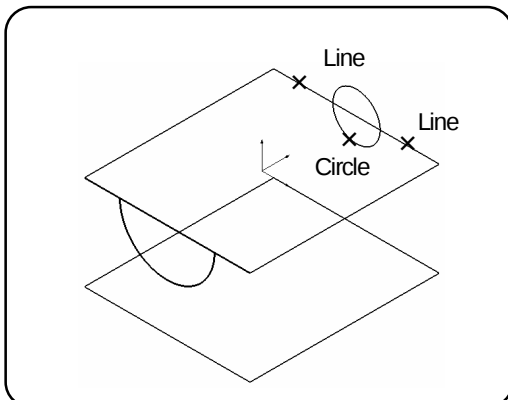
Entity number: Single

Specify the entity to be trimmed. (Circle)

Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

8 . Trim the circle of R10 to make it to be arc.



Select the [Trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

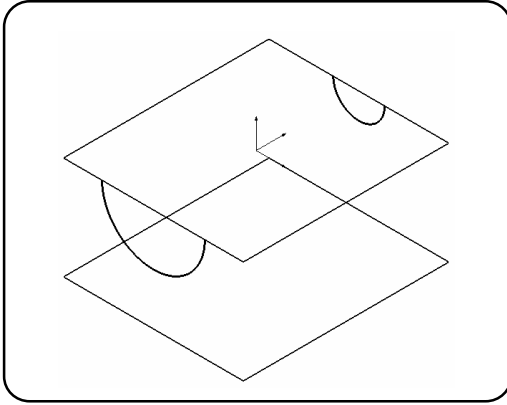
Entity number: Single

Specify the entity to be trimmed. (Circle)

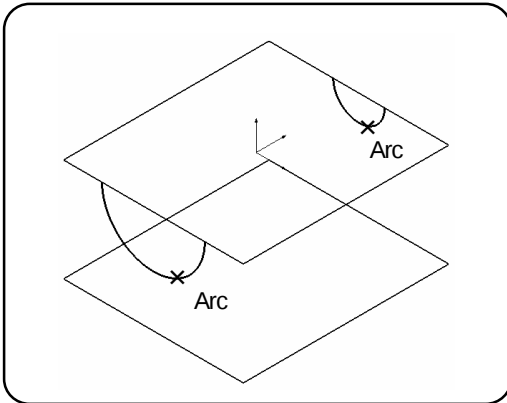
Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

9 . Wire frames have been created. Next, create surfaces.



10 . Create a cylinder part with [Cylinder Surface].



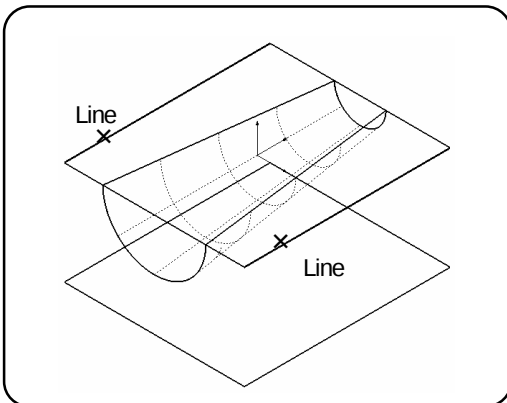
Select the [Multi surface] command.

Specify the contour. (Arc)

Specify the contour. (Arc)

Press the center button to execute the command.

11 . Create the top surface with [Multi surface] command.



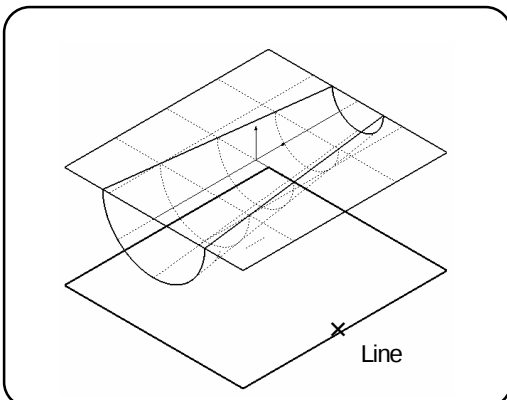
Select the [Multi surface] command.

Specify the contour. (Line)

Specify the contour. (Line)

Press the center button to execute the command.

12 . Put the lower rectangle into the selected state.

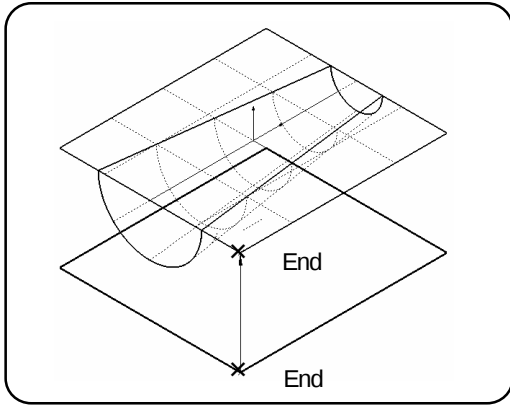


Execute the [Select(Entity)] command.

Right-click >> pop-up menu >> Chain

Specify the entity to be selected. (Line)

13 . Create side surfaces with [Cylinder surface] command.



Select the [Cylinder surface] command.

Specify the entity. (Already selected >> No need to select here).

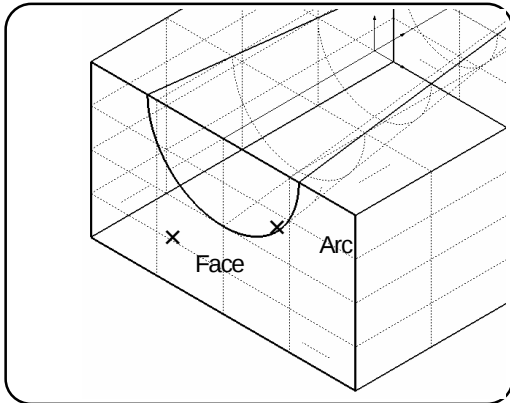
Press the center button to determine the specifying.

Specify the start point to create the cylinder surface.
(End/Intersect)

Specify the end point to create the cylinder surface.
(End/Intersect)

Press the center button to execute the command.

14 . Trim the side surface with the arc or surface contour.



Select the [Trim surface] command.

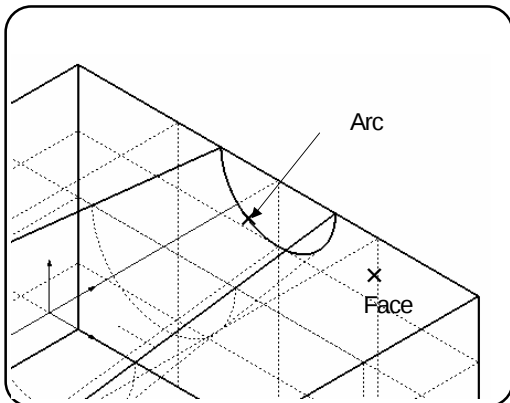
Specify the surface to be trimmed. (Face)

Press the center button to determine the specifying.

Specify the surface contour. (Arc/Face)

Press the center button to execute the command.

15 . Trim the side surface with the arc or the surface contour.



Select the [Trim surface] command.

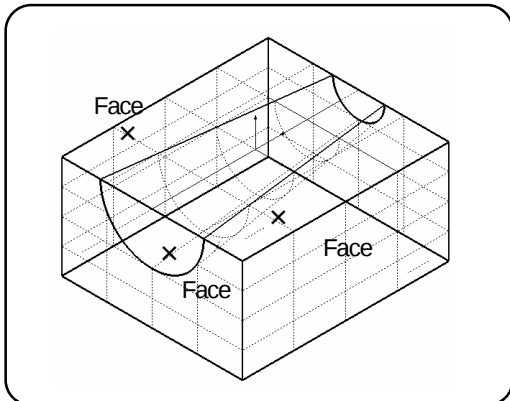
Specify the surface to be trimmed. (Face)

Press the center button to determine the specifying.

Specify the surface contour. (Arc/Face)

Press the center button to execute the command.

16 . Trim the top surface with [Cylinder surface].



Select the [Trim surface] command.

Specify parameters in the command sheet.

Plural trimming points: ON

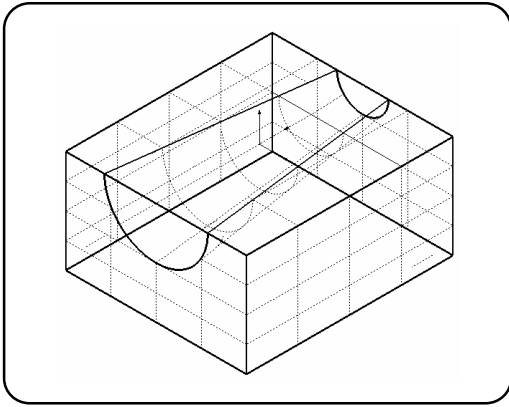
- Specify the surface to be trimmed. (Face)

Press the center button to determine the specifying.

Specify the reference surface. (Face)

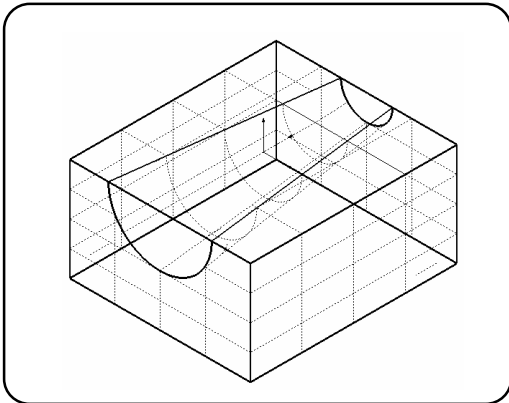
Press the center button to execute the command.

17 . Put the wire frame into the selected state.



Execute the [Select (Except surface)] command.

18 . Move the selected wire frames to layer [10].



Select the [Move/Copy] command.

Specify parameters in the command sheet.

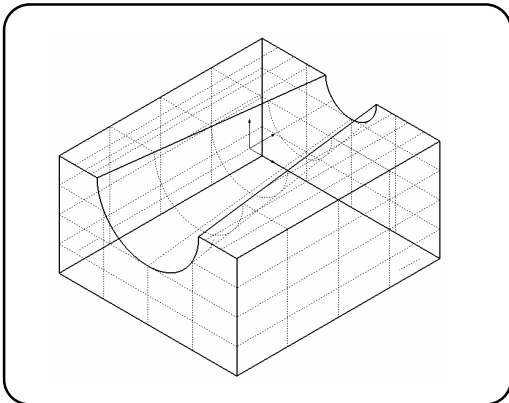
Move

Layer: 10

Select: OFF

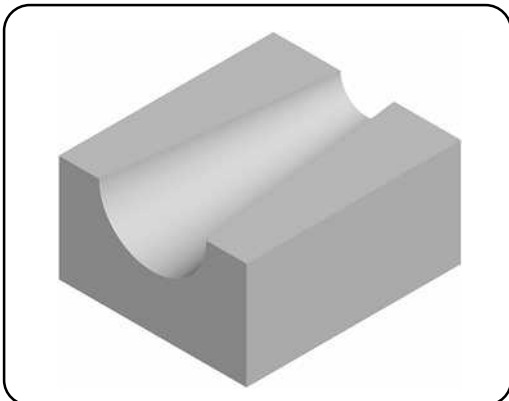
Press the center button to execute the command.

19 . Hide all layers except the active layer.



Select the [Hide(All)] command.

20 . The shape (except fillet entities) has been created !



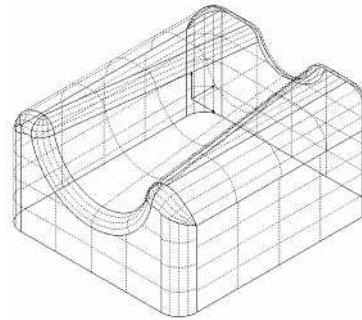
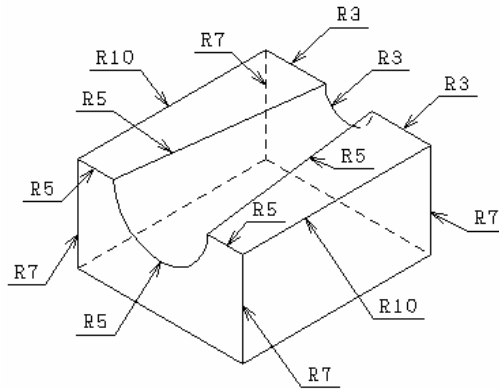
Select the [View type(Shade)] command.

Confirm the created surfaces.



Select the [View type(Wireframe)] command.

Create fillet surfaces as below.



Rule for creating fillet surfaces

When two cavity fillets/core fillets are overlapped each other, create surfaces in descending order.

21 . Create a fillet surface of R10 between the top surface A and side surface B.



Select the [Fillet surface] command.

Specify parameters in the command sheet.

<Page1> Radius: 10, Curvature continuous: OFF, All entities

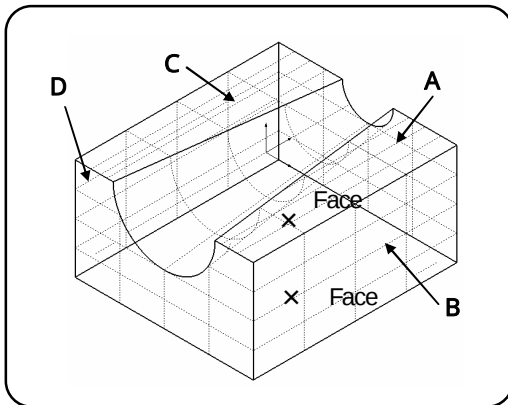
<Page2> Extend fillet: OFF, Extend surface: OFF

<page3> Trim: ON, First: ON, Second: ON

Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center button to execute the command.



Create a fillet surface of R10 between the top surface

C and side surface D with the same operation above.

22 . Create a fillet surface of R7 between the two side surfaces.



Specify parameters in the command sheet.

<Page1> Radius: 7, Curvature continuous: OFF, All entities

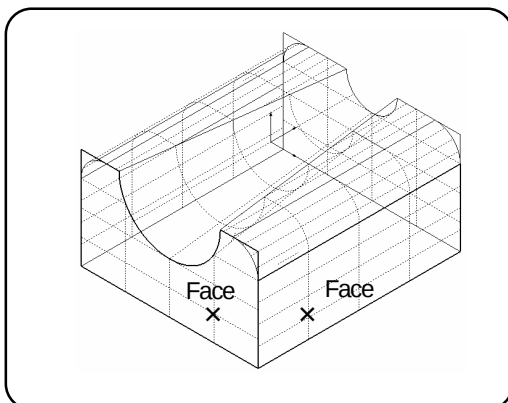
<Page2> Extend fillet: OFF, Extend surface: ON, Distance: 0

<page3> Trim: ON, First: ON, Second: ON

Specify the first surface. (Face)

Specify the second surface. (Face)

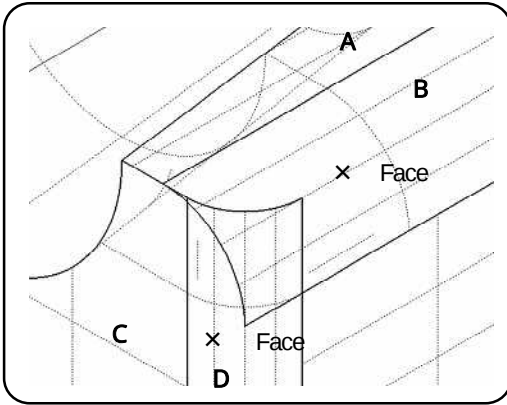
Press the center button to execute the command.



Create a fillet surface at the other three corners with

the same operation.

23 . Create a fillet surface of R5 between the surface group AB and the surface group CD.



Specify parameters in the command sheet.

<Page1> Radius: 5, Curvature continuous: OFF, All entities

<Page2> Extend fillet: ON , start: 0 end:1

Extend surface: OFF

<page3> Trim: ON, First: ON, Second: ON

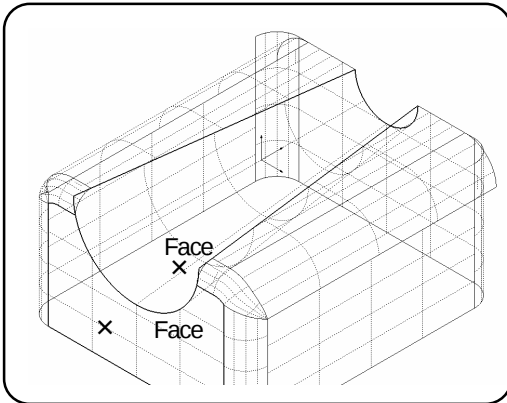
Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center button to execute the command.

Create a fillet surface at the other side with the same operation.

24 . Create a fillet surface of R5 between the cylinder surface and the side surface.



Specify parameters in the command sheet.

<Page1> Radius: 5, Curvature continuous: OFF, All entities

<Page2> Extend fillet: OFF

Extend surface: ON, Distance: 0

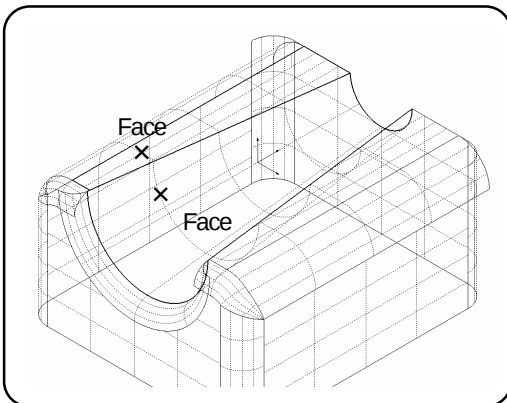
<page3> Trim: ON, First: ON, Second: ON

Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center button to execute the command.

25 . Create a fillet surface of R5 between the cylinder surface and the top surface.



Specify parameters in the command sheet.

<Page1> Radius: 5, Curvature continuous: OFF, All entities

<Page2> Extend fillet: OFF

Extend surface: ON, Distance: 5

<Page3> Trim: ON, First: ON, Second: ON

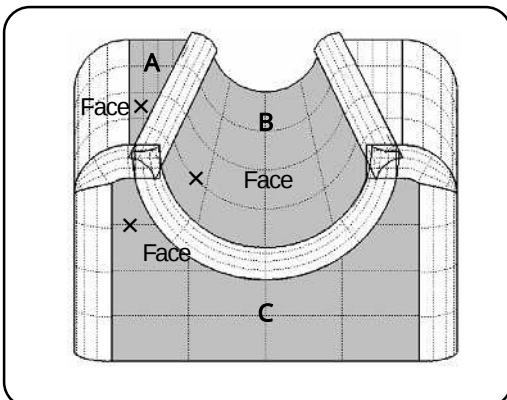
Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center button to execute the command.

Create a fillet surface at the other side with the same operation.

26. Create a fillet surface of R5 at the corner of surfaces ABC.



Select the [Corner Fillet Surface].

Specify parameters in the command sheet.

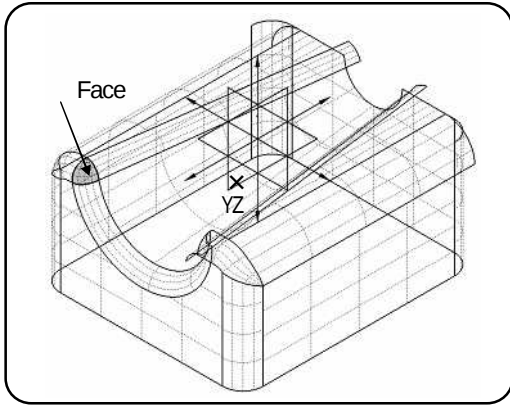
<Page1> Radius: 5

Specify the first surface. (Face)

Specify the second surface. (Face)

Specify the third surface. (Face)

27 . Copy the created corner fillet surface to the other side based on the YZ plane.



Select the [Mirror/Copy] command.

Specify parameters in the command sheet.

<Page1> Copy, Select: ON

[Select original/copied entities]

Specify the entity. (Face)

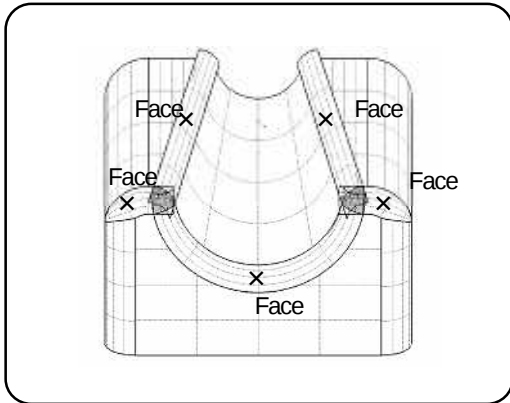
Press the center button to determine the specifying.

Specify the copying origin. >> [F12] (Work origin)

Specify the plane. (YZ plane)

Press the center button to execute the command.

28 . Trim the fillet surface with the selected corner fillet surface.



Select the [Trim surface] command.

Specify parameters in the command sheet.

<Page1> Selected contour: ON, Surface contour: ON

(Others: OFF)

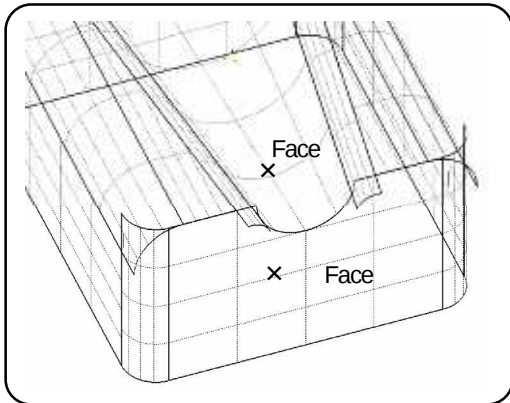
- Specify the surfaces to be trimmed (surfaces to be left) (Face)

Press the center button to execute the command.

Specify the reference surface. (Already selected >> No need to select here)

Press the center button to execute the command.

29 . Create a fillet surface of R3 between the entire top surfaces and the side surface at rear side.



Select the [Fillet surface] command.

Specify parameters in the command sheet.

<Page1> Radius: 3, Curvature continuous: OFF, All entities

<Page2> Extend fillet: OFF, Extend surface: OFF

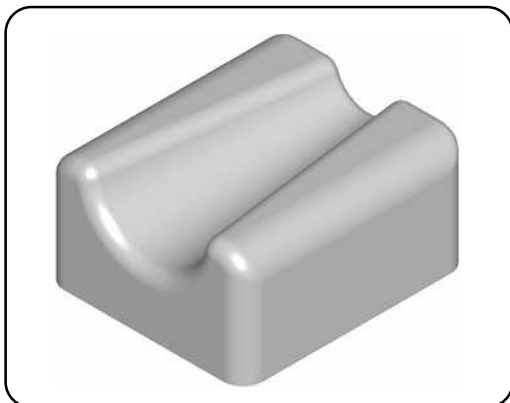
<Page3> Trim: ON, First: ON, Second: ON

Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center button to execute the command.

30 . Completed !! Confirm the shape and save the model file.



Select the [View (Shade)] command.

Confirm the created surfaces.



Select the [View (Wireframe)] command.



Select the [Save] command.

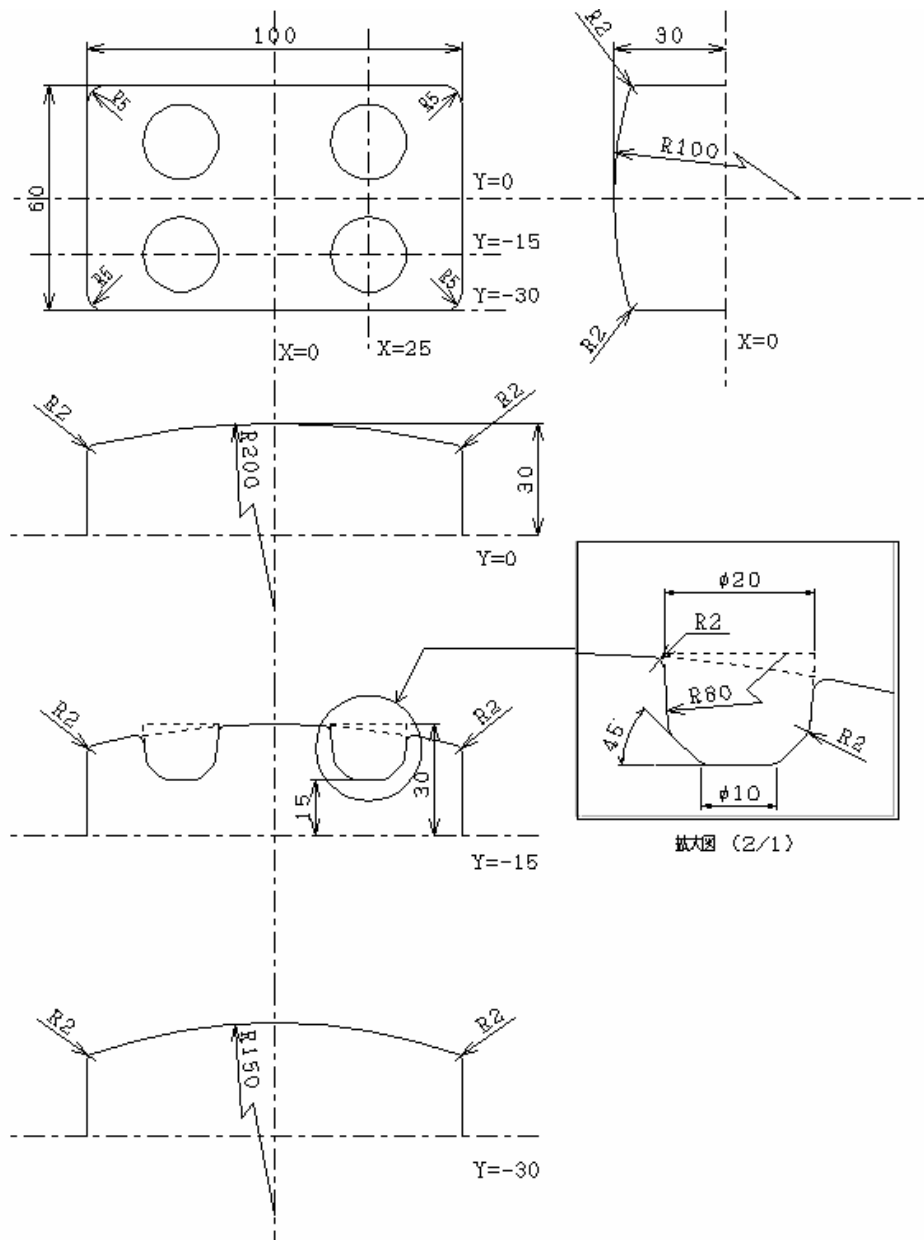
Save in Ex) D:/cam-tool/cam-tool File

File name Exercise1

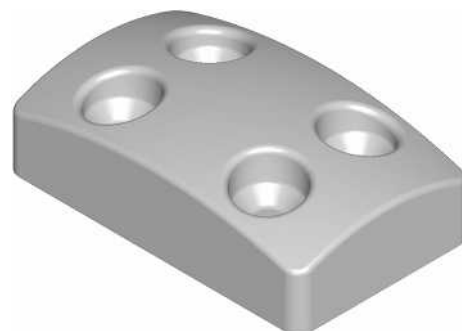
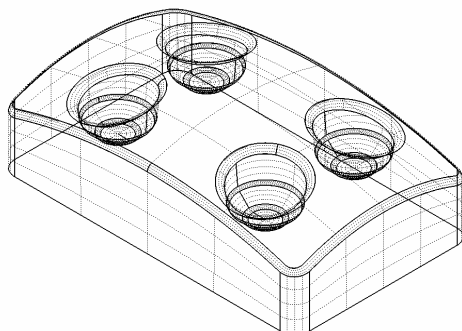
Click the [Save] button.

Exercise 2

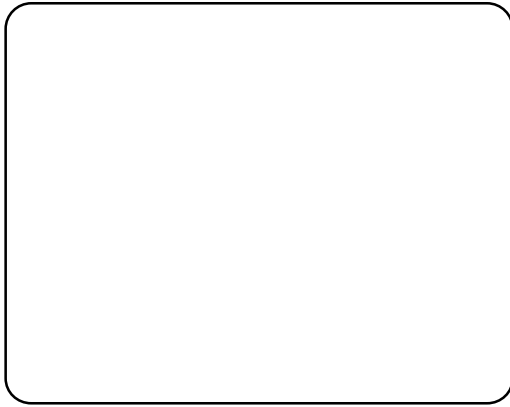
Create the following shape.



<Completed shape>

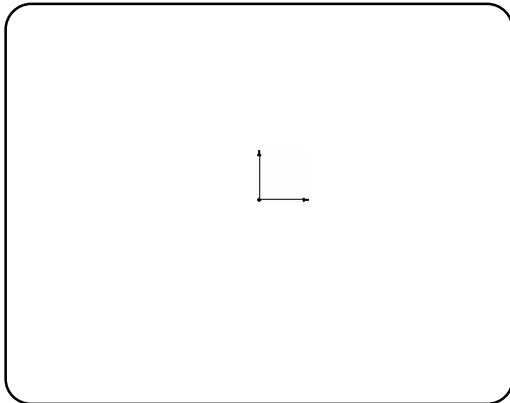


1. Open a new model file.



Select the [Open(New)] command.

2 . Create lines that cross at right angle each other.



Select the [Cross] command.

Specify parameters in the command sheet.

Specify length: ON

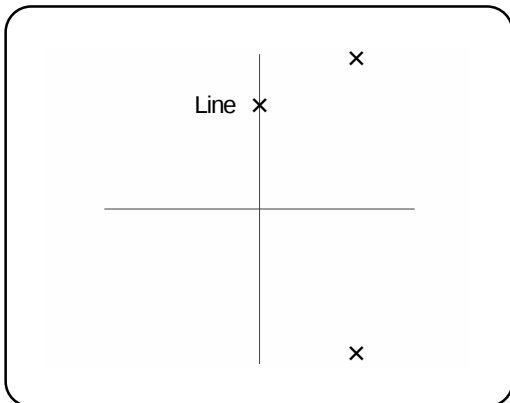
Horizontal: 200

Vertical: 200

Plane: XY plane

Specify the center point. >> [F12] (Work origin)

3 . Create an interval line at 100 mm far from the Y-axis.



Select the [Interval] command.

Specify the reference line (Line).

Specify parameters in the command sheet.

Distance: 100

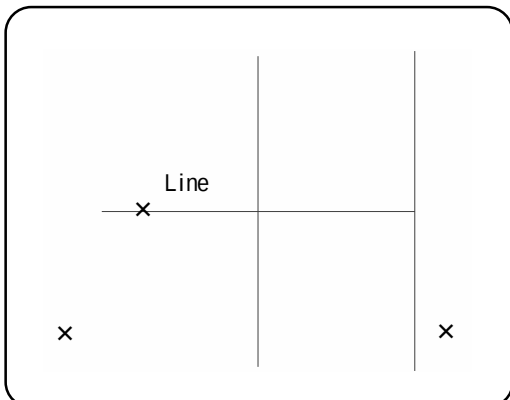
Repeat: 1

Plane: XY Plane

Specify the start point direction

Specify the end point direction.

4 . Create an interval line at 100 mm far from the X-axis, two times.



Select the [Interval] command.

Specify the reference line (Line).

Specify parameters in the command sheet.

Distance: 100

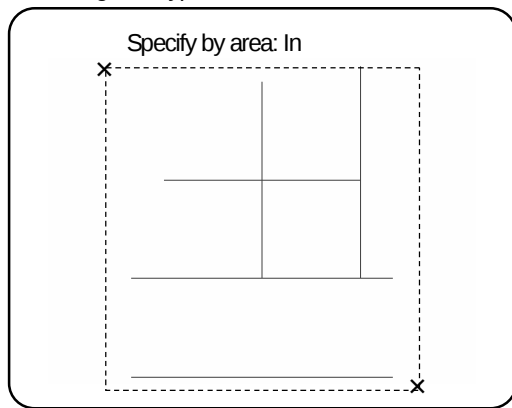
Repeat: 2

Plane: XY Plane

Specify the start point direction.

Specify the end point direction.

5 . Change line type.



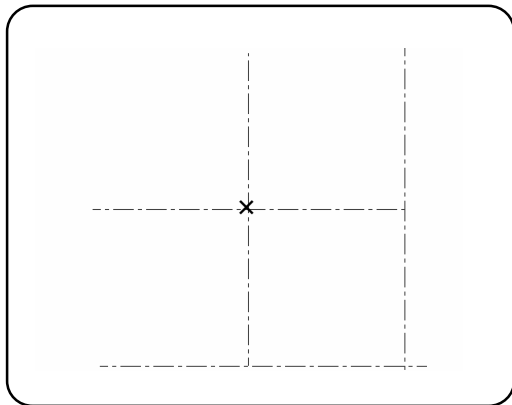
Select the [Change Attribute (Line type) command.

Center line: ON

Right-click >> pop-up menu >> Chain

Specify entities. (Specify by area)

6 . Create a rectangle.



Select the [Rectangle] command.

Specify parameters in the command sheet.

Specify length: ON,

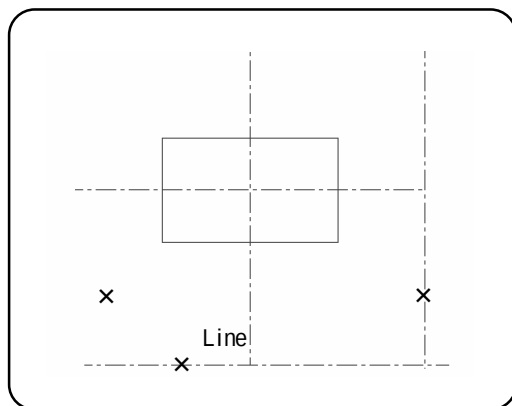
Horizontal: 100

Vertical: 60

Plane: XY plane

Specify the center point. >> [F12] (Work origin)

7 . Create an interval line at 30mm far from the X-axis.



Select the [Interval] command.

Specify the reference line. (Line)

Specify parameters in the command sheet.

Distance: 30

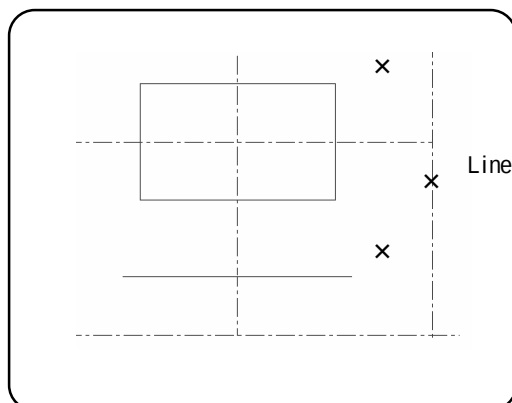
Repeat: 1

Plane: XY plane

Specify the start point direction.

Specify the end point direction.

8 . Create an interval line at 30mm far from the Y-axis.



Select the [Interval] command.

Specify the reference line. (Line)

Specify parameters in the command sheet.

Distance: 30

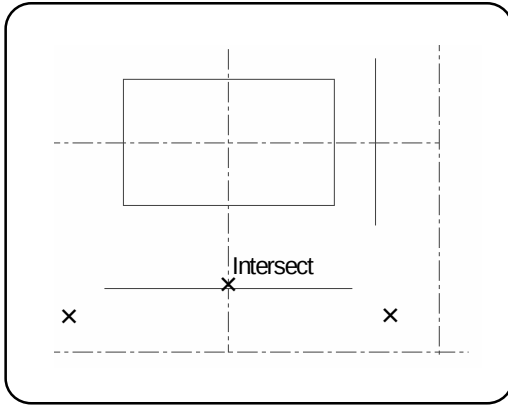
Repeat: 1

Plane: XY plane

Specify the start point direction.

Specify the end point direction.

9 . Create an arc of R200 that is tangent to the specified coordinate at 0 degree.



Select the [Arc (Radius, Tangent angle, P, S, E)] command.

Specify parameters in the command sheet.

Radius: 200

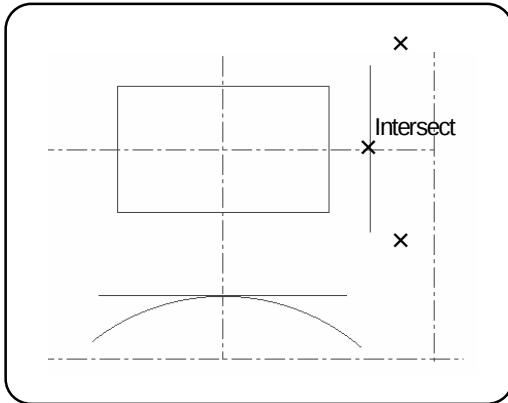
Angle: 0

Specify the tangent point. (Intersect)

Specify the start point direction.

Specify the end point direction.

10 . Create an arc of R100 that is tangent to the specified coordinate at 90 degree.



Select the [Arc (Radius, Tangent angle, P, S, E)] command.

Specify parameters in the command sheet.

Radius: 100

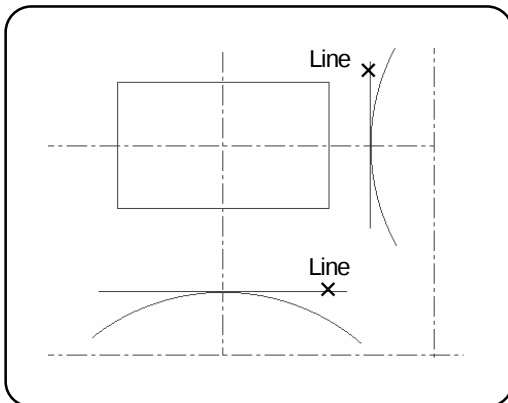
Angle: 90

Specify the tangent point. (Intersect)

Specify the start point direction.

Specify the end point direction.

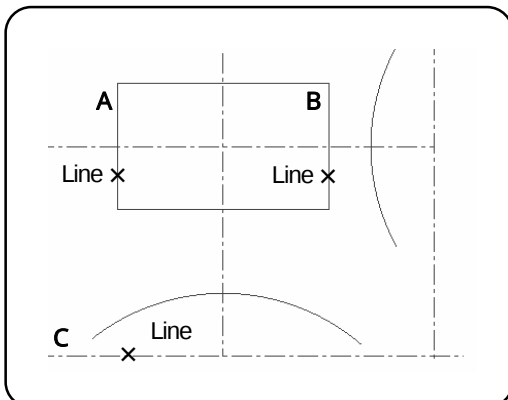
11 . Delete the unnecessary lines.



Select the [Delete] command.

- Specify the unnecessary lines. (Line)

12 . Lengthen the lines A and B to the line C.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

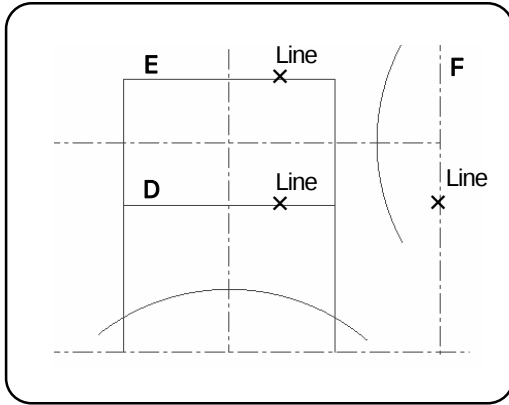
Entity number: Plural

- Specify the lines to be trimmed (lines to be left) (Line)

Press the center button to determine the specifying.

Specify the reference entity. (Line)

13 . Lengthen the lines D and E to the line F.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

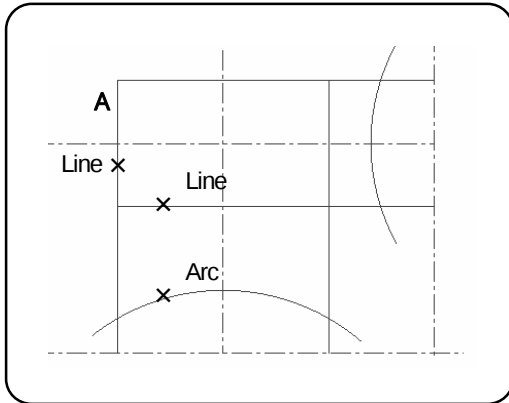
Entity number: Plural

- Specify the lines to be trimmed (lines to be left) (Line)

Press the center button to determine the specifying.

Specify the reference entity. (Line)

14 . Delete the unnecessary part of the line A.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

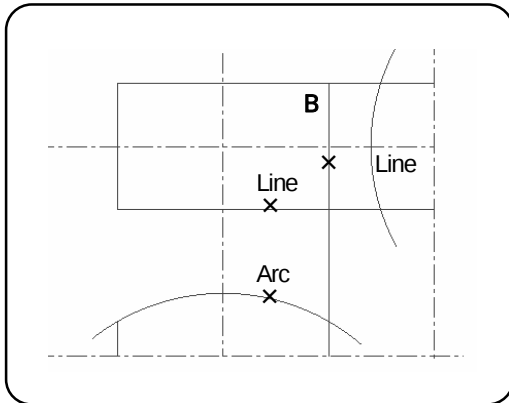
Entity number: Single

Specify the line to be trimmed (line to be left) (Line)

Specify the first reference entity. (Line)

Specify the second reference entity. (Arc)

15 . Delete the unnecessary part of the line B.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

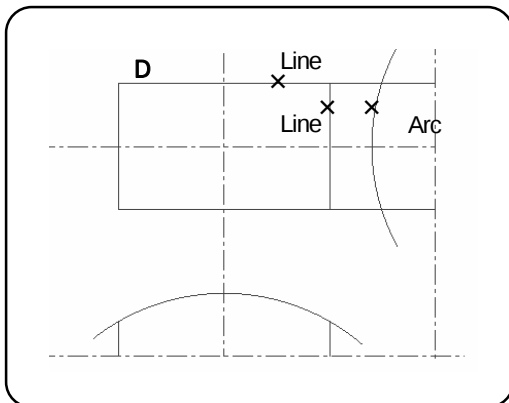
Entity number: Single

Specify the line to be trimmed (line to be left) (Line)

Specify the first reference entity. (Line)

Specify the second reference entity. (Arc)

16 . Delete the unnecessary part of the line D.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

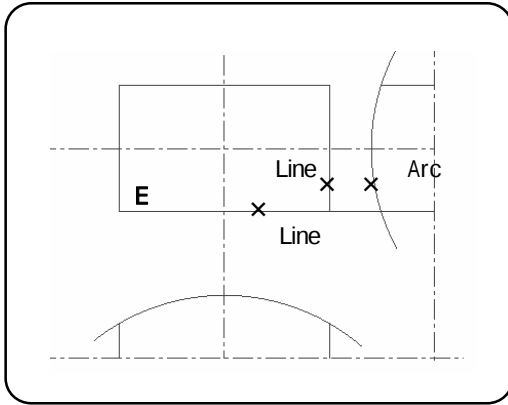
Entity number: Single

Specify the line to be trimmed (line to be left) (Line)

Specify the first reference entity. (Line)

Specify the second reference entity. (Arc)

17 . Delete the unnecessary part of the line E.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

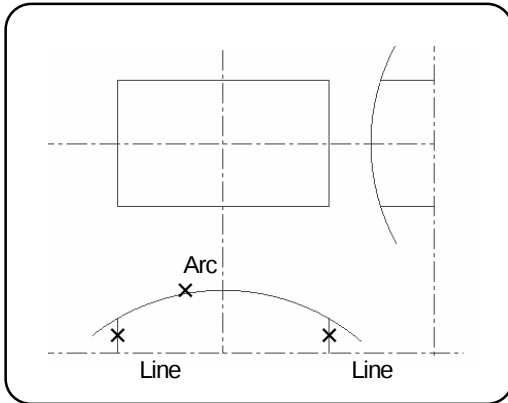
Entity number: Single

Specify the line to be trimmed (line to be left) (Line)

Specify the first reference entity. (Line)

Specify the second reference entity. (Arc)

18 . Delete the unnecessary part of the arc at front.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

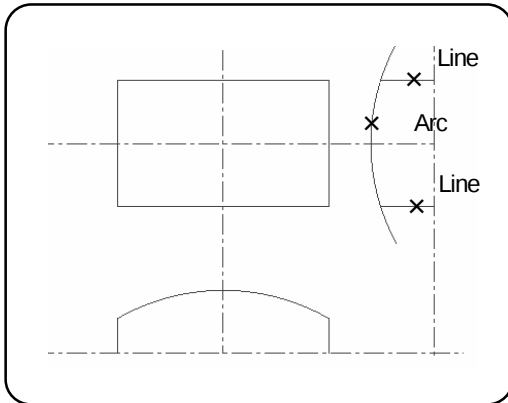
Entity number: Single

Specify the line to be trimmed (line to be left) (Arc)

Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

19 . Delete the unnecessary part of the arc at right side.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

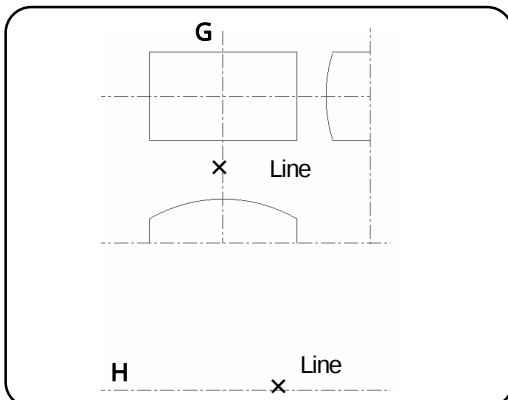
Entity number: Single

Specify the line to be trimmed (line to be left) (Arc)

Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

20 . Lengthen the line G to the line H.



Select the [trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

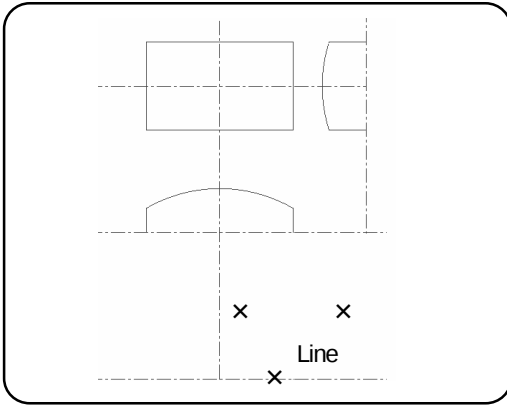
Trim: Intersection

Entity number: Single

Specify the entity to be trimmed. (Line)

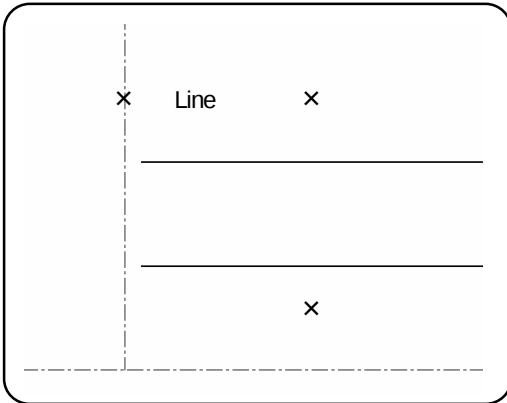
Specify the reference entity. (Line)

21 . Create an interval line at 15mm far from the reference line, two times.



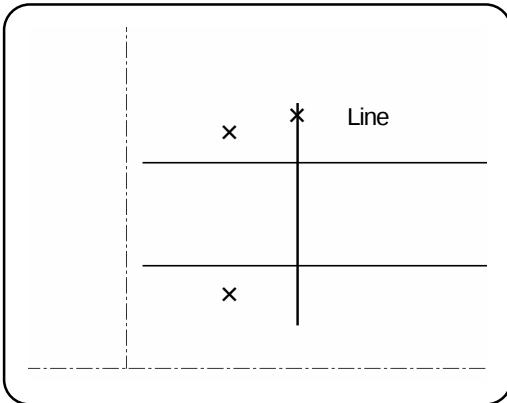
Select the [Interval] command.
Specify the reference line. (Line)
Specify parameters in the command sheet.
Distance: 15
Repeat: 2
Plane: XY Plane
Specify the start point direction.
Specify the end point direction.

22 . Create an interval line at 25mm far from the reference line.



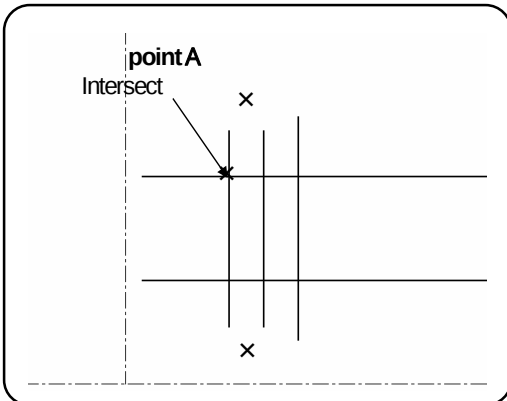
Select the [Interval] command.
Specify the reference line. (Line)
Specify parameters in the command sheet.
Distance: 25
Repeat: 1
Plane: XY plane
Specify the start point direction.
Specify the end point direction.

23 . Create an interval line at 5mm far from the reference line, two times.



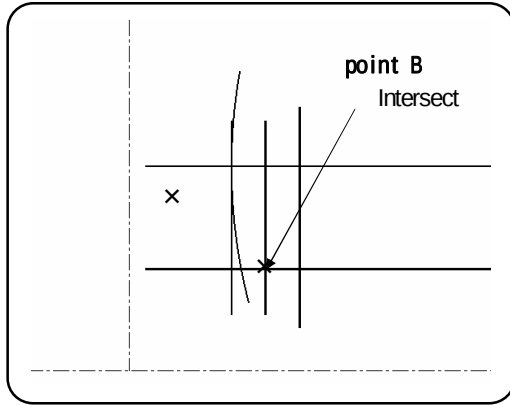
Select the [Interval] command.
Specify the reference line. (Line)
Specify parameters in the command sheet.
Distance: 5
Repeat: 2
Plane: XY plane
Specify the start point direction.
Specify the end point direction.

24 . Create an arc of R80 that is tangent to the point A at 90 degree.



Select the [Arc (Radius, Tangent angle, P, S, E)] command.
Specify parameters in the command sheet.
Radius: 80
Angle: 90
Specify the tangent point (A). (Intersect)
Specify the start point direction.
Specify the end point direction.

25 . Create an angle line of - 45 degree from the point B.



Select the [Angle] command.

Specify parameters in the command sheet.

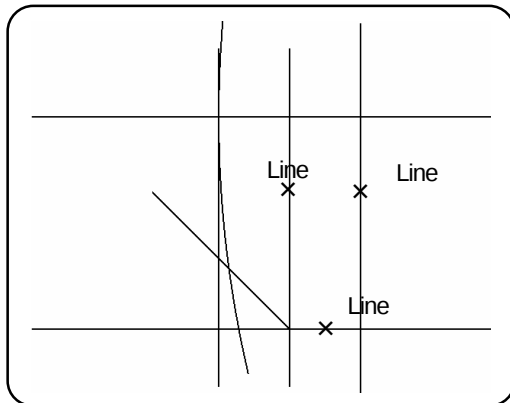
Angle: - 45

Plane: XY Plane

Specify the start point (point B). (Intersect)

Specify the end point direction.

26 . Trim the unnecessary part of lines.



Select the [Trim] command.

Specify parameters in the command sheet.

Entity: One side

Dividing entity: Two dividing points

Trim: Intersection

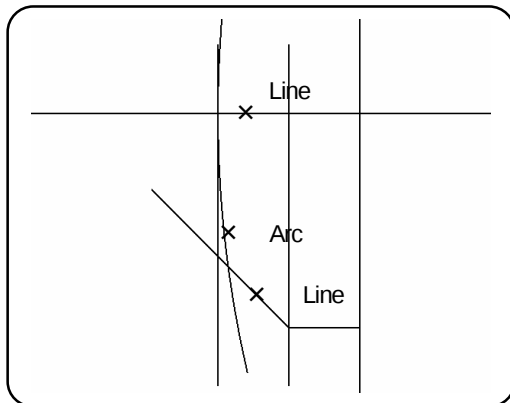
Entity number: Single

Specify the line to be trimmed (Line to be left). (Line)

Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

27 . Trim the unnecessary part of lines.



Select the [Trim] command.

Specify parameters in the command sheet.

Entity: Both sides

Dividing entity: Two dividing points

Trim: Intersection

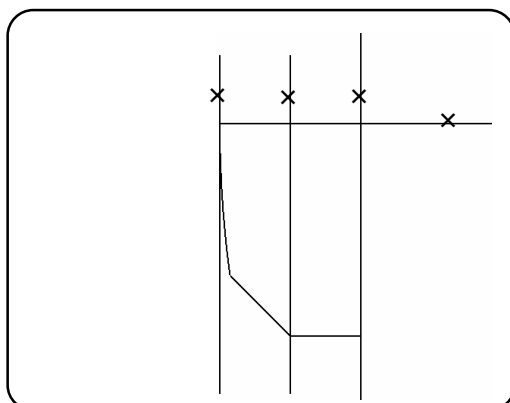
Entity number: Single

Specify the arc to be trimmed (Line to be left). (Arc)

Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

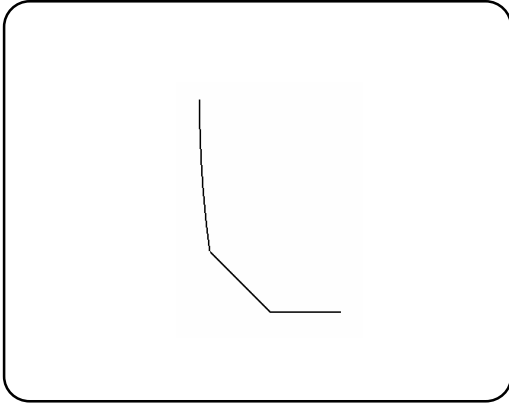
28 . Delete the reference lines.



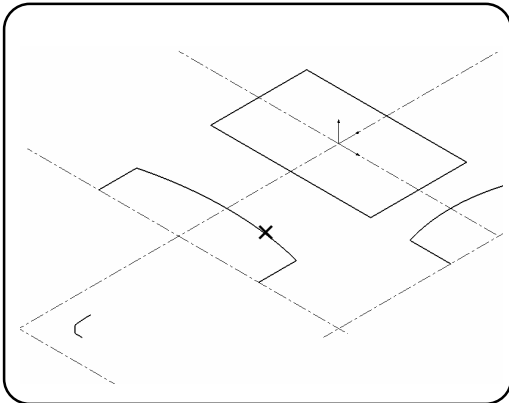
Select the [Delete] command.

- Specify the lines to be deleted. (Line)

29 . Section of a hole has been created.



30 . Put the front entities into the selected state.

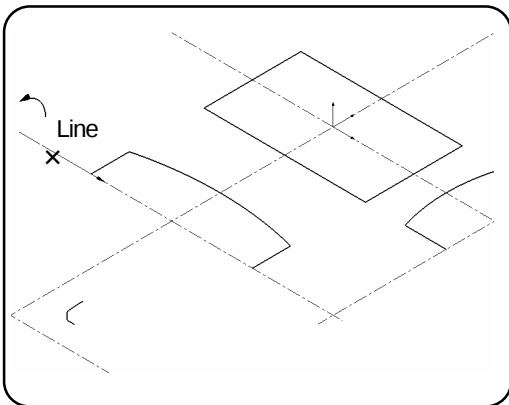


Execute the [Select(Entity)] command.

Right-click >> pop-up menu >> Chain

Specify the entity to be selected.(Arc)

31. Tilt the front entities to the ZX plane.



Select the [Rotate/Copy] command.

Specify parameters in the command sheet.

Move

Revolution: Axis

Angle: 90

Select:: ON

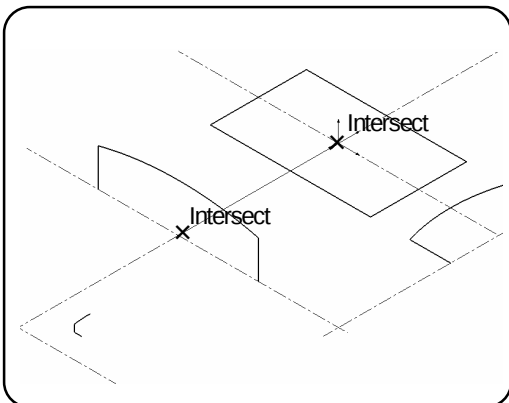
Specify the entity. (Already selected >> No need to select here).

Press the center button to determine the specifying.

Specify the line as tilting axis. (Line)

Press the center button to execute the command.

32 . Move the tilted section to the work origin.



Select the [Move/Copy] command.

Specify parameters in the command sheet.

Move, Select: OFF

Specify the entity. (Already selected)

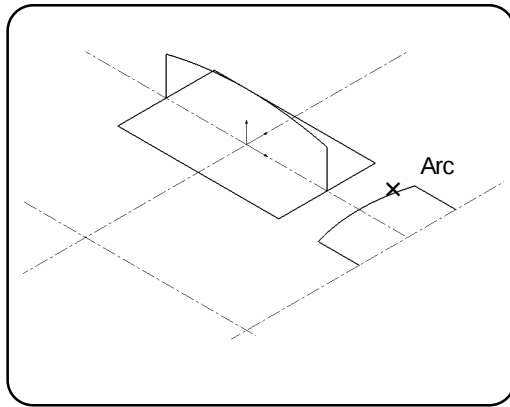
Press the center button to determine the specifying.

Specify the coordinate of the origin of moving.

Specify the coordinate of the end point of moving.

Press the center button to execute the command.

33 . Put the right section into the selected state.

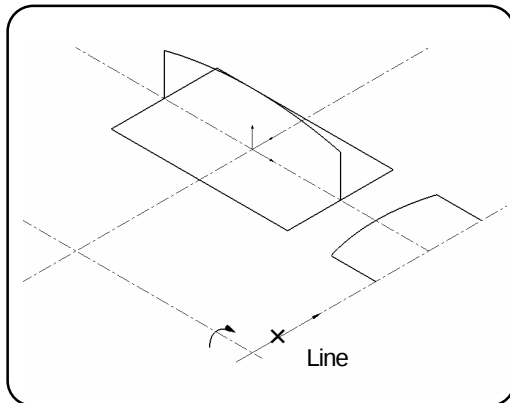


Execute the [Select (Entity)] command.

Right-click >> pop-up menu >> Chain

Specify the entities to be selected. (Arc)

34. Tilt the right entities to the YZ plane.



Select the [Rotate/Copy] command.

Specify parameters in the command sheet.

Move

Revolution: Axis

Angle: 90

Select:: ON

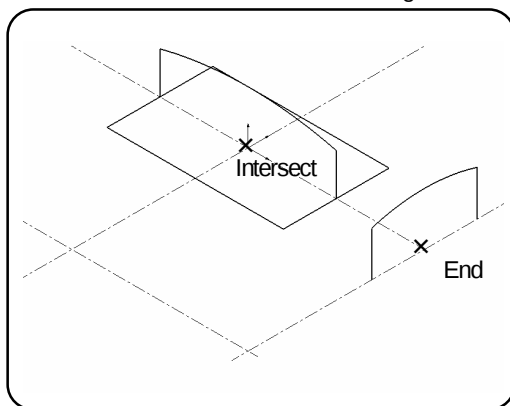
Specify the entity. (Already selected)

Press the center button to determine the specifying.

Specify the line as tilting axis. (Line)

Press the center button to execute the command.

35 . Move the tilted section to the work origin.



Select the [Move/Copy] command.

Specify parameters in the command sheet.

Move

Select:: OFF

Specify the entity. (Already selected)

Press the center button to determine the specifying.

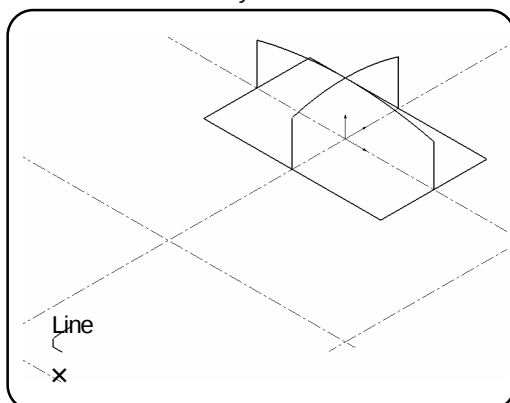
Specify the coordinate of the origin of moving the entities.

(End/Intersect)

Specify the coordinate of the end point of moving.

Press the center button to execute the command.

36 . Put the section entity of the hole into the selected state.

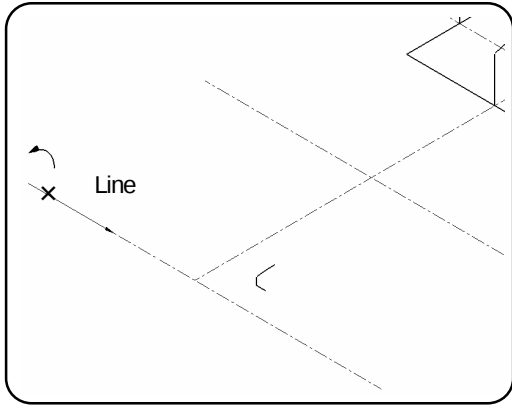


Execute the [Select (Entity)] command.

Right-click >> pop-up menu >> Chain

Specify the entity to be selected. (Line)

37. Tilt the section entity of the hole to the ZX plane.



Execute the [Rotate/Copy] command.

Specify parameters in the command sheet.

Move

Angle: 90

Select: ON

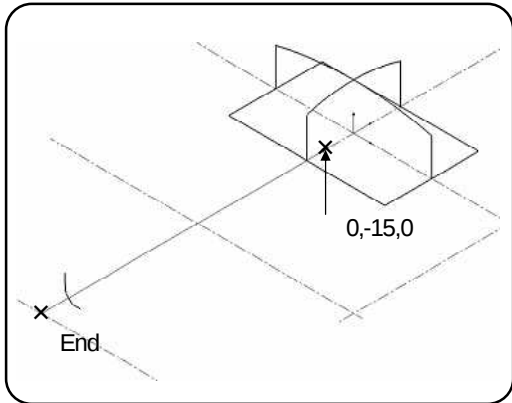
Specify the entity. (Already selected)

Press the center button to determine the specifying.

Specify the line as tilting axis. (Line)

Press the center button to execute the command.

38 . Move the tilted entity.



Select the [Move/Copy] command.

Specify parameters in the command sheet.

Move

Select: OFF

Specify the entity. (Already selected)

Press the center button to determine the specifying.

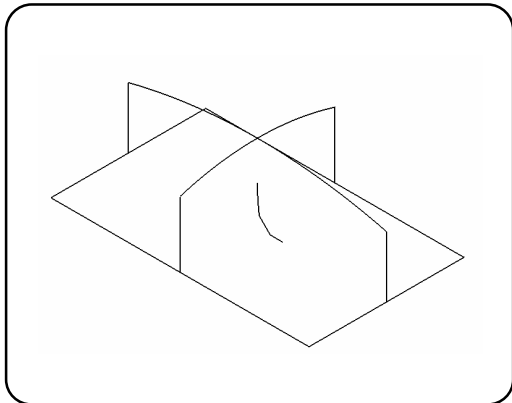
Specify the coordinate of moving origin. (End/Intersect)

Input the coordinate of moving end point.

(0, -15, 0)<Enter>

Press the center button to execute the command.

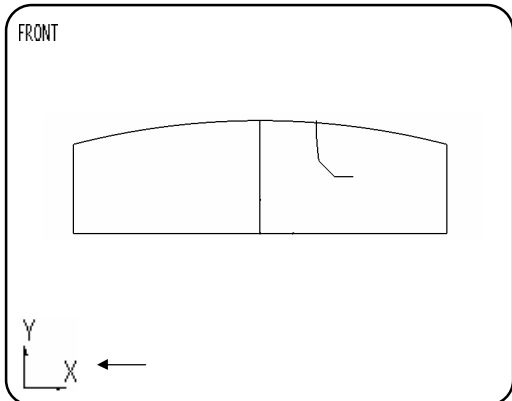
39 . Delete the center line.



Select the [Delete] command.

Delete the entities of center line. (Line)

40 . Define the ZX plane as work plane.



Select the [View direction (FRONT)] command.

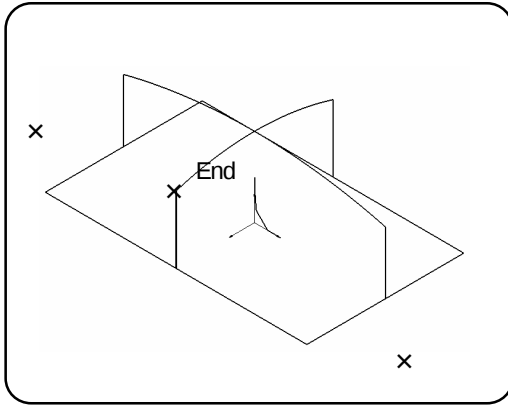
Set the FRONT view as the active view.



Select the [Define Work Plane (view)] command.

The current active view is defined as work plane.

41 . Create a circle of R150.



Select the [Arc (Radius, Tangent angle, P, S, E)] command.

Specify parameters in the command sheet.

Radius: 150

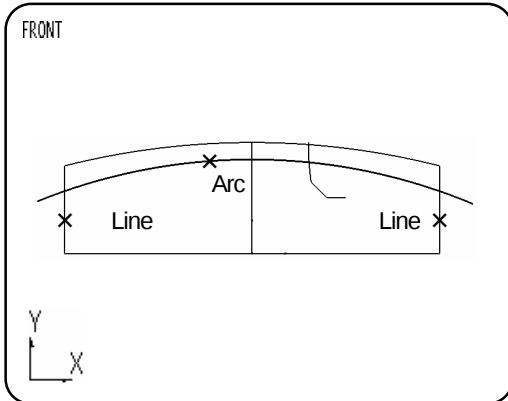
Angle: 0

Specify the tangent point. (End/Intersect)

Specify the start point direction.

Specify the end point direction.

42 . Trim the unnecessary parts in the [View direction (FRONT)].



Select the [View direction (FRONT)] command.

* When you have defined work plane, select the command pressing [Shift] key.



Select the [Trim] command.

Specify parameters in the command sheet.

Entity: One side, Dividing entity: Two dividing points

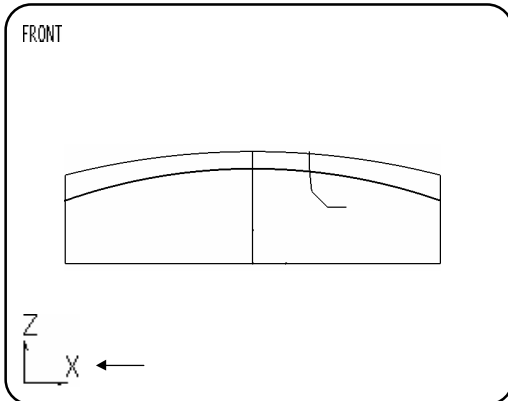
Trim: View, Entity number: Single

Specify the entity to be trimmed (to be left). (Arc)

Specify the first reference entity. (Line)

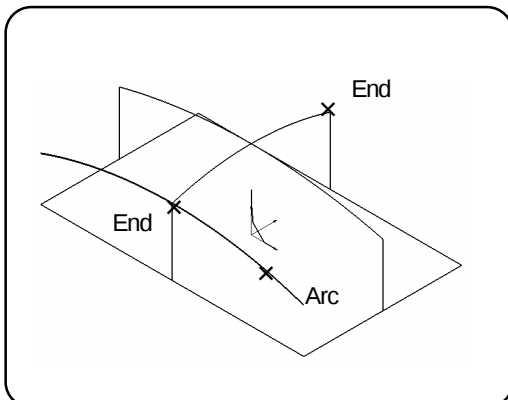
Specify the second reference entity. (Line)

43 . Restore the work plane.



Select the [Control Work Plane (Reset)] command.

44 . Copy the arc.



Select the [Move/Copy] command.

Specify parameters in the command sheet.

Copy, Select: OFF

Specify the entity to be copied. (Arc)

Press the center button to determine the specifying.

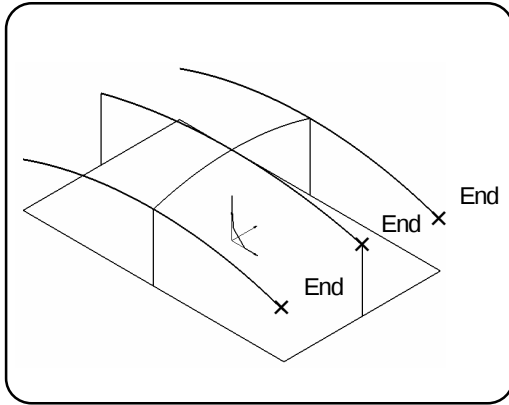
Specify the coordinate of moving origin. (End/Intersect)

Specify the coordinate of moving end point.

(End/Intersect)

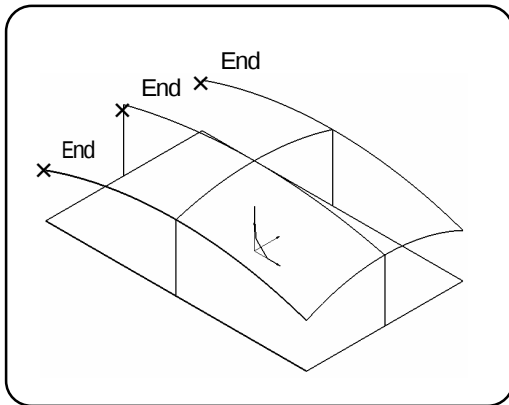
Press the center mouse button to execute the command.

45 . Create an arc by specifying three points.



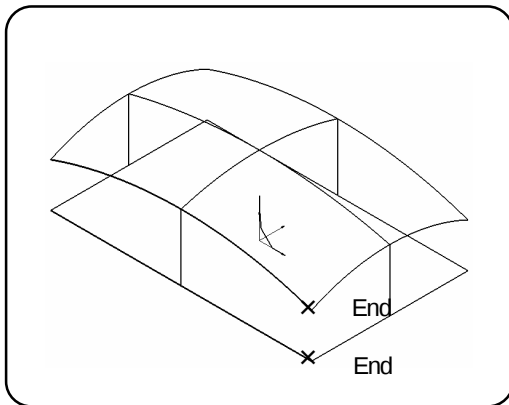
Select the [Arc (3 points)] command.
Specify the start point. (End)
Specify the passing point. (End/Intersect)
Specify the end point. (End)

46 . Create an arc at the other side with the same operation.



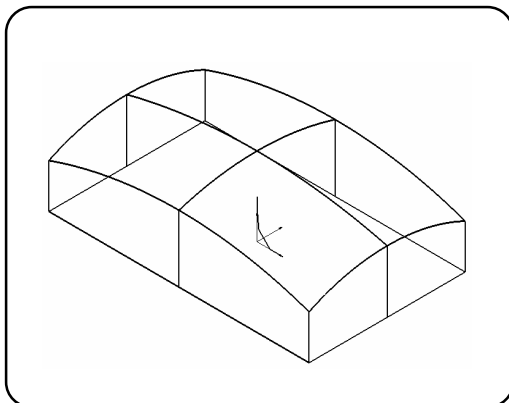
Select the [Arc (3 points)] command.
Specify the start point. (End)
Specify the passing point. (End/Intersect)
Specify the end point. (End)

47 . Connects the end points with a line.

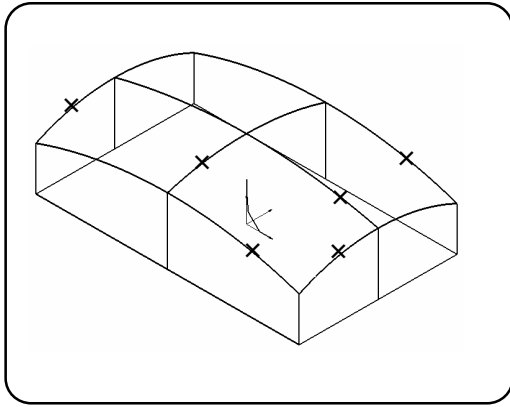


Select the [Single] command.
Specify the coordinate of start point. (End/Intersect)
Specify the coordinate of end point. (End/Intersect)

48 . Wire frames have been created. Next, create surfaces.



49 . Create a grill surface for top surface.



Select the [Grill Surface] command.

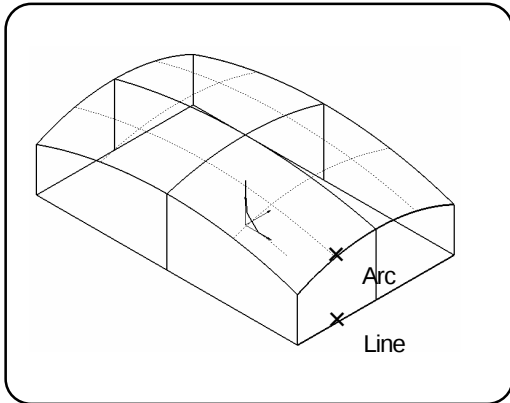
- Specify the primary curves. (Arc)

Press the center button to determine the specifying.

- Specify the crossing curves. (Arc)

Press the center mouse button to execute the command.

50 . Create a multi surface for side surfaces.



Select the [Multi Surface] command.

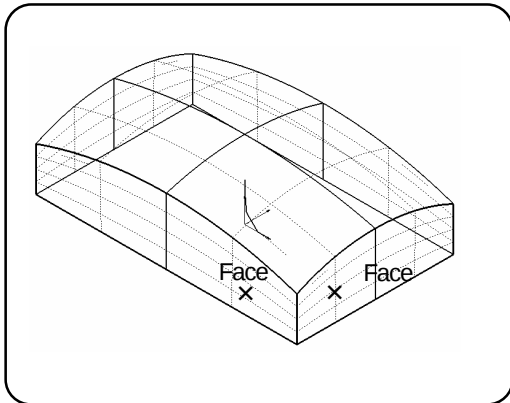
Specify the contour. (Arc/Edge)

Specify the contour. (Line)

Press the center mouse button to execute the command.

Create multi surfaces for three other sides.

51 . Create a fillet surface of R5 between the side surfaces.



Select the [Fillet Surface] command.

Specify parameters in the command sheet.

<page1> Radius: 5, Curvature continuous: OFF, All entities

<page2> Extend fillet: OFF, Extend surface: OFF

<page3> Trim: ON, First: ON, Second: ON

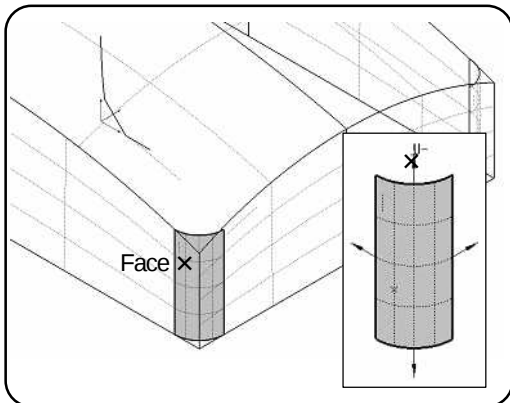
Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center mouse button to execute the command.

Create fillet surfaces at three other corners.

52 . Expand upper side of the created fillet surface by 10%.



Select the [Expand Surface] command.

Specify parameters in the command sheet.

<page1> Liner, Distance: OFF

Specify the surface to be expanded. (Face)

Specify the expanding direction.

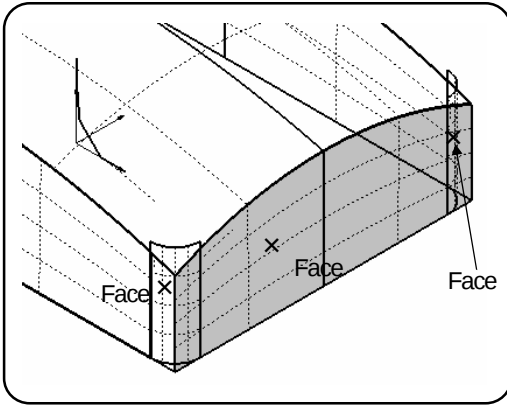
Specify the parameter for expanding.

<page1> Distance: OFF >> [0.1]

Press the center mouse button to execute the command.

Expand upper side of surfaces at three other sides by 10% with the same operation.

53 . Trim the side surface with the contour of the expanded fillet surface.



Select the [Trim Surface] command.

Specify parameters in the command sheet.

<page1> Selected contour: ON (Others: All OFF)

Specify the surface to be trimmed (to be left). (Face)

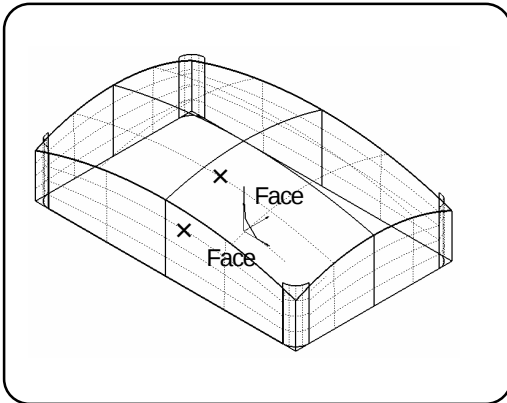
Press the center button to determine the specifying.

- Specify the reference surfaces (Mesh). (Face)

Press the center mouse button to execute the command.

Trim the surface at the other side with the same operation.

54 . Create a fillet surface of R2 between the top surface and the side surface.



Select the [Fillet Surface] command.

Specify parameters in the command sheet.

<page1> Radius: 2, Curvature continuous: OFF, All entities

<page2> Extend fillet: OFF, Extend surface: OFF

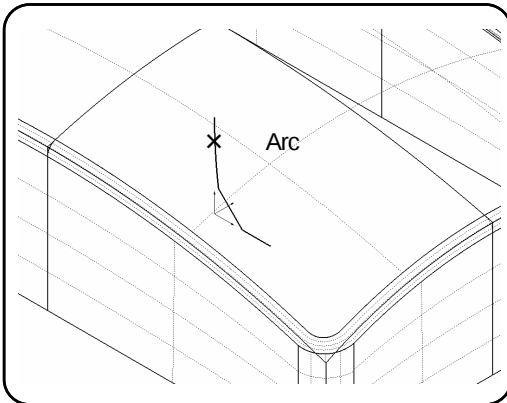
<page3> Trim: ON, First: ON, Second: ON

Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center mouse button to execute the command.

55 . Put the section of the hole into the selected status.

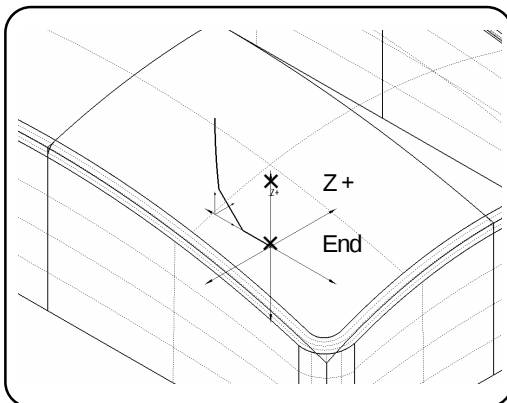


Execute the [Select(Entity)] command.

Right-click >> pop-up menu >> Chain

Specify the entities to be selected. (Arc)

56 . Create a revolved surface with the selected entity.



Select the [Revolved Surface] command.

Specify parameters in the command sheet.

<page1> Axis, Start angle:0, End angle: 360

Specify the entities. (Already selected) >> No need select.

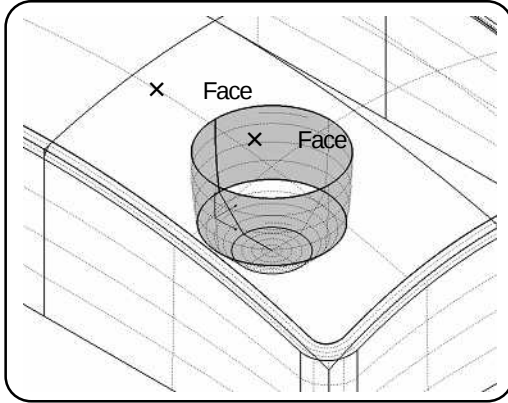
Press the center button to determine the specifying.

Specify the rotating axis. (End)

Specify the direction entity. (Z+ direction)

Press the center mouse button to execute the command.

57 . Create a fillet surface of R2 between the top surface and the side surface of the hole.



Select the [Fillet Surface] command.

Specify parameters in the command sheet.

<page1> Radius: 2, Curvature continuous: OFF, All entities

<page2> Extend fillet: OFF, Extend surface: OFF

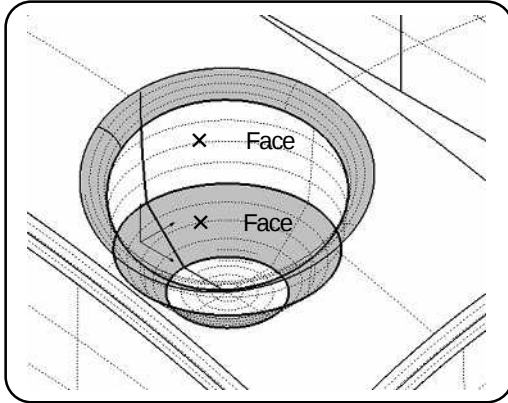
<page3> Trim: ON, First: ON, Second: ON

Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center mouse button to execute the command.

58 . Create a fillet surface of R2 between the side surfaces of hole.

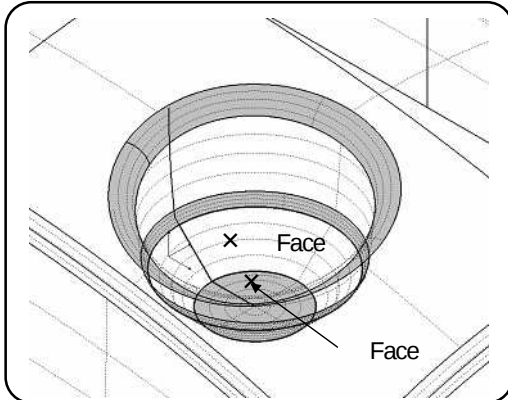


Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center mouse button to execute the command.

59 . Create a fillet surface of R2 between the side surface and the bottom surface of the hole, with the same operation.

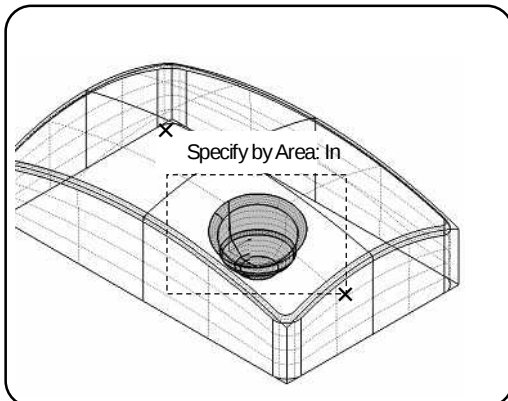


Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center mouse button to execute the command.

60 . Copy the hole shape to the other side based on the ZX plane.



Select the [Mirror/Copy] command.

Specify parameters in the command sheet.

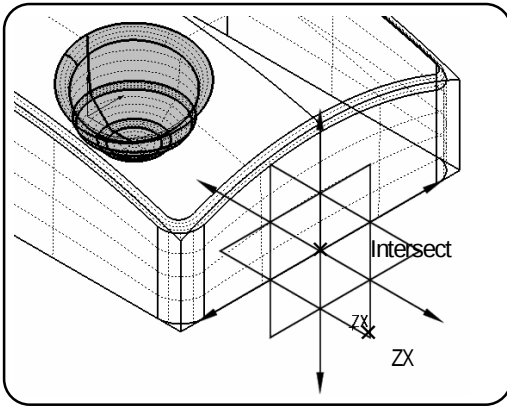
<page1> Copy, Select: ON [Select original/copied entity]

Specify the entities.

Right-click >> pop-up menu >> Specify by Area: In

Specify area.

Press the center button to determine the specifying.

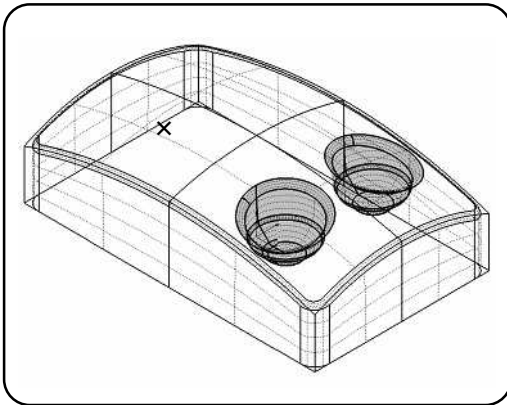


Specify the copying origin. (Intersect)

Specify the direction entity. (ZX plane)

Press the center mouse button to execute the command.

61 . Copy the selected shape of hole to the other side based on the YZ plane.



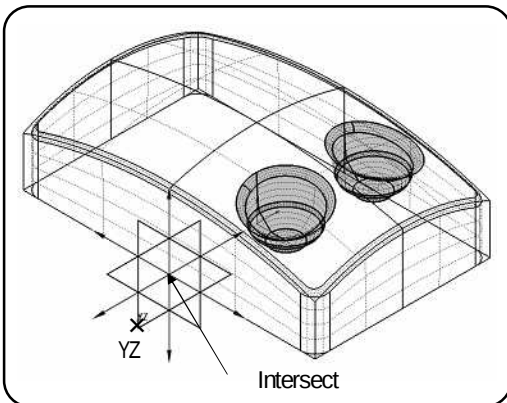
Select the [Mirror/Copy] command.

Specify parameters in the command sheet.

<page1> Copy, Select: OFF

Specify the entities. (Already selected) >> No need select.

Press the center button to determine the specifying.

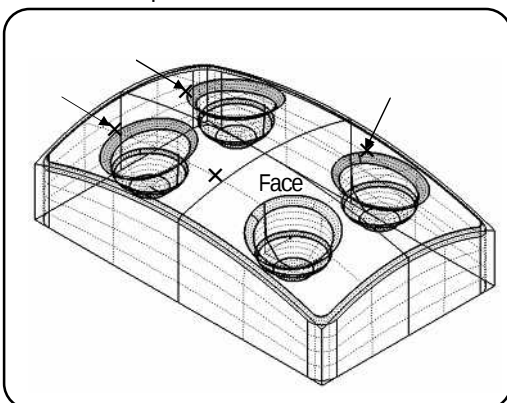


Specify the coping origin. (Intersect)

Specify the direction entity. (YZ plane)

Press the center mouse button to execute the command.

62 . Trim the top surface with the outer contour of the fillet surfaces.



Select the [Trim Surface] command.

Specify parameters in the command sheet.

<page1> Selected contour: ON (Others: All OFF)

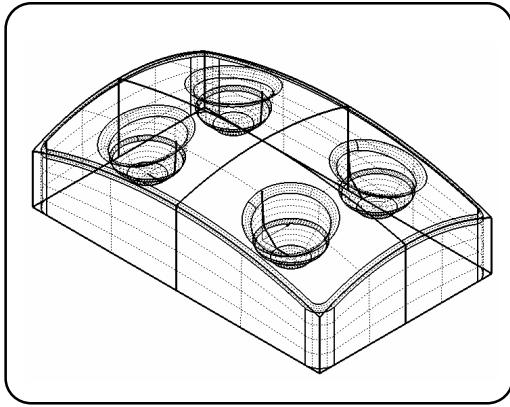
Specify the surface to be trimmed (to be left). (Face)

Press the center button to determine the specifying.

- Specify the outer contour of the fillet surfaces. (Face)

Press the center mouse button to execute the command.

63 . Move Wire frames to other layer.



Execute the [Select (Except Surface)] command.



Select the [Layer Move/Copy] command.

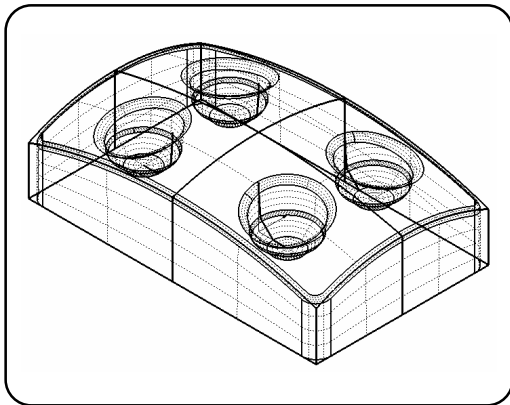
Set parameters in the command sheet.

<page1> Move, Layer10, Select: OFF

Specify the entities. (Already selected) >> No need to select here.

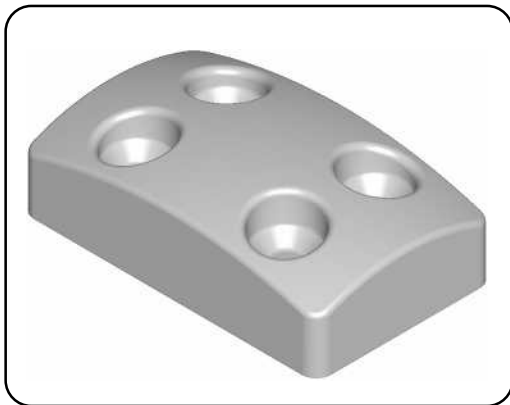
Press the center mouse button to execute the command.

64 . Hide layers other than the active layer.



Select the [Hide All layers] command.

65 . Completed! Confirm the shape and save the file.



Select the [View Type (Shade)].

Confirm the created shape.



Select the [View Type (Wire)].



Select the [Save] command.

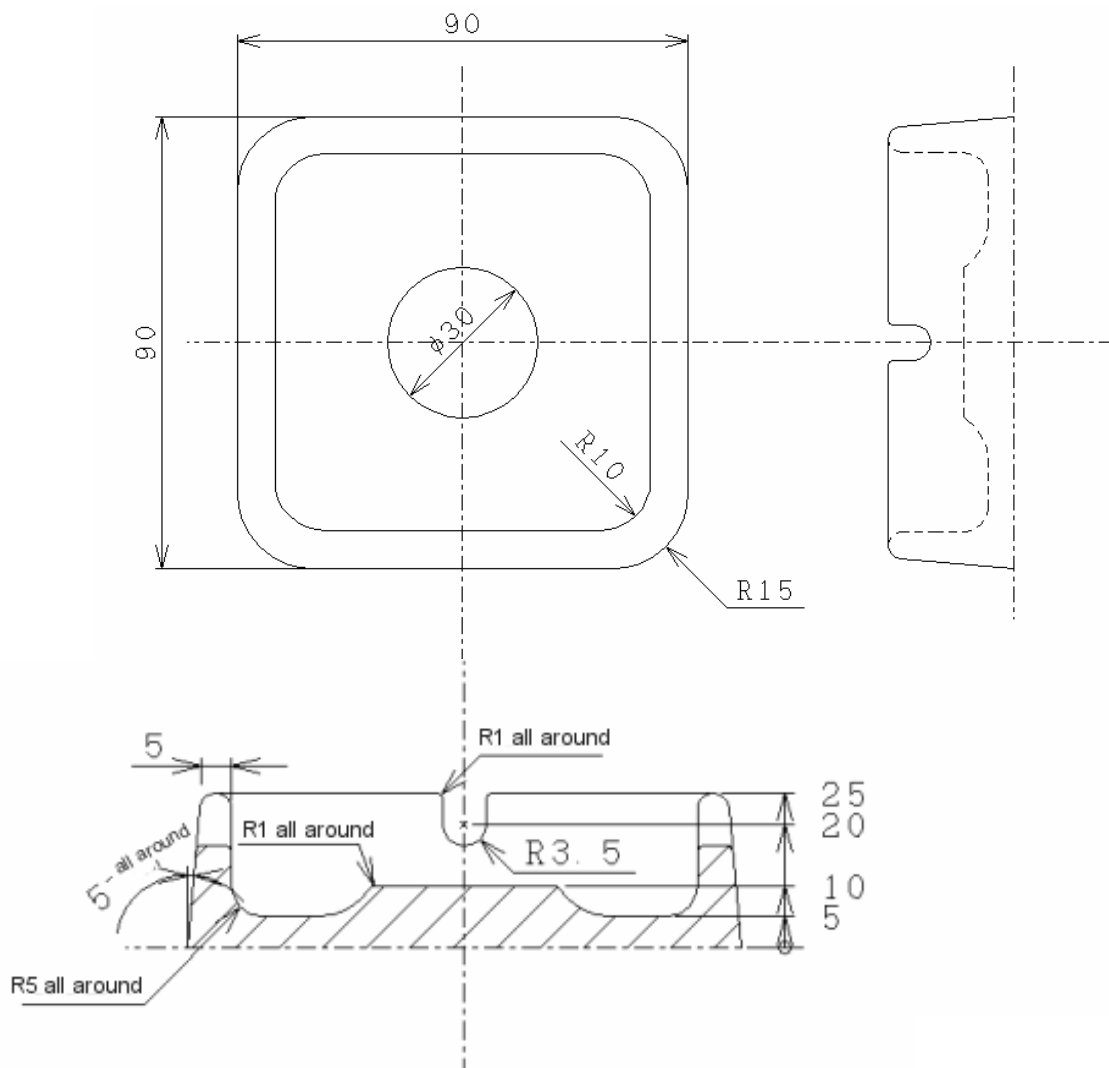
Saved in Ex.) C/cam-tool/Cam-toolFile

File name Exercise 2

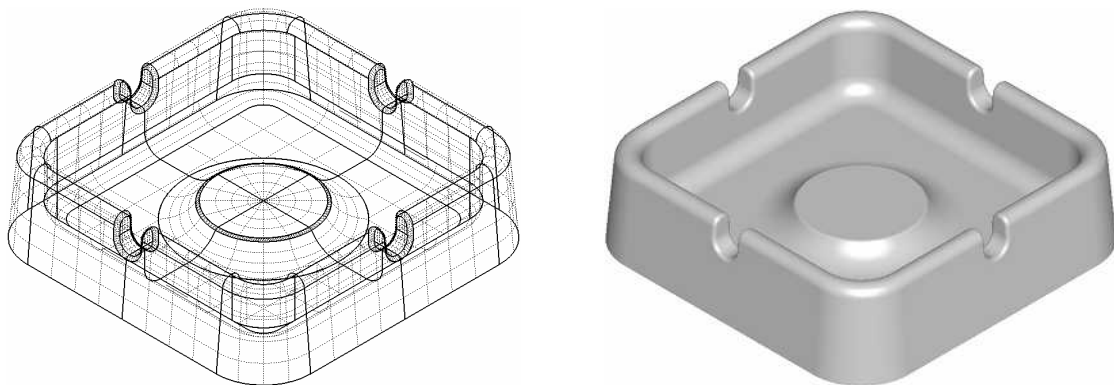
Click the [Save] button.

Exercise3

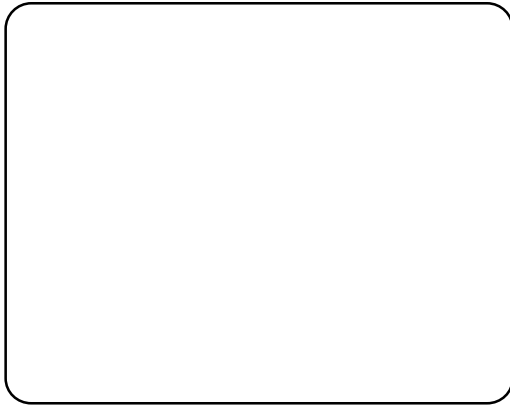
Create the following shape.



<Completed shape>

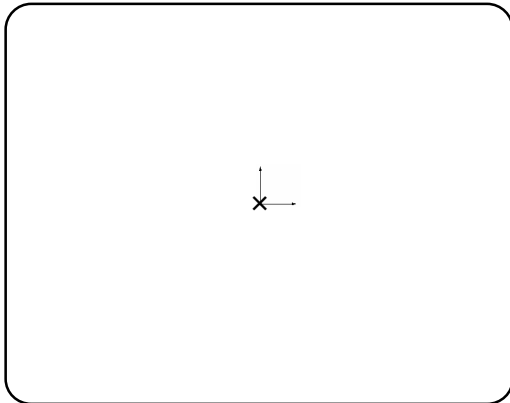


1. Open a new model file.



Select the [New] command.

2 . Create cross line of [center line].



Select the [Set Line Type (Wire)] command.

Center line: ON



Select the [Cross] command.

Set parameters in the command sheet.

Specify length: ON

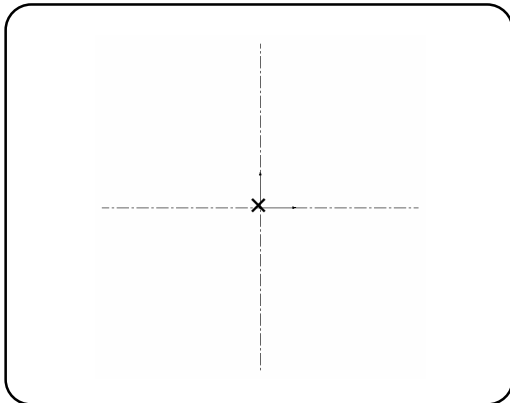
Vertical length: 100

Horizontal length: 100

Plane: XY plane

Specify the center point. >> [F12] (Work origin)

3 . Create a reference line from the work origin in the Z-axis direction.



Select the [Parallel] command.

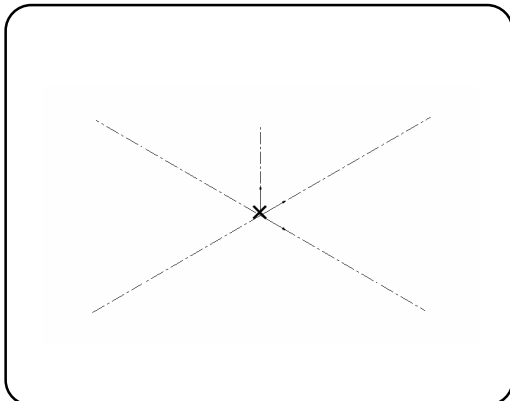
Set parameters in the command sheet.

Reference line: Z-axis

Specify the start point. >> [F12] (Work origin)

Specify the end point direction. >> [0,0,30]

4 . Create a rectangle with solid line.



Select the [Set Line Type (Wire)] command.

Solid line: ON



Select the [Rectangle] command.

Set parameters in the command sheet.

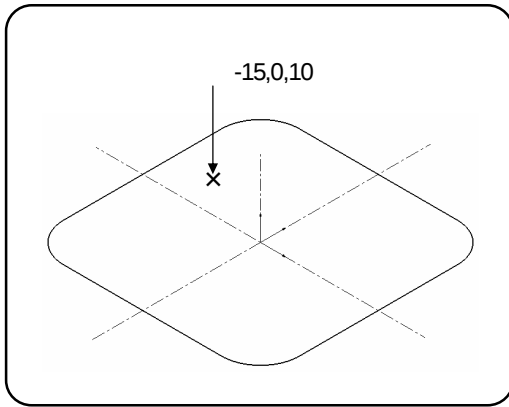
<page1> Specify length: ON, Plane: XY plane

Horizontal: 90, Vertical: 90

<page2> Radius of fillet: 15

Specify the center point. >> [F12] (Work origin)

5 . Create a line in the X-axis direction on the ZX plane.



Select the [Parallel] command.

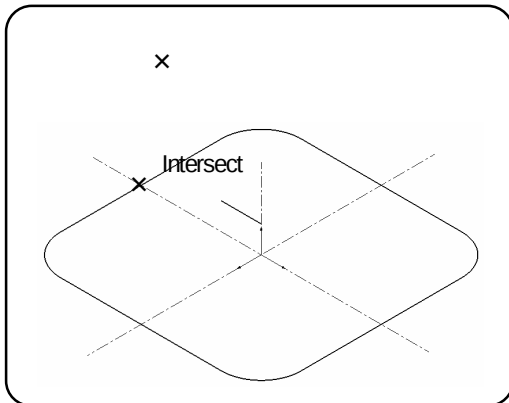
Set parameters in the command sheet.

Reference axis: X-axis

Specify the start point . >> [- 15,0,10]

Specify the end point direction. >> [F12] (Work origin)

6 . Create an angle line of 85 degree on the ZX plane.



Select the [Angle] command.

Set parameters in the command sheet.

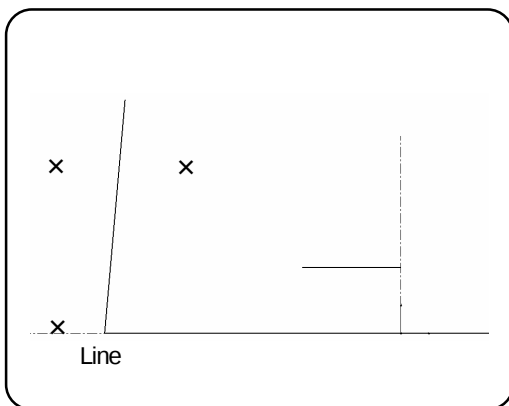
Angle: 85

Plane: ZX plane

Specify the start point. (Intersect)

Specify the end point direction.

7 . Create an interval line at 25mm far from the X-axis in the Z+ direction.



Select the [Interval] command.

Specify the reference line. (Line)

Set parameters in the command sheet.

Distance: 25

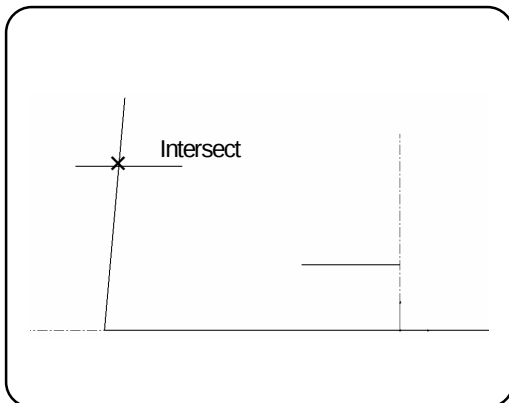
Repeat: 1

Plane: ZX Plane

Specify the start point direction.

Specify the end point direction.

8 . Create a circle of R5 on the ZX plane.



Select the [Circle (Radius, Center)] command.

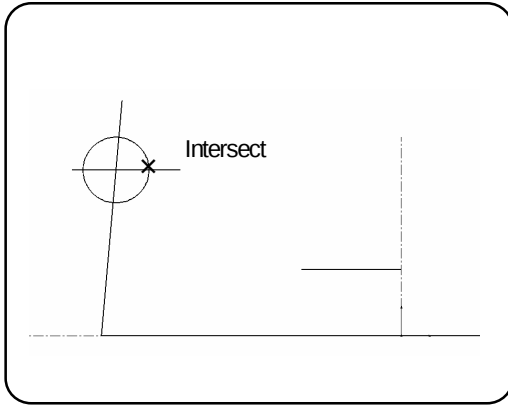
Set parameters in the command sheet.

Radius: 5

Plane: ZX Plane

Specify the center point. (Intersect)

9 . Create a line in the Z-axis direction.



Select the [Parallel] command.

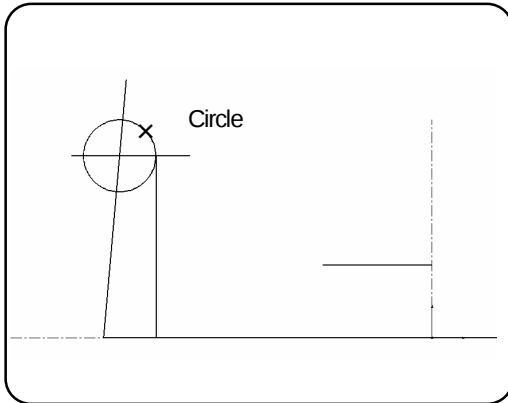
Set parameters in the command sheet.

Reference line: Z-axis

Specify the start point. (Intersect)

Specify the end point direction. >> [F12] (Work origin)

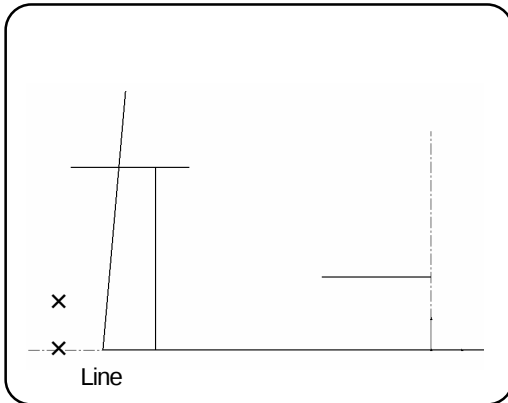
10 . Delete the circle of R5.



Select the [Delete] command.

Specify the entity. (Circle)

11 . Create an interval line at 5mm far from the X-axis in the Z+ direction.



Select the [Interval] command.

Specify the reference line. (Line)

Set parameters in the command sheet.

Distance: 5

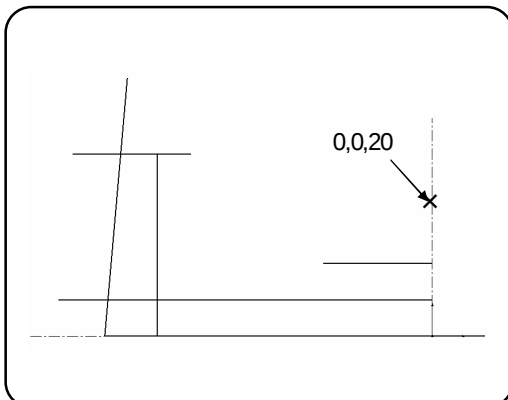
Repeat:: 1

Plane: ZX Plane

Specify the start point direction.

Specify the end point direction. >> [F12] (Work origin)

12 . Create a circle or R3.5 on the ZX plane.



Select the [Circle (Radius, Center)] command.

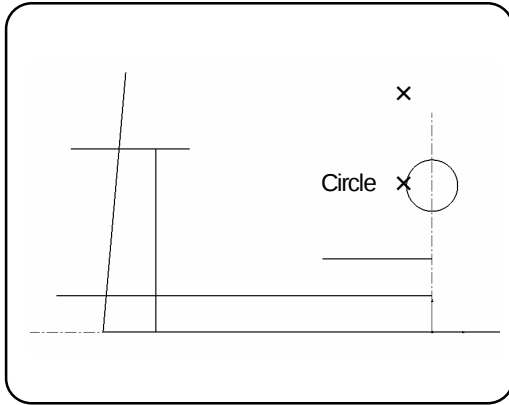
Set parameters in the command sheet.

Radius: 5

Plane: ZX Plane

Specify the center point. >> [0,0,20]

13 . Create a line from the end point of the circle in the Z-axis direction.



Select the [Parallel] command.

Set parameters in the command sheet.

Reference line: Z-axis

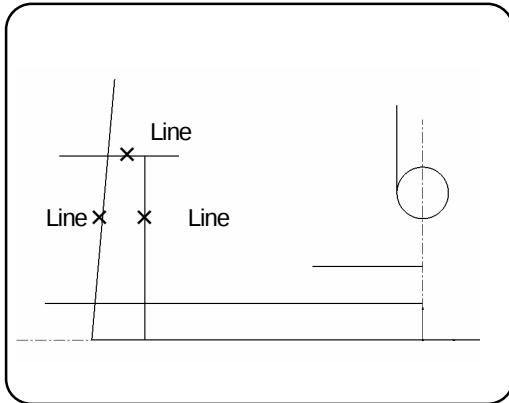
Specify the start point.

Right click >> [Input by Get data: End point]

Specify the entity. (Circle)

Specify the end point direction.

14 . Trim the unnecessary parts.



Select the [Trim] command.

Set parameters in the command sheet.

Entity: Both sides

Dividing entity: Two dividing points

Trim: Intersection

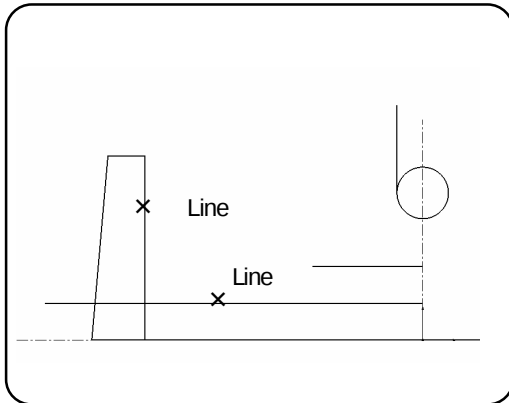
Entity number: Single

Specify the entity to be trimmed (to be left). (Line)

Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

15 . Trim the unnecessary parts with the same operation above.



Set parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

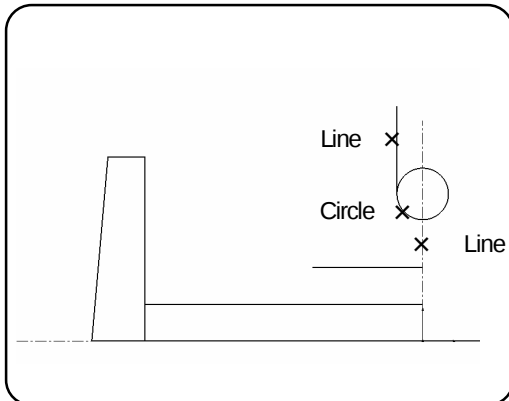
Trim: Intersection

Entity number: Single

Specify the entity to be trimmed (to be left). (Line)

Specify the reference entity. (Line)

16 . Trim the unnecessary parts of circle.



Set parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

Trim: Intersection

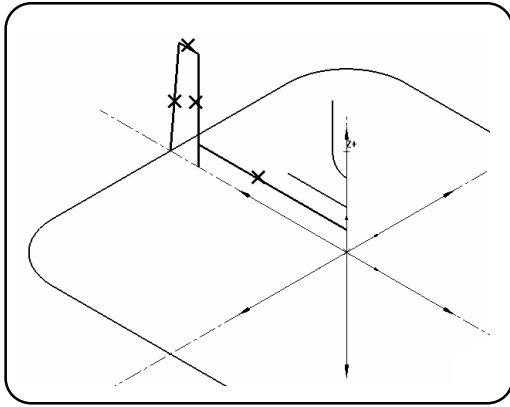
Entity number: Single

Specify the entity to be trimmed (to be left). (Circle)

Specify the first reference entity. (Line)

Specify the second reference entity. (Line)

17. Rotate the section entity and copy it.



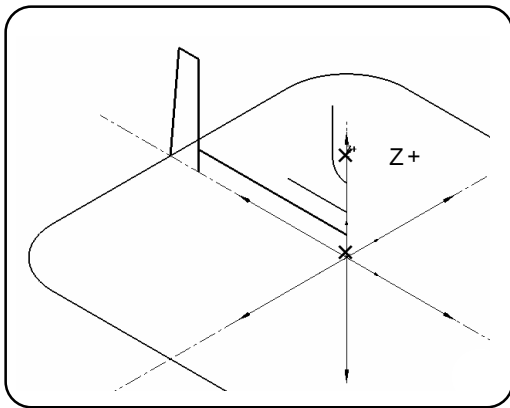
Select the [Rotate/Copy] command.

Set parameters in the command sheet.

Copy, Angle: 90, Select: OFF

- Specify the entities. (Line)

Press the center button to determine the specifying.

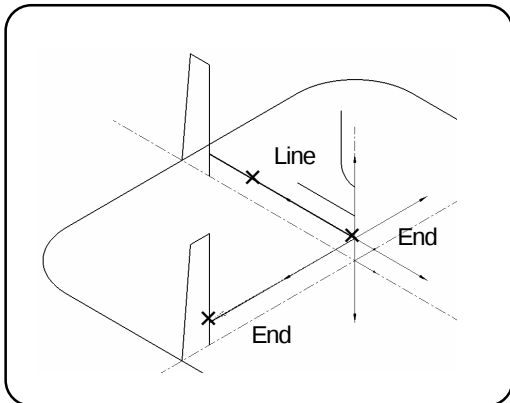


Specify the origin of rotating axis. >> [F12] (Work origin)

Specify the direction entity. >> [Z (+)]

Press the center mouse button to execute the command.

18 . Copy the line to create a rectangle.



Select the [Move/Copy] command.

Set parameters in the command sheet.

Copy, Select: OFF

Specify the entity to be copied. (Line)

Press the center button to determine the specifying.

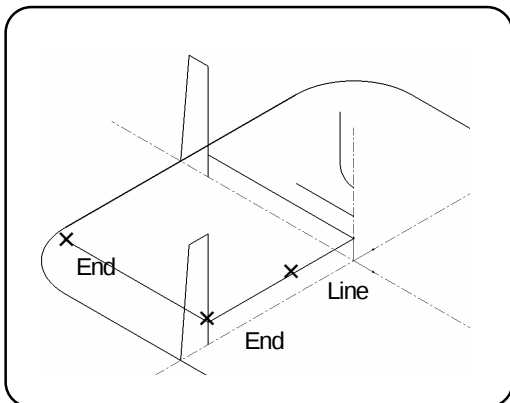
Specify the coordinate for moving origin. (End/Intersect)

Specify the coordinate for moving end point

(End/Intersect)

Press the center mouse button to execute the command.

19 . Copy the line again with the same operation.



Specify the entity to be copied. (Line)

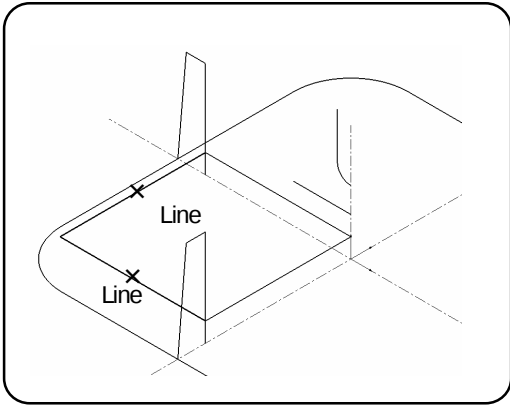
Press the center button to determine the specifying.

Specify the coordinate for moving origin. (End/Intersect)

Specify the coordinate for moving end point. End)

Press the center mouse button to execute the command.

20 . Create a fillet surface of R10 at the corner of the rectangle.



Select the [Fillet] command.

Set parameters in the command sheet.

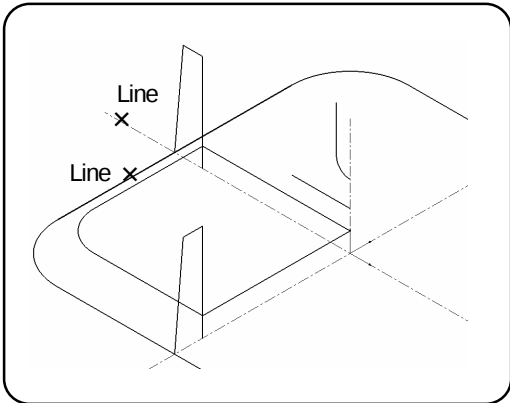
Radius: 10

Entity: Both sides

Specify the first tangent entity. (Line)

Specify the second tangent entity. (Line)

21 . Trim the unnecessary parts.



Select the [Trim] command.

Set parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

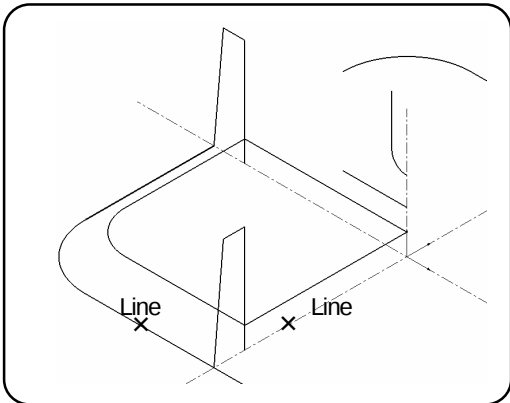
Trim: Intersection

Entity number: Single

Specify the entity to be trimmed (to be left). (Line)

Specify the reference entity. (Line)

22 . Trim the unnecessary parts with the same operation above.



Set parameters in the command sheet.

Entity: One side

Dividing entity: One dividing point

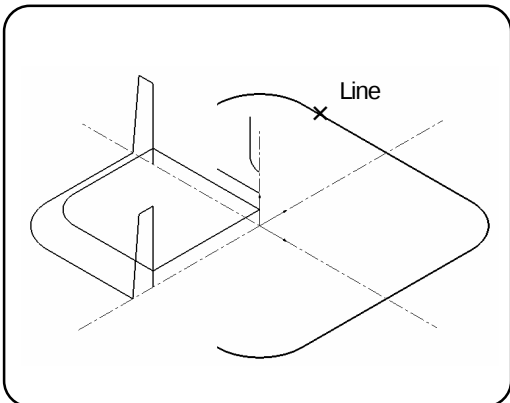
Trim: Intersection

Entity number: Single

Specify the entity to be trimmed (to be left). (Line)

Specify the reference entity. (Line)

23 . Put the unnecessary parts in the selected status, and delete them.



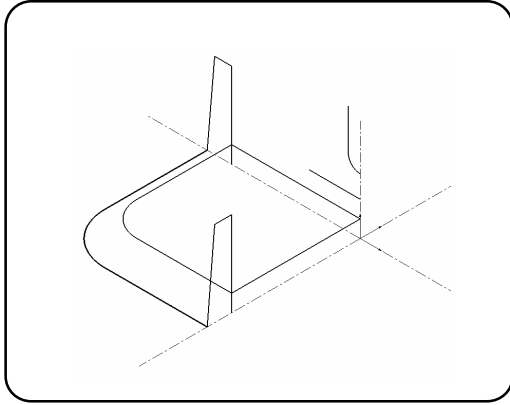
Execute the [Select(Entity)] command.

Right-click >> pop-up menu >> Chain

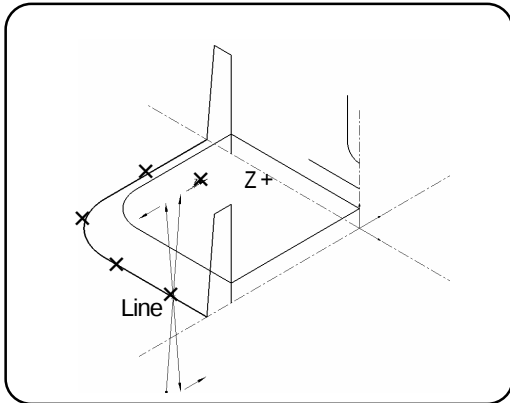
Specify the entities to be selected. (Line)

Select the [Delete] command.

24 . Wire frame entities have been created. Next, create surfaces.



25 . Create a taper surface of 5 degree for outer side.



Execute the [Select(Entity)] command.

- Specify the entities to be selected. (Line, Arc, Line)



Select the [Taper Surface] command.

Set parameters in the command sheet.

Angle:5, Z-length:25

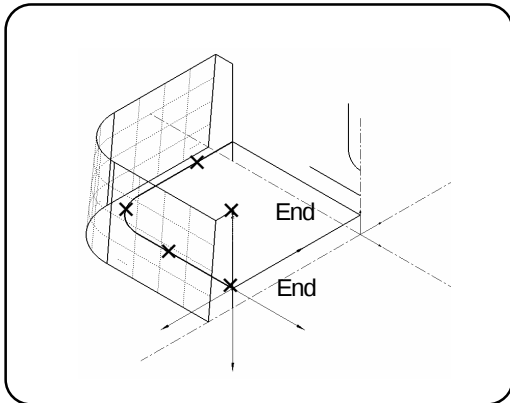
Each contour: ON, [Normal]

Specify the guide line.

Specify the direction entity (Z+ direction/Inner)

Press the center mouse button to execute the command.

26 . Create a cylinder surface for inner side.



Select the [Cylinder Surface] command.

Set parameters in the command sheet.

Start position: 0

- Specify the entities. (Line , Arc , Line)

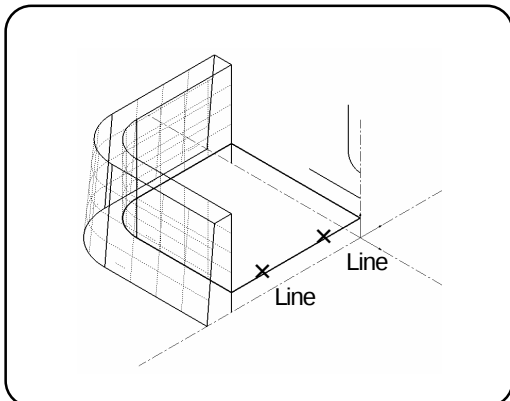
Press the center button to determine the specifying.

Specify the origin. (End/Intersect)

Specify the end point. (End/Intersect)

Press the center mouse button to execute the command.

27 . Create a multi surface for bottom surface.



Execute the [Select(Entity)] command.

Right-click >> pop-up menu >> Chain

Specify the entity. (Line) >> The closed contour is selected.

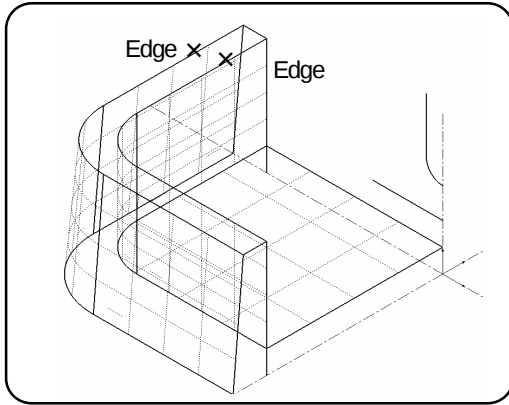


Select the [Multi Surface] command.

Specify the contour. (Line)

Press the center mouse button to execute the command.

28 . Create a multi surface for top surface.



Select the [Multi Surface] command.

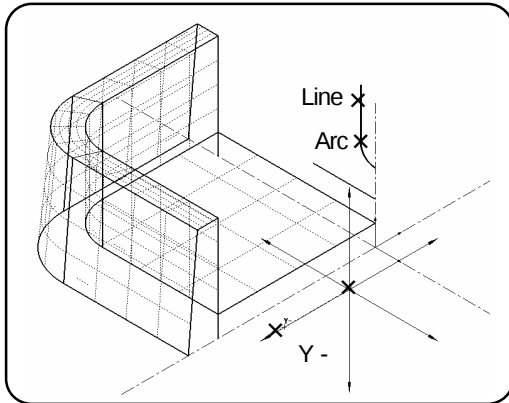
Specify the contour. (Edge)

Specify the contour. (Edge)

Press the center mouse button to execute the command.

Create multi surfaces for the other two parts with the same operation above.

29 . Create a cylinder surface for the part where cigarette is placed.



Select the [Cylinder Surface] command.

Set parameters in the command sheet.

Distance: 10, Start position: 35

- Specify the surfaces. (Line, Arc)

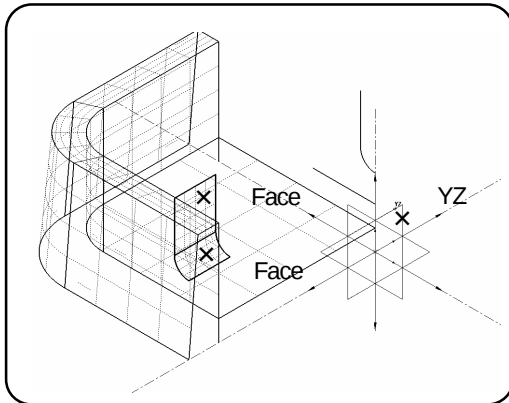
Press the center button to determine the specifying.

Click a certain position on the screen.

Specify the direction entity. (Y- direction)

Press the center mouse button to execute the command.

30 . Copy the surface where cigarette is placed, to the other side based on the specified plane.



Select the [Mirror/Copy] command.

Set parameters in the command sheet.

Copy, Select: ON, Select copied entity

- Specify the surfaces. (Face)

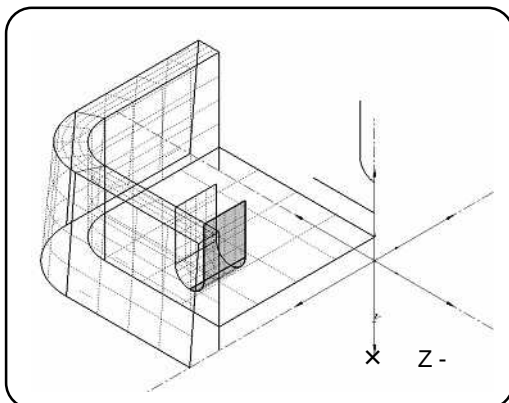
Press the center button to determine the specifying.

Specify the copying origin. >> [F12] (Work origin)

Specify the plane. (YZ plane)

Press the center mouse button to execute the command.

31 . Move the selected entities.



Select the [Rotate/Copy] command.

Set parameters in the command sheet.

Move, Revolution, Angle:90, Select: OFF

Specify the entities. (Already selected) >> No need to select here.

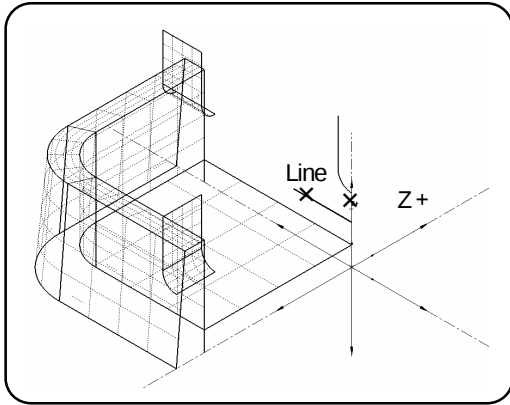
Press the center button to determine the specifying.

Specify the moving origin. >> [F12] (Work origin)

Specify the direction entity. (Z- direction)

Press the center mouse button to execute the command.

32 . Create a core shape at the center of ashtray, with [Revolved Surface].



Select the [Revolved Surface] command.

Set parameters in the command sheet.

<page1> Axis, Start angle: 0, End angle: 90

Specify the entity. (Line)

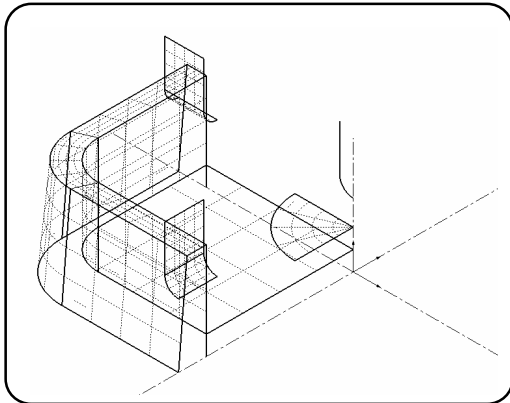
Press the center button to determine the specifying.

Specify the rotating origin. >> [F12] (Work origin)

Specify the direction entity. (Z+ direction)

Press the center mouse button to execute the command.

33 . Move the wire frame entities to layer10.



Execute the [Select (Except Surface)] command.



Select the [Layer Move/Copy] command.

Set parameters in the command sheet.

Move, Layer10, Select: OFF

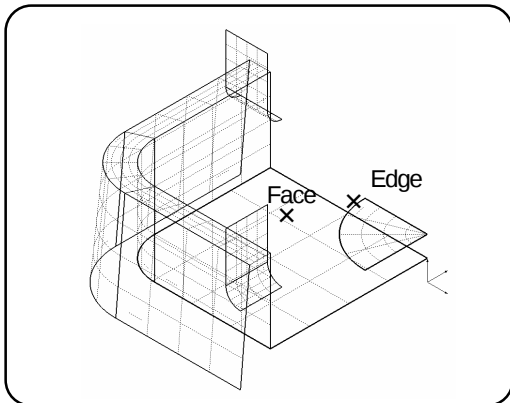
Specify the entities. (Already selected) >> No need to select here.

Press the center button to determine the specifying.



Select the [Hide All Layers] command.

34 . Create a surface or R10 between the contour of the core shape at the center of ashtray and the bottom surface.



Select the [S Fillet Surface] command.

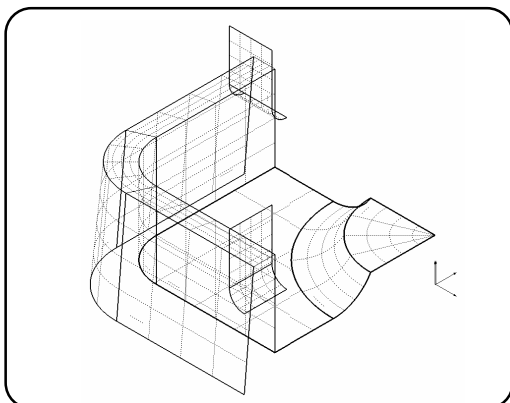
Set parameters in the command sheet.

Radius:10, Curvature continuous: OFF, All entities

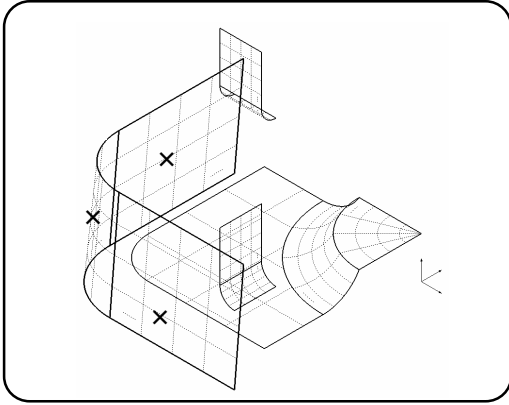
Specify the surface contour as guide line. (Edge)

Specify the surface. (Face)

Press the center mouse button to execute the command.



35 . Create surfaces that are tangent to the outer side surfaces, the top surfaces, and inner side surfaces.

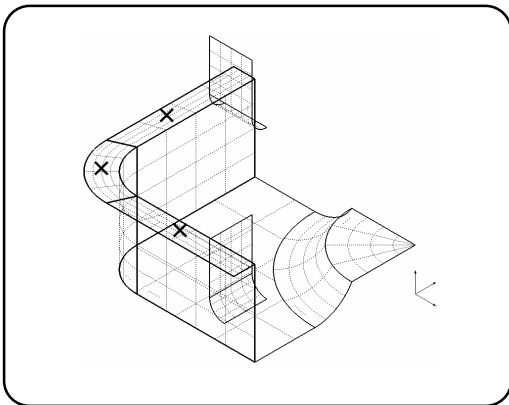


Select the [3-Tangent Fillet Surface] command.

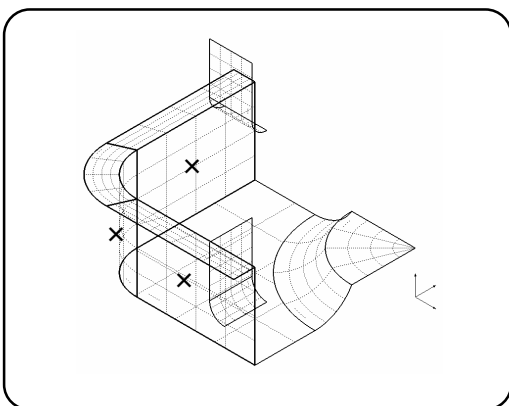
Set parameters in the command sheet.

Guide line: None, Section spacing: 0.5

- Specify the first group of surfaces (outer side). (Face)
Press the center button to determine the specifying.

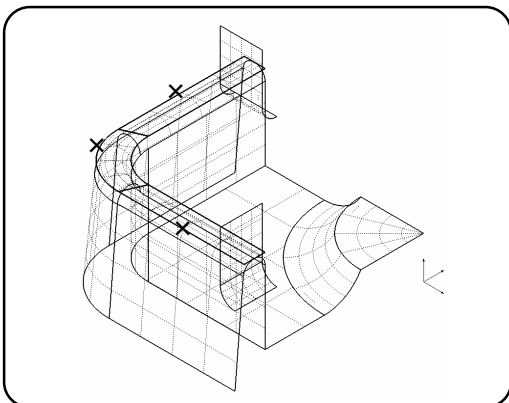


- Specify the second group of surfaces (top). (Face)
Press the center button to determine the specifying.



- Specify the third group of surfaces (inner side) (Face)
Press the center mouse button to execute the command.

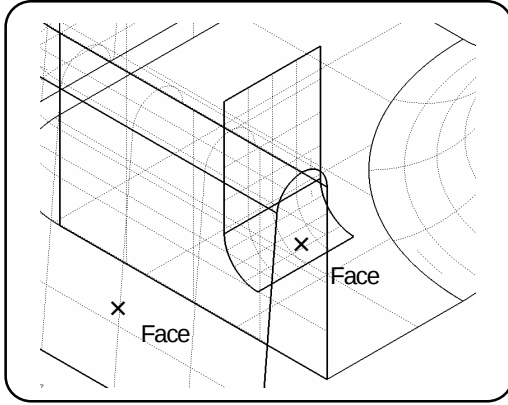
36 . Delete the unnecessary top surfaces.



Select the [Delete] command.

- Specify the surfaces. (Face)

37 . Create a fillet surface of R1 at the corner of the part where cigarette is placed.



Select the [Fillet Surface] command.

Set parameters in the command sheet.

<page1> Radius: 1, Curvature continuous: OFF, All entities

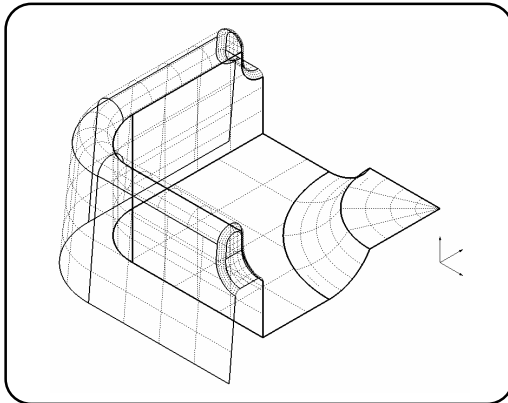
<page2> Extend fillet: OFF, Extend surface: OFF

<page3> Trim: ON, First: ON, Second: ON

Specify the first surface. (Face)

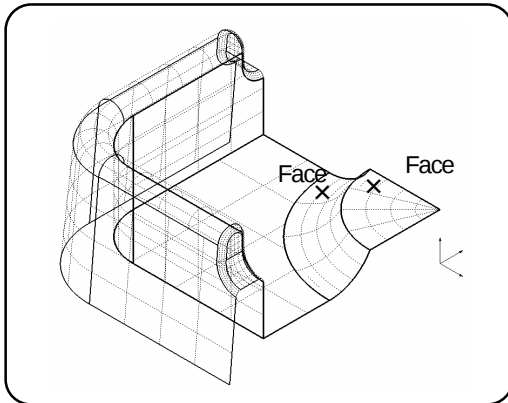
Specify the second surface. (Face)

Press the center mouse button to execute the command.



Create a fillet surface at the other side by 90 degree with the same operation.

38 . Create a fillet surface of R1 at the corner of the core shape, which is at the center of the ashtray.

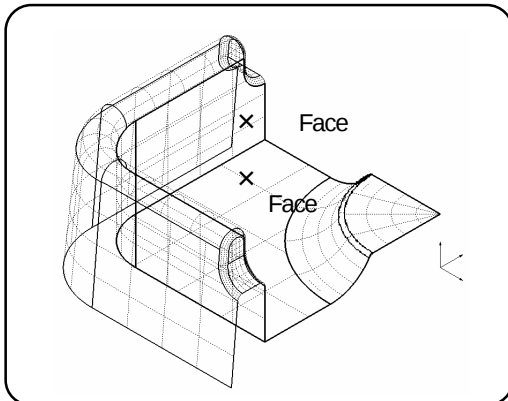


Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center mouse button to execute the command.

39 . Create fillet surface of R5 at the corner between the inner side surface and the bottom surface.



Set parameters in the command sheet.

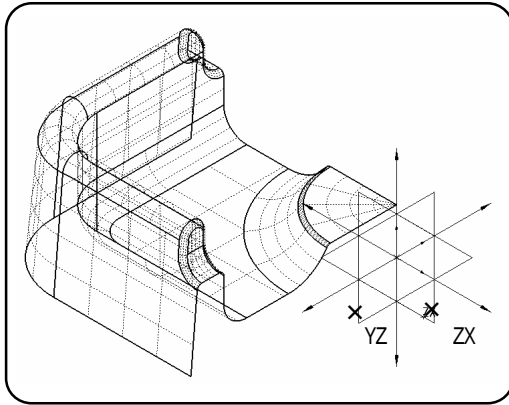
<page1> Radius:5

Specify the first surface. (Face)

Specify the second surface. (Face)

Press the center mouse button to execute the command.

40 . Copy all surfaces to the other side based on the ZX plane and the YZ plane.



Execute the [Select (All)] command.

Select the [Mirror/Copy] command.

Set parameters in the command sheet.

Copy, Select: OFF

Specify the entities. (Already selected) >> No need to select here.

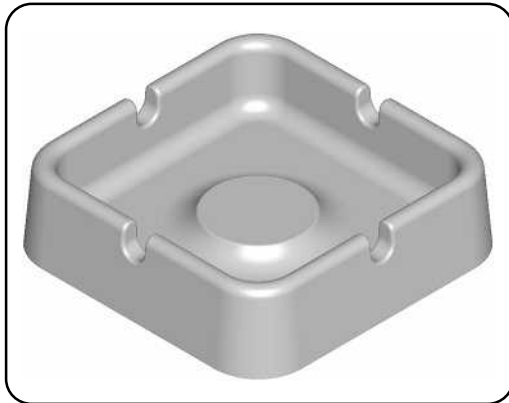
Press the center button to determine the specifying.

Specify the copying origin. >> [F12] (Work origin)

- Specify the plane.(YZ plane, ZX plane)

Press the center mouse button to execute the command.

41 . Completed! Confirm the shape and save the file.



Select the [View Type (Shade)].

Confirm the created shape.



Select the [View Type (Wire)].



Select the [Save] command.

Saved in Ex.) C: /cam-tool/Cam-toolFile

File name Exercise 3

Click the [Save] button.

Chapter 10

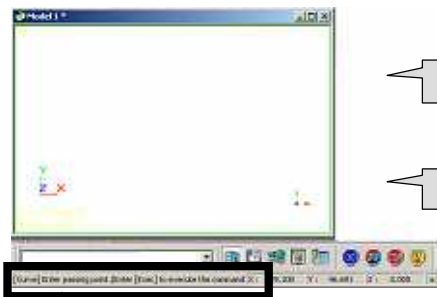
Translate data / Verify surface

Open the IGES file below.

CD-ROM/Intensive Course/sample03.igs

- Follow the [command message] displayed at the lower left of the screen, for your operation.

Ex.) Create a [Single] line by inputting [absolute coordinate value].



[Curve] Specify the start.

Left-click at the coordinate of start.

[Curve]Enter passing point.

Left-click at the coordinate of end.

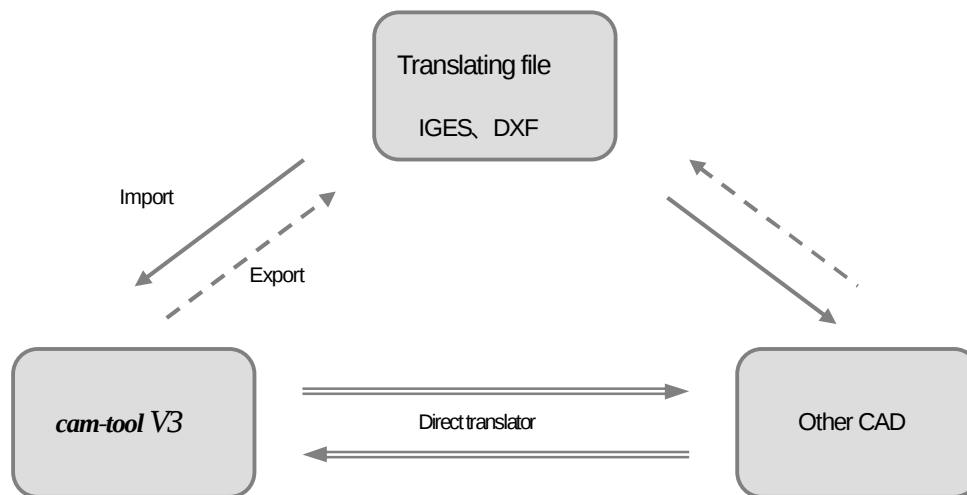
Command message

What is data translation?

There are two methods to translate data. You can convert data into translation file such as [IGES] or [DXF], and import the converted data into cam-tool V3. Or, if you use translating software, you can open data directly in cam-tool V3.

■ Translating file

In *cam-tool V3*, data is translated by [IGES] or [DXF] as the standard function.



■ Direct translator

Other CAD data can be opened directly using our translating software.

GRAPHIC PRODUCTS provide the following soft wares as option functions.

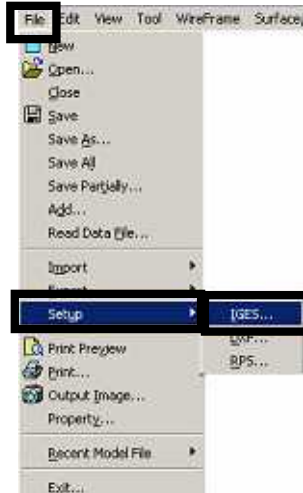
- CATIA V4 (IN/OUT)
- CATIA V5 (IN/OUT)
- Pro/ENGINEER (IN/OUT)
- I-DEAS (IN/OUT)
- UNIGRAPHICS (IN/OUT)

... Other files

Settings for IGES translation

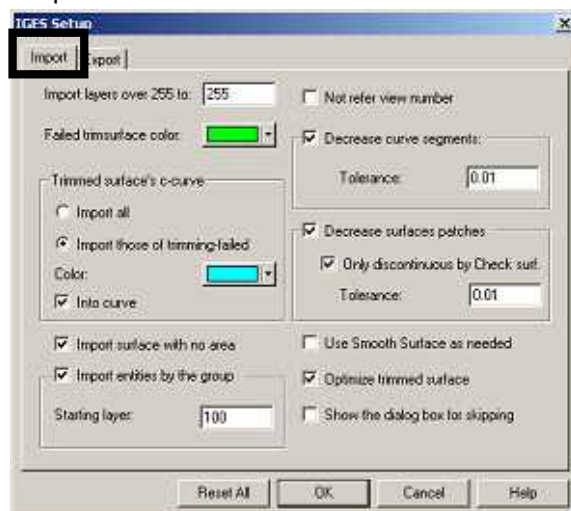
Set conditions to translate IGES data (Import/Export IGES data).

- Select from the menu

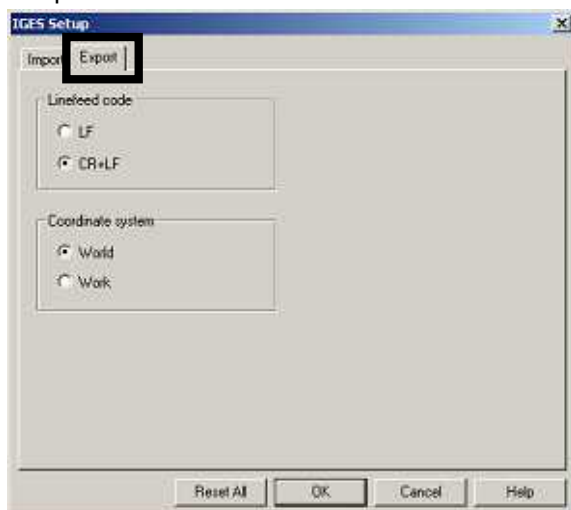


Click [Menu bar] >> [File] >> [Setup] >> [IGES]

<Import>



<Export>



- Import layers over 255 :

If entities are placed in the layer other than [1-255], specify the layer number.

- Failed trim surface color:

Specify the color of the surface that has not been trimmed successfully.

- Trimmed surface's C-CURVE (Trimmed contour curve)

Import all: Trim contours of all of trimmed surfaces are imported as curves.

Import those of trimming-failed: Only trim contour that has not been trimmed successfully is imported as curves.

Color: Specify the color of when trim contour is imported as curve.

Into curve: Modify trim contour including connecting space, and convert the contour into curve.

- Import surface with no area

ON: If a surface whose area is [0], and it is consisted of trim contours, is included in the data, it is imported as the original surface.

OFF: A surface whose area is [0] is not imported.

- Import entities by the group

ON: If group information is included in the data, entities are imported by each group separately.

Starting layer: The first group is imported in the [Starting layer].

And next group is imported in the next layer.

OFF: Entities are imported in their respective layer.

- Not refer view number

ON: Not import view number of the directory.

OFF: Import view number of the directory.

- Decrease curve segments:

ON: Reduce the segment number of curves within the specified tolerance. Data can be reduced.

OFF: Import curves without reducing the segment number of curves. Data can be reduced.

- Decrease surfaces patches

ON: Reduce the patch number of surfaces within the specified tolerance.

OFF: Import surfaces without reducing the patch number of surfaces.

- Only discontinuous by Check surf.

ON: Only surface that is detected as invalid surface by [Check Surface], will be the target for this function.

OFF: All of surface will be target for this function.

- Use Smooth Surface as needed

ON: Only surface that is detected as invalid surface by [Check Surface], will be modified automatically.

OFF: Import surfaces without modifying surfaces.

- Optimize trimmed surface

ON: When original surface of the trimmed surface is imported, the original surface is contracted based on the trimmed contour.

OFF: When original surface of the trimmed surface is imported, the original surface is not contracted

- Show the dialog box for skipping

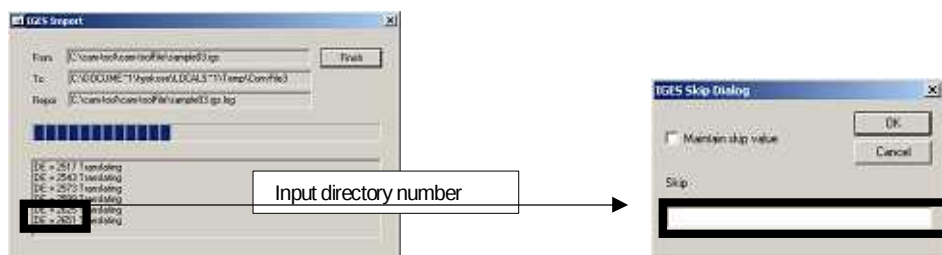
ON: [Skip dialog] is displayed when importing.

OFF: [Skip dialog] is not displayed when importing.



What is IGES skip dialog?

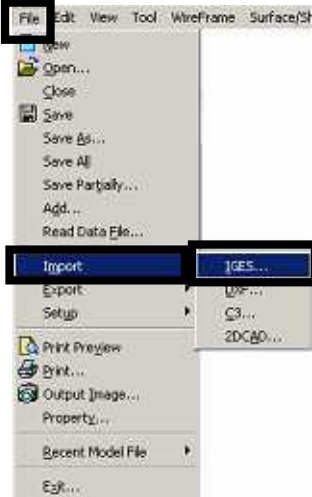
If the data has invalid information, conversion may not be end successfully. In that case, input the directory number which has the invalid information on this [Skip dialog box] to skip the entities (not convert the entities). It leads to reducing conversion time.



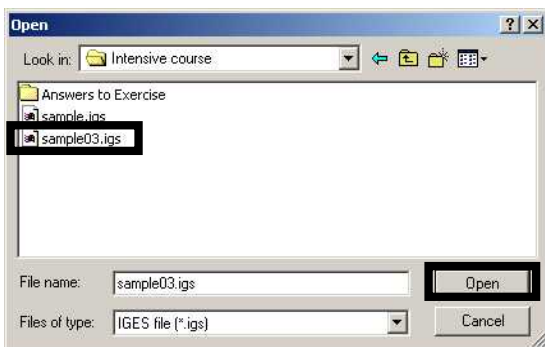
How to import IGES data

Open an IGES file as a model file of *cam-tool* V3.

■ Select from the menu



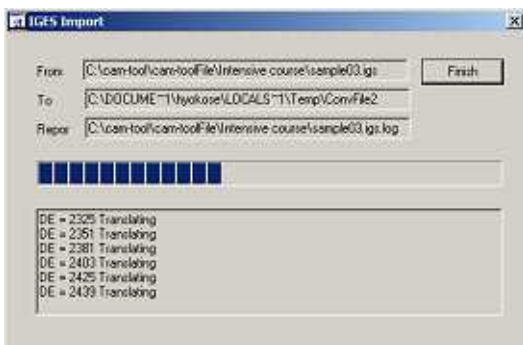
Click [File] >> [Import] >> [IGES]



Select a IGES file.

/Intensive Course/sample03.igs

Click the [Open] button.



The file is opened after conversion is completed.



[IGES file (*.igs)] is set for [Files of type] in the [Open] dialog, so that files with extension [.igs] are displayed.

What operations are needed after data translation?

Operating flow of processes needed after data translation is described with the sample shape.

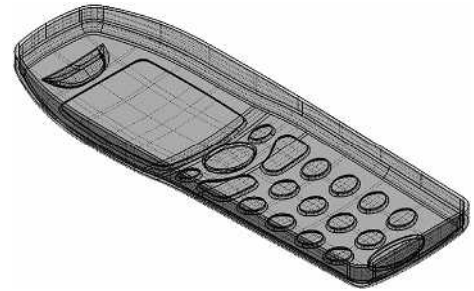
1. Data translation (Verify surface/Modify surface after conversion)



2. Developing data to cutting shape (Operations for CAM calculation)



3. Analyze shape



Operating flow after IGES data translation

Verify surfaces imported by [data translation] with the following operations. If invalid surface is detected, modify the surface. Otherwise, invalid surface may cause creating invalid [CL data].

Check surface (Confirm the consistency of each surface)

Self interference → Create a new surface (>> Create surface)

High density points → Correct Surface Patch: Delete high density points

... When cannot be modified (>> Create surface)

Hard tangency → Correct Surface Patch: Correct hard tangency

... When cannot be modified (>> Create surface)

Incorrect contour → Untrim Surface/Trim Surface (>> Modify surface)

Spike/cut → Correct Surface Contour, Correct spike/cut

... When cannot be modified (>> Create surface)

Micro edge → Correct Surface Contour, Correct micro edge

... When cannot be modified (>> Create surface)

Adjacent Surface Info (Confirm the consistency with adjacent surfaces)

Open edge → Create surface (>> Create surface)

Overlapped surface → Trim surface overlapped area (>> Modify surface)

Gap → Expand surface and Trim surface (>> Modify surface)

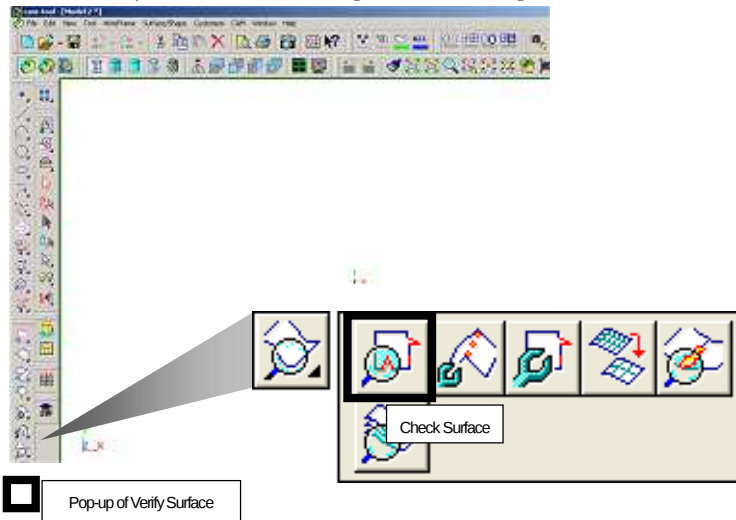
Non continuous → * Confirm that modifying the surfaces, which are not connected smoothly, is necessary.

Check overlapped surfaces (Confirm if the same surfaces exist or not.)

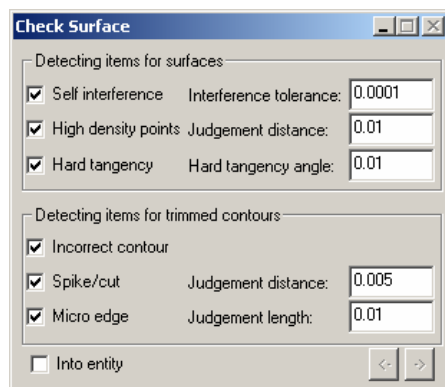
Overlapped surface → Delete overlapped surfaces

How to detect invalid surface/trim contour <Check Surface>

Check invalid part of surfaces with [Check Surface] command.



<Command sheet>



Items for verifying surface

Self interference: Detects self-interference parts on the original surface.

- Interference tolerance: Detects the seam that is equal to or larger than this value.

High density points: Detects high density points on the original surface.

- Judgement distance: Detects a space between patches which is equal to or smaller than this value.

Hard tangency: Detects hard tangency between patches on the original surface.

- Hard tangency angle: Detects hard tangency when the angle between patches is equal to or larger than this value.

Items for verifying trim contour

Incorrect contour: Detects a trimmed surface that has invalid trim contours.

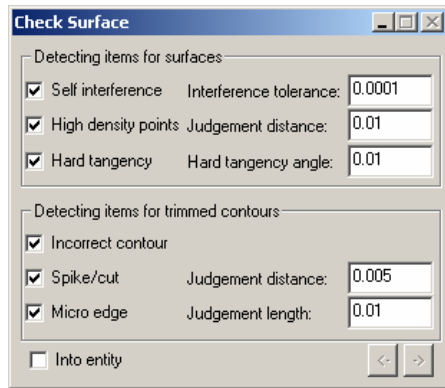
Spike/cut: Detects [spike/cut] parts on the outer contour of the trimmed surface.

- Judgement distance: Tolerance for detecting spiked/cut areas.

Micro edge: Detects micro edges whose length is equal to or smaller than "Judgement length".

- Judgement length: Tolerance for detecting micro edge.

<Operation>

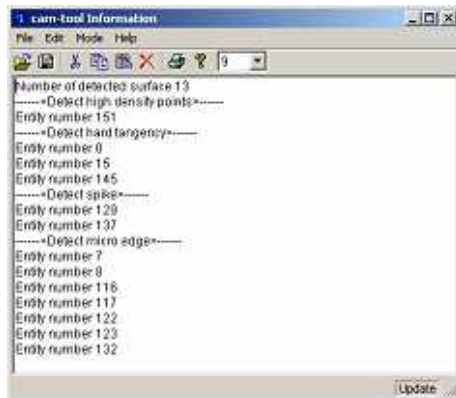


Select the [Check Surface] command.

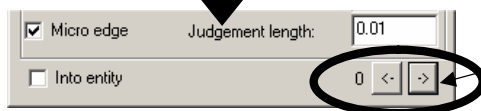
Specify parameters in the command sheet.

Ex.) All ON (Only Into curve: OFF)

Select all entities, and press the center mouse button to execute the command.



The result is displayed on the information window, and the detected parts are selected.



Click button, and the detected entity will be zoomed in and displayed in turn.

If [Self-interference] detected? >> Recreate surface

If [High density points] detected? >> [Correct Surface Patch] command (page.206)

If [Hard tangency] detected? >> [Correct Surface Patch] command (page.207)

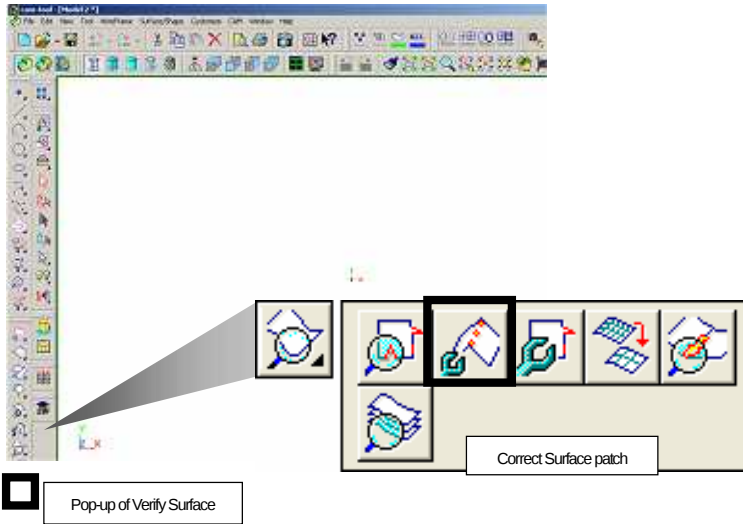
If [Invalid contour] detected? >> Recreate surface

If [Spike/cut] detected? >> [Correct Surface Contour] command (page.209)

If [Micro edge] detected? >> [Correct Surface Contour] command (page.210)

How to modify invalid surface automatically <Correct Surface Patch>

Modify invalid surface depending on the detected items with [Check Surface].



<Command sheet>

Correct Surface Patch

Unused patch
☒ Delete

High density points
☒ Delete Judgement width: 0.01

Hard tangency
☒ Correct Method: Tangent line continuity
 Hard tangency angle: 0.01

Save original invalid surfaces
☐ Save Layer: 255

Tolerance: 0.001

Unused patch: Detect unused patches and delete them.

High density points: Delete
 · Judgement width Detect high density points and delete them.
 Delete patch, which width between patch lines is smaller than this value.

Hard tangency: Modify hard tangency.
 · Method: Select from [Tangent line continuity], [Curvature continuity], [Divide]
 · Hard tangency angle: The surface is divided if the angle of hard tangency is larger than this value.

Tolerance: Specify tolerance.
 When distance between the original surface and the modified surface is longer than this value, modification will not be done.



When the distance above is longer than the specified tolerance, an error message is displayed on the information dialog. In that case, change the tolerance, and execute the command again.

How to delete [unused patch]

Information of unused surface can be deleted as [unused patch].

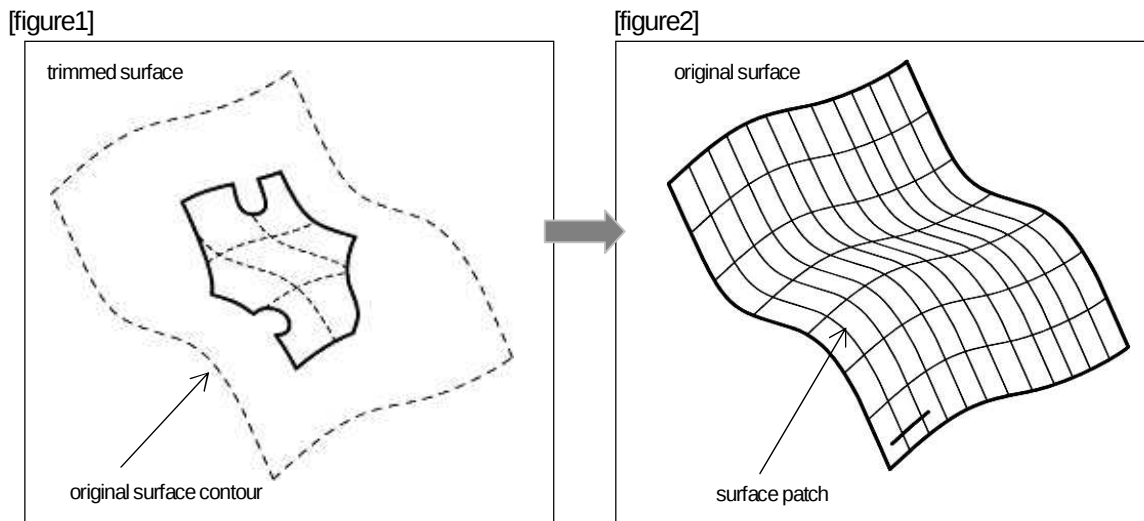
<Operation>

Select the [Correct Surface Patch] command.

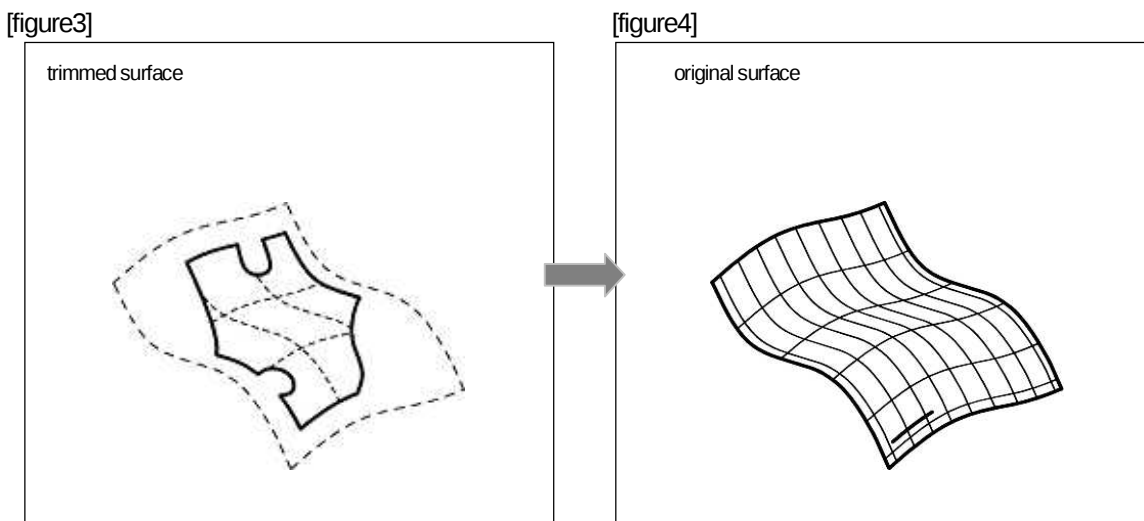
Specify parameters in the command sheet. >> Unused patch: ON

Select all surfaces and press the center mouse button to execute the command.

- Although [figure1] is the trimmed surface, it has the information of original surface in [figure2]. (Confirm with [Untrim] command.)



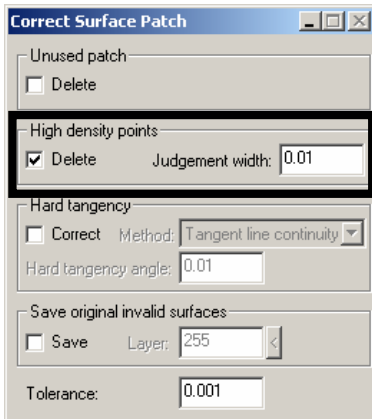
- When [Correct Surface Patch] command is executed with [Unused patch] ON, unused patches are deleted. As a result, the trimmed surface is modified so as to be the surface in [figure3], and it has the information of original surface in [figure4]. Unnecessary information is deleted, and model file data can be reduced.



How to delete [High density points]

Detects high density points on the original surface and delete them.

<Operation>



Select the [Correct Surface Patch] command.

Specify parameters in the command sheet.

High density points, Delete: ON

• Judgement width: Ex.) 0.01

Tolerance: Ex.) 0.001

Select target surfaces and press the center mouse button to execute the command.

(If already selected, no need to select here)

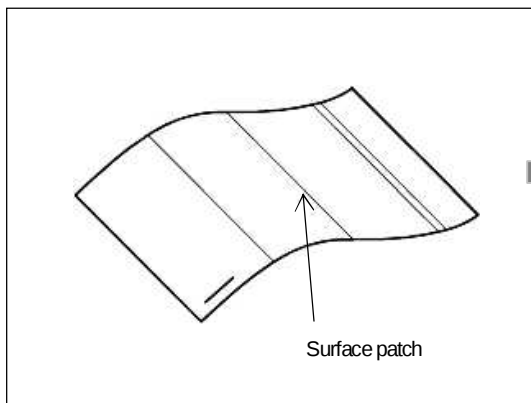
If modification is done normally, the selected status will be canceled.

*If modification failed, the selected status will not be canceled.

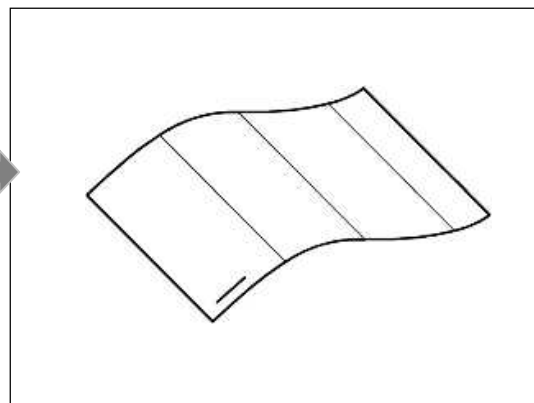
Change tolerance and execute the command again.

There are high density points in [figure1]. Delete high density points which distance between the patches is shorter than the specified [Judgement width], with [Correct Surface Patch, High density points: ON], so as to be the surface in [figure2].

[figure1]



[figure2]



Convenient usage of the command

If you execute the [Check Surface] command with only [High density points] function ON, only the detected [high density points] will be in the selected status. And then, execute [Correct Surface Patch, High density points: ON]. Operations of specifying entities can be omit.

How to delete hard tangency

Detects hard tangency between patches in the original surface, and delete them.
There are three methods to modify surface.

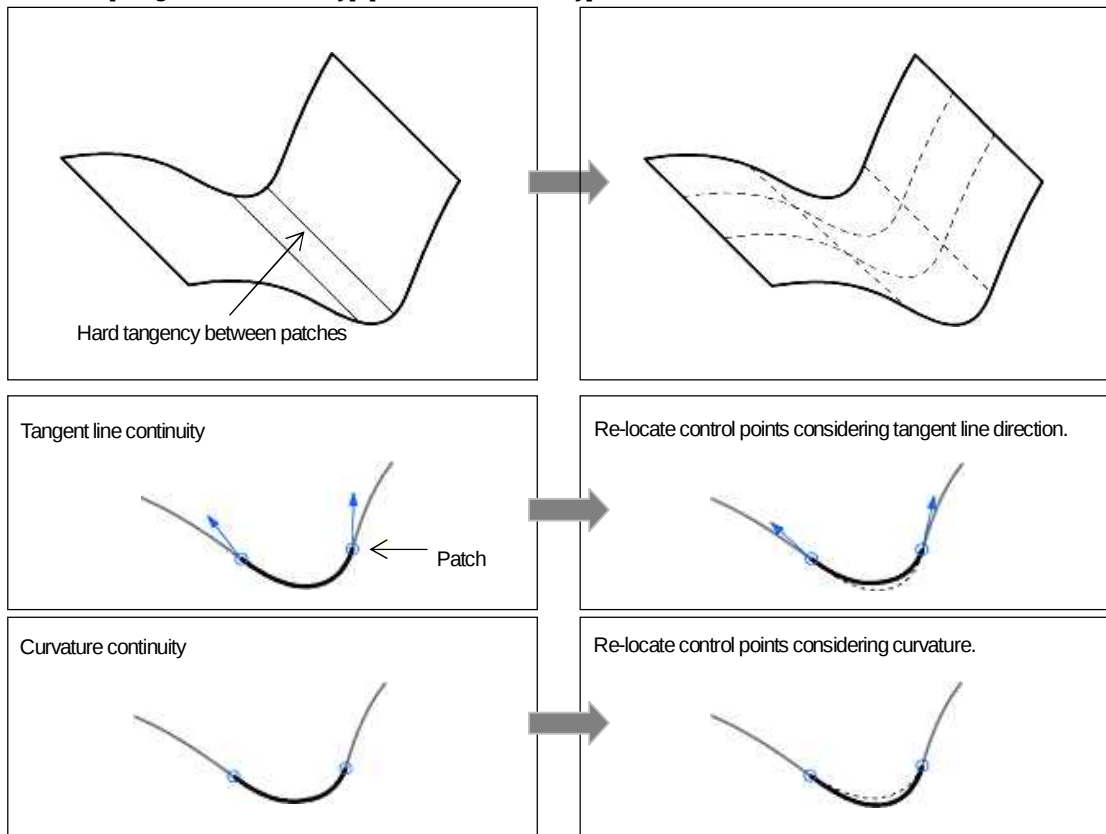
<Operation>

Select the [Correct Surface Patch] command.

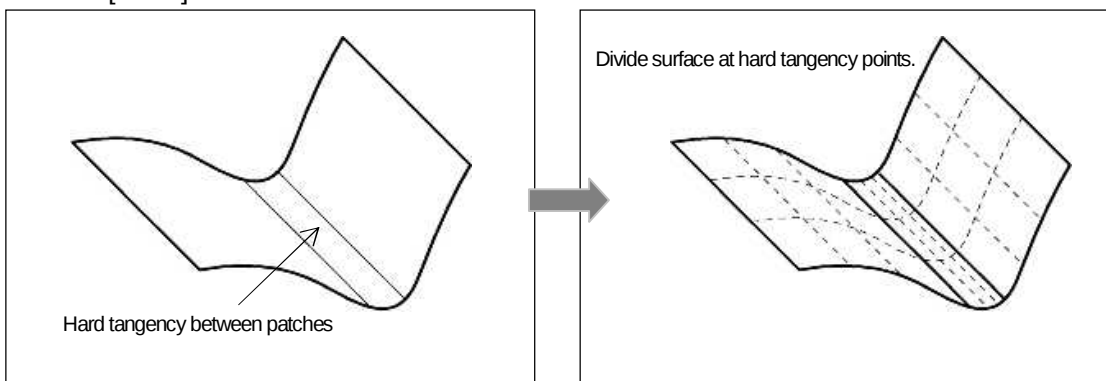
Specify parameters in the command sheet. >> Hard tangency: ON

Select all surfaces and press the center mouse button to execute the command. * If [Check surface] has been executed in advance, it is no need to select here.

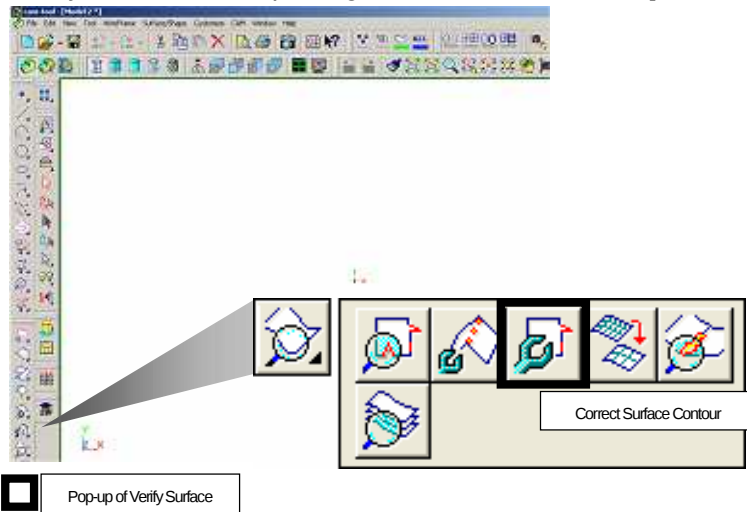
• Method: [Tangent line continuity], [Curvature continuity]



• Method: [Divide]



Modify invalid surface depending on the detected items with [Check Surface].



<Command sheet>

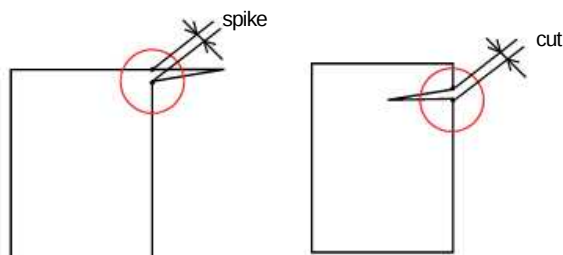


Spike/cut

- Judgement distance

Modify spike/cut areas.

Modify spike/cut that is smaller than this value.



Micro edge

- Judgement length

Modify micro edge

Modify contour (micro edge) that is smaller than this value.

Tolerance

Specify tolerance.

When distance between the original surface and the modified surface is longer than this value, modification will not be made.



When the distance above is longer than the specified tolerance, an error message is displayed on the information dialog. In that case, change the tolerance, and execute the command again.

How to modify spike/cut

Detect spike or cut part of trimmed contour, and delete them.

<Operation>



Select the [Correct Surface Contour] command.

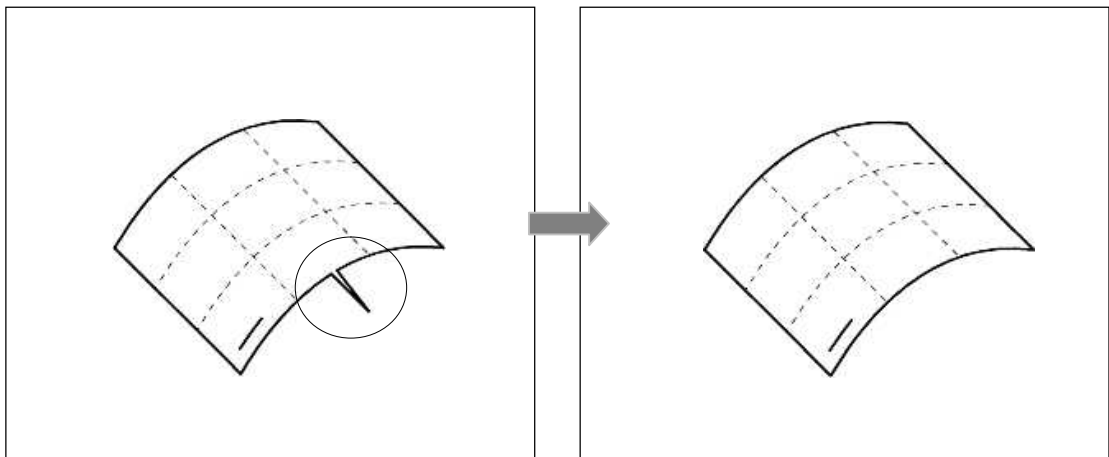
Specify parameters in the command sheet.

Spike/cut: ON

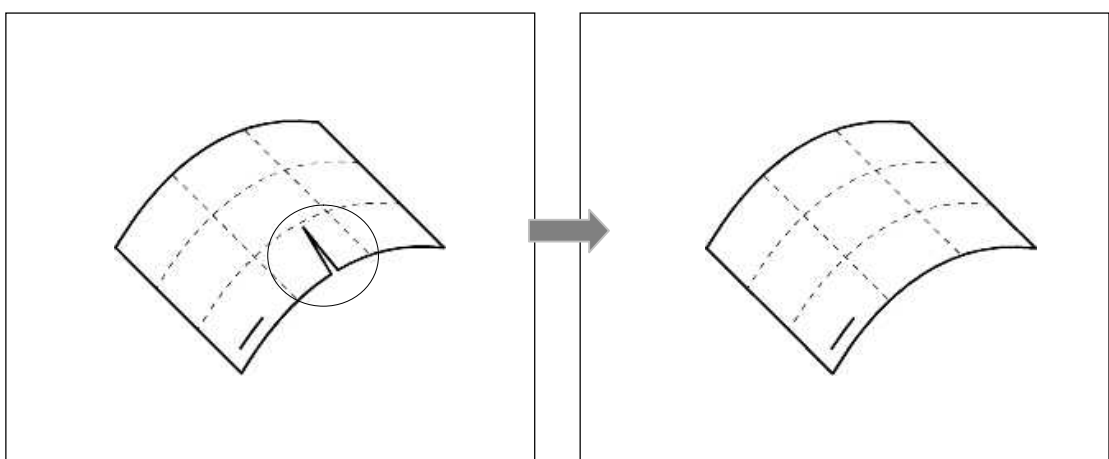
Select target surfaces and press the center mouse button to execute the command.

* If [Check surface] has been executed in advance, it is no need to select here.

• Modify spike



• Modify cut



How to modify micro edge

Detect [short surface contour] of trimmed contour, and delete them.

<Operation>



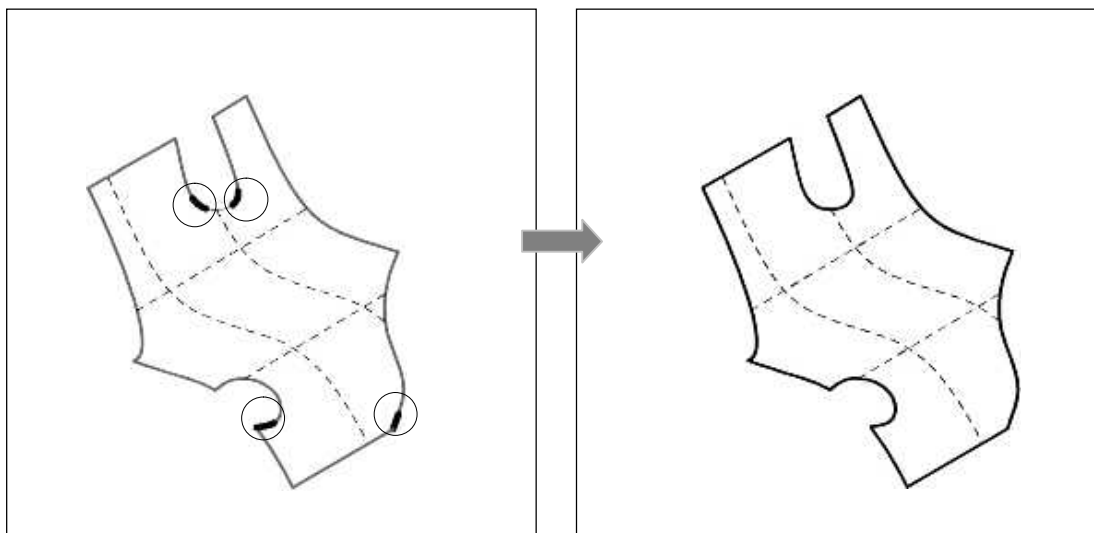
Select the [Correct Surface Contour] command.

Specify parameters in the command sheet.

Micro edge: ON

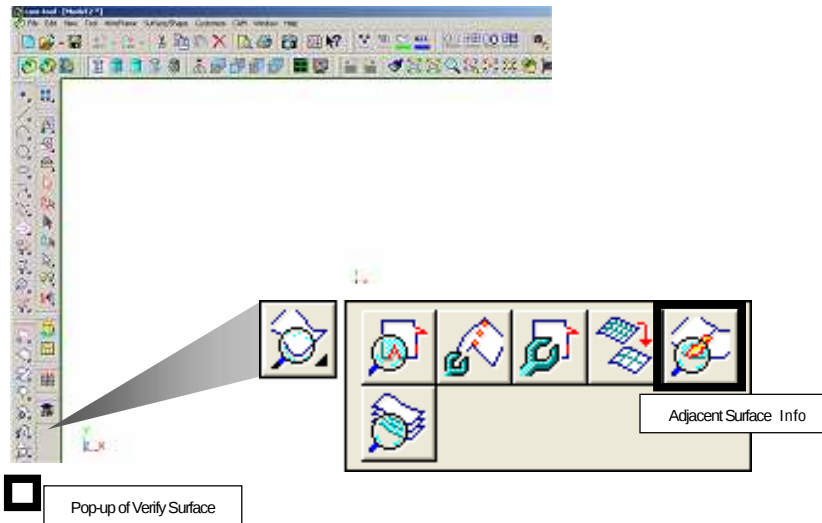
Select target surfaces and press the center mouse button to execute the command.

* If [Check surface] has been executed in advance, it is no need to select here.



How to confirm if there is a part which has no adjacent surface, or gap/ overlapped part between surfaces <Adjacent Surface Info>

Check gap or overlapped part of adjacent surfaces with [Adjacent Surface Info].



<Command sheet>



Open edge: Detect surface contour that no surface exist adjacent to.

- Loop: Detect loop of open edge.

Overlapped surface: Detect overlapped part of adjacent surfaces.

- Interference tolerance: Detect overlapped part of adjacent surfaces that is larger than the specified value.
(Interference tolerance < Shell tolerance)

Gap: Detect gap between adjacent surfaces.

- Gap tolerance: Detect gap between adjacent surfaces that is larger than the specified value.
(Gap tolerance < Shell tolerance)

Non continuous: Detect boundary of adjacent surfaces which has larger angle than the specified value.

- No continuous angle: Detect boundary which has larger angle than the specified value.
(No continuous angle < Max. no continuous angle)

Shell tolerance: When the distance between contours of adjacent surfaces is smaller than the specified value, those surfaces will be target for [Adjacent Surface Info] function.

Retry shelling: Shell processing is repeated to the maximum value of the specified value for shell tolerance. Detecting precision is improved by this function.

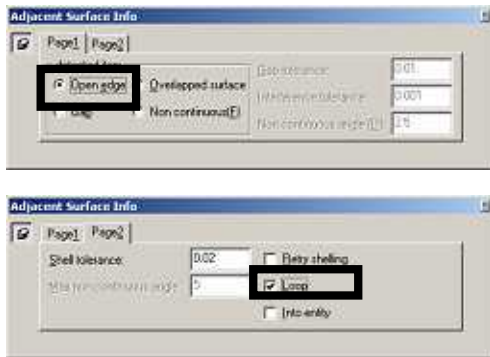
Max. no continuous angle: When [No continuous] function is executed, smaller angle than the specified value will be target for the detecting.

Into entity: Convert the detected result into entity.

How to confirm if a part that has no surface exist

Detect surface contour that has no adjacent surface as [Open edge].

<Operation>



Select the [Correct Surface Contour] command.

Specify parameters in the command sheet.

(page1) Open edge: ON

(page2) Loop: ON

Select all surfaces and execute the center mouse button to execute the command.

After you confirm the result, create a surface with creating surface command.



Confirm the status of adjacent surfaces with the same operation above.

You do not need to modify all of detected parts. Confirm the detected result and determine the modification is necessary.

How to confirm gap between adjacent surfaces

Command sheet [Gap]: ON

>> To modify the detected gap, expand surface by [Expand Surface] or [Untrim Surface] command, and then trim the surface again.

How to confirm overlapped part between adjacent surfaces

Command sheet [overlapped part]: ON

>> To modify the detected overlapped part, trim the surface with [Trim Surface] command.

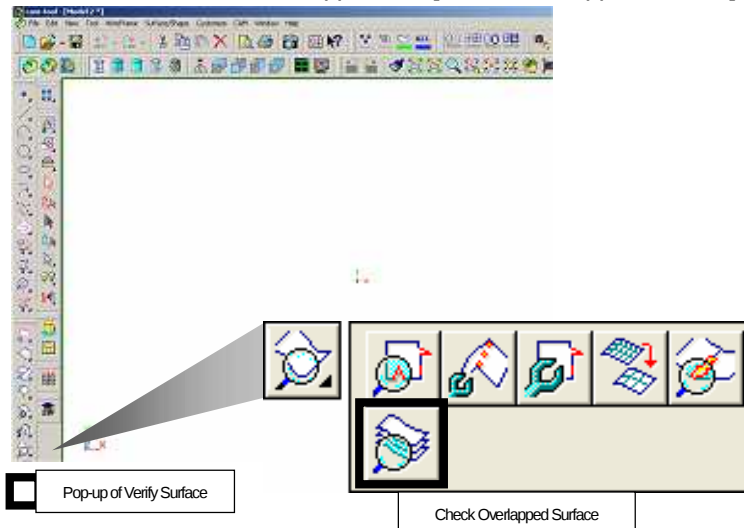
How to confirm [No continuous] part between adjacent surfaces

Command sheet [No continuous]: ON

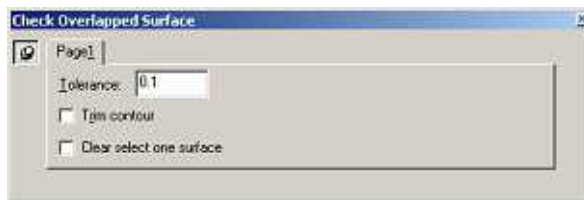
>> To modify the detected no continuous part, place a fillet surface with [Fillet Surface] command.

How to detect the overlapped surface <Check Overlapped Surface>

Confirm if surfaces are overlapped with [Check Overlapped Surface].



<Command sheet>

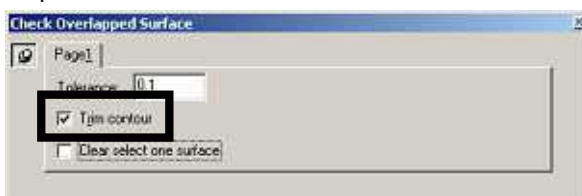


Tolerance: Detect overlapped surfaces which the distance between surfaces is smaller than the specified value.

Trim contour: ON Detect overlapped surfaces by trimmed contour.
OFF Detect overlapped surfaces by original contour.

Clear select one surface: ON Selected status of only one of the detected overlapped surface will be canceled.
OFF Selected status of all of the detected overlapped surfaces will be canceled.

<Operation>



Select the [Check Overlapped Surface] command.

Specify parameters in the command sheet.

Trim contour: ON

Select target surfaces and press the center mouse button to execute the command.

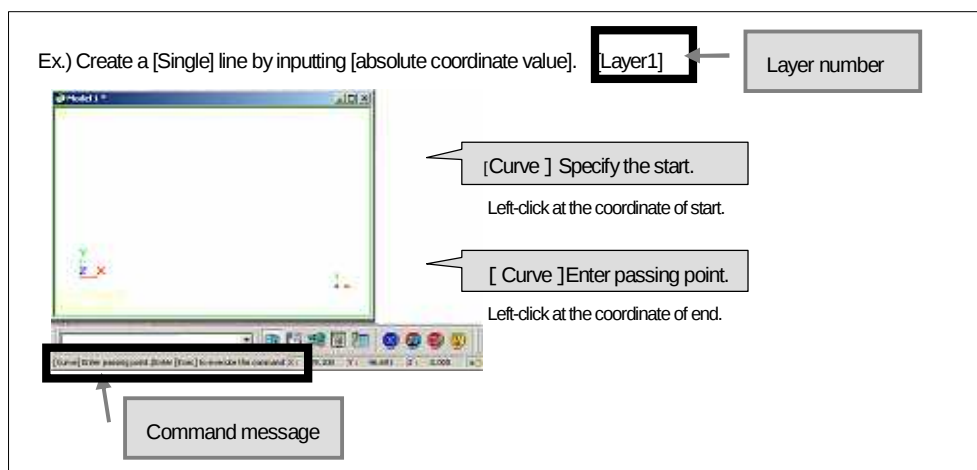
Chapter 11

Develop to cutting shape / Analyze shape

Open the model file below.

CD-ROM/Intensive Course/CADmodel3.gmd

- If [Layer number] is written in the description, change the active layer number.
- Follow the [command message] displayed at the lower left of the screen, for your operation.



Operating flow of developing product shape to cutting shape

If the shape you imported by data translation is [product shape], develop it to [cutting shape]. Operating flow of developing product shape to cutting shape is described.

1. Divide shape into cavity and core

The shape is automatically divided into cavity shape and core shape.



2. Check connected/unconnected surfaces

Check if the surfaces are connected or not to the divided cavity/core shape.



3. Move unconnected surface

Surfaces, which are not connected to the each shape are moved to the right side.



4. Create parting surface

Create parting surface with creating surface command.



5. Confirm fillet radius

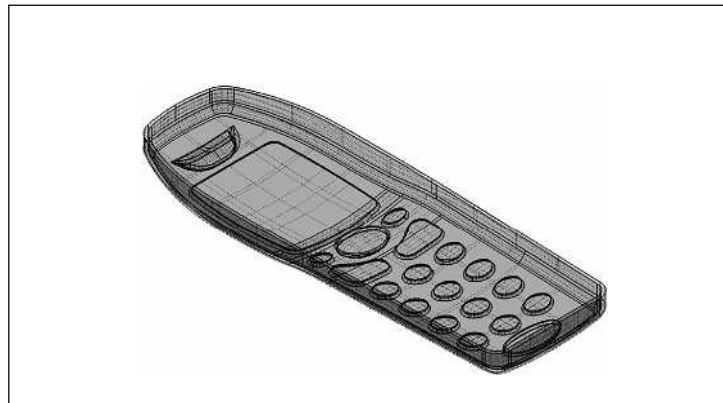
Confirm fillet surface radius of the shape.



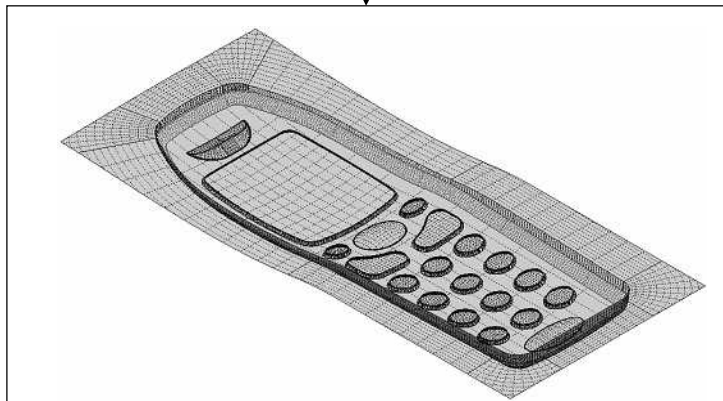
6. Confirm draft info of shape

Confirm whether the surface has proper angle that is needed for opening the model shape.

Product shape



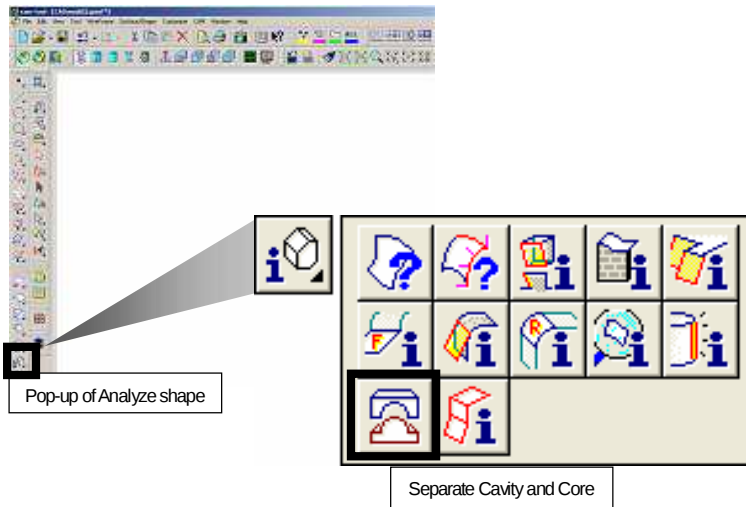
Cutting shape



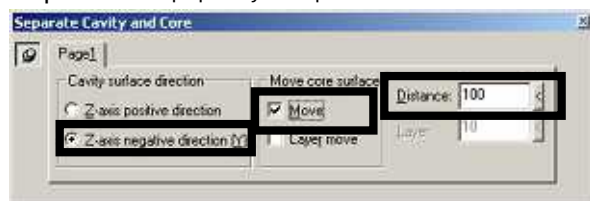
How to divide shape into cavity shape and core shape

<Separate Cavity and Core>

A model is divided into [Cavity shape] and [Core shape], and core shape is selected. And the core shape can be moved in the active layer or moved to the specified layer simultaneously. [Layer1]



<Operation> Pop-up Analyze shape



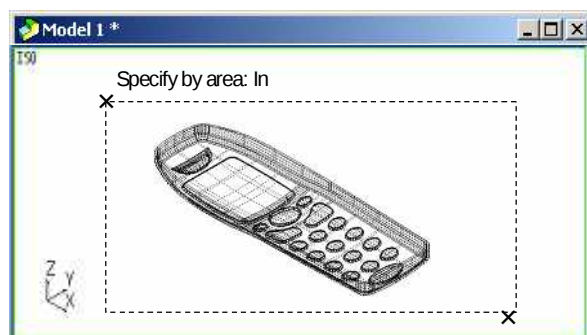
Select the [Separate Cavity and Core] command.

Specify parameters in the command sheet.

Cavity surface direction: Z-axis negative direction

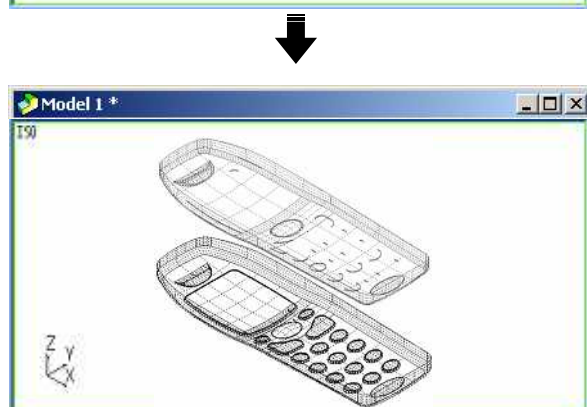
Move core surface: Move

Distance: 100



[Separate Cavity and Core] Pick surfaces.

Specify surfaces by using [Specify by area: In] and press the center mouse button to determine the specifying.



The core shape is moved and selected.

* Information of divided surfaces is displayed on the output window.

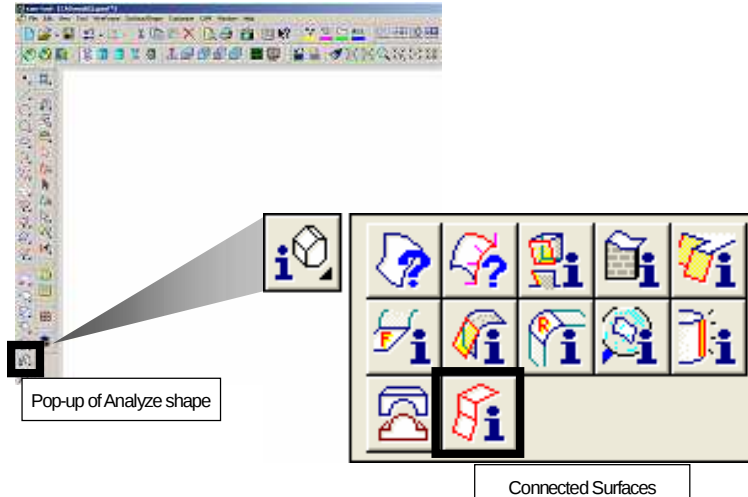
Execute the [Select (None)], and confirm the divided result.

*In this model file, the core shape is displayed in [white], and

the cavity shape is displayed in [light blue].

How to check connected surfaces <Connected Surfaces>

After you execute the [Separate Cavity and Core], check if there are surfaces that are not connected to the divided cavity/core shape. Detected connected surfaces or unconnected surfaces can be moved in the active layer or moved to the specified layer.



<Operation>



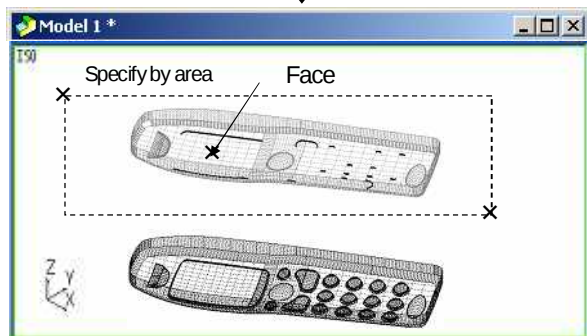
Select the [Connected Surfaces] command.

Specify parameters in the command sheet.

Clear: Connected

Unconnected surfaces: Move

Distance: -100



[Connected Surfaces] Pick a surface.

Specify the surfaces using [Specify by area: In], and press the center mouse button to determine the specifying.

[Connected Surfaces] Pick one of the selected surfaces for the start.

Click the surface as the base surface for detecting connected surfaces. (Face)

* Be sure to click a surface that can be clearly recognized as cavity or core shape.



Unconnected surfaces are selected or moved.

*Detecting information is displayed on the output window.

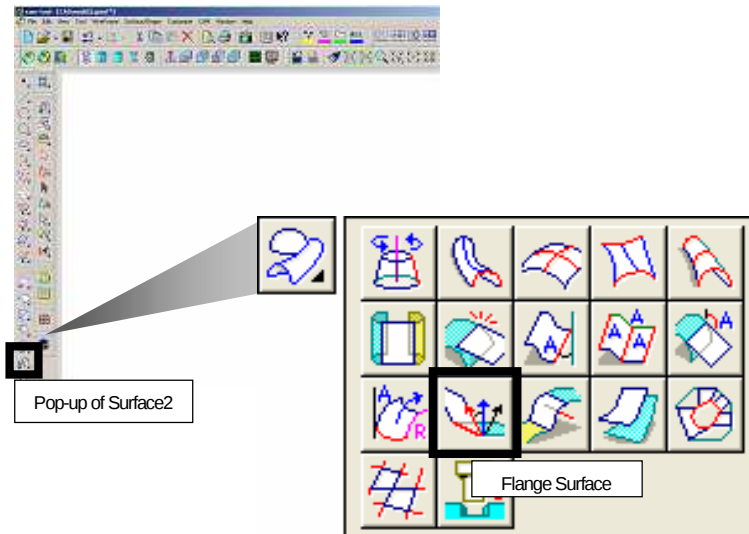
Execute the [Select (None)] to confirm the result.

Execute the [Connected Surfaces] for the cavity shape as the same operation. Distance: 100

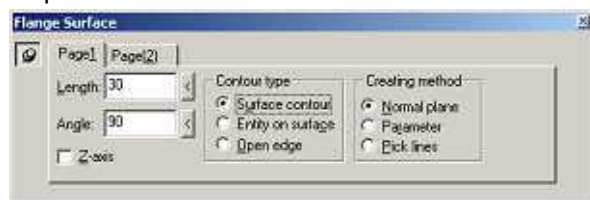
How to create parting surface (1) <Flange Surface>

Create parting surface. There are various commands to create a parting surface.

As an example, create [Flange surface] with [surface contour]. [Layer2]



<Operation>

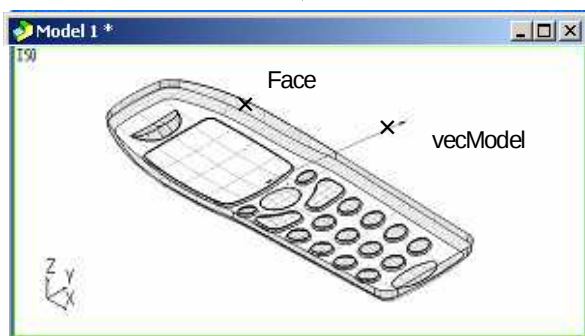


Select the [Flange Surface] command.
Specify parameters in the command sheet.

Surface contour: ON

Length: 30

Angle: 90

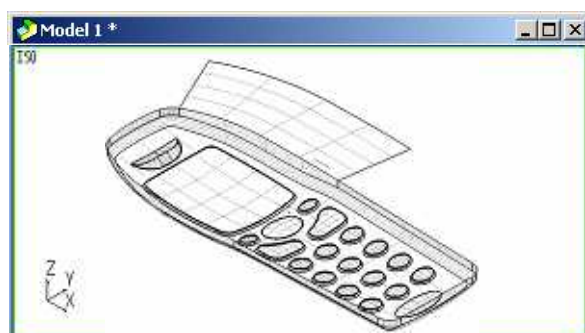


[Flange Surface] Pick an edge.

Click the surface contour. (Face)

[Flange Surface] Specify correcting direction.

Click the direction to create a flange surface.
(vecModel)



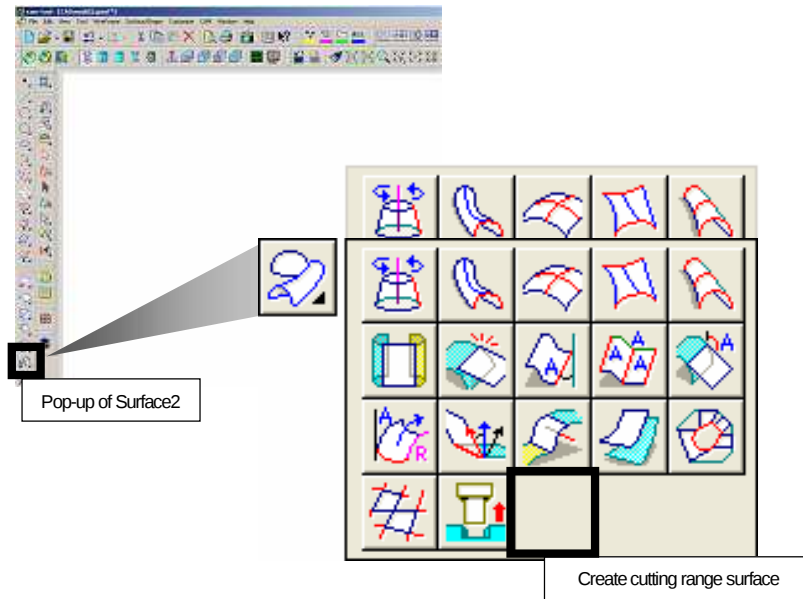
Press the center button to execute the command.

* Create other parting surfaces all around the shape with the same operation. (Completed shape = [Layer3])

How to create parting surface (2) <Create cutting range surface>

Create parting surface. There are various commands to create a parting surface.

As an example, create [Cutting Range surface] with [surface contour]. [Layer2]



<Operation>



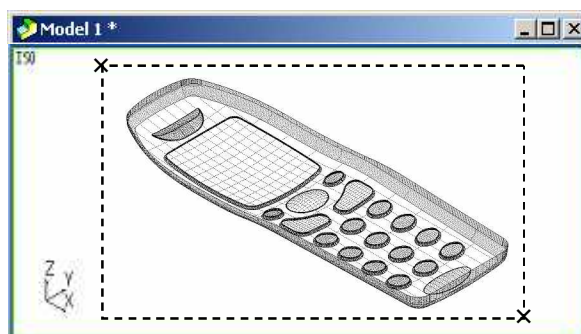
Select the [Create Cutting Range Surface] command.
Specify parameters in the command sheet.

Create tangent surface: OFF

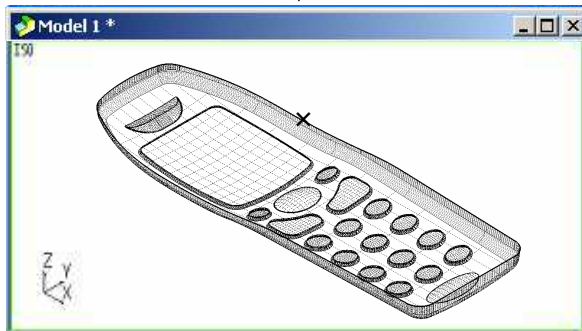
Create horizontal surface: ON

Offset amount: 20

Layer move: OFF

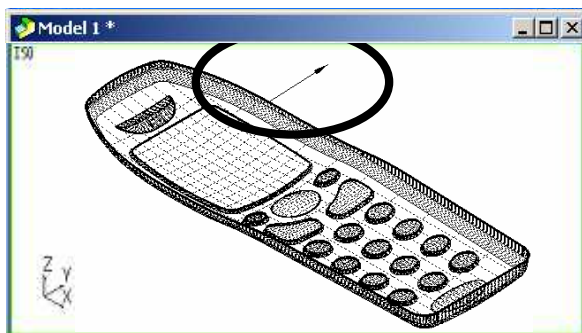


Specify the shape by [Specify by area: In] and press the center mouse button to determine the specifying.



[Create Cutting Range Surface] Pick a temporary entity.

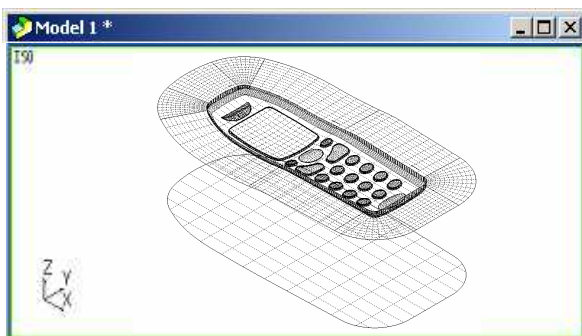
Specify the temporary entity.



[Create Cutting Range Surface] Specify a taper direction..

Click the direction to create surfaces.

* Click the arrow to flip the direction.

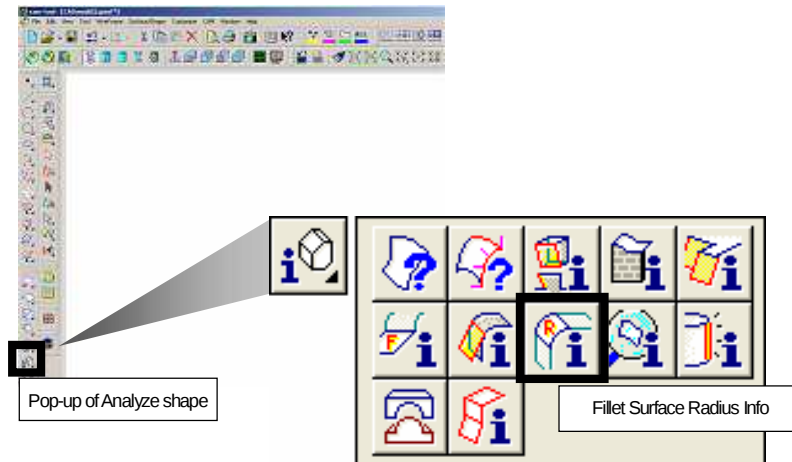


Press the center button to execute the command.

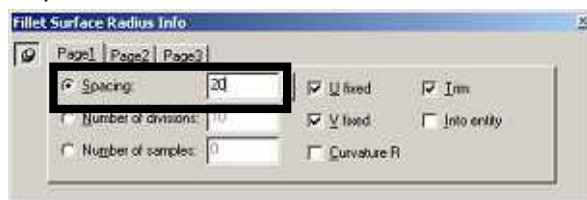
How to confirm fillet surface radius <Fillet Surface Radius Info >

Various information of the shape can be confirmed with [Analyze Shape] commands.

Confirm the fillet radius information of the shape, and use the detected [Minimum radius value] for CAM operation including selecting tool. [Layer5]



<Operation>



Select the [Fillet Surface Radius Info] command.

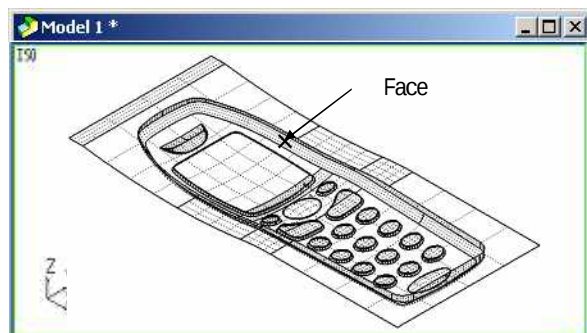
Specify parameters in the command sheet.

(page1) Spacing: 20



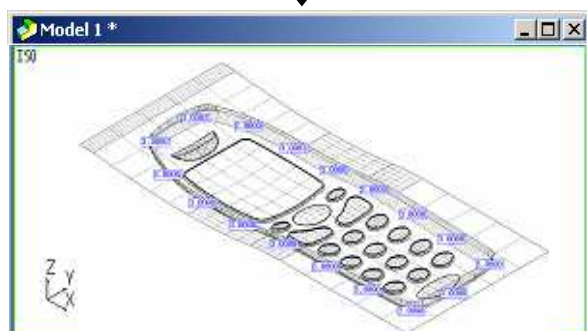
(page2) Concave R only: ON

Min R: ON



[Fillet Surface Radius Info] Pick a surface.

Press the center button to execute the command.



Radius of fillet surfaces are displayed on the information window.

* Interval between the displayed radius can be changed by specifying value for [Spacing] in the command sheet.



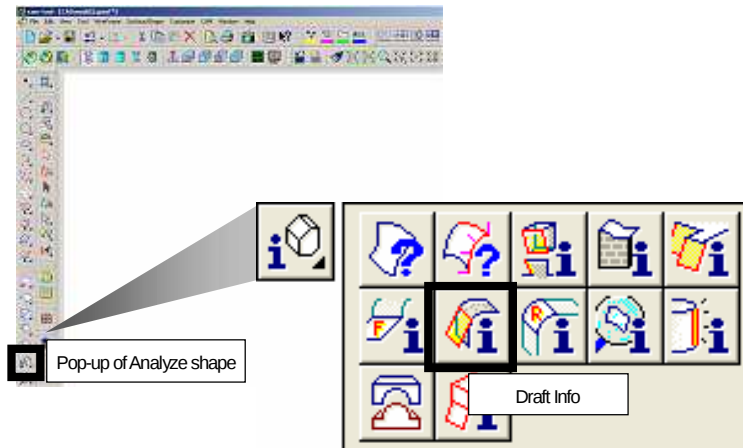
Curvature Radius Shading

[Menu bar] >> [View] >> [View type] >> [Curvature Radius Shading]

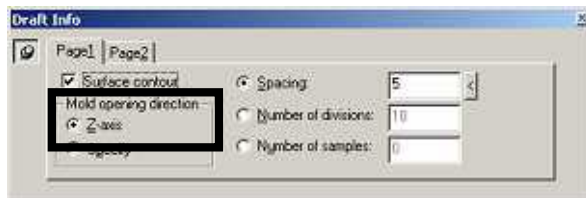
Range of curvature of the shape can be confirmed by color.

How to confirm section angle of the shape <Draft Info>

The angle of section of the shape is displayed. Confirm that the surface has the proper angle, which is required when the mold shape is pulled out. [Layer6]



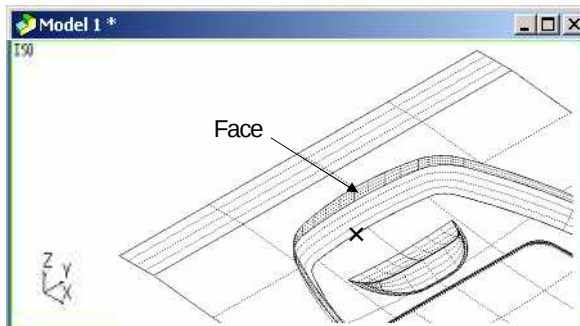
<Operation>



Select the [Draft Info] command.

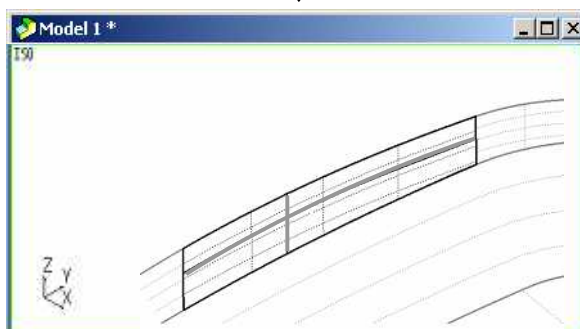
Specify parameters in the command sheet.

Mold opening direction: Z-axis

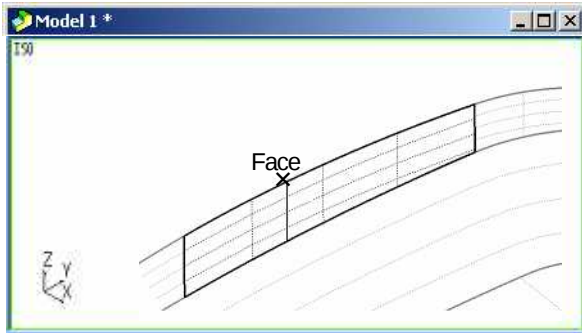


[Draft Info] Pick a surface.

Click the entity and press the center mouse button to determine the specifying. (Face)

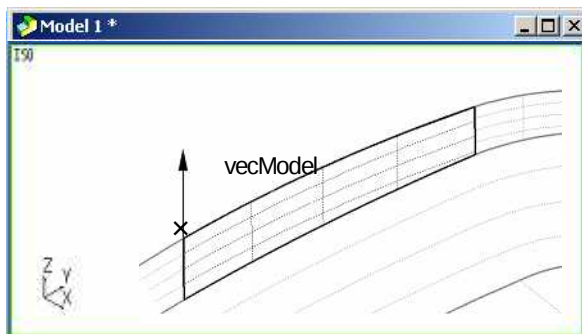


Parameter lines are displayed.



[Draft Info] Specify output location of parameter line.

Specify the upper surface contour. (Face)

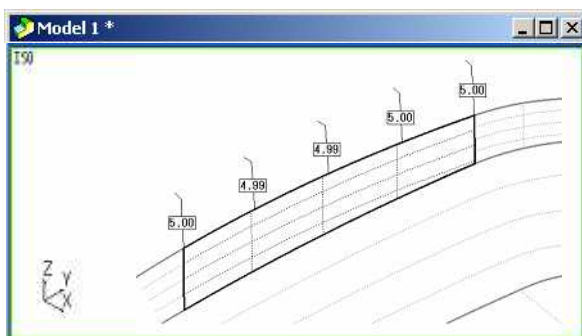


[Draft Info] Set the reference vector.

Specify the direction to pull out the mold shape.
(vecModel)

* Click the arrow to flip the direction.

Point the arrow at the top this time.



Press the center button to execute the command.



Angle shading

[Menu bar] >> [View] >> [View type] >> [Angle Shading]

Flat part, wall, under-cut area, or range of the specified angle can be confirmed by color.