



October 31, 2002

# INTAD

# Training Materials

designed for

# PTC's

# Pro/DESKTOP



## Book 1

## Basic to Intermediate

For use with PTC's Pro/DESKTOP  
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PTC  
Shaping Innovation

## Training Program Concepts Outline/Checklist

Design & Technology in Schools Program



- Sketches •  Generating
  - Modifying
  - Constraints
- Transform and  Duplicate functions
- Extrude Profile •  Taper
  - Thin
- Round Edges •  Constants &  Variable radius
- Chamfer Edges •  Equal and  unequal setback
- Shell
- Work-planes •  Selecting
  - Generating
  - Offset
  - Angled
- Redefining Features
- Revolve Profile
- Sweep Profile along a Profile
- Holes
- Draft Faces
- Lofting
- Sweep Profile along a Helix
- Feature Modification •  Deform Face
  - Pattern
  - Mirror
  - Scale
- Assembly •  Working with Components
  - Transform
  - Duplicate
  - Use
- Drawings •  Generate drawing
  - Add views
  - Section Views
  - Dimensioning
  - Adding Text
- Album •  Rendering
  - Adding Textures
  - Adding background images
  - Importing backgrounds and textures
  - Studio functions
  - Printing and  print resolution
- Saving files - the importance of saving associated files

**Pro/DESKTOP** is a 3d/2d Modelling Environment.

It enhances a different way of thinking about Graphics problem, but still the student must analyse the drawing problem in order to efficiently & effectively solve the problem.

Pro/DESKTOP works on:

1. Faces being selected
2. Workplanes either New or already in use
3. New Sketch

A Workplane may be used for many Sketches, BUT a Sketch can be used for only one operation.

These notes have been used in a class situation and corrected as needed, BUT there may still be small errors. E-mail [nevobrien@uq.net.au](mailto:nevobrien@uq.net.au) for mistakes.

There is a copy of these notes on the INTAD web site under Pro/DESKTOP  
<http://www.intad.asn.au>

Many of the competencies are covered by these brief notes.

The notes are “cooking recipes” & seem to be most suitable for middle secondary students.

INTAD is looking for other “lessons” to increase our knowledge store. We need to share.

Several web sites to visit:

<http://www.ptc.com>

<http://www.prodesktop.net>

<http://www.tep.org.uk>

If you find any other sites, inform INTAD.

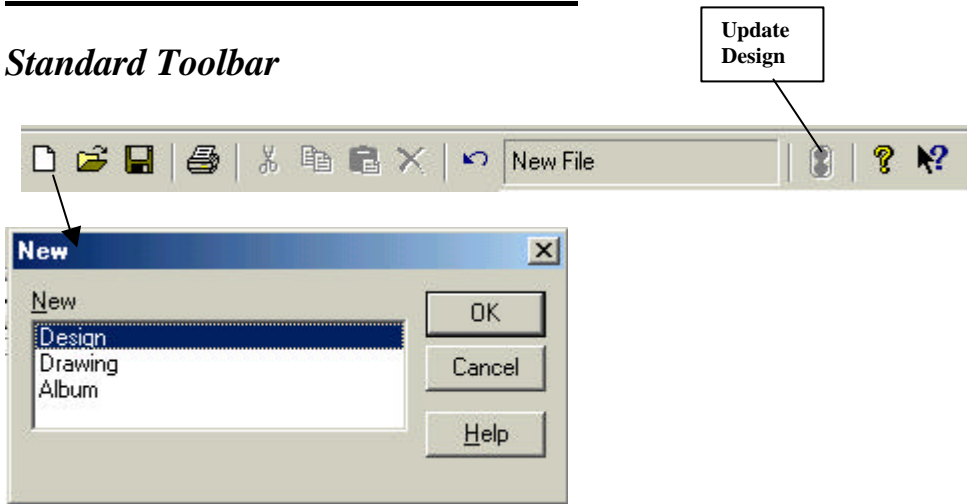
Pro/DESKTOP has tutorials in its help system plus a Getting Started Guide (68 pages) & a User Guide (362 pages) usually at:

**C:\Program Files\PTC\ProDESKTOP 2000i2\Program\Help**

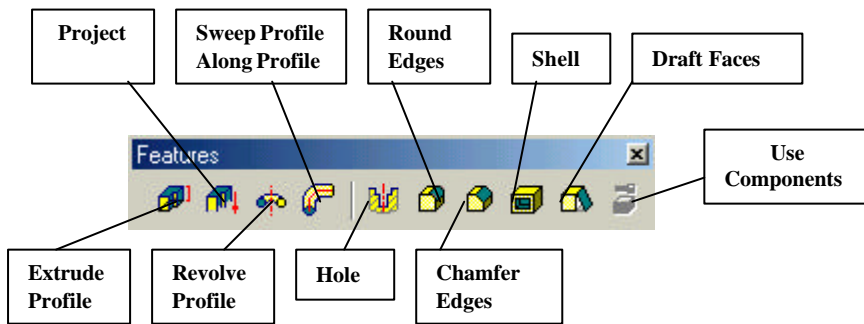
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# Pro/DESKTOP 2000i<sup>2</sup> Toolbars

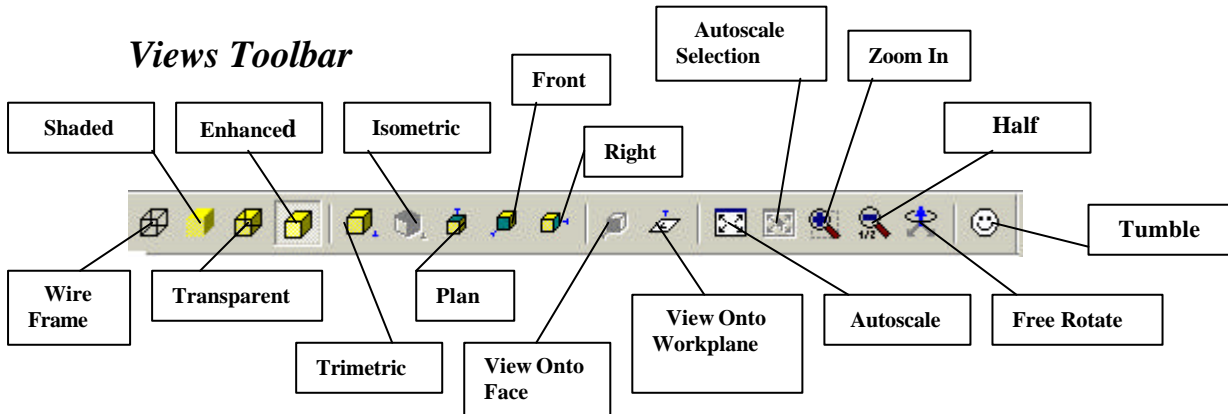
## Standard Toolbar



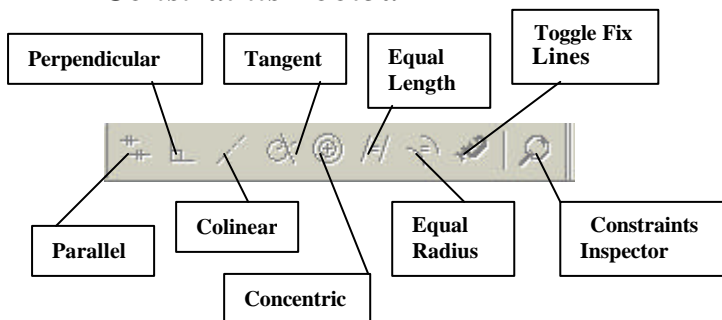
## Features Toolbar

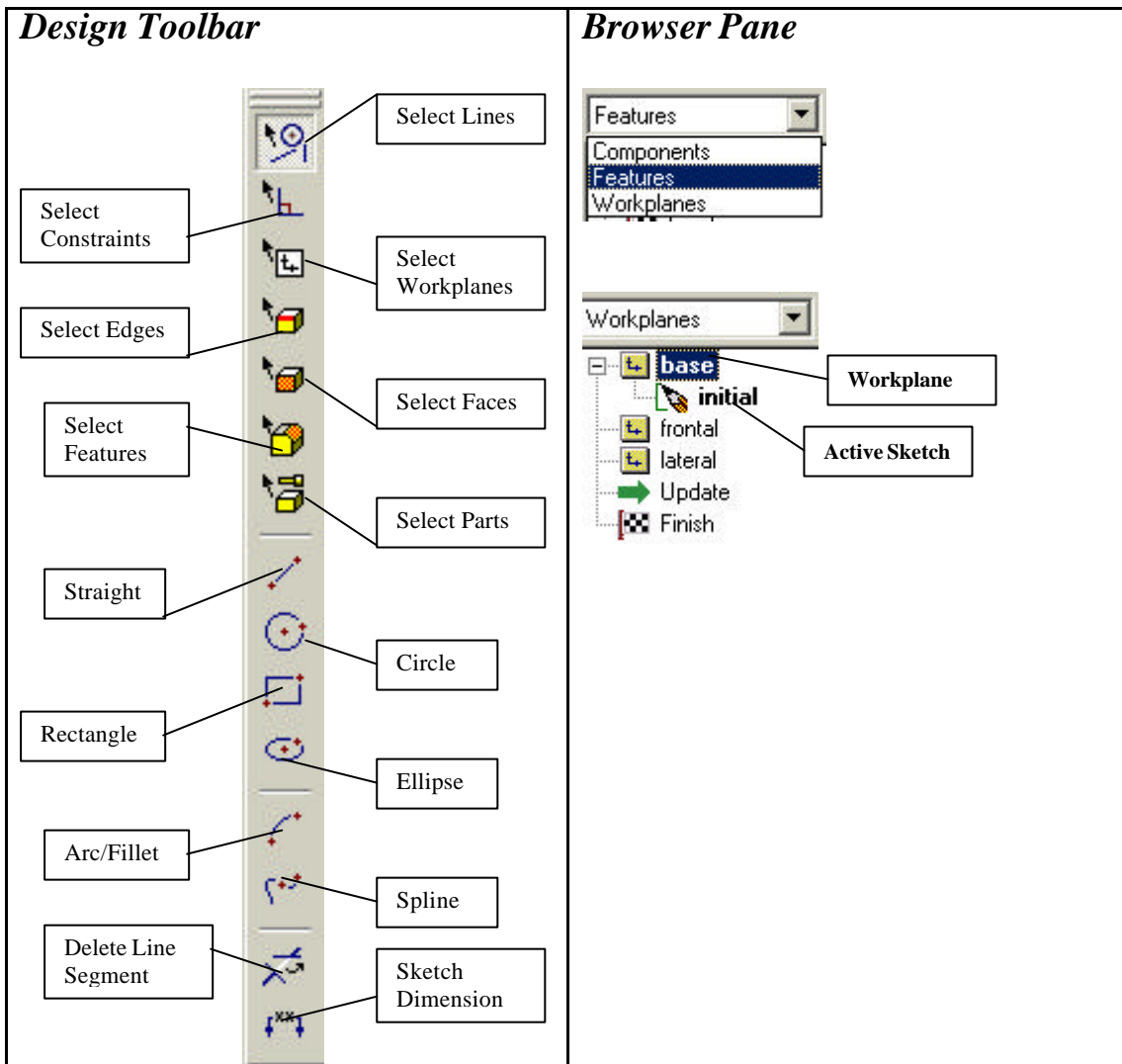


## Views Toolbar



## Constraints Toolbar




















































































## Menus






File	Select	Assembly
<ul style="list-style-type: none"> <li>New... Ctrl+N</li> <li>Open... Ctrl+O</li> <li>Close</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Save Ctrl+S</li> <li>Save Copy As...</li> <li>Import</li> <li>Export</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Close All</li> <li>Save Session</li> <li>Save Session As...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Print Setup...</li> <li>Print... Ctrl+P</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Send To</li> <li>Properties</li> </ul> <hr/> <ul style="list-style-type: none"> <li>1 Coat Hanger</li> <li>2 C:\My Documents\...\TShirt</li> <li>3 C:\My Documents\...\Gun</li> <li>4 C:\My Documents\...\Toy album</li> <li>5 C:\My Documents\...\Torch 3</li> <li>6 cog assembly</li> <li>7 C:\My Documents\...\shaft</li> <li>8 C:\My Documents\...\track</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Exit</li> </ul>	<ul style="list-style-type: none"> <li>Lines L</li> <li>Constraints N</li> <li>Workplanes W</li> <li>Edges E</li> <li>Faces F</li> <li>Features A</li> <li>Parts P</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Edges of Selected Faces</li> <li>Faces of Selected Edges</li> <li>Parent of Selected Part</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Add Connected Lines Ctrl+E</li> <li>Add Loop Edges</li> <li>Add Tangent Edges</li> <li>Add Tangent Faces</li> <li>Add Like Components</li> <li>Add Completely Dependent Features</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Remove Inactive Objects</li> <li>Synchronize Browser Shift+B</li> </ul>	<ul style="list-style-type: none"> <li>Add Component...</li> <li>Replace Component...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>New Part in Context</li> <li>Open Part in Context</li> <li>New Driven Design</li> <li>Open Layout</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Align</li> <li>Mate</li> <li>Offset...</li> <li>Center Axes</li> <li>Orient...</li> <li>Fix Components</li> <li>Hide Components</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Product Structure...</li> <li>Set Component Color...</li> </ul>
<h3>Edit</h3> <ul style="list-style-type: none"> <li>Can't Undo Ctrl+Z</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Cut Ctrl+X</li> <li>Copy Ctrl+C</li> <li>Paste Ctrl+V</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Delete Del</li> <li>Select All Ctrl+A</li> <li>Invert Selection</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Shift Point-to-Point</li> <li>Transform...</li> <li>Duplicate... Ctrl+D</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Properties Alt+Enter</li> <li>Pick Up Properties... Ctrl+Shift+C</li> <li>Apply Properties Ctrl+Shift+V</li> </ul>	<h3>Line</h3> <ul style="list-style-type: none"> <li>Straight S</li> <li>Circle C</li> <li>Rectangle R</li> <li>Ellipse I</li> <li>Arc T</li> <li>Spline B</li> <li>Delete Segments D</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Mirror...</li> <li>Offset Chain...</li> <li>Project</li> <li>Convert To Straight</li> <li>Add Text Outline...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Toggle Construction Ctrl+G</li> <li>Toggle Sketch Filled Ctrl+Shift+F</li> <li>Toggle Sketch Rigid Ctrl+Shift+R</li> </ul>	<h3>Workplane</h3> <ul style="list-style-type: none"> <li>New Workplane... Ctrl+L</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Transform Axes...</li> <li>Reposition Axes</li> <li>Reverse</li> <li>Redefine...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>New Sketch... Ctrl+K</li> <li>Activate Sketch</li> <li>Show All Sketches</li> <li>Hide Other Sketches Ctrl+H</li> <li>Delete Empty Sketches</li> </ul>
<h3>View</h3> <ul style="list-style-type: none"> <li>Autoscale Shift+A</li> <li>Autoscale Selection Shift+S</li> <li>Half Scale Shift+H</li> <li>Zoom In Shift+Z</li> <li>Manipulate Space</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Wire Frame F9</li> <li>Shaded F10</li> <li>Transparent F11</li> <li>Enhanced F12</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Go To</li> <li>Rotate</li> <li>Tumble Shift+U</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Toolbars</li> <li>Dialog Bar</li> <li>Status Bar</li> </ul>	<h3>Constraint</h3> <ul style="list-style-type: none"> <li>Dimension Z</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Parallel</li> <li>Perpendicular</li> <li>Collinear</li> <li>Tangent</li> <li>Concentric</li> <li>Equal Length</li> <li>Equal Radius</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Toggle Fixed Ctrl+F</li> <li>Toggle Reference Ctrl+R</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Inspector...</li> </ul>	<h3>Tools</h3> <ul style="list-style-type: none"> <li>Components Browser Shift+C</li> <li>Features Browser Shift+E</li> <li>Workplanes Browser Shift+K</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Variables... Alt+1</li> <li>Design Rules... Alt+2</li> <li>Configurations... Alt+3</li> <li>New Measurement</li> <li>Palette...</li> <li>Animation...</li> <li>Solver...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>New Drawing...</li> <li>Options...</li> </ul>
<h3>Help</h3> <ul style="list-style-type: none"> <li>Contents and Index F1</li> <li>What's This? Shift+F1</li> <li>Tutorials</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Tip of the Day...</li> <li>PTC on the Web</li> </ul> <hr/> <ul style="list-style-type: none"> <li>About Pro/DESKTOP...</li> </ul>	<h3>Feature</h3> <ul style="list-style-type: none"> <li>Extrude Profile...</li> <li>Project Profile...</li> <li>Revolve Profile...</li> <li>Sweep Profile</li> <li>Loft Through Profiles...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Round Edges...</li> <li>Chamfer Edges...</li> <li>Draft Faces...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Modify Solids...</li> <li>Shell Solids...</li> <li>Insert Holes...</li> <li>Use Component...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Pattern...</li> <li>Unpattern</li> <li>Mirror...</li> <li>Redefine...</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Update Design F5</li> <li>Update and Propagate Ctrl+F5</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Restore All</li> </ul>	<h3>Windows</h3> <ul style="list-style-type: none"> <li>New Window</li> <li>Set Color...</li> <li>Split</li> <li>Update Links</li> </ul> <hr/> <ul style="list-style-type: none"> <li>Cascade</li> <li>Tile</li> <li>Arrange Minimized</li> </ul> <hr/> <ul style="list-style-type: none"> <li>1 Design1 *</li> <li>Windows...</li> </ul>

## Pro/DESKTOP 2000i<sup>2</sup> Keyboard Commands

Sketch Commands			
Menu	Command	Button	Hot Keys
<b>Line</b>	Straight		S
	Circle		C
	Rectangle		R
	Ellipse		I
	Arc		T
	Spline		B
	Delete Segment		D
	Toggle Construction		Ctrl + G
	Toggle Sketch Filled		Ctrl + Shift + F
	Toggle Sketch Rigid		Ctrl + Shift + R
<b>Constraint</b>	Dimension		Z
	Parallel		
	Perpendicular		
	Collinear		
	Tangent		
	Concentric		
	Equal Length		
	Equal Radius		
	Toggle Fixed		Ctrl + F
	Toggle Reference		Ctrl + R
	Inspector		
<b>Select</b>	Lines		L
	Constraints		N
Design Commands			
<b>Feature</b>	Extrude Profile		
	Project Profile		
	Revolve Profile		
	Sweep > Sketch Path		
	Insert Holes		
	Round Edges		
	Chamfer Edges		
	Shell Solids		
	Draft Faces		
	Use Component		
	Update Design		F5
	Update & Propagate		Ctrl + F5
<b>Tools</b>	Components Browser		Shift + C
	Features Browser		Shift + E


























	Workplanes Browser		Shift + K
<b>Workplane</b>	New Workplane		Ctrl + L
	New Sketch		Ctrl + K
	Hide Other Sketches		Ctrl + H
<b>Select</b>	Workplanes		W
	Edges		E
	Faces		F
	Features		A
	Parts		P
	Add Connected Lines		Ctrl + E
	Synchronize Browser		Shift + B
<b>Drawing Commands</b>			
<b>Select</b>	Annotations		A
	Features		F
	Views		V
	Add Connected Lines		Ctrl + E
	Synchronize Browser		Shift + B
<b>Center Line</b>	Common Plane		
	Common Axis		
	Mid-Plane		
	Pitch Circle		
	Center Points		
	Phantom Intersection		
<b>Dimension</b>	Linear		
	Angular		
	Diametric		
	Radial		
	Note	<b>A</b>	
	Geometric Tolerance		
	Datum Feature		
	Design Variables		
	Surface Finish		
	Part Reference Balloon		
	Insert Callout Note		Insert
	Move Callout L/R/Up/Down		Arrows
	Start New Line		Enter
	Shoulder Right		Ctrl + Right Arrow
	Shoulder Left		Ctrl + Left Arrow
<b>Drawing</b>	Update Views		F5
	New Sketch		Ctrl + K
	Hide Other Sketches		Ctrl + H
<b>Table</b>	Cycle Row Up		Shift + Up Arrow








	Cycle Row Down		Shift + Down Arrow
	Cycle Column Left		Shift + Left Arrow
	Cycle Column Right		Shift + Right Arrow
<b>General Commands</b>			
<b>File</b>	New		Ctrl + N
	Open		Ctrl + O
	Save		Ctrl + S
	Print		Ctrl + P
<b>Edit</b>	Undo		Ctrl + Z
	Cut		Ctrl + X
	Copy		Ctrl + C
	Paste		Ctrl + V
	Delete		Delete
	Select All		Ctrl + A
	Duplicate		Ctrl + D
	Properties		Alt + Enter
	Pick Up Properties		Ctrl + Shift + C
	Apply Properties		Ctrl + Shift + V
<b>View</b>	Auto Scale		Shift + A
	Autoscale Selection		Shift + S
	Half Scale		Shift + H
	Zoom In		Shift + Z
	Manipulate		Space
	Wire Frame		F9
	Shaded		F10
	Transparent		F11
	Enhanced		F12
<b>View&gt;Go To</b>	Isometric		Shift + I, Home
	Trimetric		Shift + T, End
	Plan		Shift + P
	Front Elevation		Shift + N
	Right Elevation		Shift + R
	Onto Face		Shift + F
	Onto Workplane		Shift + W
	Previous		Alt + Left Arrow
	Next		Alt + Right Arrow
<b>View&gt;Rotate</b>	Spin Left/Right		Left/Right Arrow
	Tilt Up/Down		Up/Down Arrow
	Turn Counter Clockwise		Page Up
	Turn Clockwise		Page Down
	Tumble		Shift + U

<b>Tools</b>	Variables		Alt + 1
	Design Rules		Alt + 2
	Configurations		Alt + 3
<b>Tools&gt;Macro</b>	Record New Marco		
	Stop Recording		
	Marcos		Alt + F8
	Resume Marco	I>	Alt + F9
	Visual Basic Editor		Alt + F11
<b>Special Control Features</b>			
<b>Keystroke</b>	<b>+ Selection</b>	<b>= Action</b>	
Shift	Modifying end point of a line	Extend or trim a line	
Shift	Straight line being created	Parallel to workplane axis	
Double Click	Object not on active plane	Activities its sketch	
Double Click	Selected object	Open properties dialog box	
Ctrl + Double Click	Object	Select feature & sync browser	
Esc	New Object	Open selection command	
Shift	Section or projected view	View will align to its principal view	
Shift	Add note	Note is placed without leader	
Shift	Radial dimension	Diameter dimension is placed	
Shift	Angular dimension	Major angle is dimensioned	
<b>Mouse Items</b>			
Manipulate Scene	 Spacebar (On/Off)	<ul style="list-style-type: none"> <li>- Hold down Left mouse button to Rotate in 3 dimensions</li> <li>- Shift + Hold down Left mouse button to drag/adjust position on screen</li> <li>- Shift + Ctrl + Hold down Left mouse button to Rotate in one plane</li> </ul>	
Zoom in/out	Scroll Wheel on mouse	Rotate Scroll Wheel to enlarge or reduce scale	

## Pointer Shapes

From Help File

	Create
	Split Bar Move
	Sketch Object Translate (Move Unchanged)
	Feature Translate (Move Unchanged)
	Viewpoint Pan, Split Window Bar Move
	Point Set
	Viewpoint Tumble
	Viewpoint Rotate
	Feature Rotate
	Length Constrain
	Separation Constrain
	Component Translate (Move Unchanged)
	Viewpoint Zoom
	Material Fill
	Segment Delete
	Select
	Drag Handle Move
	Drop Add (Palette)
	Drop Move (Palette)
	No Drop (Palette)
	Window Resize
	Circle Resize
	Arc Bend
	2-Point Arc Create
	Tangent Arc Create

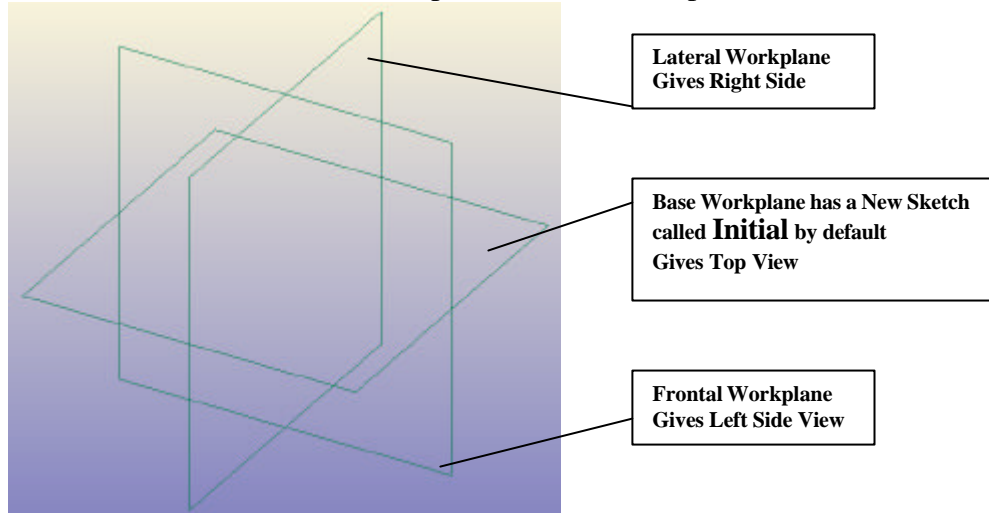
	Vertex Fillet
	Nonvertex Fillet
	Endpoint Move
	Spline Point Move
	Spline Extend
	Spline Point Add
	Spline Point Delete



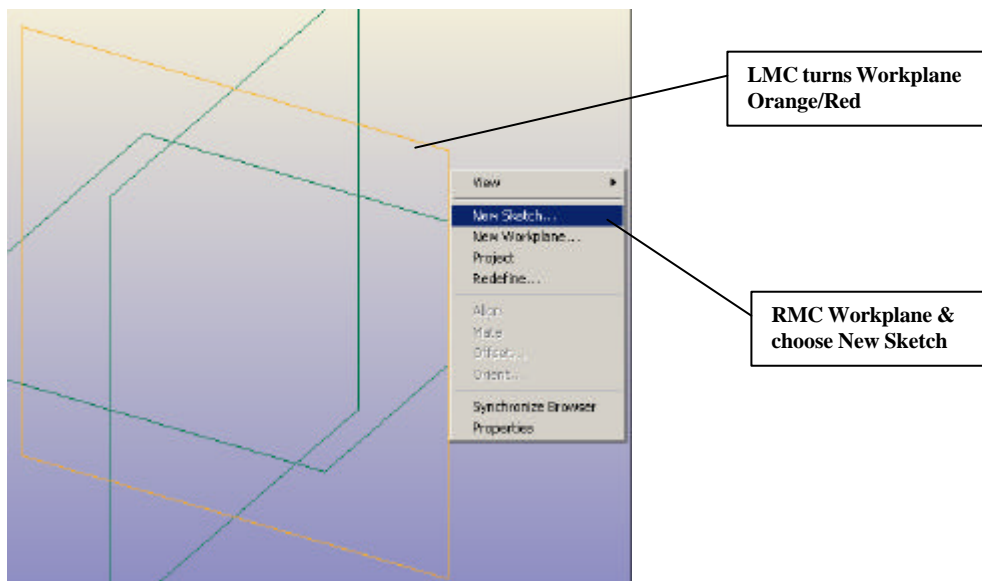
## Workplanes:

1. It is essential that students be introduced to the notion of Workplanes & New Sketches at a very early stage, if not the first stage.

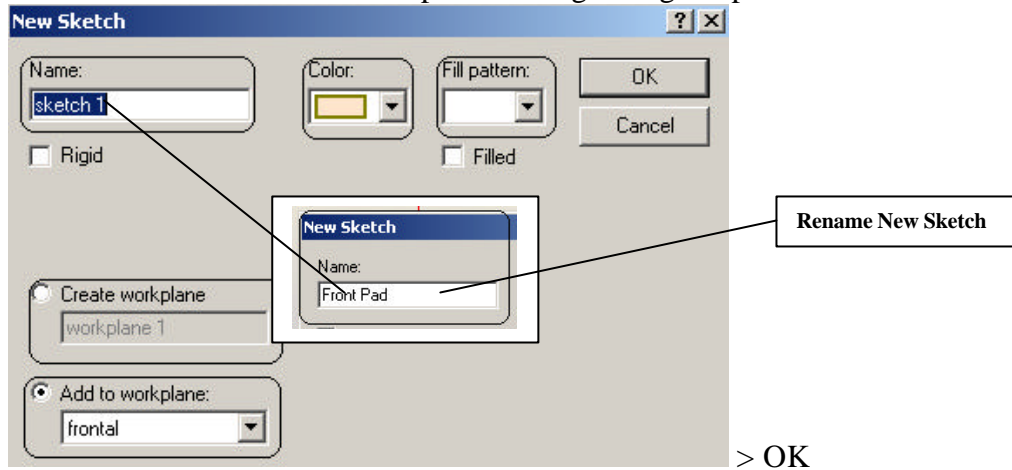
2. Use  to show the three Workplanes that are set-up in Pro/DESKTOP.



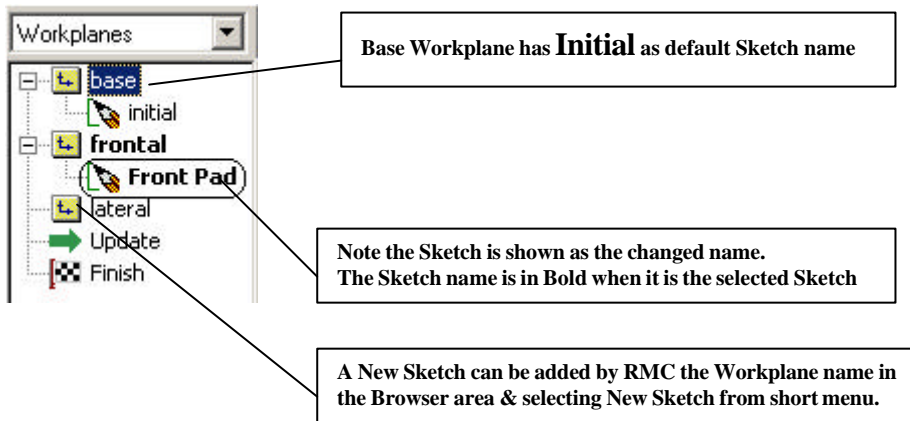
3. Only the Base Workplane has a New Sketch already on it. Left Mouse Click (LMC) any Workplane > it goes orange/red > Right Mouse Click (RMC) and choose New Sketch from menu OR goto Workplane > New Sketch



- Rename the New Sketch to a more meaningful name than Sketch 1 as this will allow the identification in a complex drawing & is good practice.



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Notes:

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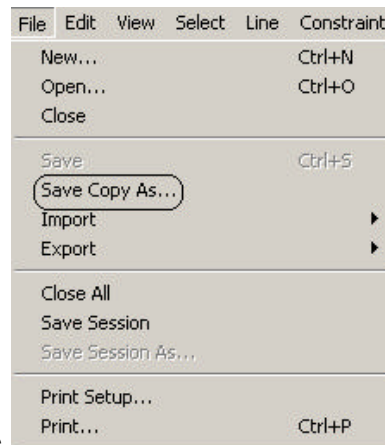


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## Save Files for Moving from Present location:



1. Use File **Save Copy As...**. This will allow the Saving of the file & associated files to a new location such as a floppy disk instead of the computer's C:/
2. Specify a Filename, Folder location and Drive letter as needed > OK
3. For Example: all files to do with an Assembly design would be saved to the new location. Pro/DESKTOP uses an internal referencing system to keep track of the associated files. This allows Pro/DESKTOP files to be kept to a small file size as the Assembly file has the required location/filename etc as part of its file information.
4. Similar for e-mailing the files.

### Notes:

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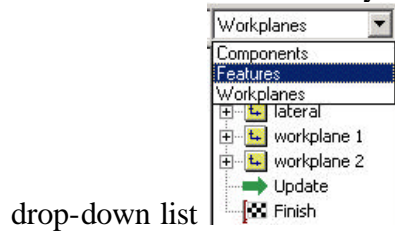
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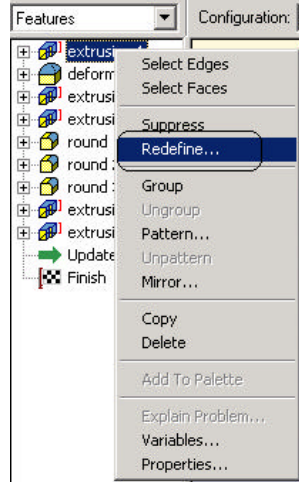
## Redefining Features:

1. Features can be redefined by selecting the Features in the Select Browser



drop-down list

2. Right Mouse Click (RMC) the Feature that requires changing > select



Redefine from menu

3. Change as needed > OK
4. Drawing might require an Update  after redefine is used.

Notes:

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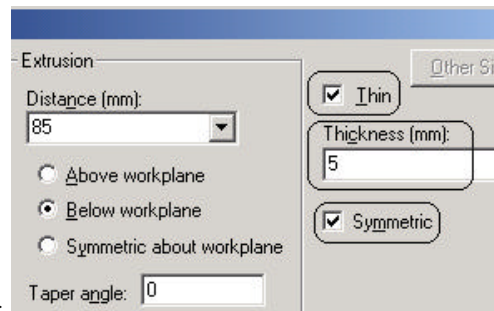
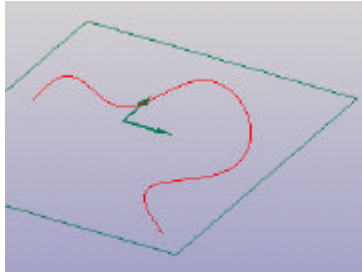
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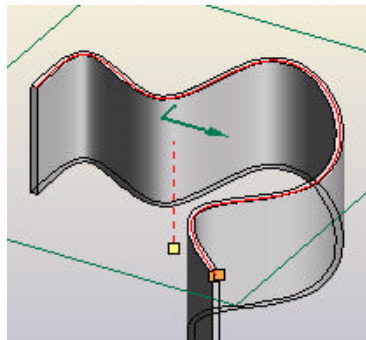
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## Thin:

1. For some reason the following profile was drawn using the Spline tool 

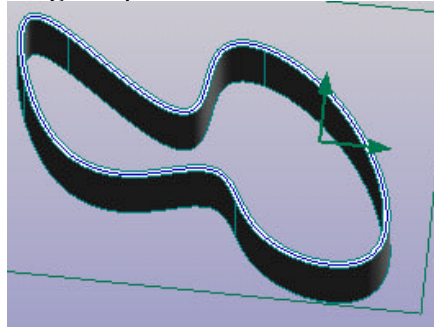


3. An Extrude Profile >



4. Gives this

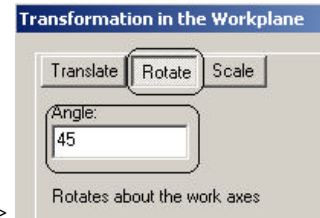
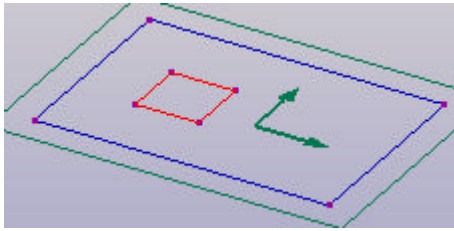
5. There are many irregular profiles that can be formed by using the Spline &



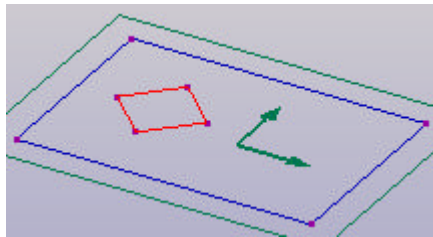
Thin command.

## Transform & Shell:

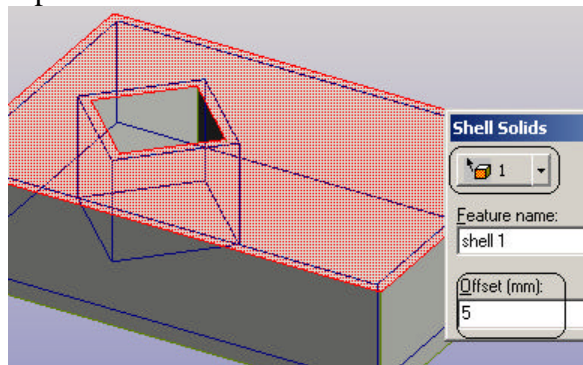
1. Draw a Rectangle & a Square (hold down Shift to obtain Square)



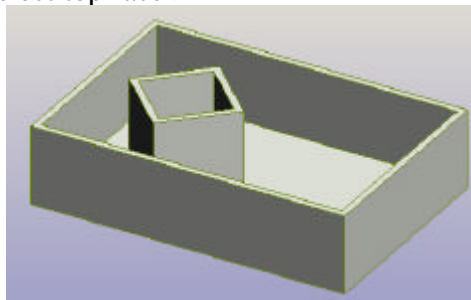
2. Right Mouse Click (RMC) Square > choose Transform > > OK



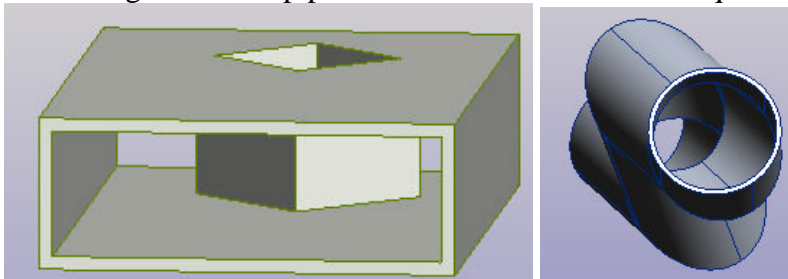
3. Rotated Square
4. Extrude Profile > Below Workplane > OK



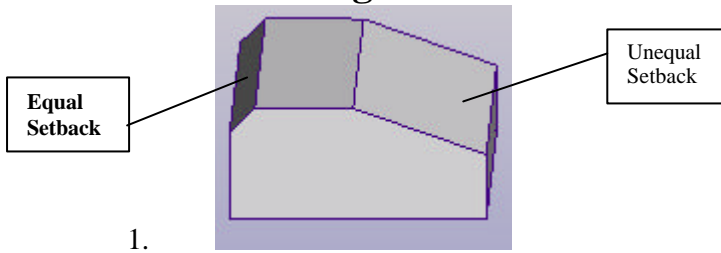
5. Shell  > select top face > > OK



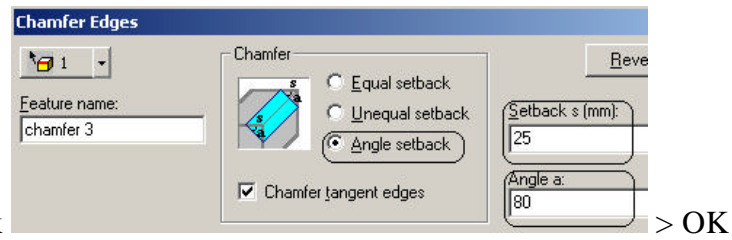
6. Shelled solid
7. Shelling to make a pipe or similar: select Faces as required & Shell as 5 & 6 above.



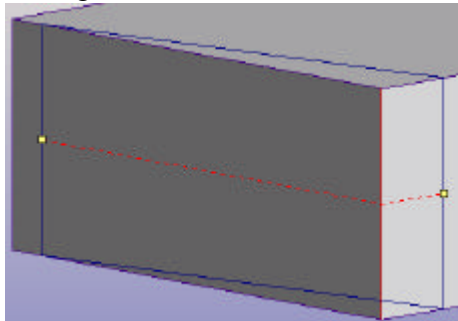
## Chamfer Edges:



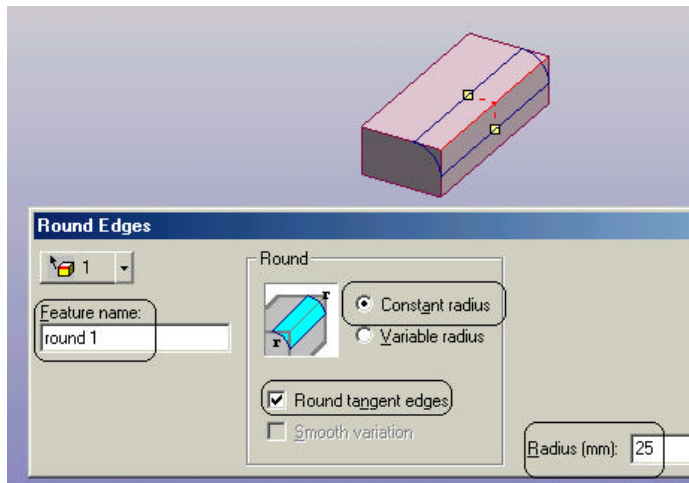
1.



2. Angle Setback

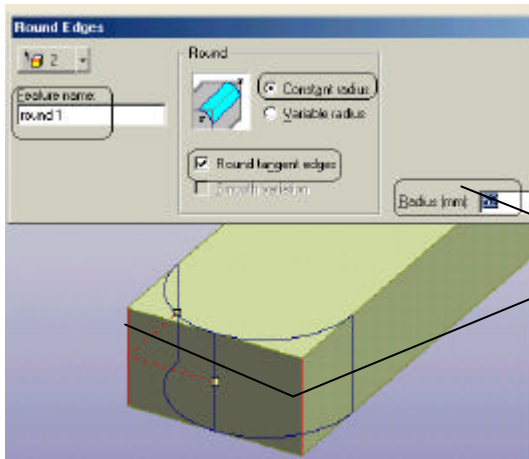


## Round Edges



1.

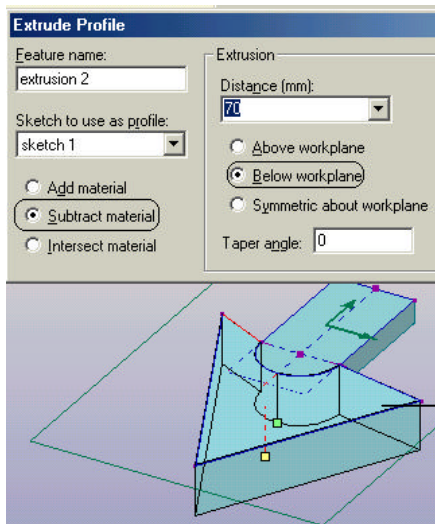
2. How to make an end into a semi-circle using Round Edges



Select Edges to be curved into Semi-circle.  
Use Round Edges icon and input the Radius  
OK

> OK

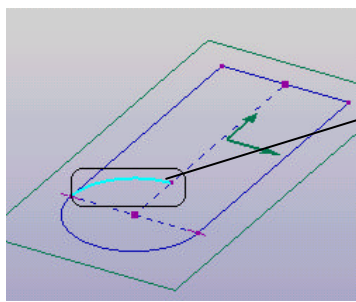
Semi-Circle using Cutting Plane



Notice that the Cutting Plane must be a closed figure

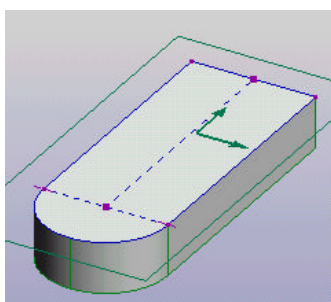
1. > OK

Semi-Circle using Normal Viewpoint



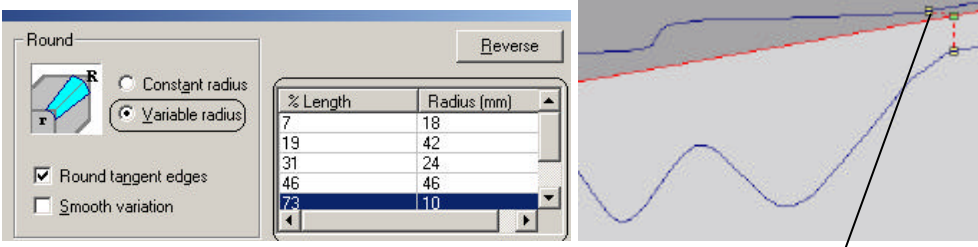
**Plan** for the use of Circle  
Remove any stray lines/curves

1.



2 Finished Object

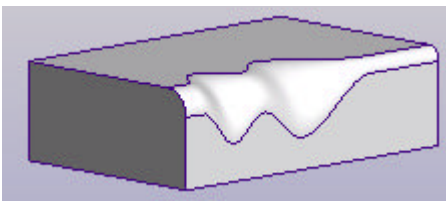
## Round Edges Variable Radius:

1. 



% Length	Radius (mm)
7	18
19	42
31	24
46	46
73	10


> OK

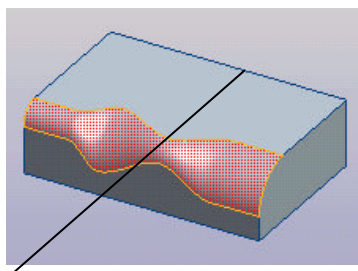
LMC & drag the Yellow/Green indicators to apply various Radii

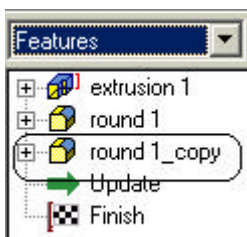
2. 

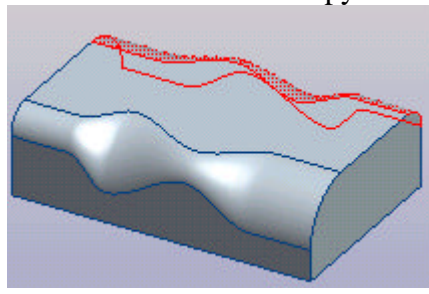
3. If there is a need to have another edge with the same edge treatment, use the following:

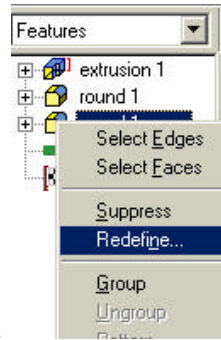
Select the Rounded Face  . Note: The normal Copy icon becomes active  . Click Copy.

Select the required Edge & use Paste icon  .

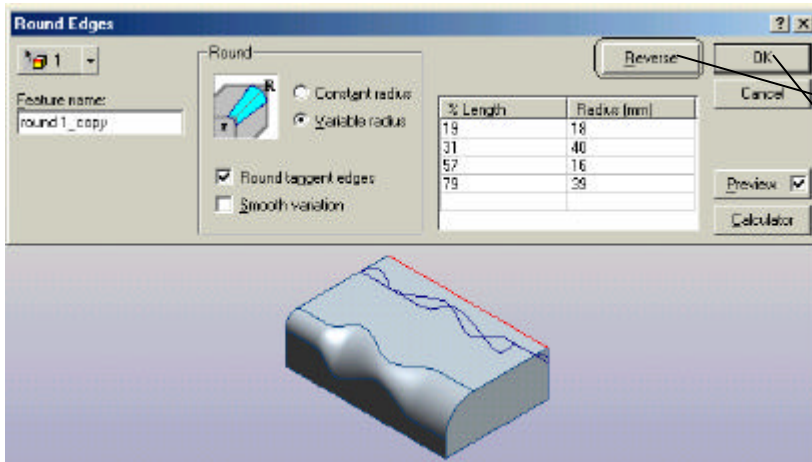


The Features browser shows that the copy has taken place  as well

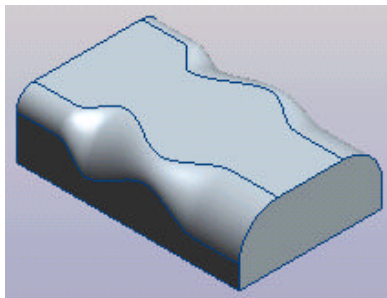
as the Design. 



RMC the Round 1\_copy & select Redefine.



LMC the Reverse button  
Then OK



Final Product.

Only one at a time can be completed in this manner.

Notes:

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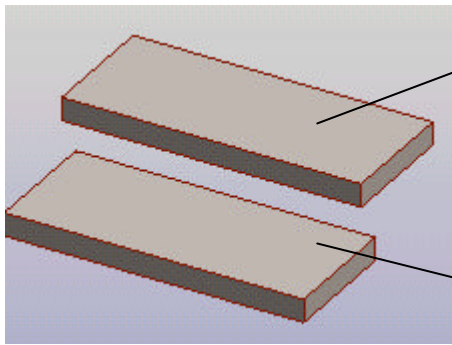
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## “Joining” two pieces of Steel/Timber:

Files for this activity are at [www.intad.asn.au](http://www.intad.asn.au) follow to Pro/DESKTOP page

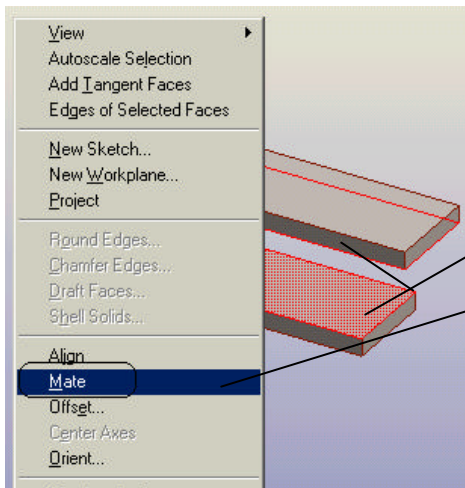



### Second Piece:

Assembly > Add Component > choose Component > OK

### First Piece:

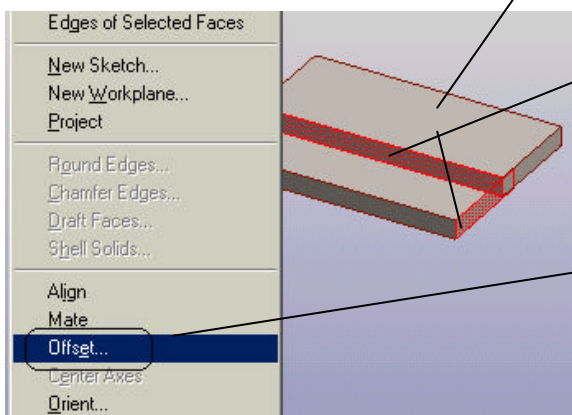
Assembly > Add Component > choose Component > OK  
Assembly > Fix Components




Select two faces 

Right Mouse Click (RMC) > choose Mate

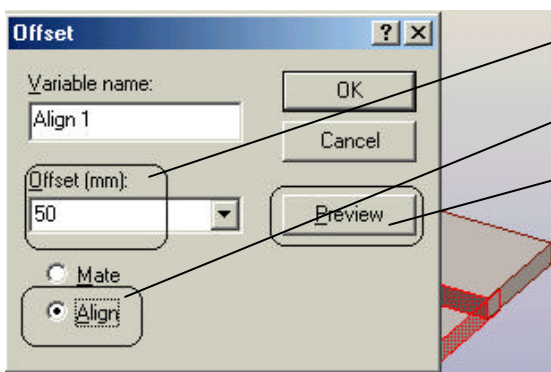
Pieces come together as such



**These two selected Faces need to be a 90° to each other & 50 mm from each other.**

Use Select Parts  to move them in relation to each other as the Offset command does some 'odd' things: seems to go the wrong way

RMC > Offset




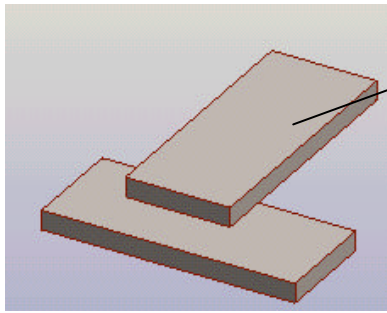
Set Offset to 50

Check Align button is selected

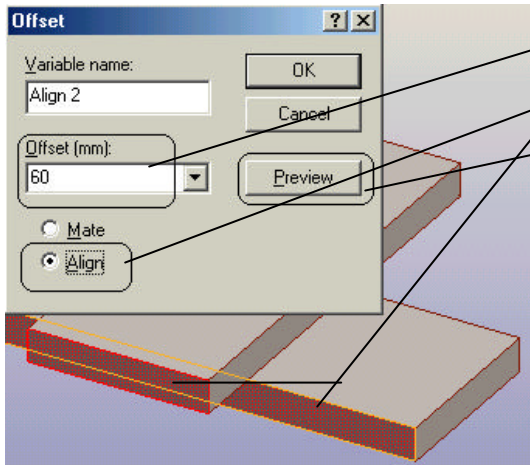
Use Preview to ensure that the 'odd' thing mentioned above does not happen before you click OK



If the Preview is not correct, Cancel & use the

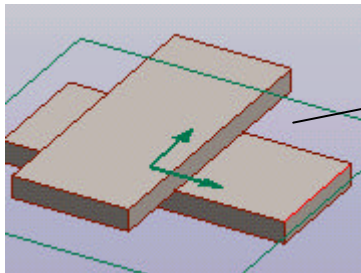
Select Parts  to readjust the position



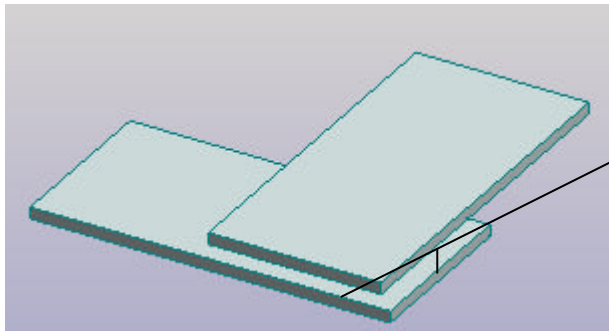
The non-Fixed part will be rotated to Align itself & will be Offset the distance away as specified



Select two required Faces   
 Set Offset to 60  
 Check Align button is selected  
 Use Preview to ensure that the 'odd' thing mentioned above does not happen before you click OK  
 If the Preview is not correct, Cancel & use the Select Parts  to readjust the position

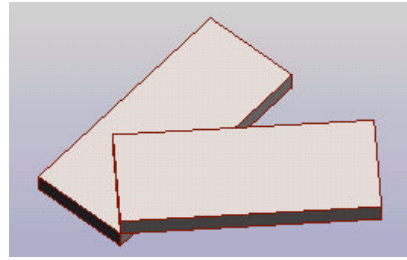



Final Product

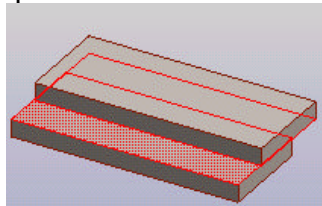


Shows Offset used for both sets of Faces. Needed to use a -15 Offset for this exercise

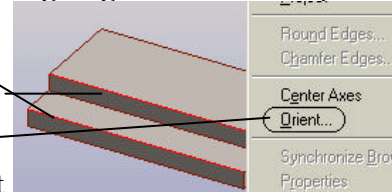
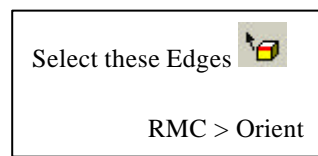
## Angular “Joining” of Steel/Timber:



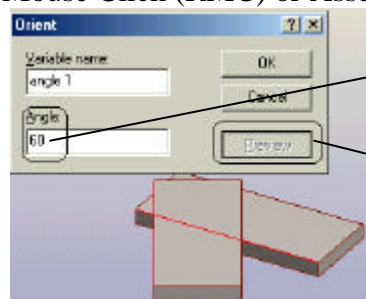
1. Draw the parts as required.
2. New Design
3. Assembly > Add Component > choose part > OK
4. Assembly > Fix Component
5. Assembly > Add Component > choose part > OK
6. Select  the top face & the bottom face of the parts & Assembly > Mate



7. Looks like this
8. Select  a long Edge of the top part & a long Edge of the bottom part. Right




Mouse Click (RMC) or Assembly > Orient

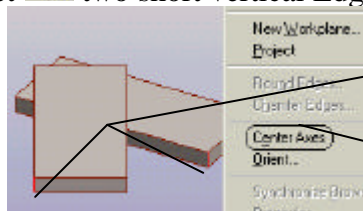


Type in an Angle

Use Preview to get an idea if the Angle is going to be correct

9. > OK

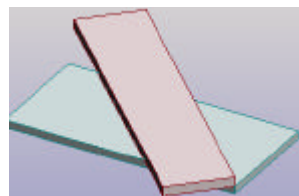
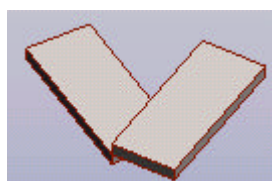
10. Select  two short vertical Edges as shown > RMC or Assembly > Centre



Select two short vertical Edges as required

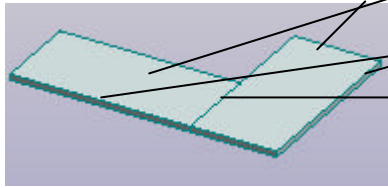
Use Centre Axis

Axis

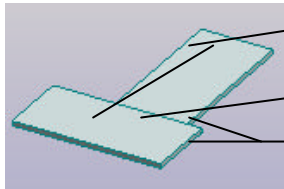


11. Final Product  showing Angular Joining for Steel/Timber.

## Butt

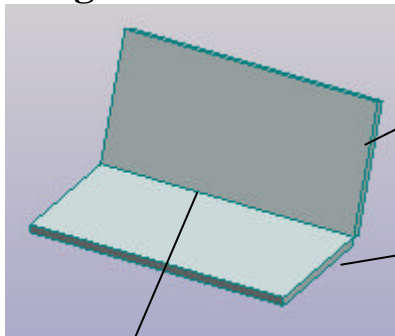


- Select & RMC & Align Top Faces
- Select & RMC & Align these Faces
- Select & RMC & Mate these Faces

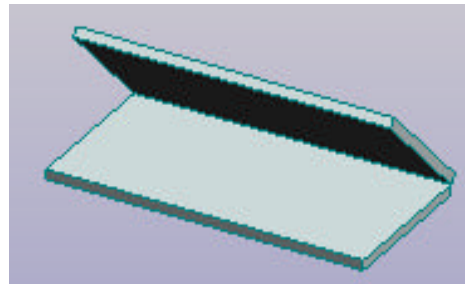


- Select & RMC & Align Top Faces
- Select & RMC & Mate these Faces
- Select & RMC & Offset > 15 > Align > OK these Faces

## Hinge Action

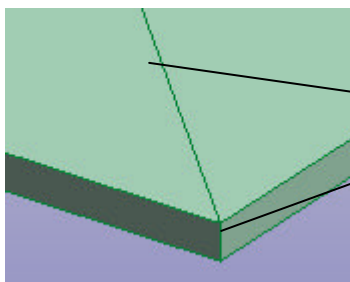
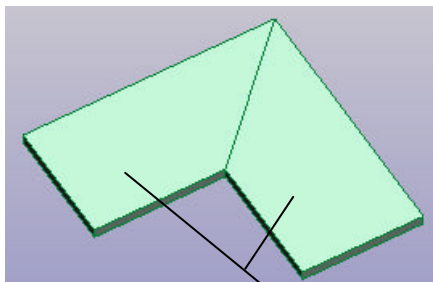


- Select & RMC & Align these end Faces



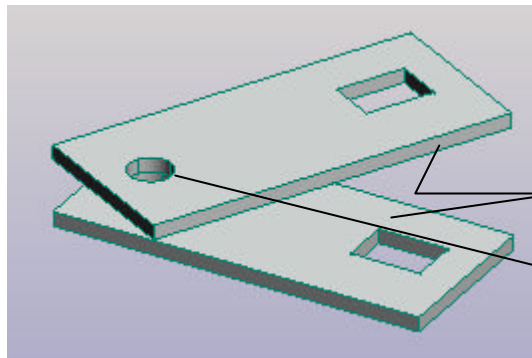
- Select & RMC & Centre Axis these Edges

## Mitre



- Select & RMC & Align Top Faces
- Select & RMC & Centre Axis on these Edges

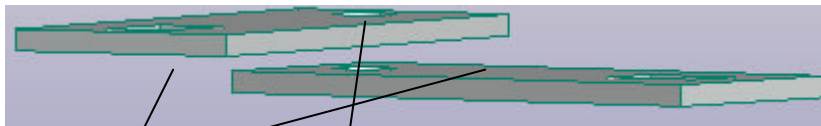
## Holes Centred & Rotation



Use for surfaces in contact with each other

Select & RMC & Mate this top Face & this underside Face

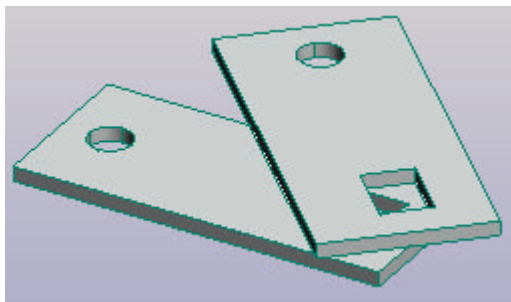
Select & RMC & Centre Axis of two Holes



Select Faces & RMC & use Offset > 25 > Align > OK these Faces

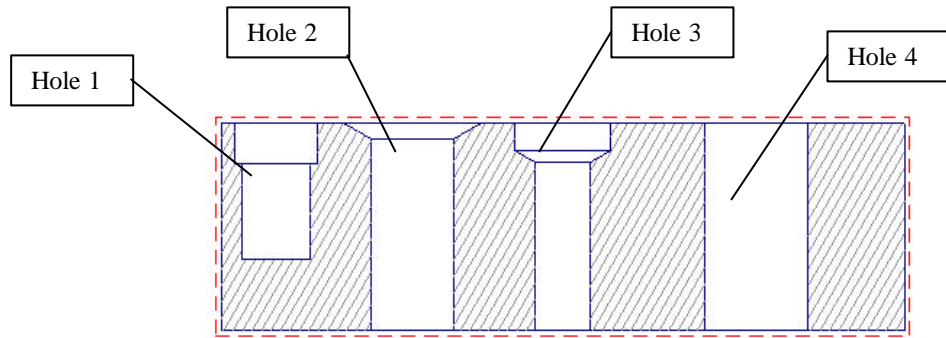
Select & RMC & Centre Axis of two Holes

Use for surfaces which are not in contact with each other e.g. a set of drawers opening around a central post.



A non-circular Hole can be used for rotation

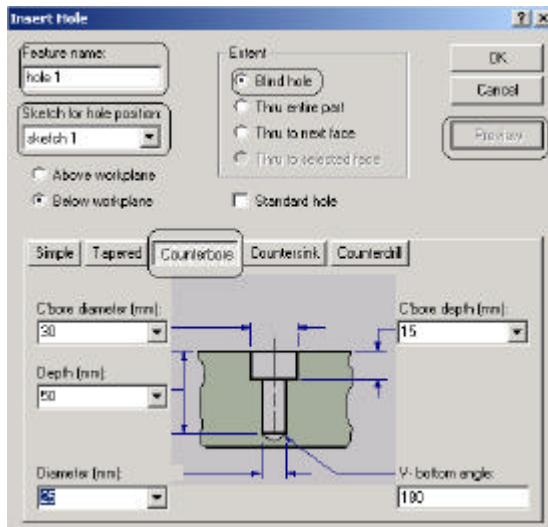
by using an Edge of the Hole to Centre Axis > OK



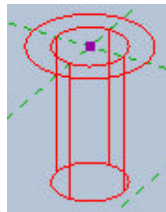
**Holes/Threads:**

A - A

1. Draw a circle in the required place > use

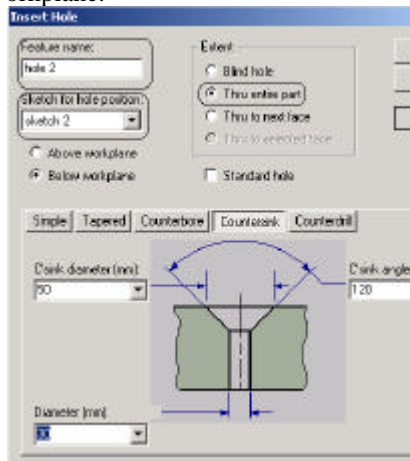


2. > OK

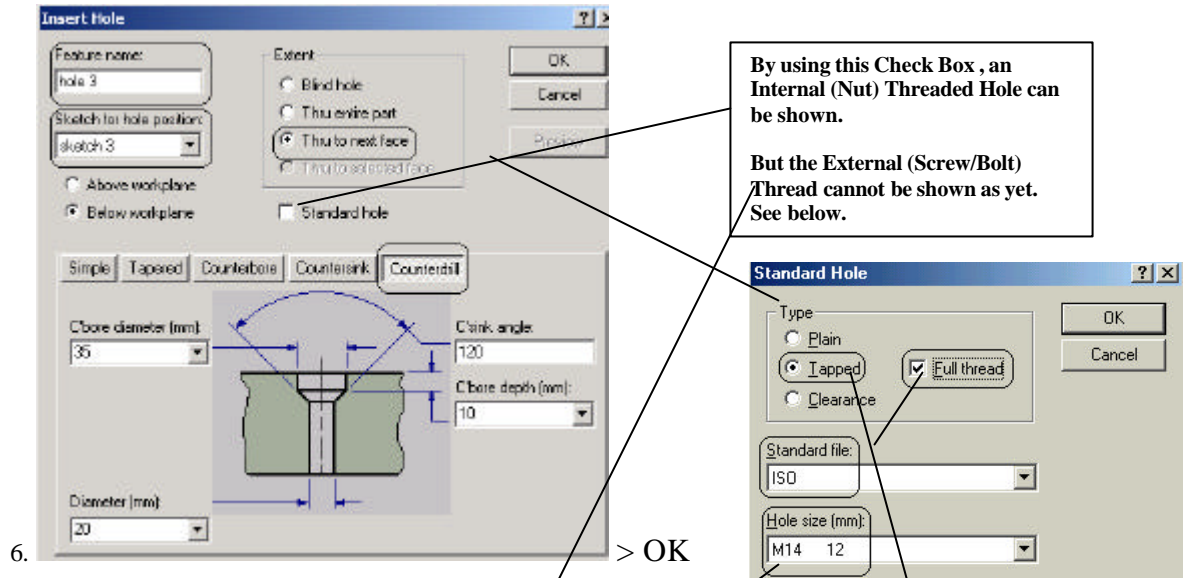


3. The Hole can be previewed

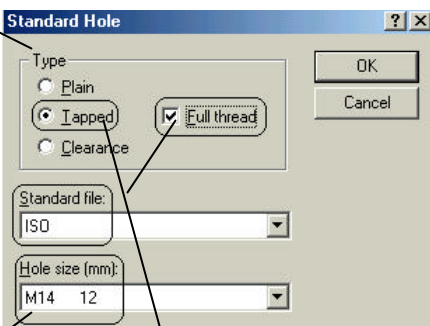
4. Because each hole is at a different depth, a New Sketch needs to be added to the Base Workplane.



5. > OK




By using this Check Box , an Internal (Nut) Threaded Hole can be shown.  
But the External (Screw/Bolt) Thread cannot be shown as yet. See below.



Choose Tapped, Full Thread  
Thread size & drilling size

thomaspope  
Member since Jan-7-02  
Oct-29-02, 09:33 AM (EST)

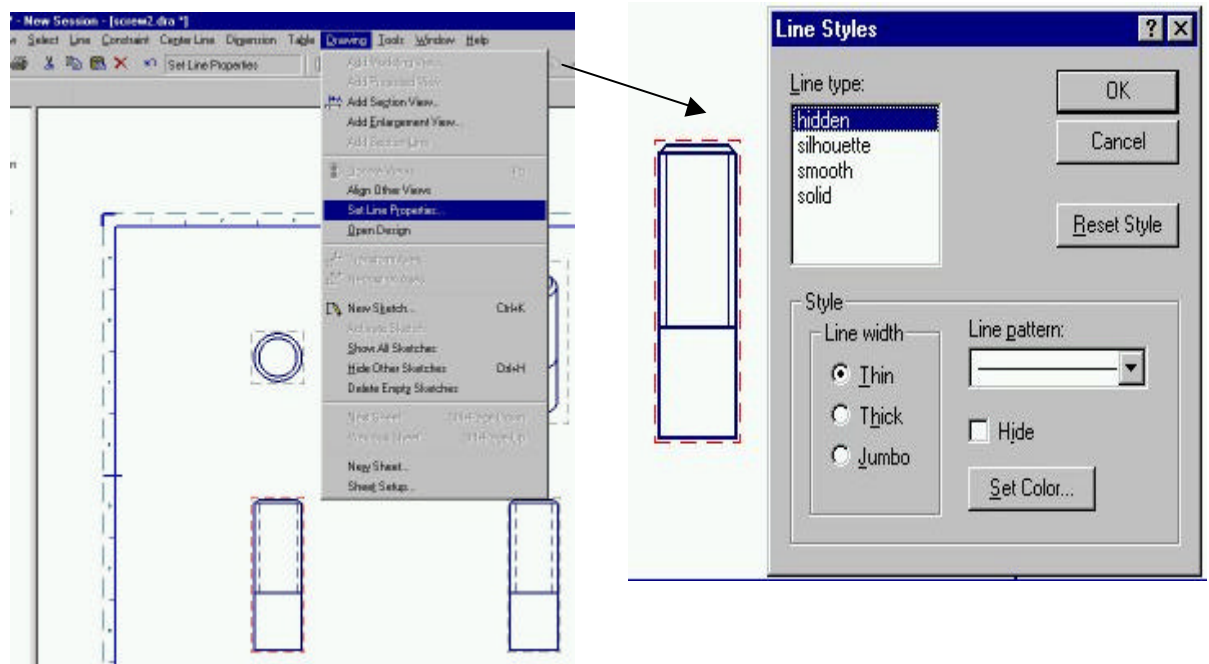
3. "Creating a male thread"

Oh! I understand now. You want the lines to look thin and not dashed. You have to change the line style. First, select the view(s)  you want to change. Then select Drawing and Set Line Properties... and change hidden to a thin line instead of dashed.

Please note that the lines are hidden for the thread detail and changing the line properties changes hidden lines for the entire view, not just for the threads.

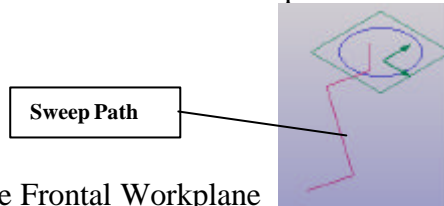
Two Design files Assembled-eg for M36 thread, drill size 32. Two cylinders of correct sizes & Chamfer 2 on M36, Assemble

In reply to a query of mine on Topic 499, prodesktop.net, discussion forum, on male threads. Edited message.



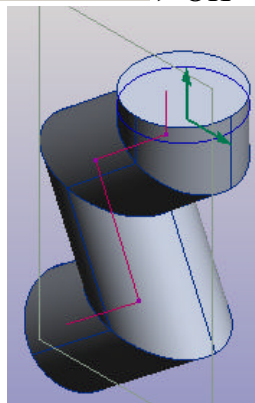
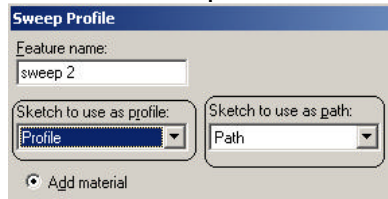
## Sweep Profile along Sketch Path:




1. Draw a Circle on the Base Workplane > RMC Initial rename Profile > 

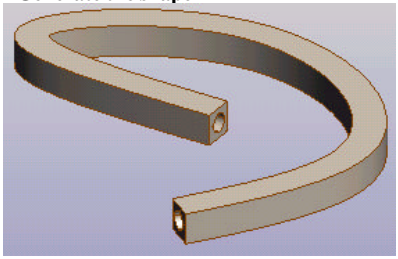


> select the Frontal Workplane RMC > choose New Sketch  
 > rename Path > OK. Note: Path & Profile MUST be at 90° except as here.

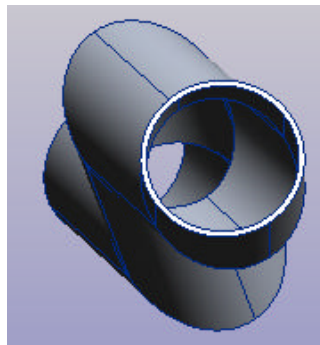
2. Draw a sweep path
3. Feature > Sweep Profile > Along Sketch Path >



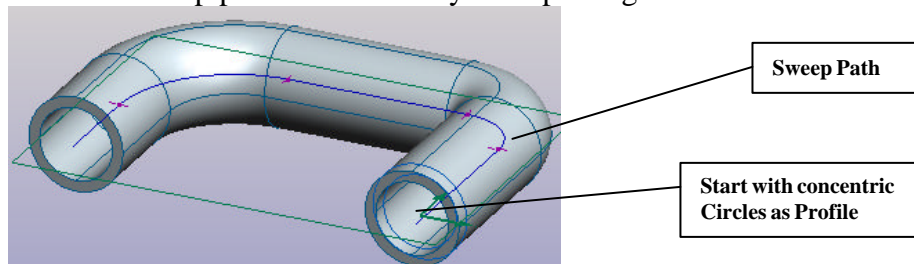
- Use the same Workplanes as before, but draw the Path using the Spline  tool
- The Path & Profile will NOT be at 90° 
- Generate the shape 



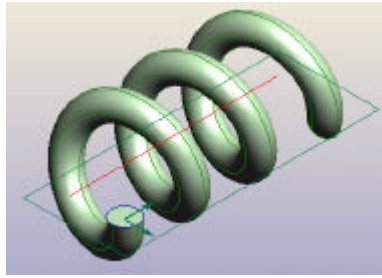
4. Finished Product .
5. Select both end faces & Shell @ wall thickness of 2 > OK



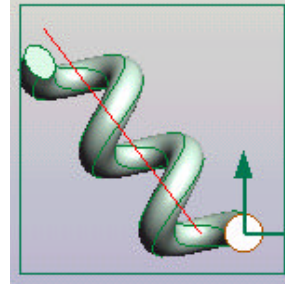
6. Sometimes a “pipe” can be done by Sweep along a Sketch Path





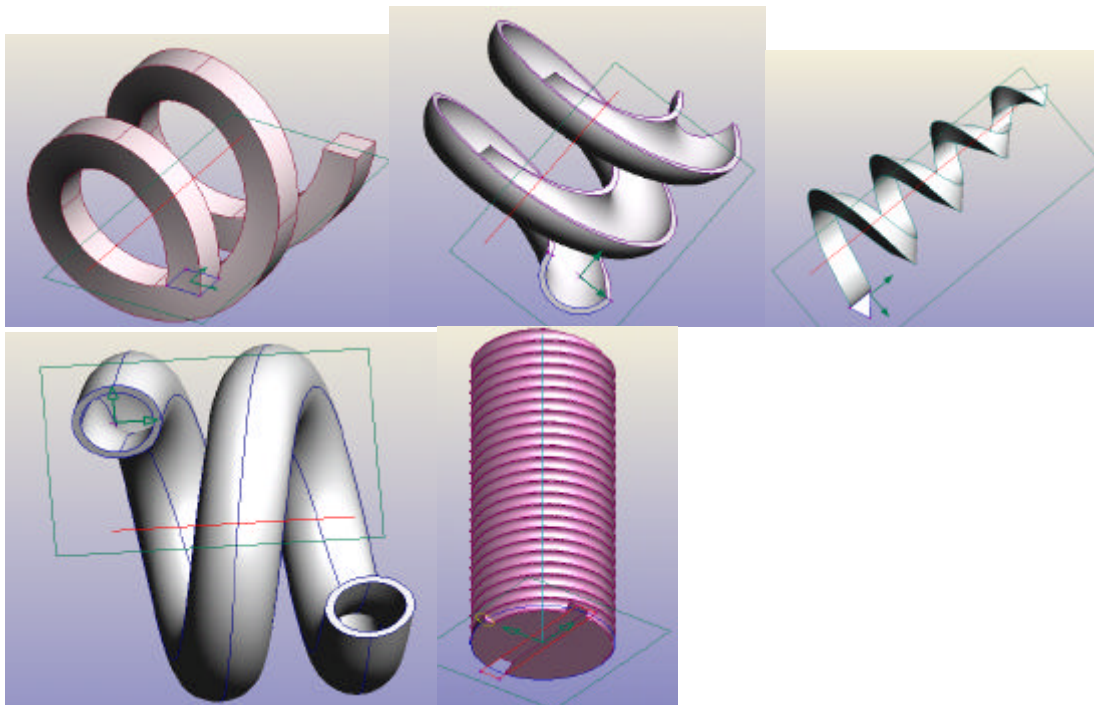
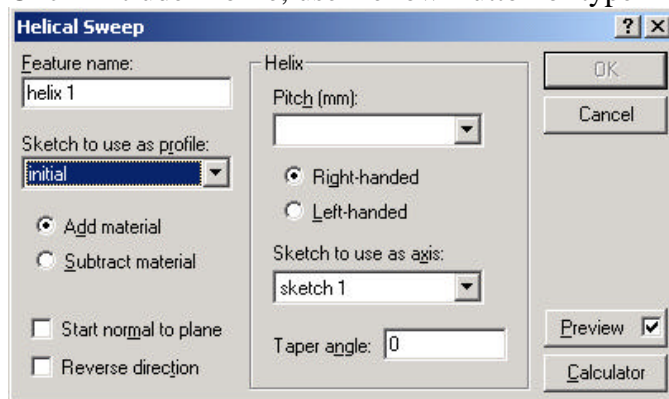
# Helixes



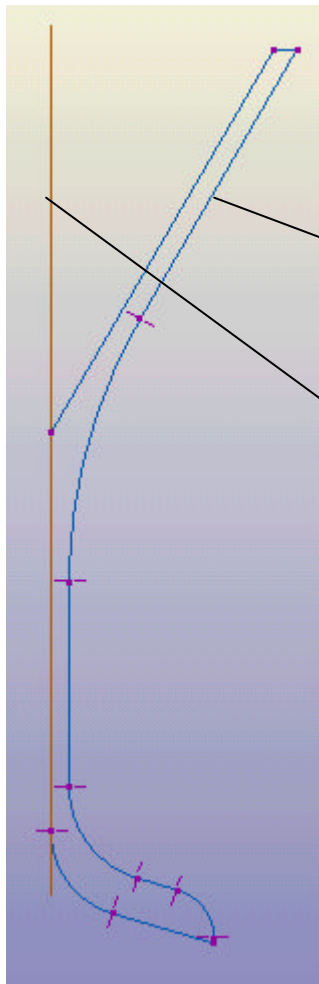
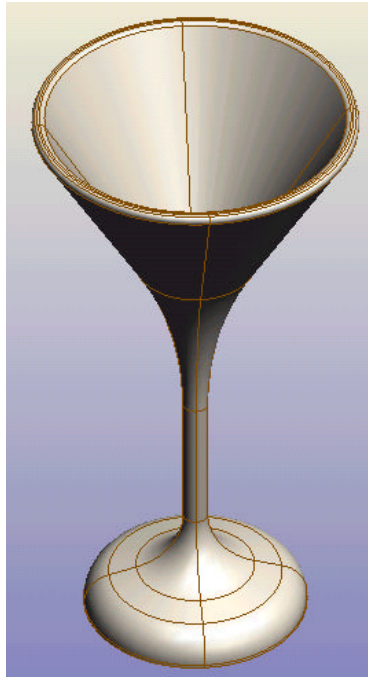
Note: The Axis line can be at an angle



1. New  > New Design, OK
2. Draw a rectangle  on the Workplane
3. Workplane > New Sketch
4. Draw, holding down SHIFT key, a line.
5. Feature > Sweep Profile > Along Helix
6. Click Extrude Profile, use Yellow Button or type in a Pitch measurement, OK



Glass



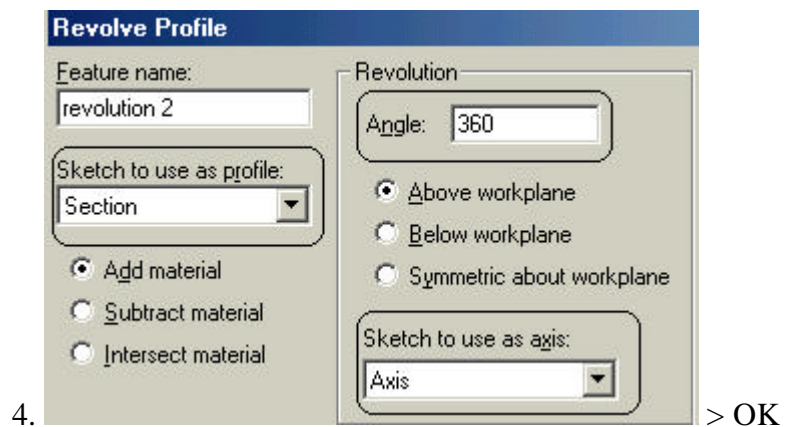
Use frontal/lateral workplane, put a New Sketch on that workplane, then draw the Section of the Glass


Place another New Sketch on the same Workplane to use as the rotational Axis

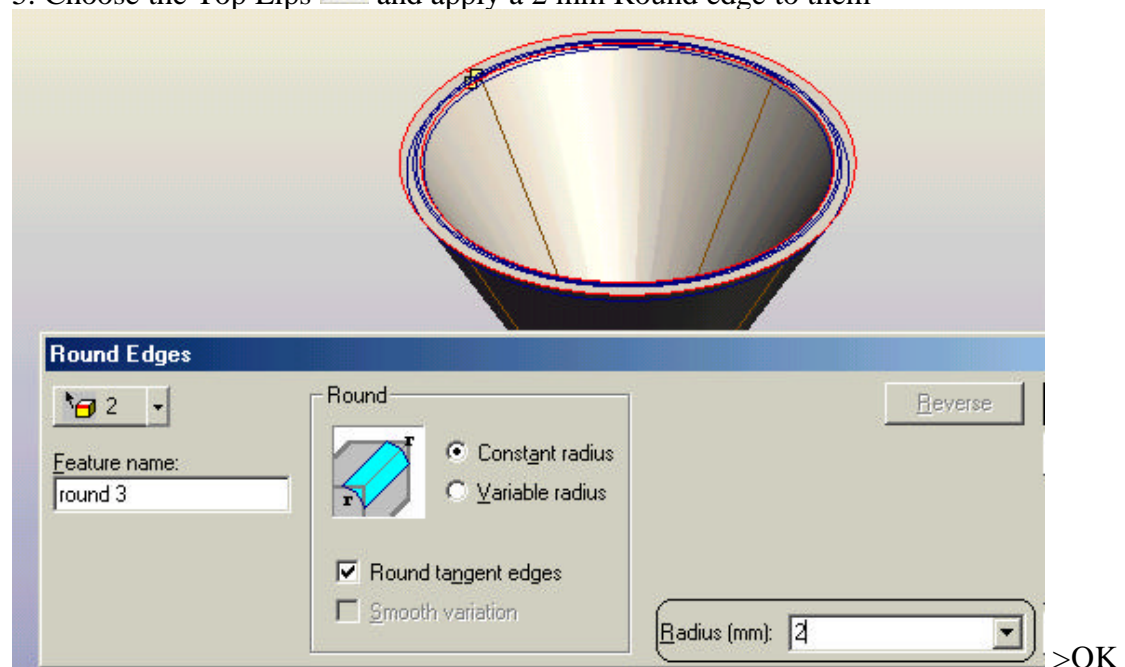
1.

2. View Isometric

3. Feature > Revolve Profile 



5. Choose the Top Lips  and apply a 2 mm Round edge to them








6. Save as Glass.

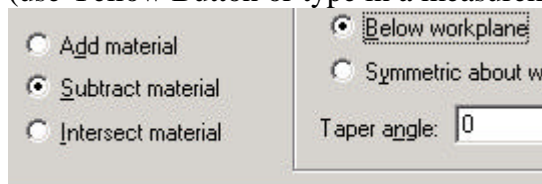
7. Goto New > Album > OK



# Table



## BODY

1. New  > New Design, OK
2. Draw a rectangle  on the Workplane
3. Draw a smaller rectangle inside the original. Dimension  the inside rectangle, note these dimensions for the INSERT piece.
4. Click Extrude Profile  BELOW Workplane. Click/Hold/Drag Yellow Button down, OK
5. Select the Left hand face of this extruded part , goto Workpalne > New Workplane, Plane of Object, OK, goto Workplane > New Sketch, OK
6. Draw a rectangle on this sketch so that it lies inside top/right/left edges and outside bottom edge and click Extrude Profile, click radio buttons Below Workplane and Subtract Material, fully Extrude it until it is clear of back face, (use Yellow Button or type in a measurement), OK






7. Select Right Hand face. Repeat 5 & 6 above.
8. Save your work as Table Body




## Insert Piece

1. New  > New Design, OK
2. Draw a rectangle, use the Dimensions from above to constrain the Insert size.
3. Click Autoscale 
4. Extrude Profile, Below Workplane, OK
5. Save your Work as Table Insert.


## ASSEMBLY

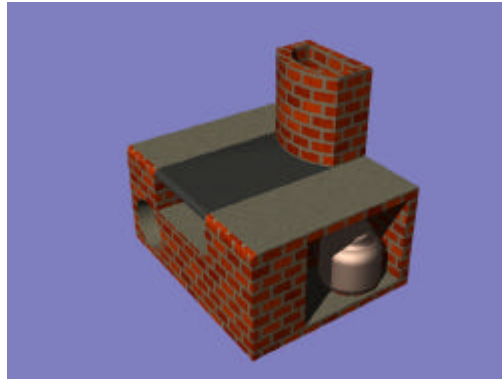
1. New  > New Design, OK
2. Assembly > Add Component, click Table Body, Open
3. Assembly > Fix Components.
4. Assembly > Add Component, click Table Insert, Open
5. Select Faces  - top of Body and hold down the SHIFT key + left click top of Insert. Both should go reddish colour.
6. Assembly > Align
7. Select Faces  - an edge of Insert and SHIFT + left click corresponding part of Body. Both should go reddish colour
8. Assembly > Mate.
9. Repeat 7 & 8 for other edge at 90°.
10. Save as Table Assembly.

## ALBUM

1. New  > Album, OK
2. Image > New Image, click Table Assembly, OK, Update 
3. Drop down the Images box  and choose Materials
4. Click + on Non-metal. Left click and drag Wood, Varnished to the Image until the wire frame of the Body is showing. Let left mouse button go. Update.
5. Left click and drag Glass to the Image until the wire frame of the Insert is showing. Let left mouse button go. Update.
6. Image > Image Properties, Studio > Lighting. Choose Room Lighting, OK. Update.
7. Save as Table Album.





## Extension

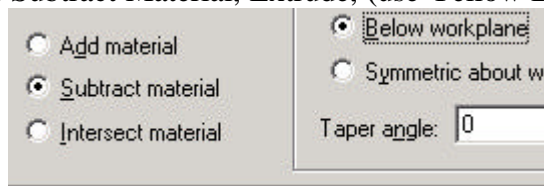
1. Design an Insert as above but ensure the Checkerboard design in the Album area fits it on an 8 X 8 pattern. Save.
2. Design an indoor Games Table along the same idea as above. Save.
3. Assemble the two components.
4. Save.
5. Goto Album and import the above assembly.
6. Drag & drop the material for the Games Table part.
7. Click Select Faces icon  before dragging & dropping the Checkerboard pattern onto the Insert.
8. Save.





# Barbeque



## BODY

1. New  > New Design, OK
2. Draw a rectangle  on the Workplane
3. Click Extrude Profile  BELOW Workplane. Click/Hold/Drag Yellow Button down, OK
4. Workplane > New Sketch
5. Draw a small Rectangle on the back edge for Fume Funnel. Extrude up. OK
6. Select the top face of this extruded part , goto Workpalne > New Workplane, Plane of Object, OK, goto Workplane > New Sketch, OK
7. Draw a smaller rectangle on this sketch and click Extrude Profile, click radio buttons Below Workplane and Subtract Material, Extrude, (use Yellow Button






- or type in a measurement) OK
7. Select Left Hand face, Workplane > New Workplane, Plane of Object, OK. Workplane > New Sketch, OK
  8. Place a three rectangles on this face. One for the BBQ plate to cover and two smaller ones for storage. Dimension  the TOP edge of the large rectangle, note this dimension + the depth of the Extrusion for BBQ Plate size. Goto Extrude Profile  and extrude back to nearest point of Fume Funnel, OK.
  9. Save your work as BBQ Body




## PLATE

1. New  > New Design, OK
2. Draw a rectangle, use the Dimensions from above to constrain the Plate size.
3. Click Autoscale 
4. Extrude Profile, Below Workplane, OK
5. Save your Work as BBQ plate

## ASSEMBLY

1. New  > New Design, OK
2. Assembly > Add Component, click BBQ Body, Open
3. Assembly > Fix Component
4. Assembly > Add Component, click BBQ Plate, Open
5. Select Faces  - top of Body and hold down the SHIFT key + left click top of Plate. Both should go reddish colour.
6. Assembly > Align
7. Select Faces  - an edge of Plate and SHIFT + left click corresponding part of BBQ Body
8. Assembly > Mate.
9. Repeat 5 & 6 for front edge of Plate & BBQ Body.
10. Save as BBQ Assembly.

## ALBUM

1. New  > Album, OK
2. Image > New Image, click BBQ Assembly, OK, Update 
3. Drop down the Images box  and choose Materials
4. Click + on Non-metals. Left click and drag Bricks to the Image until the wire frame of the Body is showing. Let left mouse button go. Update.
5. Click + on Metals. Left click and drag Cast Iron to the Image until the wire frame of the Plate is showing. Let left mouse button go. Update.
6. Image > Image Properties, Studio > Lighting. Choose Room Lighting, OK. Update.
7. Save as BBQ Album.

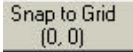




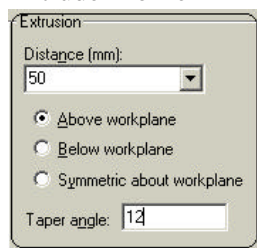
## Extension (diy)


1. Design a Gas Bottle for the BBQ.
2. Remove one of the storage parts from the front of the Body.
3. Place a rectangular compartment in the right hand end.
4. The Gas Bottle would need a round indent in the floor of this compartment for its base.
5. Save.

## Tap Handle





- Using the Base workplane, draw a circle @  and dimension it at Diameter 80.
- Autoscale  to see a larger view.
- Extrude Profile  - Above Workplane, Distance 50, Taper Angle 12, OK.

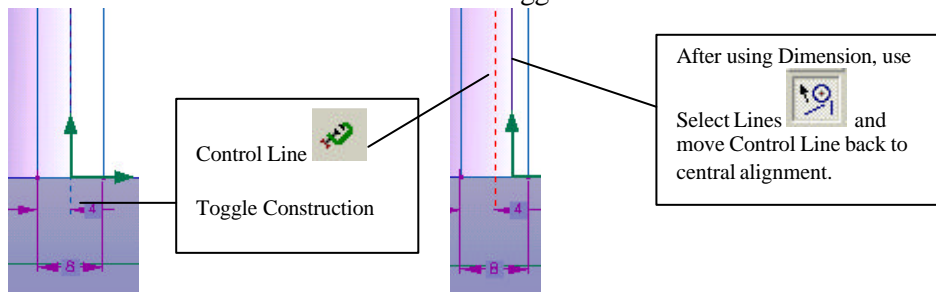


- Autoscale to see a smaller view.
- Select top face . Goto Feature > Modify Solids > Deform Face > Height 15

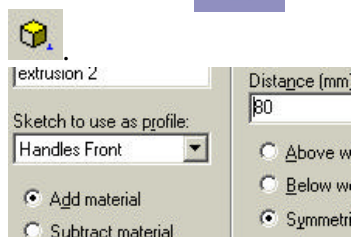



OK.

- Select Workplanes . Left Mouse Click (LMC) Frontal (goes Red). Right Mouse Click (RMC) Workplane. Select New Sketch > rename Handles Front. OK.
- View Onto Workplane .
- Draw a Rectangle of size 8 wide & full height. Constrain it by Dimensioning. Hint: Draw a control Line down the middle & Toggle Construction it to be a hidden line.

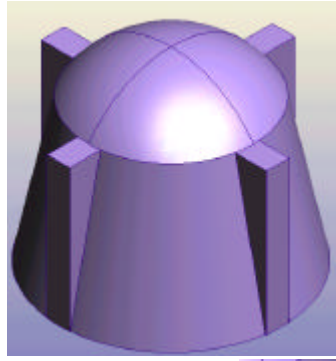


- Goto View Isometric .



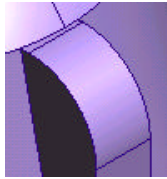
- Extrude Profile . OK.

11. Using Lateral Workplane, REPEAT 6 to 10 **BUT** rename New Sketch > Handles Lateral



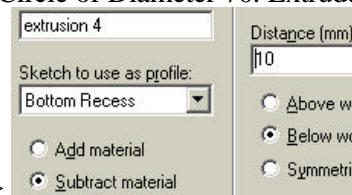
12. View Isometric Looks like this.

13. Select Edge  & Round  to  Radius 10.  OK



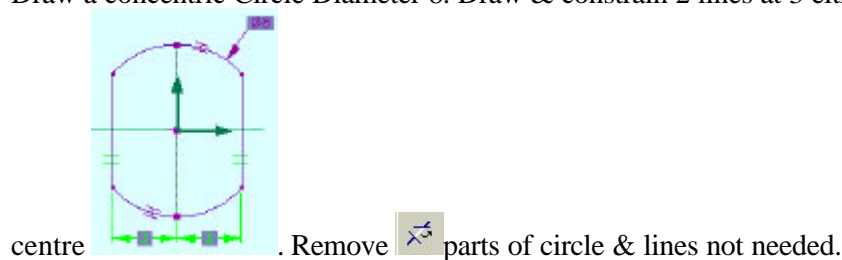
Select one Edge  
Shift LMC other  
edges

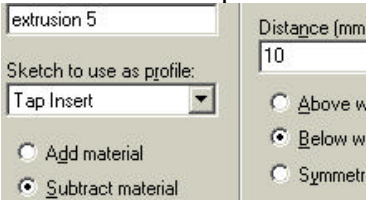
14. Round other edges as you see fit.  
15. Move around so that the bottom face is visible.  
16. Select the Bottom Face > RMC > select New Sketch > rename Bottom Recess OK  
17. Draw a concentric Circle of Diameter 70. Extrude Profile > to a depth of 10 >



Subtract Material > OK

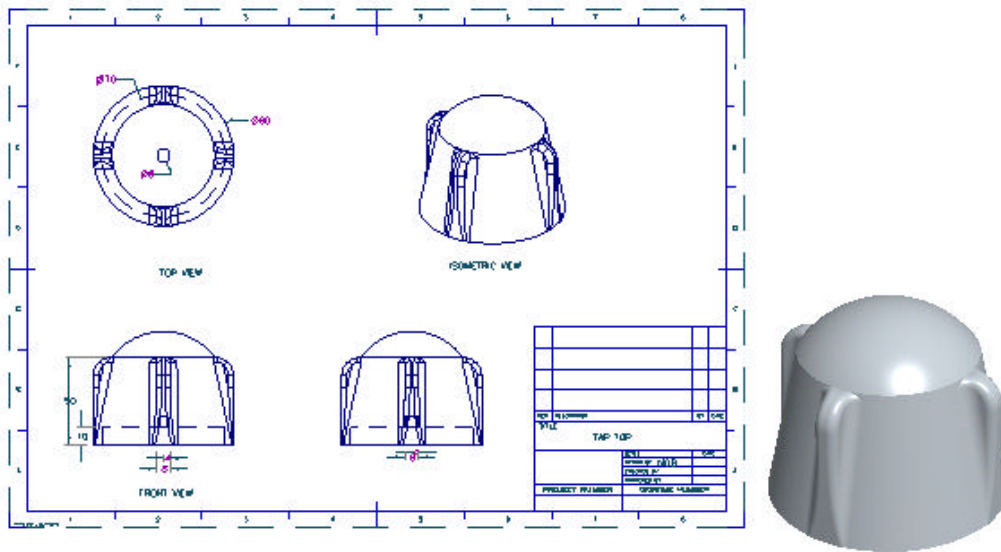
18. Select bottom Face of 17 above > RMC > select New Sketch > rename Tap Insert OK  
19. Draw a concentric Circle Diameter 8. Draw & constrain 2 lines at 3 either side of



20. Extrude Profile >  OK

21. Save as Tap Handle.

# Drawing



This one will give third angle orthographic with isometric view


Choose Existing Design

Select the open file

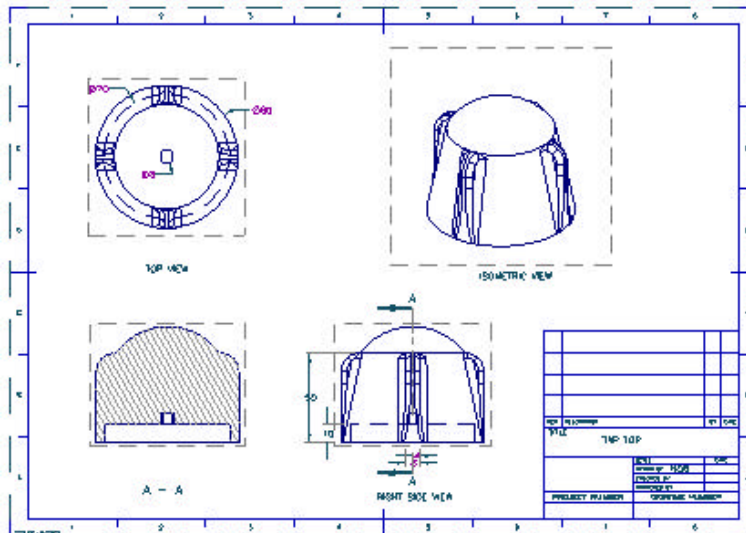
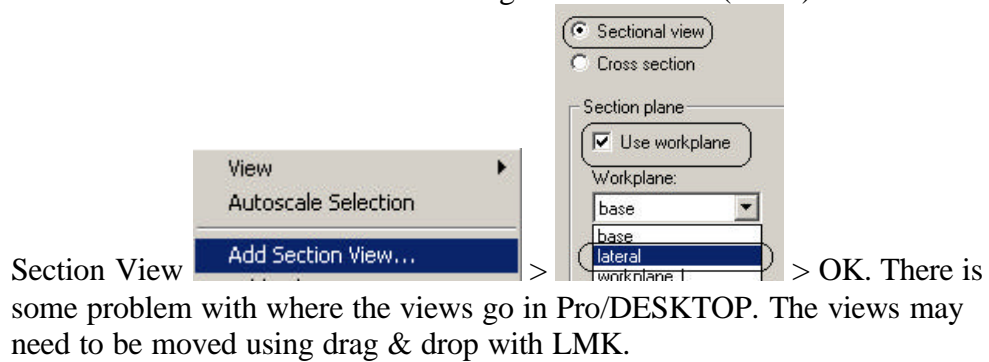
2. Use Tools > New Drawing > OK

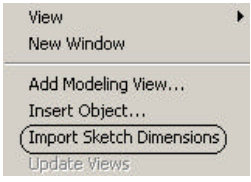
3. Select  and the modelling containers show as broken lines > Left Mouse

Click (LMC) the right side view > container goes red > press Delete key



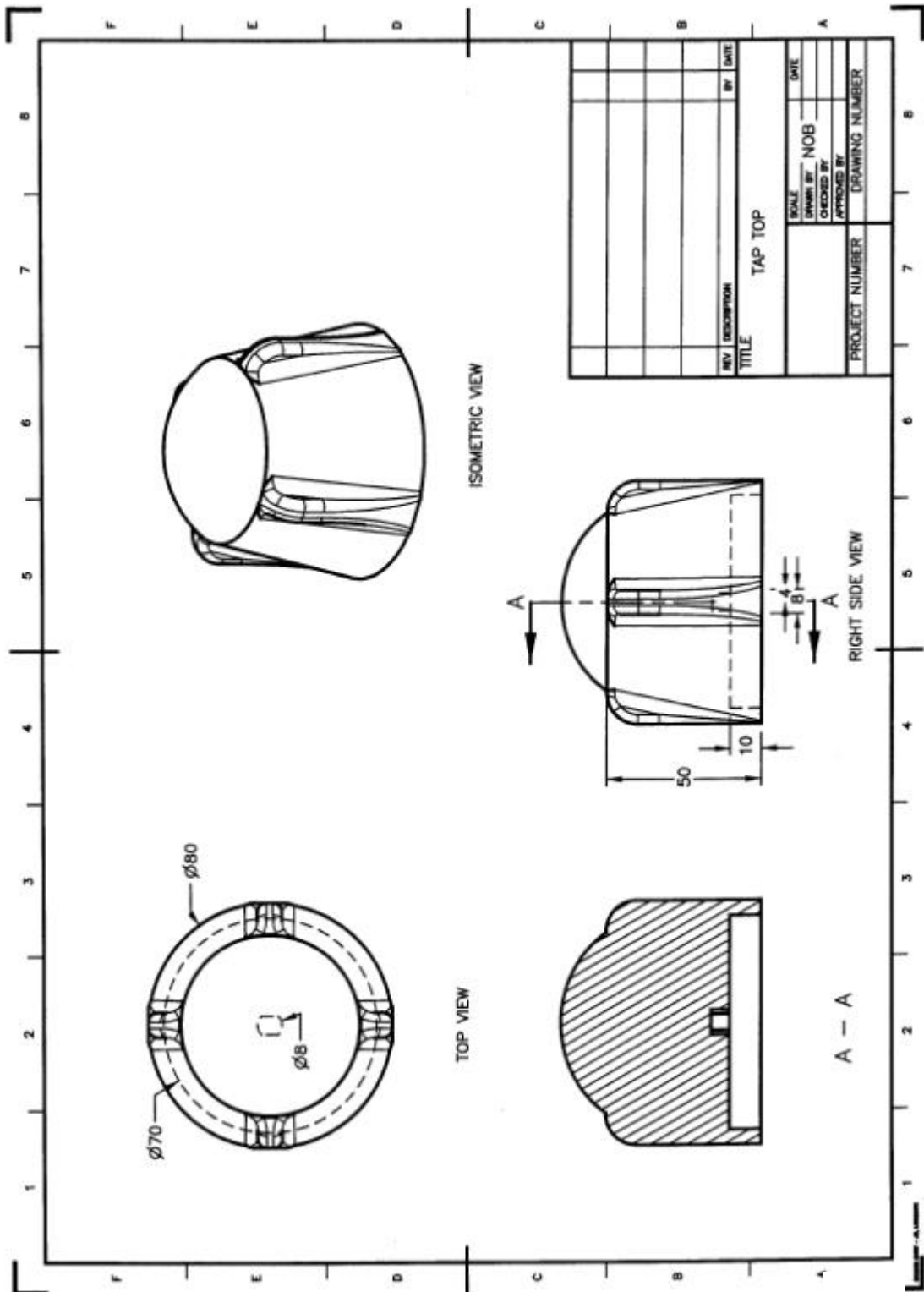
- LMC the left side view container > Right Mouse Click (RMC) > select Add

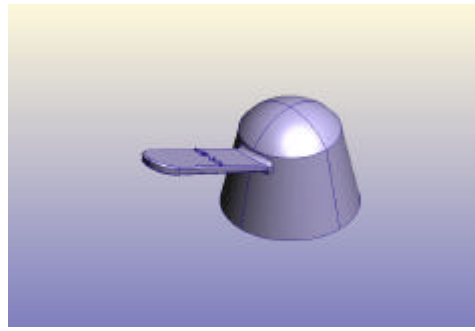


- Import the dimensions by RMC > 
- Use Shift+Z to zoom in on dimensions to change to zero decimal places & Shift+A to Autoscale back.
- Use **A** to drag a labelling box (hold down the **Shift** key to remove leader when dragging), double LMC box & Capitals for subtitles & name block entries.
- Save as Tap Handle

## Print

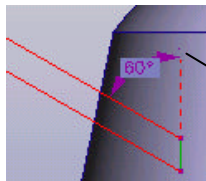
- File > Print Setup > change to Landscape > OK
- File > Print Preview to check > Close
- File > Print > (check printer type, change through Setup) > OK
- LMC the Print True Scale check box (be careful as this will print True Scale & leaves off the Border)





## *Tap Lever*

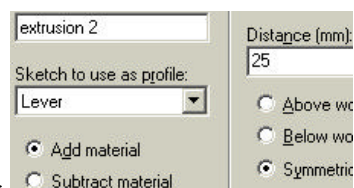
1. Follow the **Tap Handle** sheet for everything except the Handle part.
2. Select Workplanes > Left Mouse Click (LMC) either Frontal or Lateral Workplanes turns red > Right Mouse Click (RMC) > select New Sketch rename to Lever > OK
3. View Onto Workplane
4. Draw a rectangular profile using the Line tool. Ensure that the lines are parallel to each other & the angle is  $60^\circ$ . Hint: use a vertical control line reset to toggle



construction to constrain the angle.

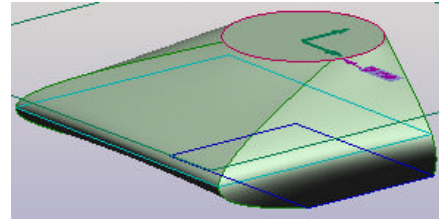
Vertical control line  
toggle construction

5. View Isometric.

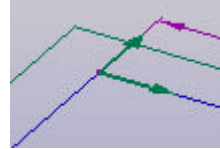


6. Extrude Profile >  Add material  Subtract material  Symmetric OK
7. Select top face of Lever. RMC > New Sketch > OK
8. View Onto Workplane.
9. Goto Line > Add Text Outline > type <C H> to give instruction on which way to turn the Lever for Hot or Cold water. The text size/font types will need some adjustment to fit onto the Lever width.
10. View Isometric
11. Extrude Profile the lettering to 1, Above Workplane, Add Material, OK
12. Round off all the edges as needed.
13. Save as Tap Lever.

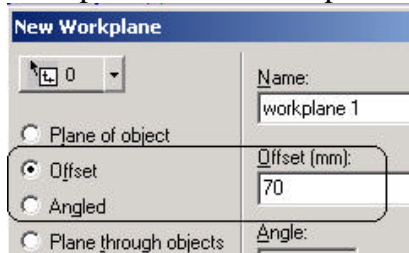
## Lofting Through Profiles



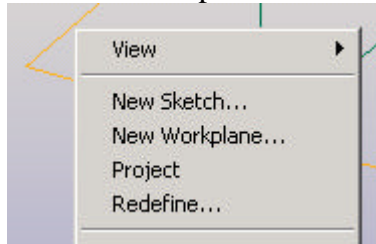
1. Use the Base Workplane (it has a sketch on it already called Initial)



2. Draw a Rectangle & Dimension it at 200 X 150.
3. Workplane > New Workplane > Offset 70 > OK




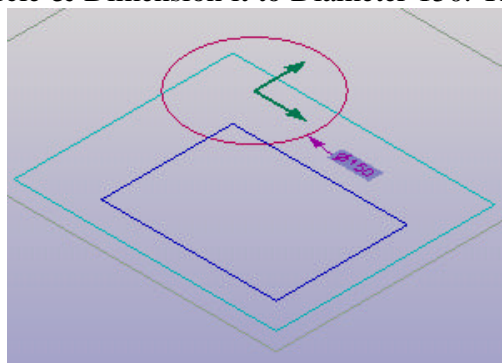
4. The new Workplane is red > Right Mouse Click (RMC) > select New Sketch



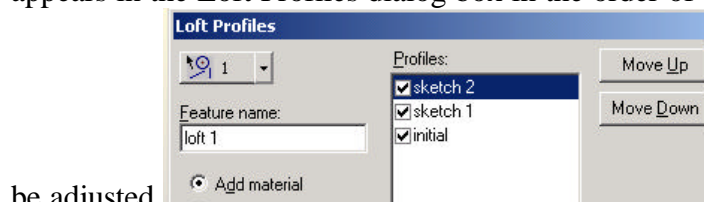
> OK Note: its name is Sketch 1

5. Draw a Rectangle & Dimension it at 300 X 250
6. REPEAT steps 3 & 4 except make the Offset 50. New Sketch name is Sketch 2

7. Draw a Circle & Dimension it to Diameter 150. There may be a need to 

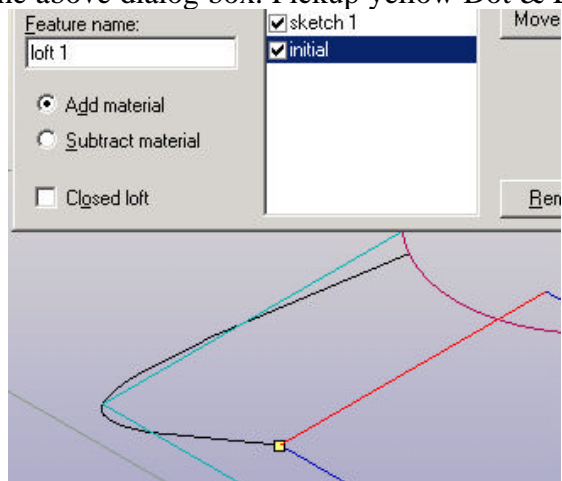


8. Looks like
9. Feature > Loft Through Profiles > Left Mouse Click (LMC) each item & it appears in the Loft Profiles dialog box in the order of selection. This order can



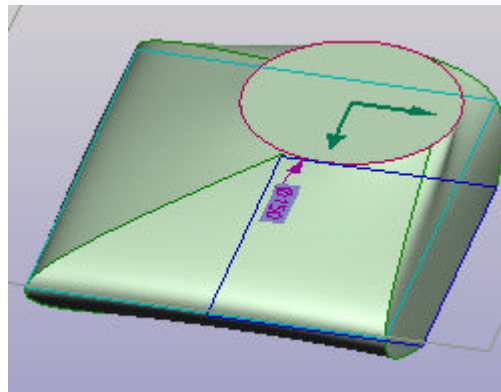
be adjusted

10. To move the point from which the Profile starts the Profile must be LMC inside the above dialog box. Pickup yellow Dot & LM drag to where you want




it to be.

> OK

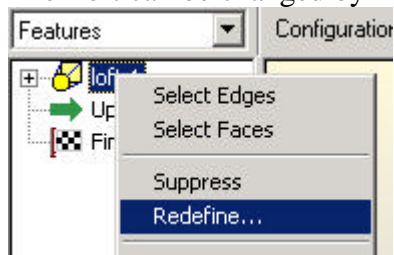


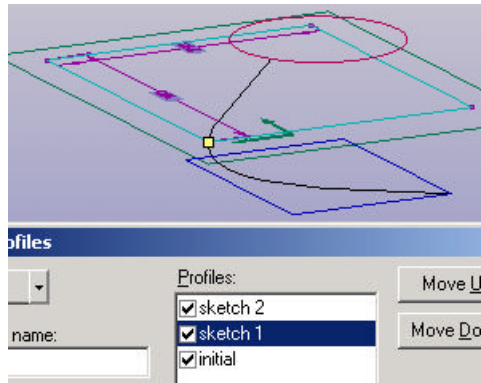
11. Looks like this.

12. Save as Bottle 1

13. All Sketches can be moved & the drawing Updated 

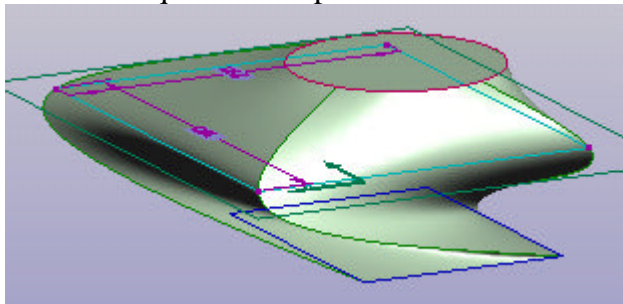
14. The Loft can be changed by RMC the Loft 1 in the Features Browser





15. Move the Yellow Dot to here

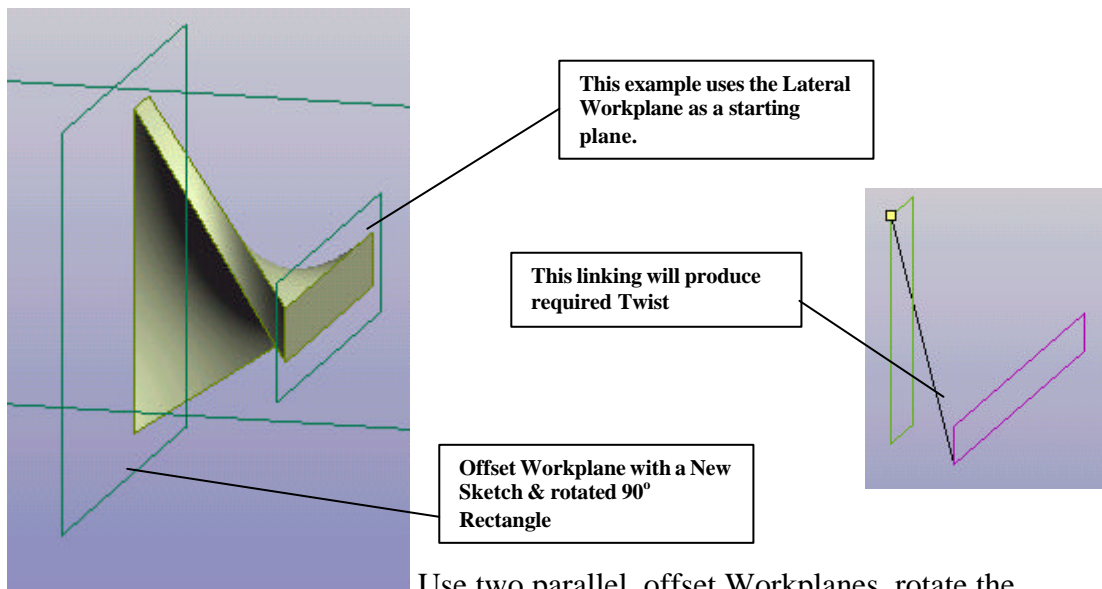
16. This will require OK & produces a twist in the Object



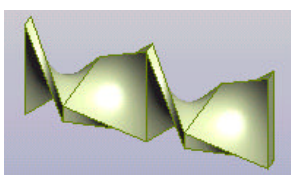
17. Save as Bottle Twist

**Another Twist that could be useful:**

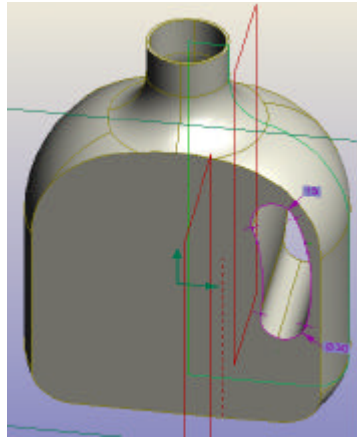
To “move” from a horizontal orientation to a “vertical” orientation:



Use two parallel, offset Workplanes, rotate the Rectangular Face 90° then Loft so that the Twist is as shown.

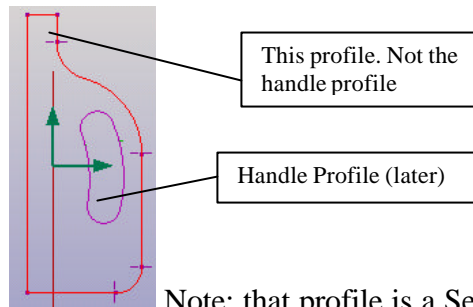


Assembly of several to form decorative finish

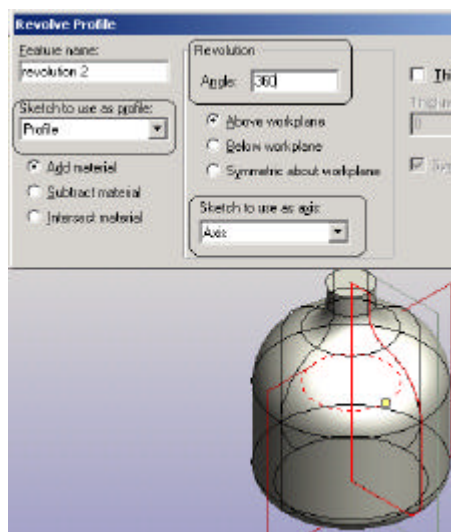




### ***Bottle with Handle***

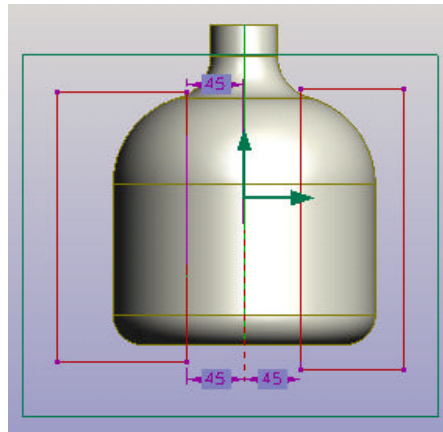
1. Use the frontal or lateral workplane. RMC > New Sketch > rename Profile



2. Draw & Revolve a similar profile Note: that profile is a Section through centre of Bottle.
3. Add New Sketch to same Workplane > rename Axis
4. Draw a Line to be used as the Axis of Rotation
5. View Isometric

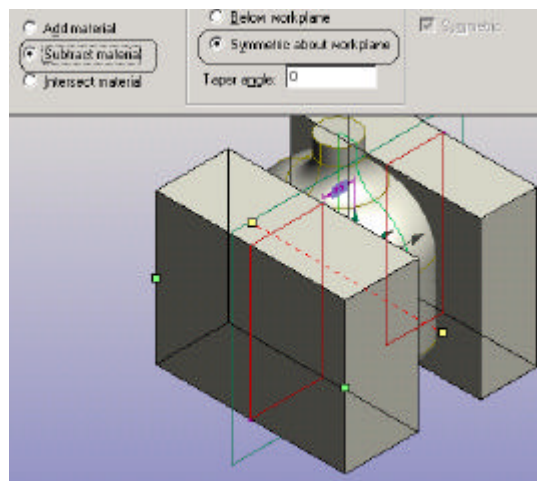


6. Feature > Revolve Profile > OK
7. Save as Container Round
8. Select Workplanes , LMC Lateral Workplane > RMC Workplane > New Sketch > rename Side Removal > OK
9. View Onto Workplane 



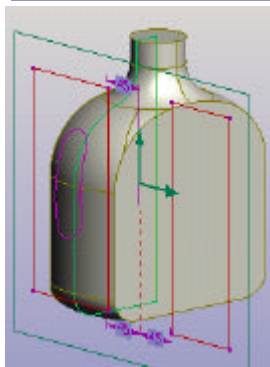
10. Draw the Rectangles shown.

11. View Isometric





12. Extrude Profile

> OK






13. Looks like this

14. Select  one of the Cut-off Faces > RMC > New Sketch > rename Handle > OK

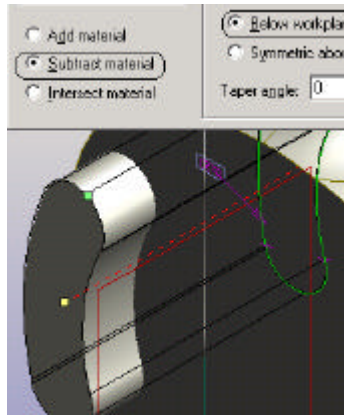
15. View Onto Workplane 

16. Draw two Circles & Dimension to 30. Draw Tangent Lines to each Circle. Check tangency by LMC an item & Shift + LMC the other so both are Red >



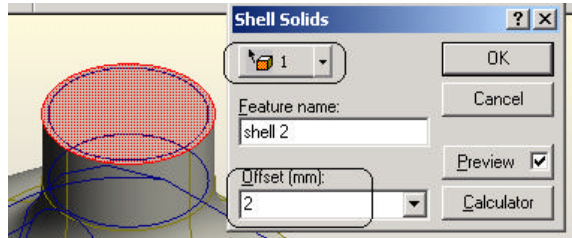
17. Use the Spline Tool  to make the Straight Lines form a curve. Select both arcs as above in 16, then select Equal Radius . Clean up all unneeded parts of the circles with the Delete Line Segment 

18. View Isometric



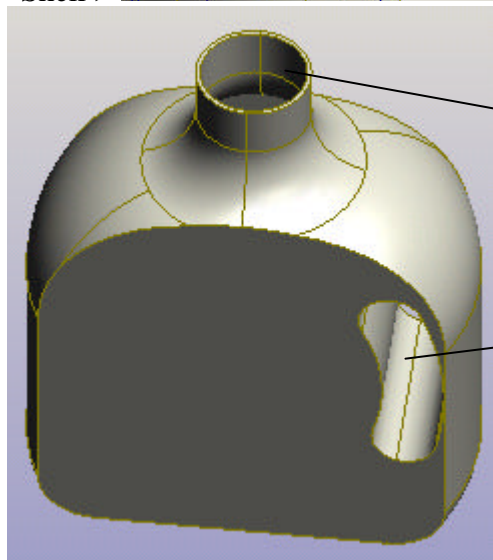
19. Extrude Profile

> OK



20. Select Top Face > Shell >

> OK



Note the "Hollow" inside

Handle goes all the way through.

21. Finished Product

22. Save

Notes:

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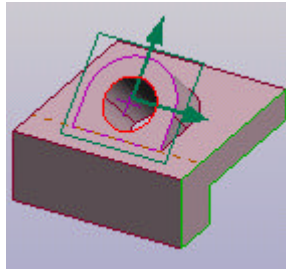
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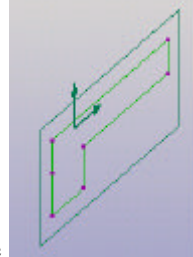
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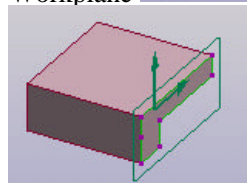
## Machined Part




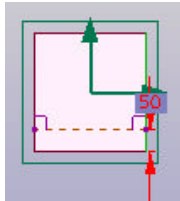
1. Select  the Lateral Workplane > Right Mouse Click > select New Sketch.



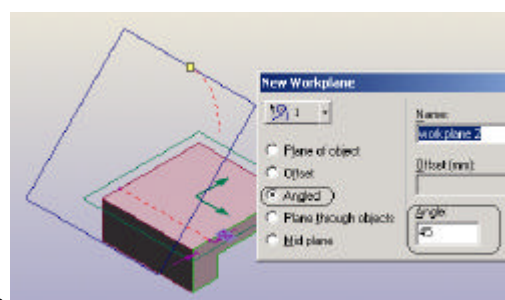
2. Draw some type of shape on the Workplane



3. Extrude Profile > to some width > OK
4. Select Top Face & RMC > select New Sketch > rename Profile > OK
5. View Onto Workplane 
6. Draw a Line & Dimension it 50 away from bottom face > RMC > Toggle Construction

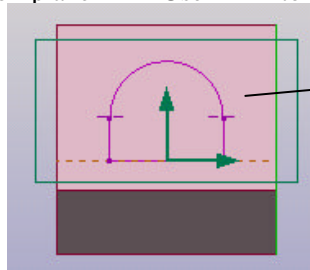


7. View Isometric



8. Workplane > New Workplane > OK
9. RMC Workplane > select New Sketch > rename Profile > OK

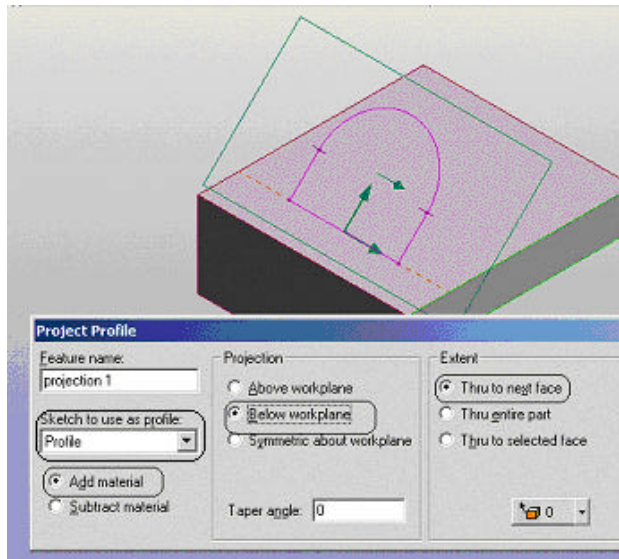
10. View Onto Workplane  Use  to see lines.



Use a Rectangle & a circle  
Check for Tangency Remove  
any unneeded lines/curves

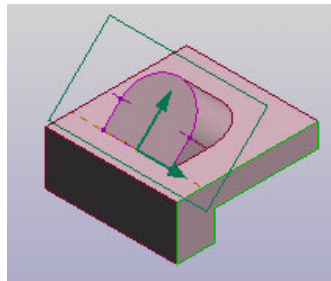
11. Draw a profile

## 12. View Isometric



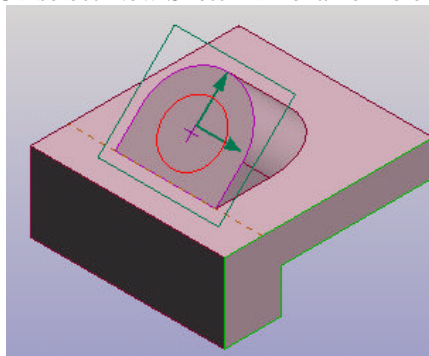
## 13. Project Profile &gt;

&gt; OK

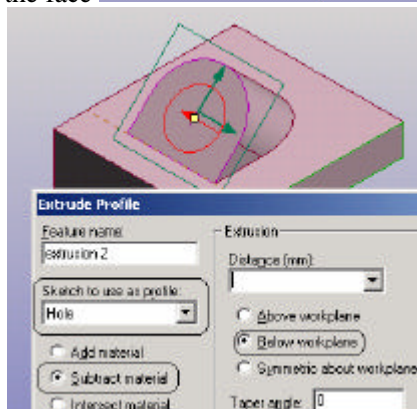


## 14. Looks like this

## 15. Select Profile Face &gt; RMC &gt; select New Sketch &gt; rename Hole &gt; OK



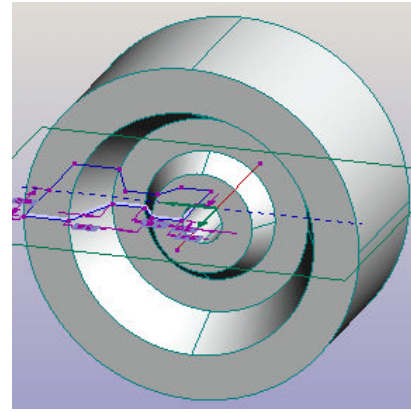
## 16. Draw a Circle on the face



## 17. Extrude Profile &gt;

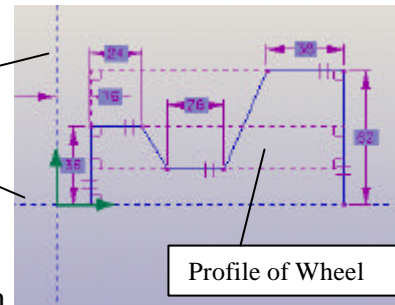
&gt; OK

## 18. Save as Machined Part

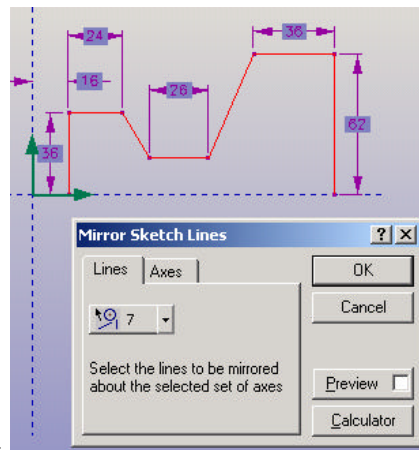


## Mirror & Revolve Profile of Wheel:

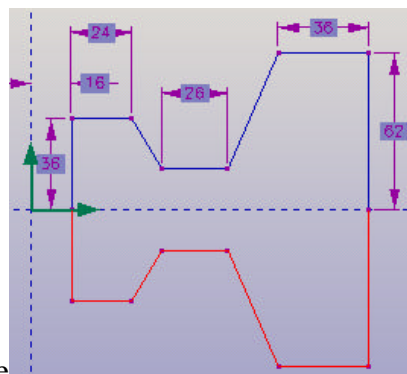
Control Lines  
Toggle Construction



1. Draw two control lines & toggle construction



2. Line > Mirror > select the lines Shift + LMC > select the Axes > OK

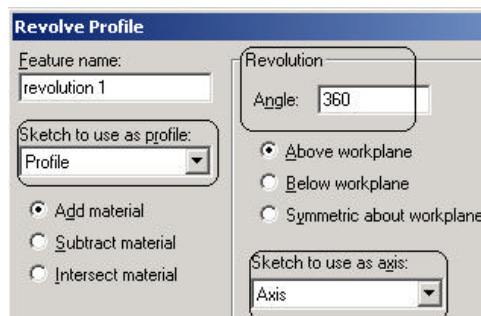
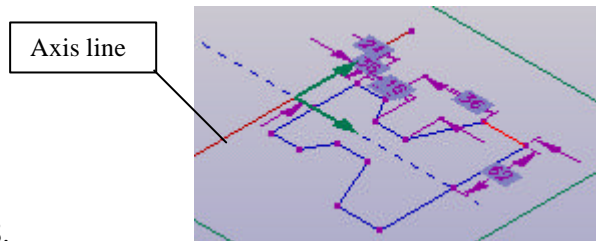


3. Profile is complete



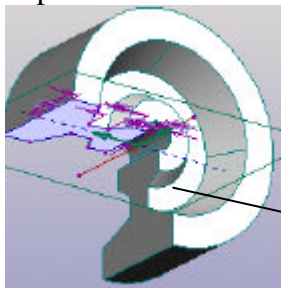
4. Line > Toggle Sketch Filled ensures that sketch is closed

5. View Isometric
6. Workplane > New Sketch > rename Axis > OK
7. Draw a Line through control line




9. Feature > Revolve > Profile > > OK

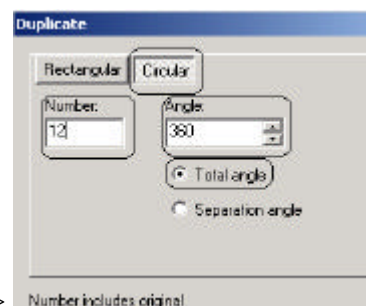
10. A partial “section” can be done by making the angle 270°



Choose this face >  
RMC > New Sketch >  
rename Slots

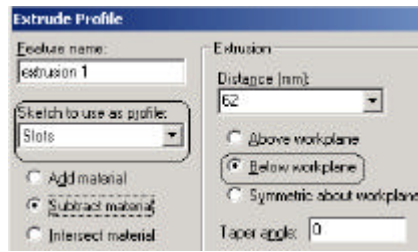
11. Select face as above 

12. View Onto Workplane

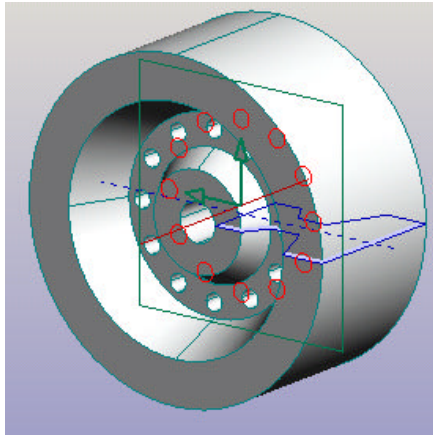


13. Draw a Circle > RMC > Duplicate > > OK

14. View Isometric

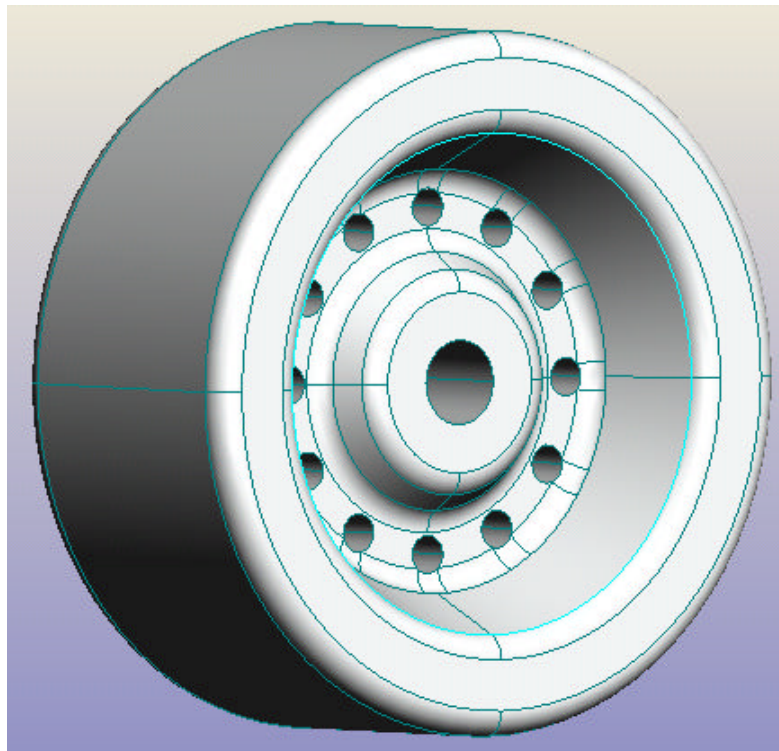


15. Extrude Profile > > OK

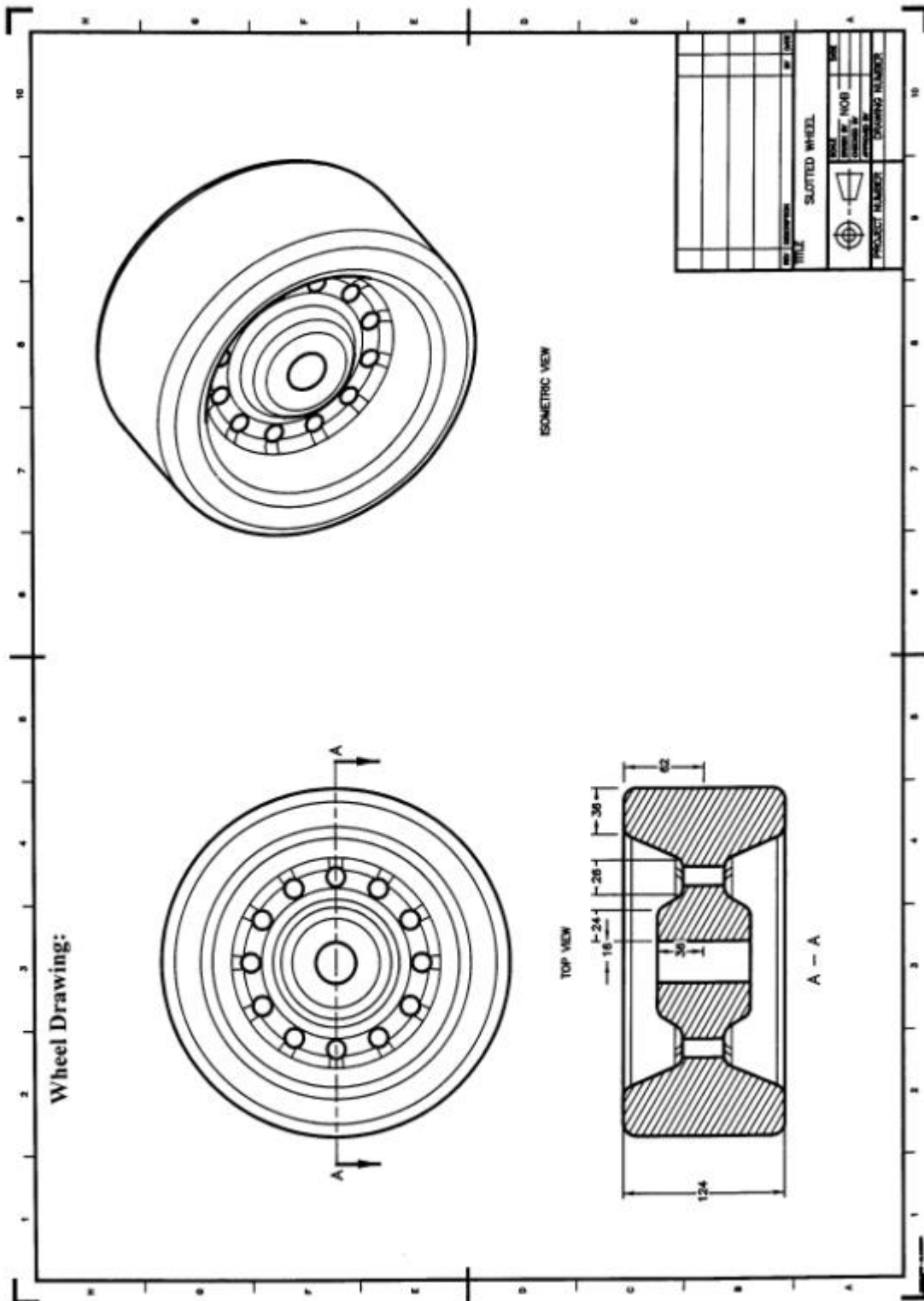


16.

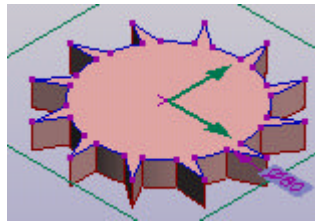
17. Round Edges  > select all edges that have same radius > 10 > OK



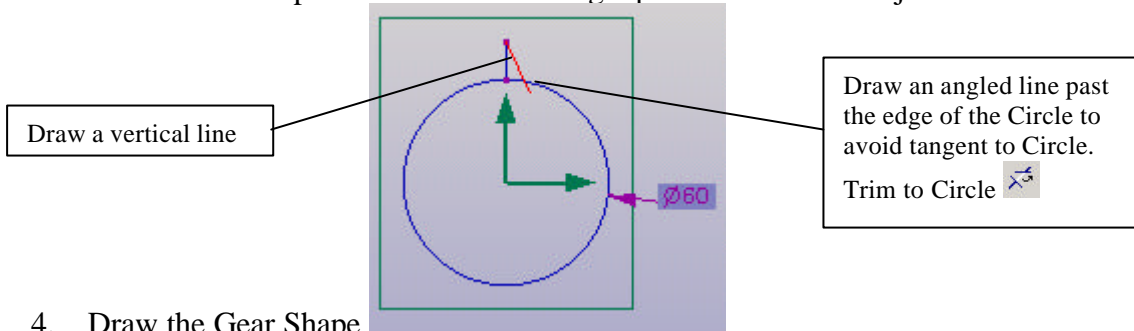
18. Final Product



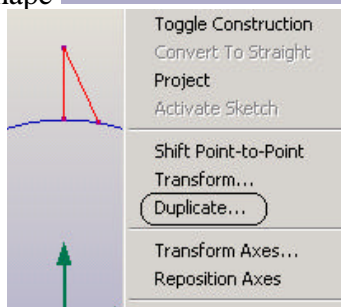
# Gear Simple:



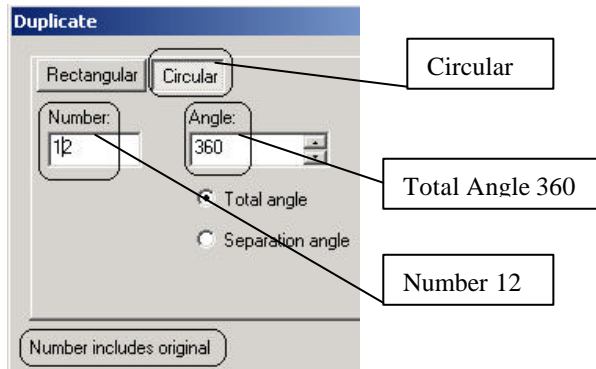
1. Draw a Circle on the Base Workplane & dimension to 60 diameter
2. Use Autoscale to see a reasonable size Design
3. View onto Workplane to view a single plane view of the object



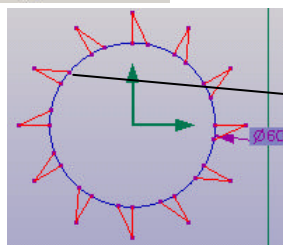
4. Draw the Gear Shape



5. Select Both Lines RMC > Duplicate

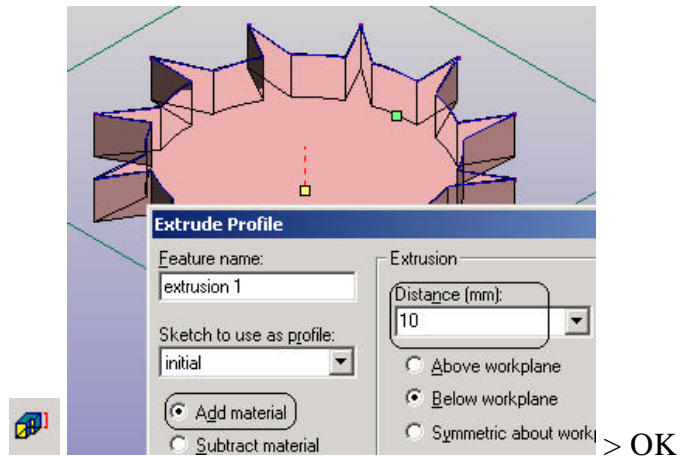


6. > OK

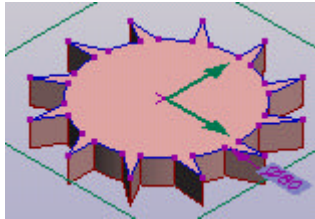


7. Looks like this
8. View Isometric

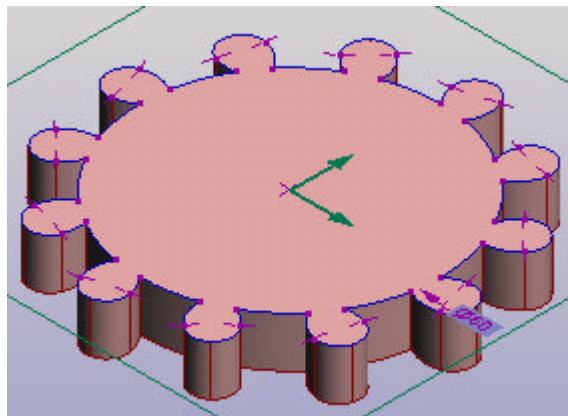
## 9. Extrude Profile



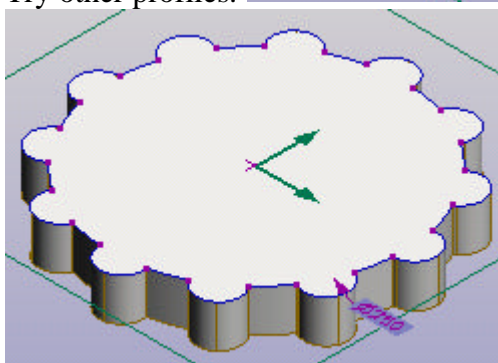
## 10. Final Product



## 11. Save



Try other profiles:



Files for this activity are at [www.intad.asn.au](http://www.intad.asn.au)  
follow to Pro/DESKTOP page

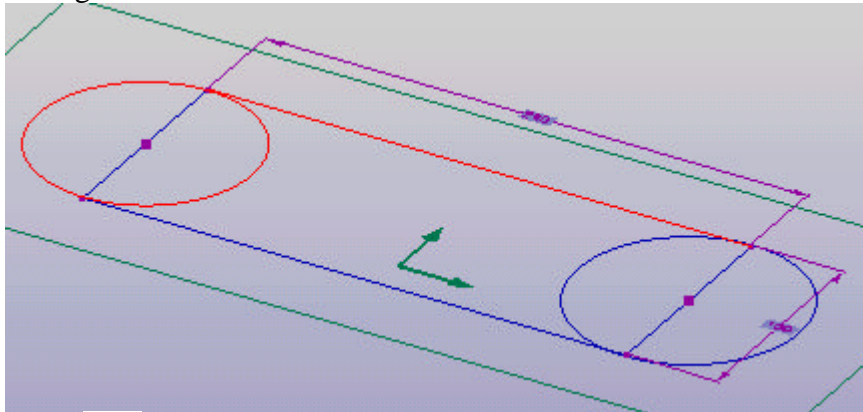
## SPANNER






or

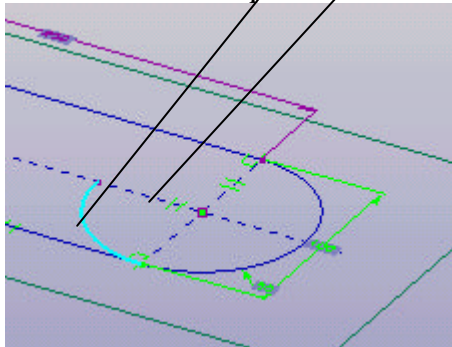


1. Draw a Rectangle & Dimension to 250 X 120. Draw a circle of Diameter 120 at each end.
2. Left Mouse Click (LMC) an edge of the Rectangle & Shift LMC a Circle. Both go red colour.

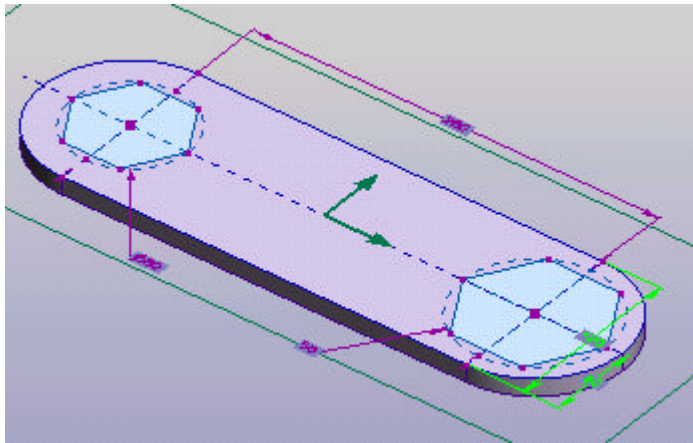


Click  to ensure Tangency. Do the same to each edge & circle.

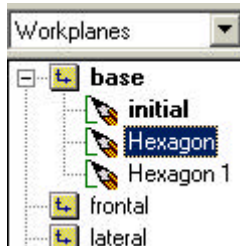
3. Place Lines as below & Toggle Construction (Ctrl G) then Fix 
4. Remove unneeded parts of the Circles using 



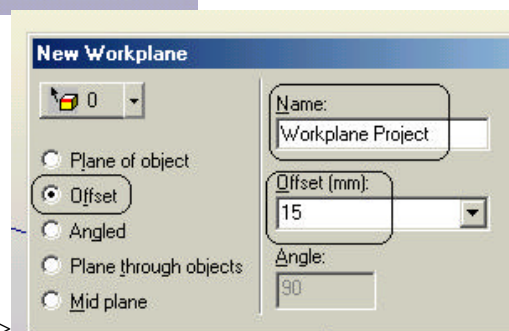
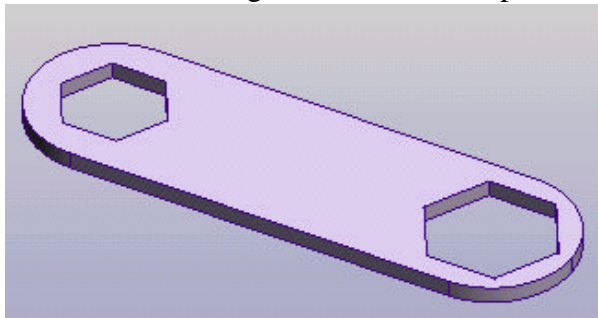
5. Extrude the object Below Workplane > 12 > OK
6. If the Palette is not turned on, goto Tools > Palette
7. LMC & Drag the Hexagonal shape from Shapes tab of the Palette. Drop the centre point onto the centre point of the end curves.



Note: Each Hexagon is a closed Sketch & places a Sketch for itself on the respective Workplane.

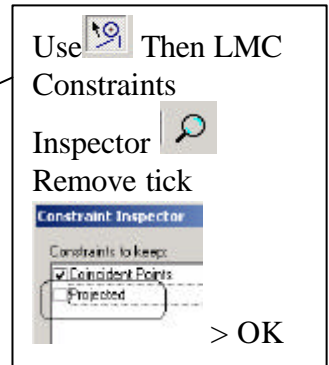
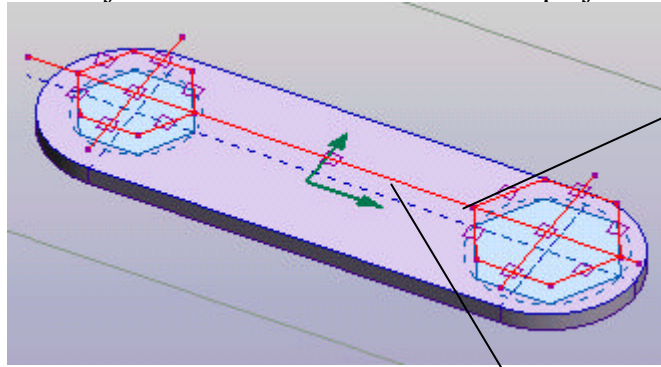


8. Dimension one of the Hexagonal circles to 40 & the other to 50.
9. Extrude each Hexagon to Below Workplane > Subtract Material > 12 > OK



10. Workplane > New Workplane > OK
11. Right Mouse Click (RMC) > New Sketch > rename Project > OK
12. Carefully choose the lines to Project.

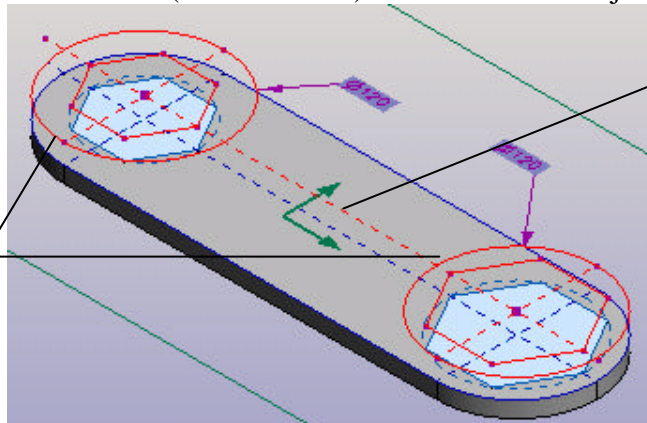
13. Use Line > Project & these selected lines will be projected to the Project



Sketch

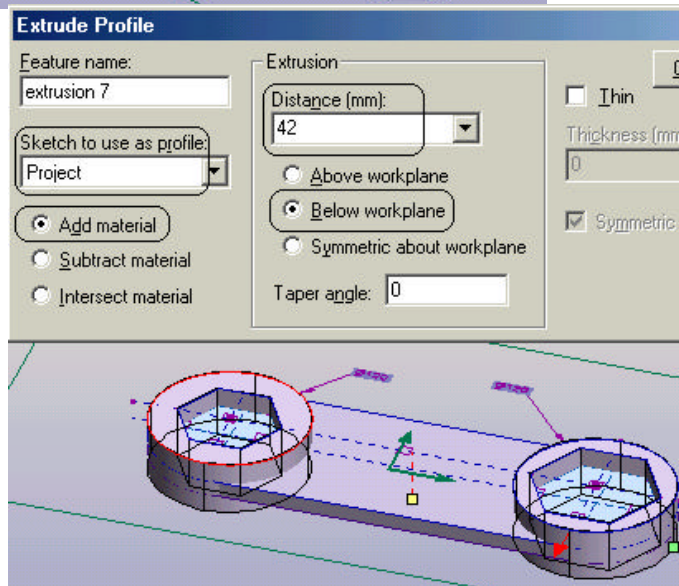
14. Carefully select the projected Construction Lines & return them to Construction Lines

15. Add Circles (Diameter 120) to each end on Project Sketch



Change Lines back to Construction Lines using Toggle Construction

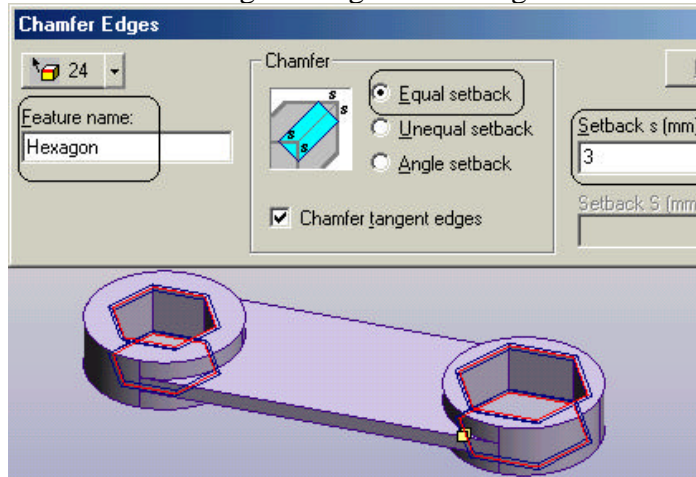
Add Circles  
120 Dia



16. Extrude

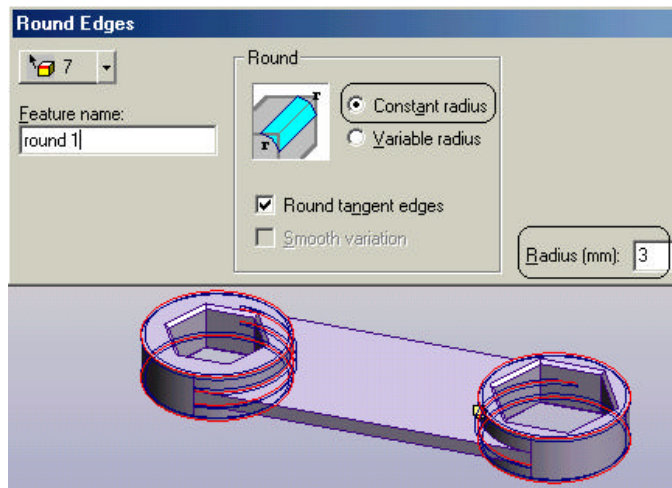
> OK

17. Select ALL Hexagonal edges  using Shift LMC & use Chamfer tool 



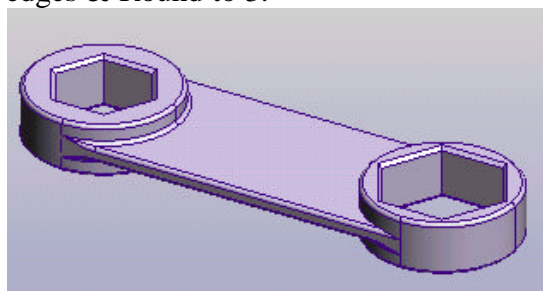
> OK

18. Select ALL curves & Round to 3



> OK

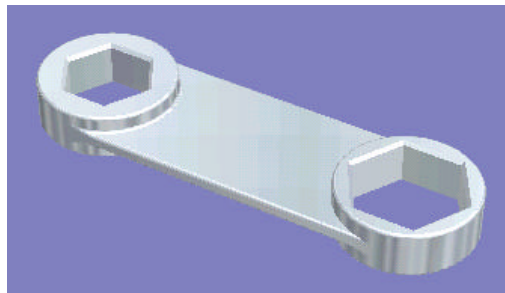
19. Select the long edges & Round to 3.



20. Looks like this

SAVE


21. Place into a New Album & use Chromium Plate, Dull



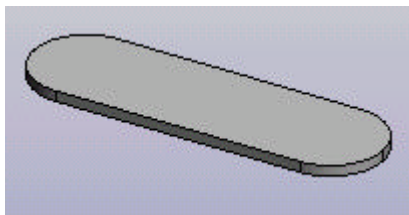
22. So now there is a Spanner.
23. Load up that file & use File > Save Copy as > give it a different file name > OK
24. Close the original file as the Assembly file is based on it & open the new file.



26. LMC & Drag the Finish Flag to below Extrusion 1
- 
- Note that

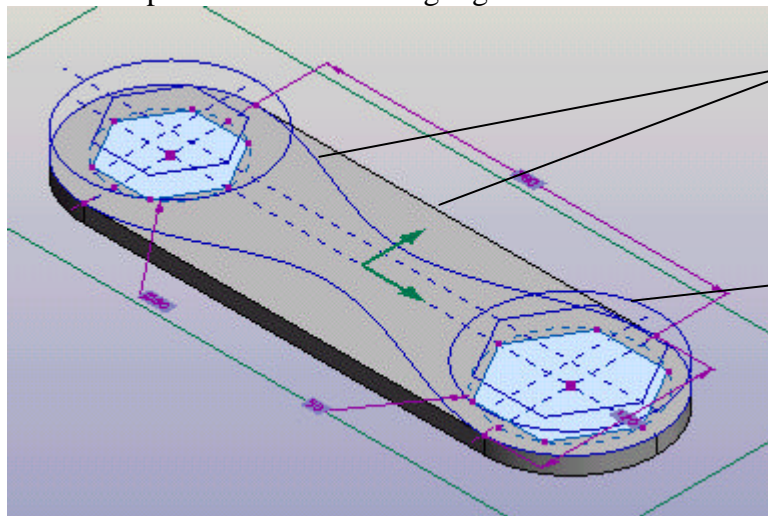
the Update Flag also moves up & the Update (green) Button  is ready to bring the Design to where you want it to be. This is a way of making file sizes smaller for moving larger files. Just open the file up at its new location & move the Finish Flag to the bottom of Features & LMC the Update Icon to

regenerate the design.



Note: Moving the Finish Flag reduces the file size from 407K to 110 K

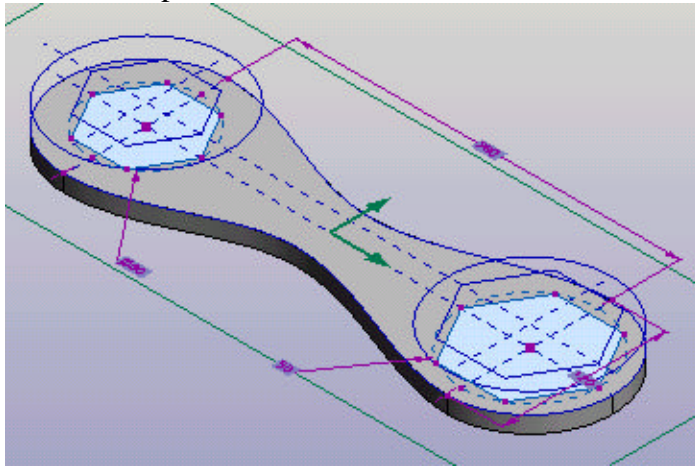
27. LMC the Spline Tool . Prehighlight the Line & move it in to form a curve



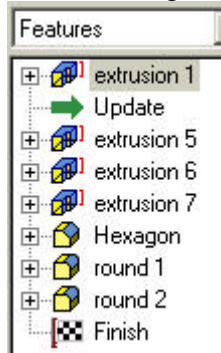
The Spline Tool allows an edge to be dragged into a curve such as these edges.


Notice **ALL** the other geometry of the design is still shown in place, but just not active at present

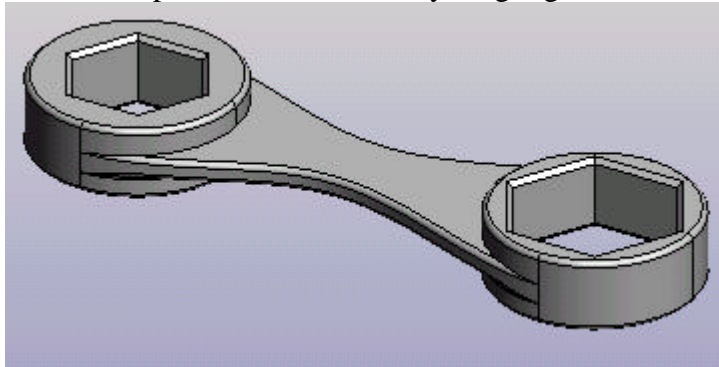
28. Click the Update Icon 



29. LMC & Drag the Finish Flag to the bottom of the Features Browser



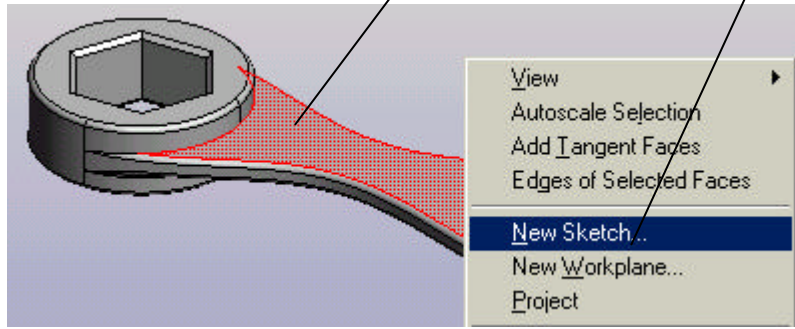
30. LMC the Update Icon  & everything regenerates to the finished article.



31. Save

Select this Face & RMC choose New Sketch > OK

65

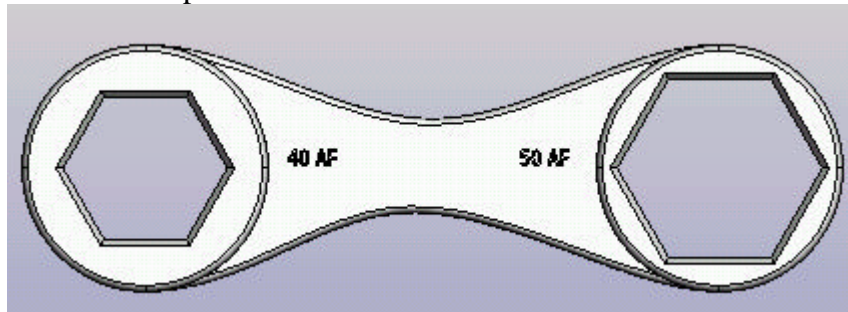


32.


33. Line > Add Text Outline > Size 8 > print 40 AF (8 spaces) 50 AF Use the Preview option to see if the Text is in the correct position > OK

34. LMC & Drag Text to final position. Extrude > Above Workplane > Add Material > size 1 > OK

35. This is the Top View



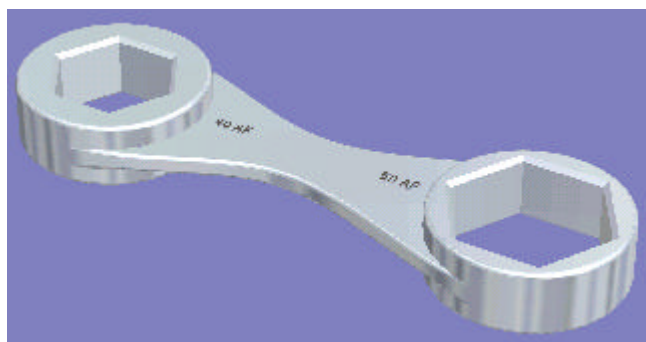
36. Save

37. Go to New Album. View > Zoom In to see letters at a reasonable size. Use the Select Faces  & LMC & Drag a suitable material over to each letter (just



the top face of each one) then Update . View > Autoscale > Update

38.



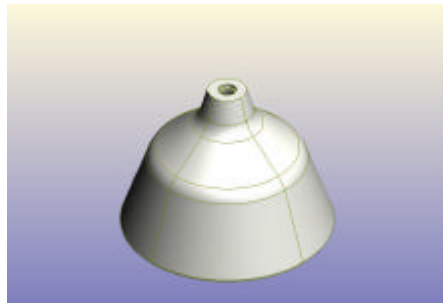
SAVE



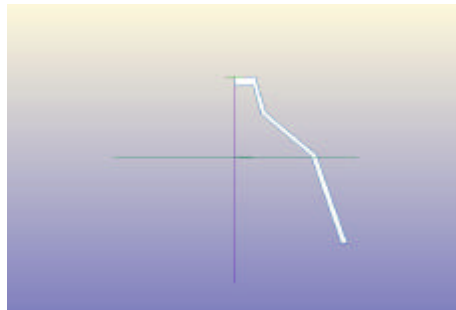


## Reading Lamp

This consists of three design files. Whilst the three parts, as described here, are not very detailed, other sections of these materials need to be adapted to ensure a correct outcome.



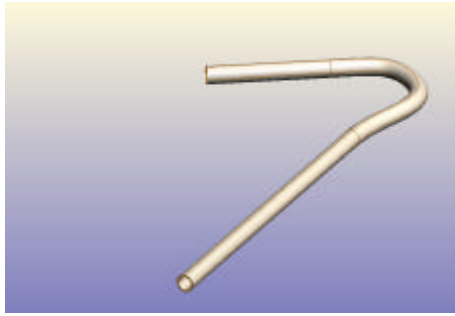
**Shade:**



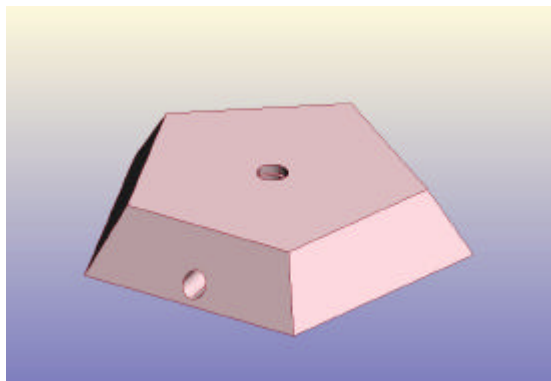
1 Draw & Revolve Profile

2 Select top & place hole (note the hole sizes so that the stem & base can contain matching sizes) in it using the Hole feature .

3 Save.

**Stem:**

1. Use Base Workplane for Path & Frontal Workplane for two Circles the correct size for Profile.
2. Sweep Profile > along Sketch Path
3. Save.

**Base:**

1 Use Tools > Palette > Shapes. Click & drag to drawing area. Note that the Palette items already have a Sketch included in them (have a look at the Workplane Browser for Sketch name) The Palette items will mate with the active Workplane.

2 Extrude & (maybe) taper sides

3 Place correct size hole  for Stem in base

4 Save

**Assembly:**

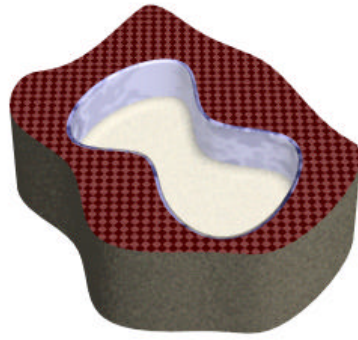
1 Assemble the Base (fixed) & Stem using Align Centres & Mate the matching bottom of the holes.

2 Repeat with the Stem & Shade.

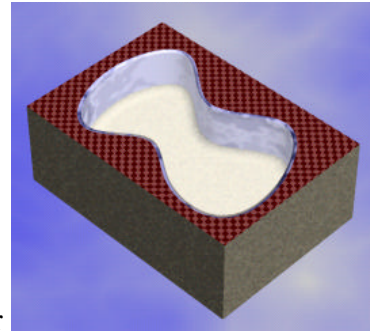
3 Save

4 Open Assembly in the Album area & proceed to have fun.

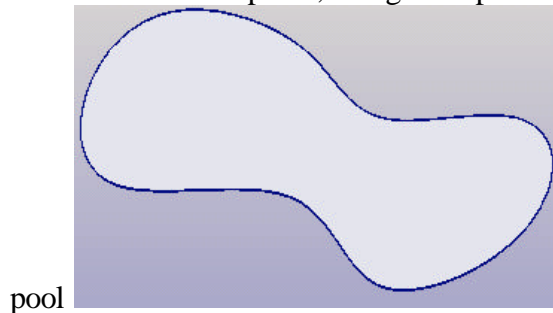
## Pool Freeform



or

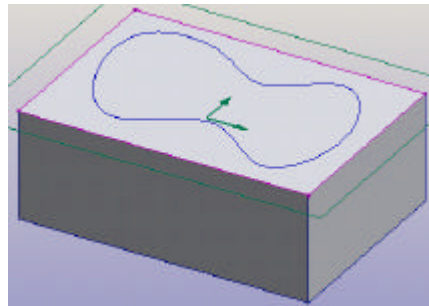


1. On the Base Workplane, using the Spline Tool  to draw the outline of the



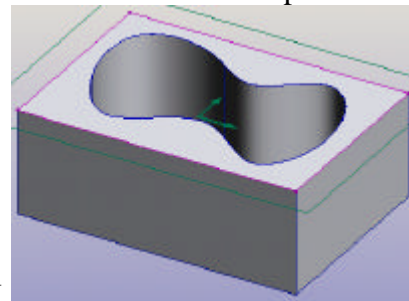
Files for this activity are at  
[www.intad.asn.au](http://www.intad.asn.au) follow to  
 Pro/DESKTOP page

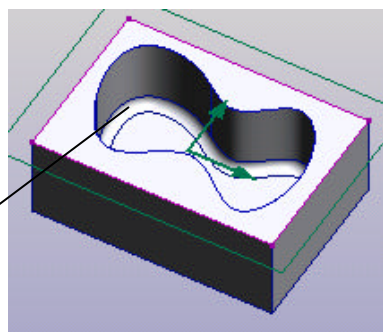
2. Save it as Pool. Use Save Copy as > Pool Insert
3. Place a New Sketch onto the Base Workplane. Using the Spline or Rectangle Tool, draw an outline outside of the Pool outline & Extrude it to a suitable



4. Go to the Initial Sketch & Extrude the Pool outline to a suitable depth >

Subtract Material > Below Workplane > OK






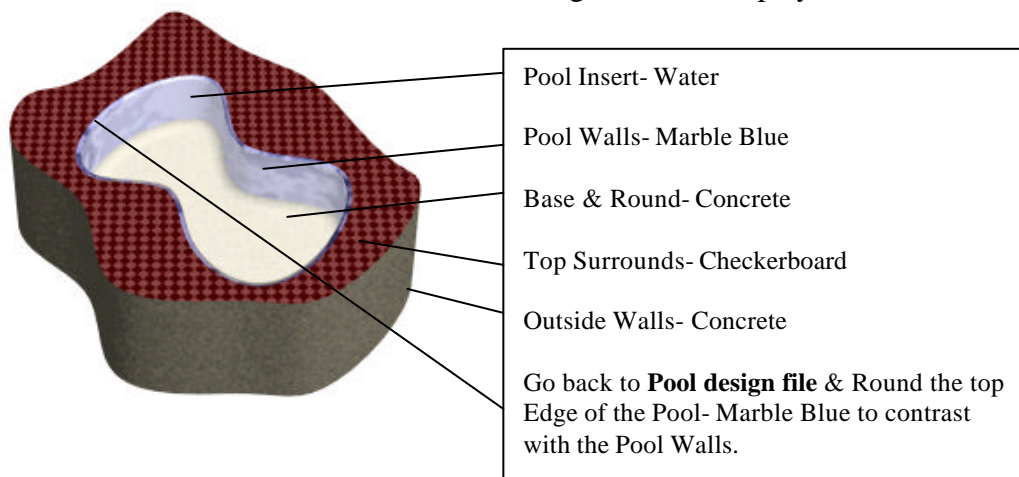
5. Select & use Round Edges to round the base
6. Save as Pool
7. Open Pool Insert file & Extrude > Below Workplane > 2 > OK
8. Save

### Assembly:

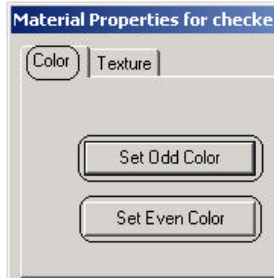
1. New Design
2. Assembly > Add Component > Pool > OK
3. Assembly > Fix Component
4. Assembly > Add > Component > Pool Insert > OK. Note: That as the Pool outline was saved in both files from the original outline the Pool Insert fits neatly into the Pool hole.
5. Select the Top Faces & RMC > Align
6. Save as Pool Assembly.

### Album:

1. New Album > OK
2. Image > New Image > Pool Assembly > OK
3. Use Faces  & Materials Browser to change material display on each face

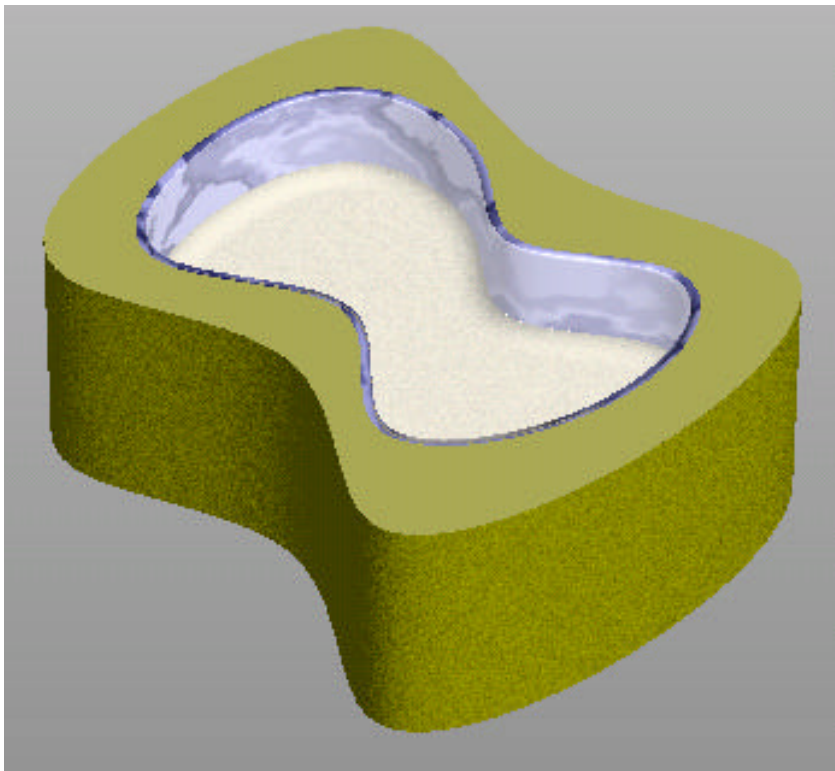


- The Checkerboard colours are changed by double LMC on Face &



Set Texture to 0.2 > OK

- Update
- Save as Pool Album



Notes:

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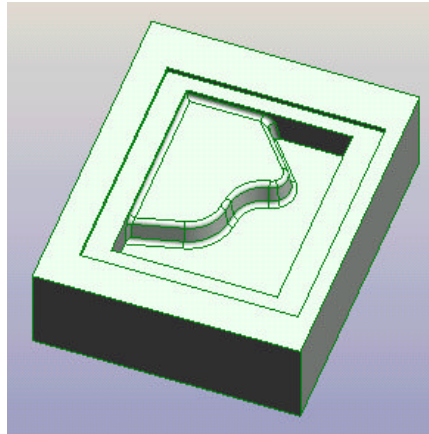
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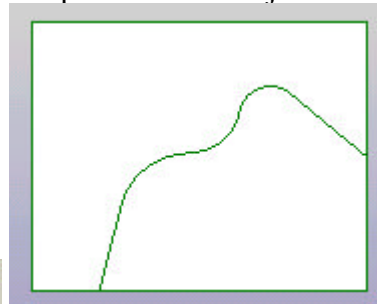
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Files for this activity are at [www.intad.asn.au](http://www.intad.asn.au) follow to Pro/DESKTOP page

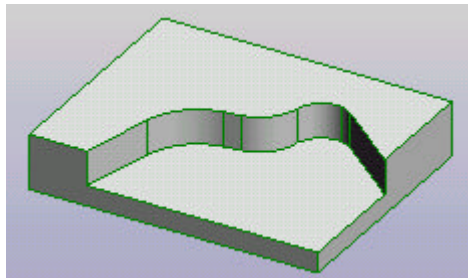
## MOULD

1. Draw a rectangle & extrude it.
2. Switch to the Plan View & construct the profile of the object. Curve the



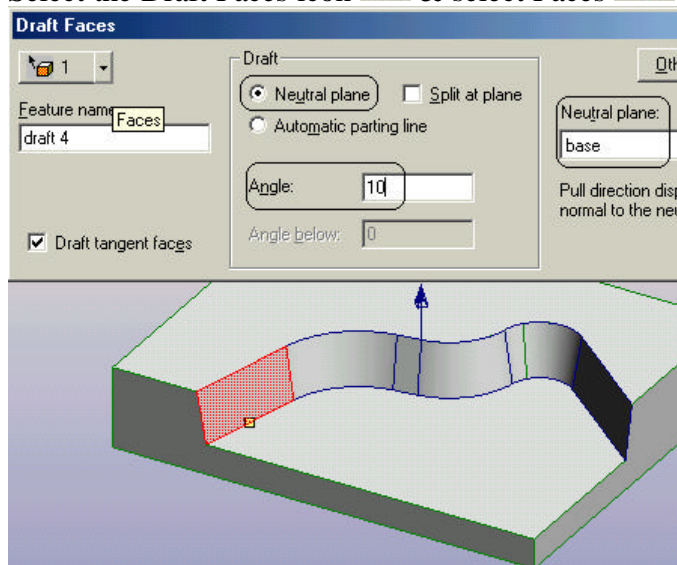
corners using the Arc/Fillet tool 

3. Switch to Isometric View & extrude to a suitable depth (refer to your



requirements)

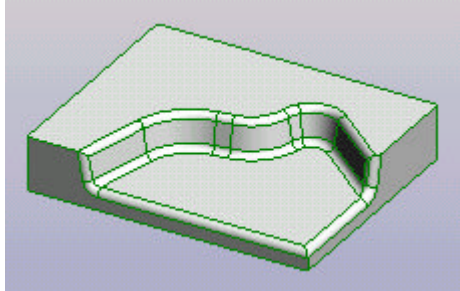
4. Select the Draft Faces icon  & select Faces 





> OK. The angle is to

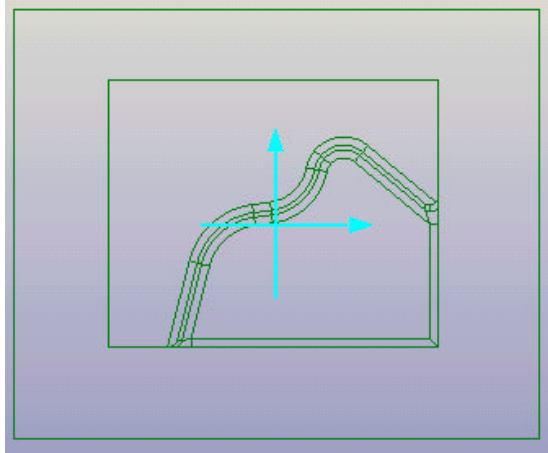
allow easy removal of the part from the mould after the forming operation is finished, so an angle of  $10^\circ$  is rather large but is used here so that the angle can be clearly seen.

5. Select the edges & Round them to 5 mm radius as shown




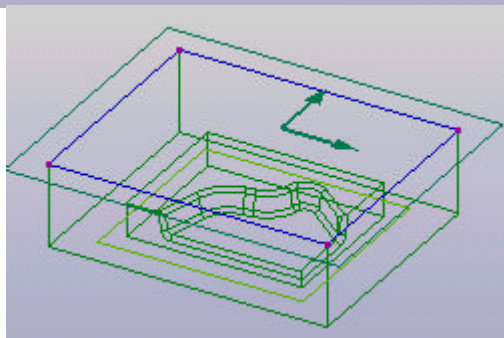
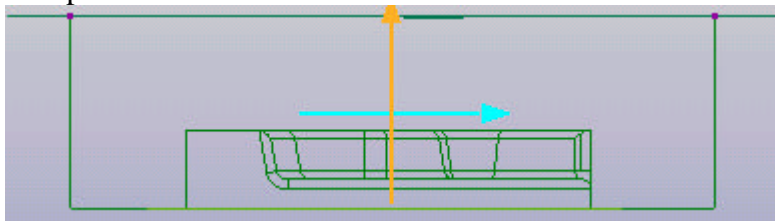
6. SAVE as Mould
7. Start a New Design file
8. Draw a rectangular block into which the original mould will fit
9. Assembly > Add Component > select Mould > OK


10. Use the Wire Frame to view both objects 
11. Use the Plan View & the Select Parts tool  to move the Mould Component

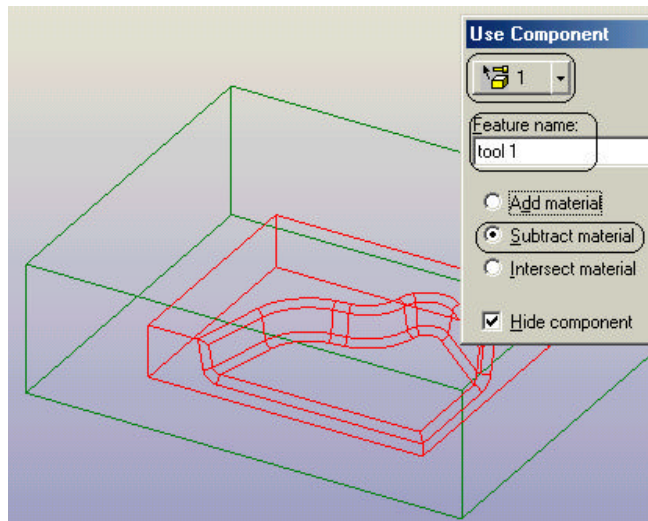


into the middle of the Block



12. Use the Front View & the Select Parts tool  to ensure that the Mould Component is level with the bottom of the Block

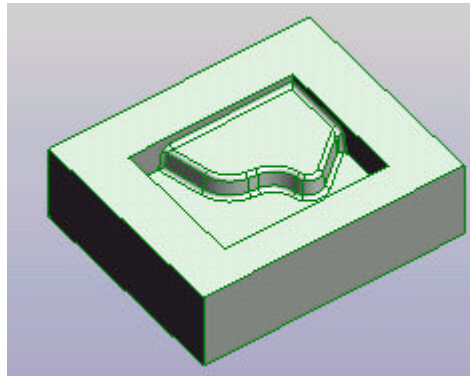


13. View Isometric
14. Feature > Use Component 

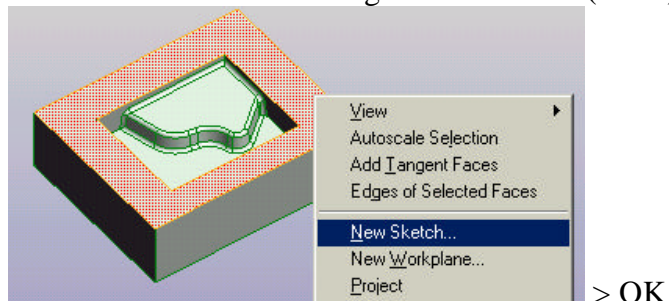


15. > OK

16. Use the Enhanced icon  & Manipulate scene  to view the bottom face

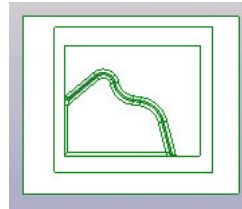


17. Select the bottom face & Right Mouse Click (RMC)



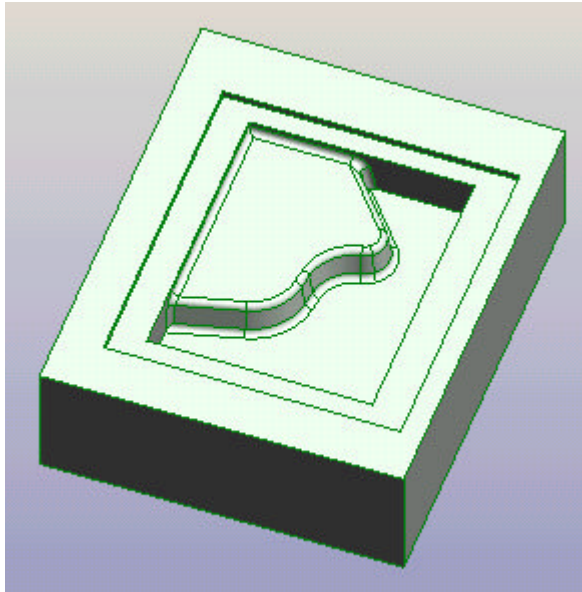
> OK

18. Use View onto Workplane to view onto that face & draw a Rectangle so that it



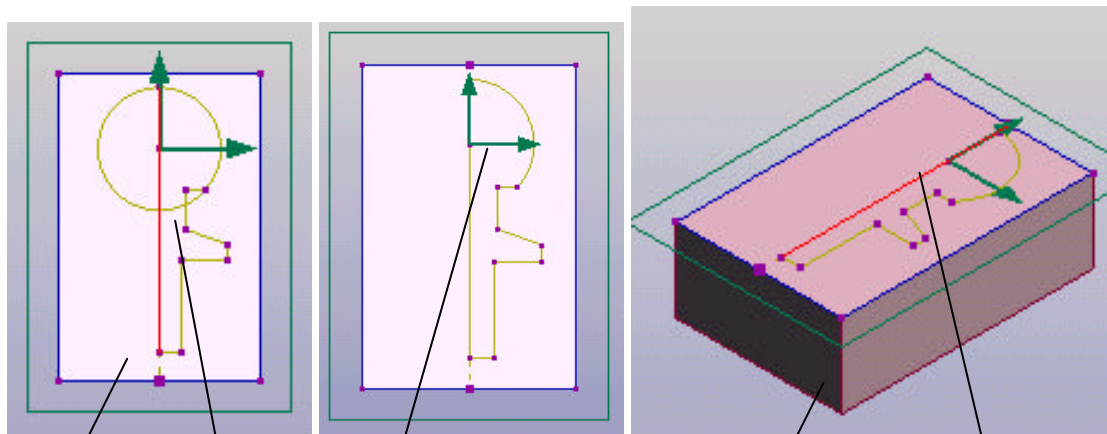
is symmetrical about the Mould Component.

19. View Isometric & extrude that Rectangle > Below Workplane > Subtract Material > 5 > OK



- 20.
21. Before use there are several other edges that would need Rounding.
22. Further processing such as holes for the vacuum action to take place & pins/holes for a “cap” to fit over the mould.
23. A male mould could be made so that a different method of manufacturing could be used.

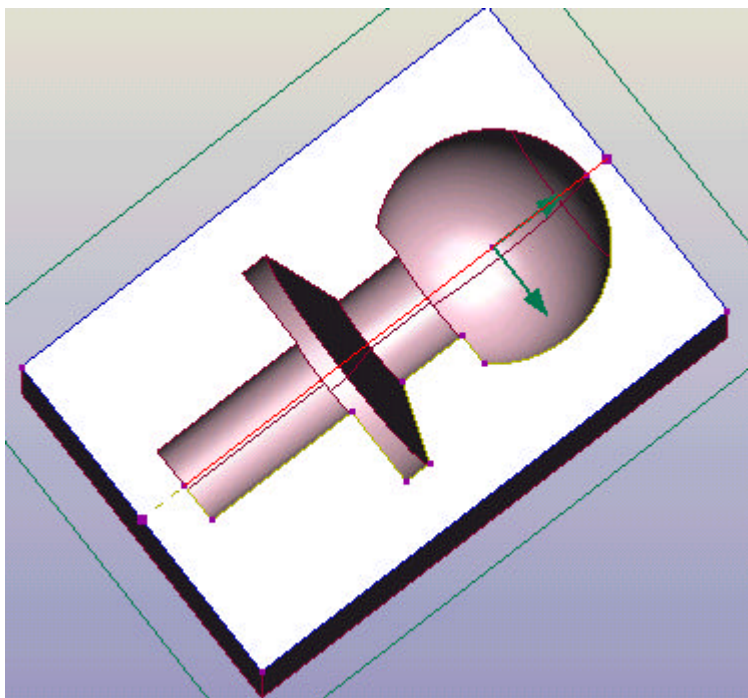
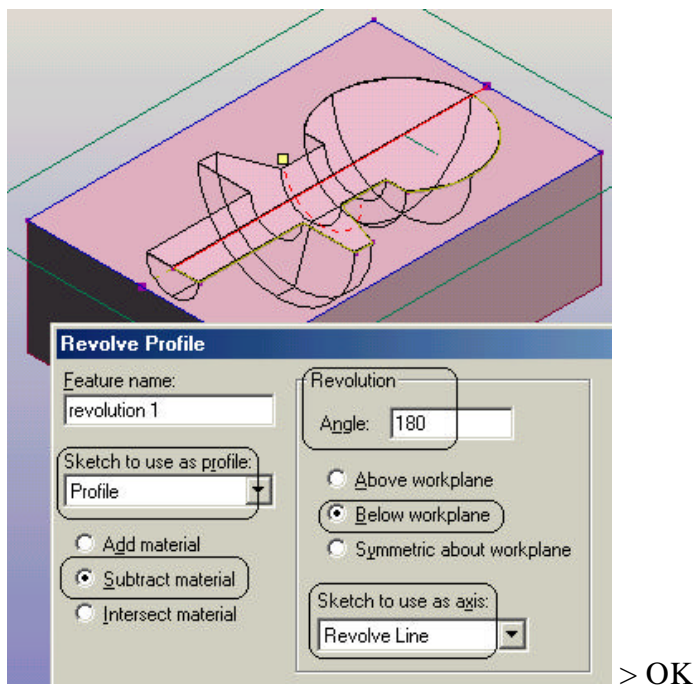
## Using Revolve Profile to form half a Mould



Draw & Extrude a Rectangular Block on Base Workplane  
 New Sketch (Profile)  
 Draw the Profile on Profile Sketch

Use  to clean up the Profile

View Isometric  
 New Sketch (Revolve Line)  
 Draw a Line as the revolving Line



The Half Mould (Towball)

**Notes:**

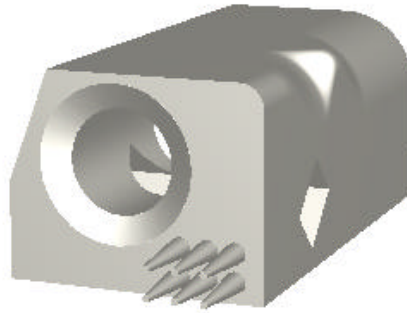
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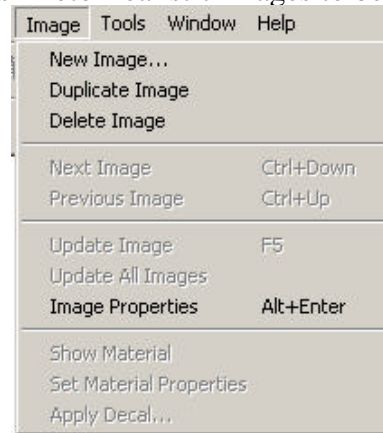
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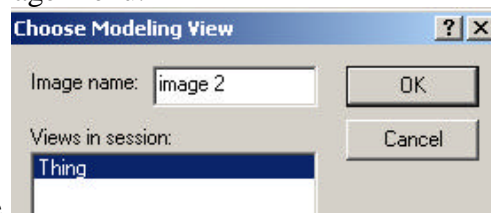
## Album:



1. Shown is a Thing
2. The Album area allows Photo Realistic images to be coloured, duplicated,



using the Image menu.



3. New Image

## Note:

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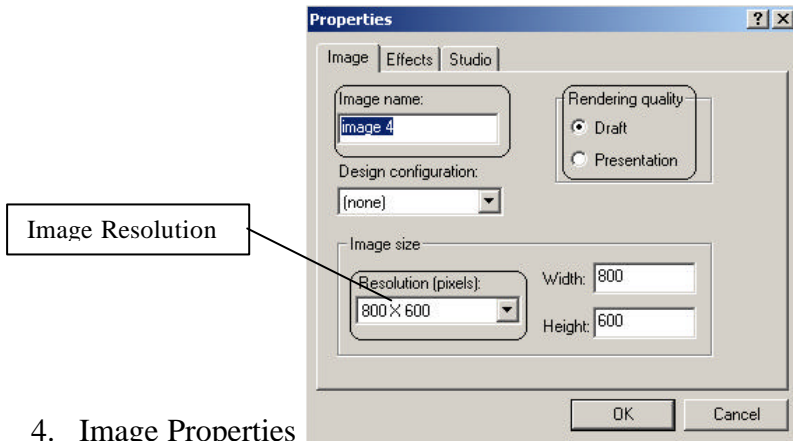
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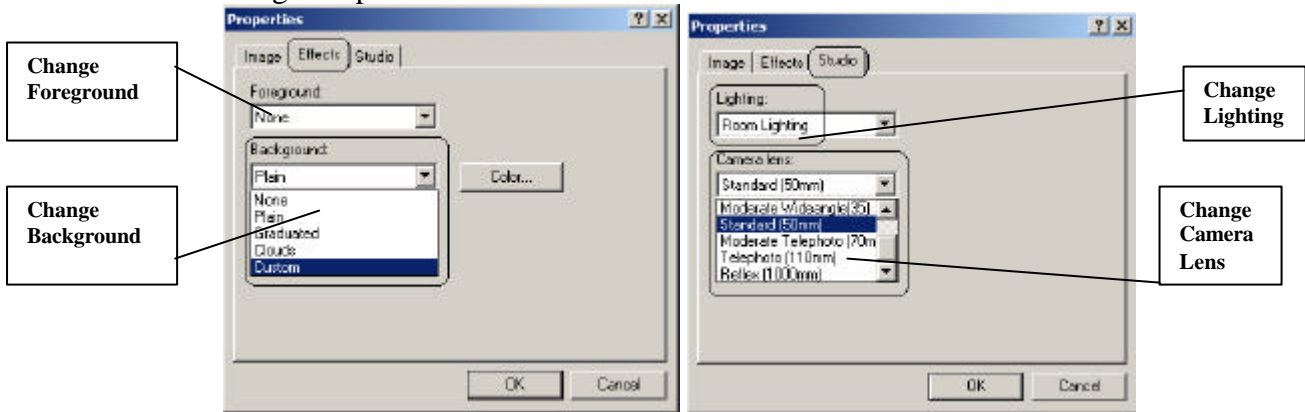
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4. Image Properties



5. Change items to see what effect they have in the Album.

Notes:

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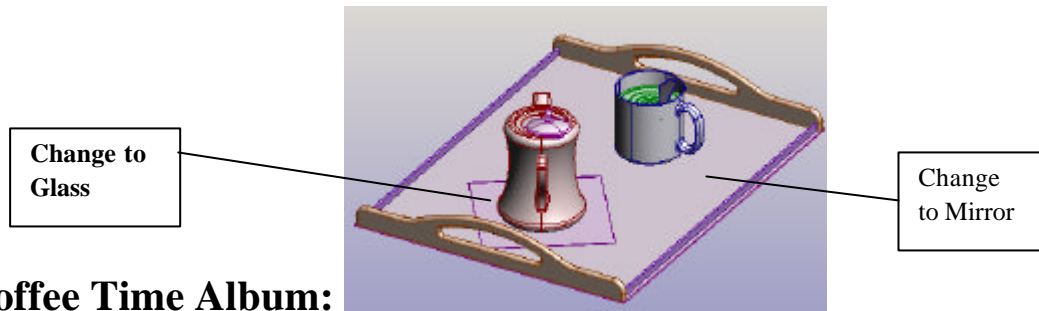
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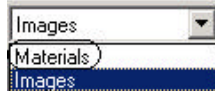
### Coffee Time Album:

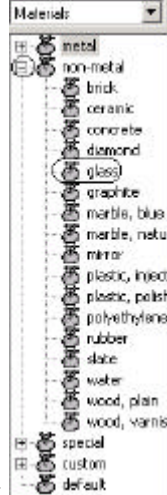
1. This is one of the sample drawing provided with Pro/DESKTOP and is used in the Tutorials in the Help menus.

Usually **C:\Program Files\PTC\Prodesktop2000i\Samples**

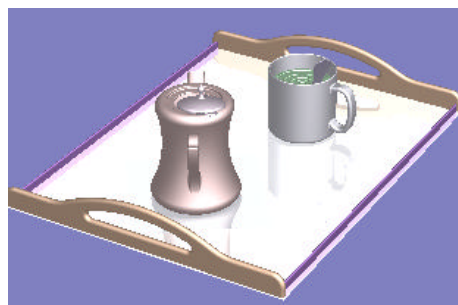
2. File > New > Album > OK

3. Image > New Image > choose Coffee Time > OK > Update 

4. Goto > 

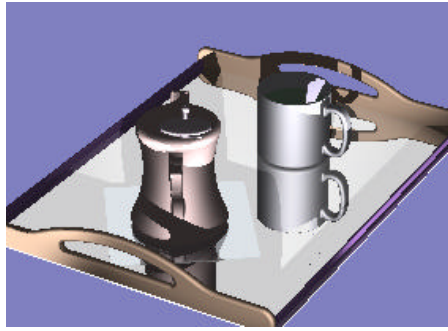
5. non-metal > + > - >  > LMC & drag the Mirror Bag to the Part until the correct Wire Frame appears > then let the mouse key go

6. Change the Base to Mirror & the Tile to Glass > Update



7. Note the Shadows

8. Change the Lighting &/or the Camera in Image > Image Properties & see what other different Shadows that can be produced.



9. Floodlights

10. Enjoy

Notes: \_\_\_\_\_

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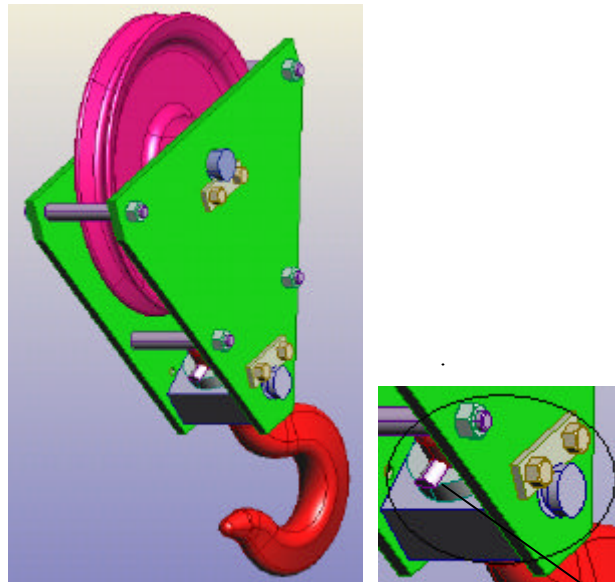
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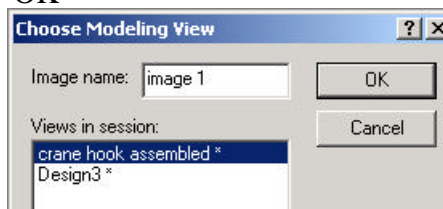
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## Crane Hook Album:



Much of this detail is hidden in a normal view.

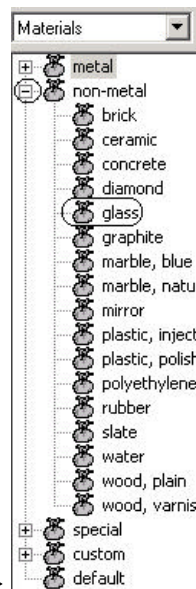
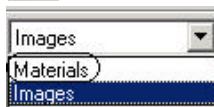
1. File > New > Album > OK



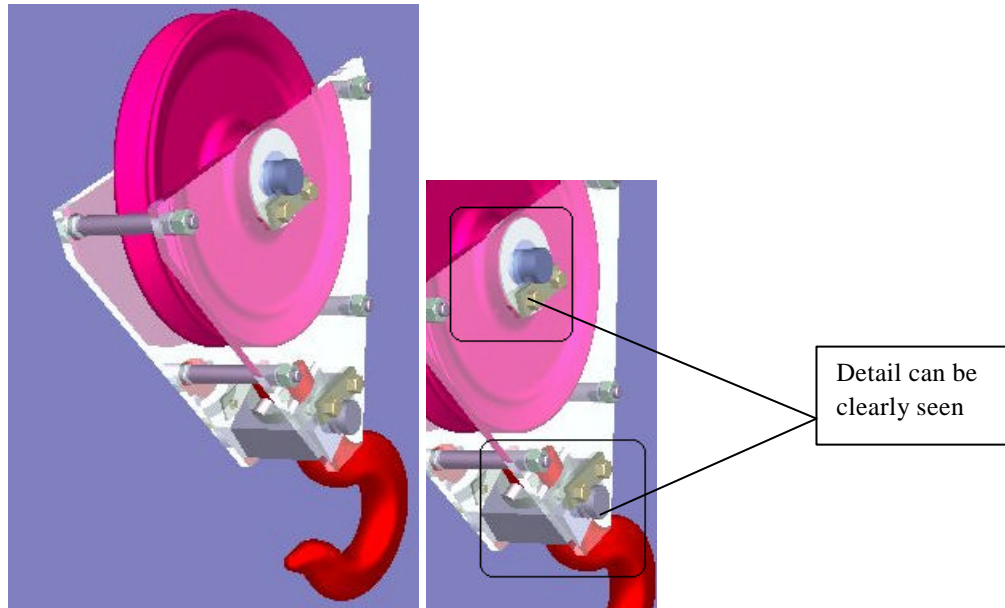
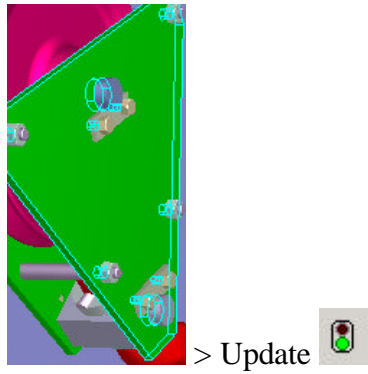
2. Image > New Image >

3. Update 

4. Goto >

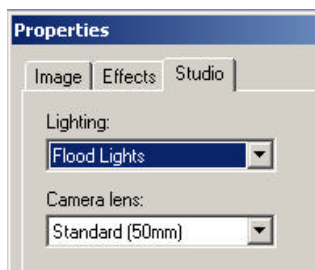


5. non-metal > + > - > > LMC & drag the Glass Bag to the Part until the correct Wire Frame appears > then let the mouse key go



6.

7. Notice what Image &gt; Image Properties &gt; Studio &gt; Lighting &gt; Floodlights

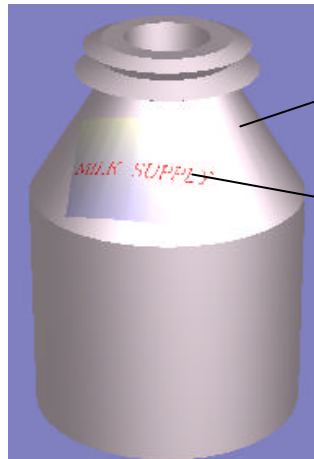


&gt; OK does to Crane Hook



8. Save as Crane Hook Album

## Logo Decal

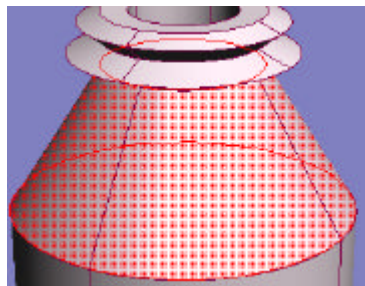


- Finish the Material choice & Update before Select Faces

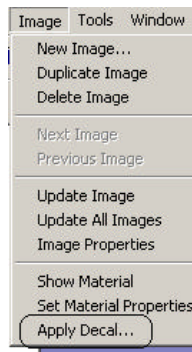
- This Decal Bitmap is an **EXPORTED** bitmap Pro/DESKTOP Design. Other programs could allow more artist freedom in the design & saving options.

Files for this activity are at [www.intad.asn.au](http://www.intad.asn.au) follow to Pro/DESKTOP page

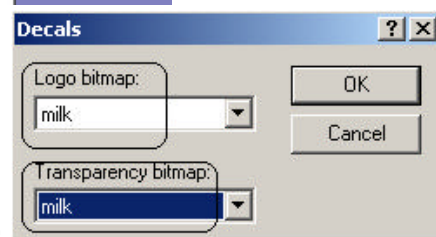
1. Select a Face



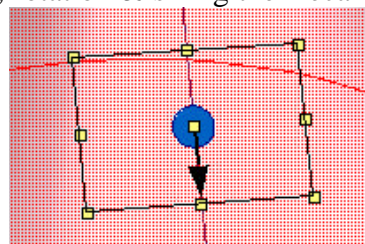
2. Selected Face goes reddish



3. Image > Apply Decal becomes available



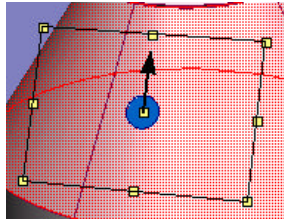
4. Choose the Logo & Transparency Bitmaps > OK (In this case, both Bitmaps were the same)
5. A box appears. By carefully moving, rotation & sizing the Decal box can be



positioned to give the required view.

The Blue circle will allow rotation & positioning. The sizing icon is the same one as for positioning instead of the normal MS Windows double headed arrow.

Sometimes there will be a need to use the (keyboard) arrow keys to identify where the Decal box has placed itself.



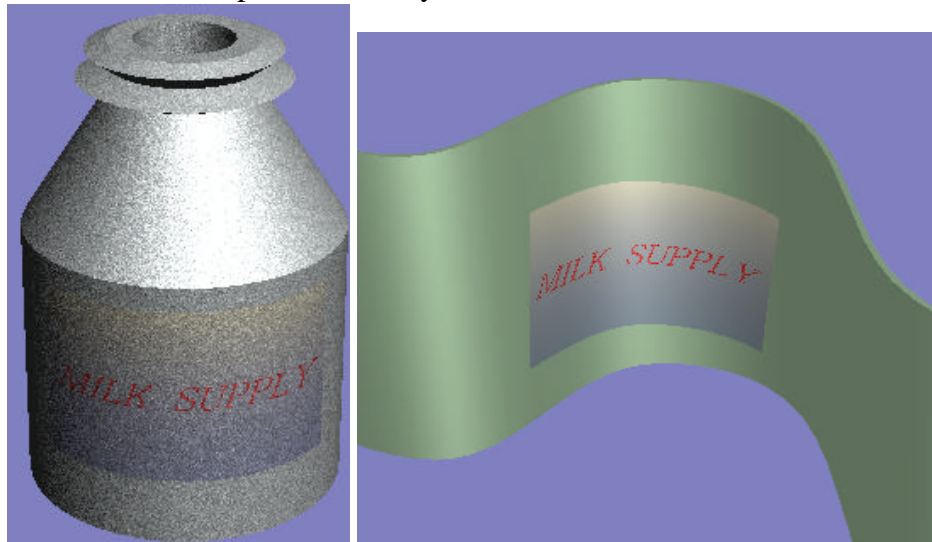
6. Rotated, positioned & resized

7. Use Update to finish 



8.

9. The Decal can be put on a variety of surfaces




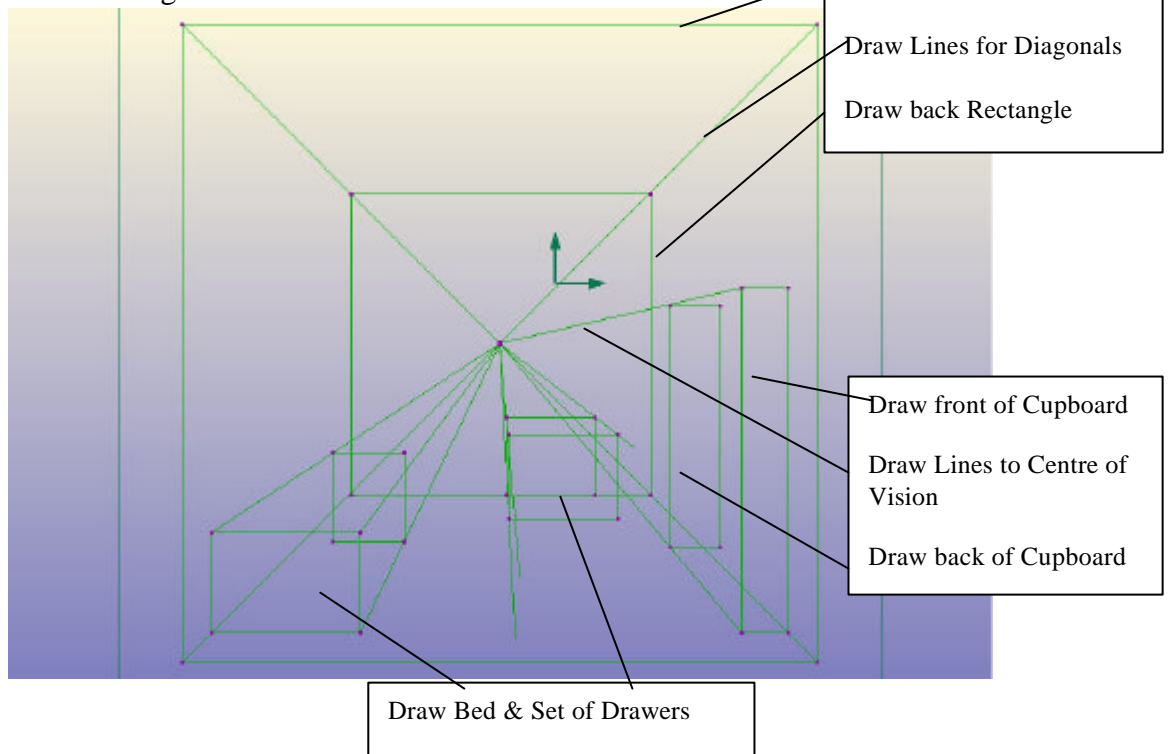
10.


### Extension:

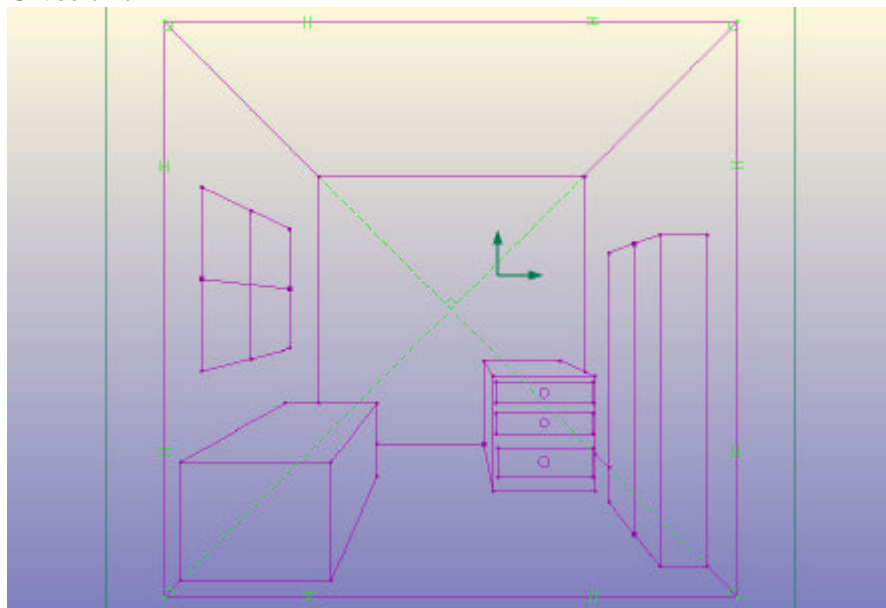
1. Design your own Logo on paper.
2. Use Pro/DESKTOP or another program to do an electronic copy of it.
3. Save it as a .bmp (bitmap) file to the folder:  
C:\Program Files\PTC\ProDESKTOP 2000i2\Program\Bitmaps
4. Pro/DESKTOP does not give an option of looking into any other folder.
5. The other program may allow the saving of a Mask & a Demask file.

## Single Point Perspective Room (sort of):

1. Use Base Workplane & View onto Workplane 
2. Draw a design file as shown



- 3.
4. Remove unneeded Lines 
5. Gives this

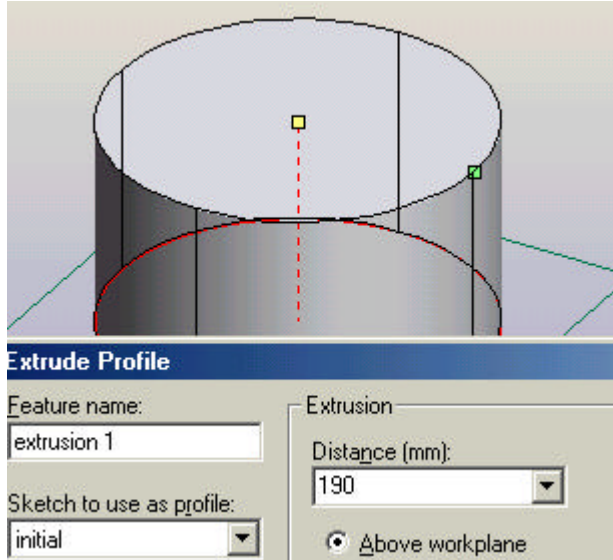




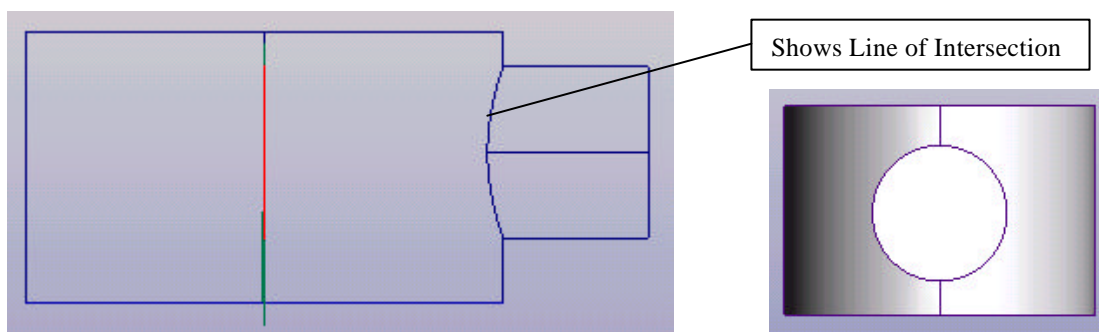
# Solid Geometry / Intersecting Solids

## Cylinder to Cylinder

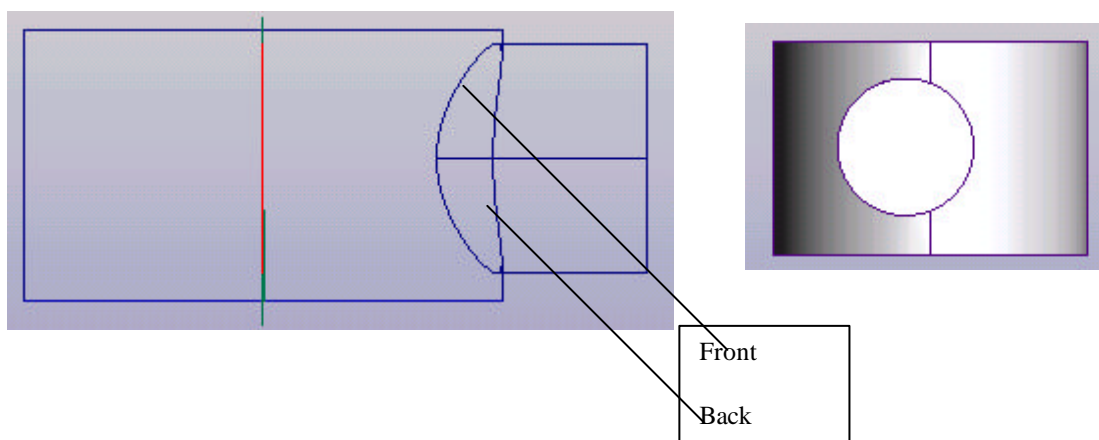
1. Draw a cylinder @ 0,0 Snap to Grid  
(0, 0) on Base Workplane



2. Put New Sketch on Frontal Workplane > Cylinder Horz > OK  
3. Draw Circle on center line of Cylinder & extrude. View Wire Frame

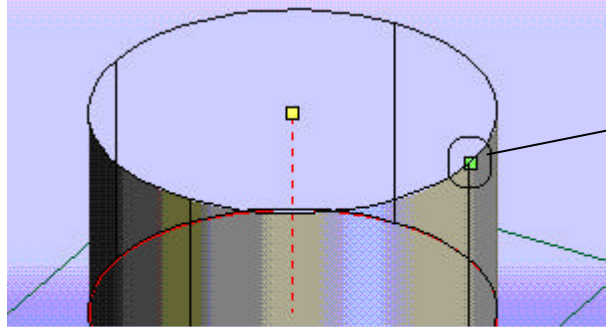


4. Move Horizontal Cylinder to show Line of Intersection of offset centers.



## Right Cylinder to Right Cone

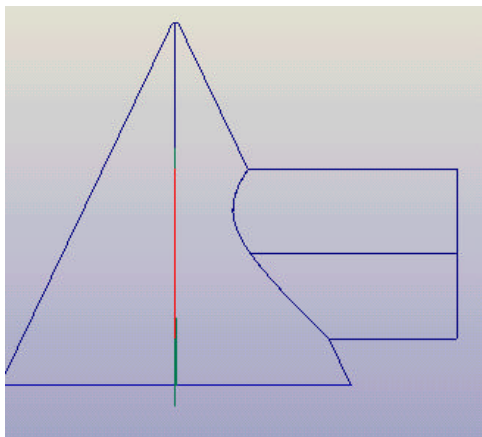
1. Reuse the original Cylinder & turn it into a Cone.



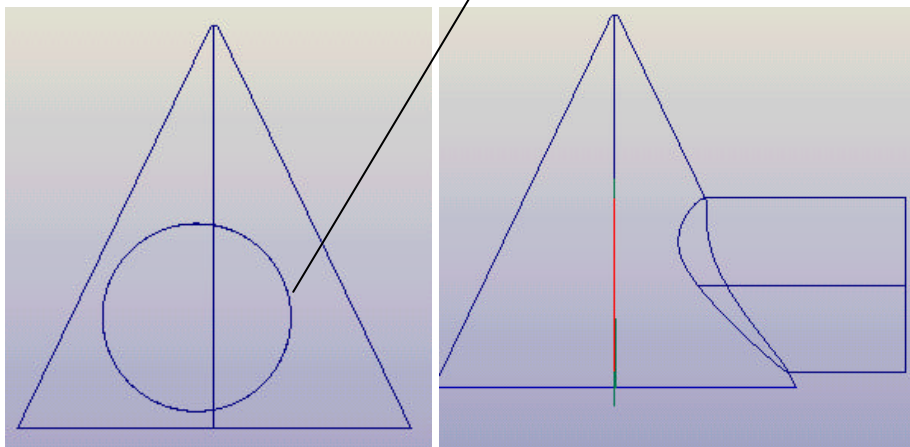
Drag Green Button inwards for Cone to form.



2. Update & Wire Frame shows students Line of Intersection between Cone & Cylinder

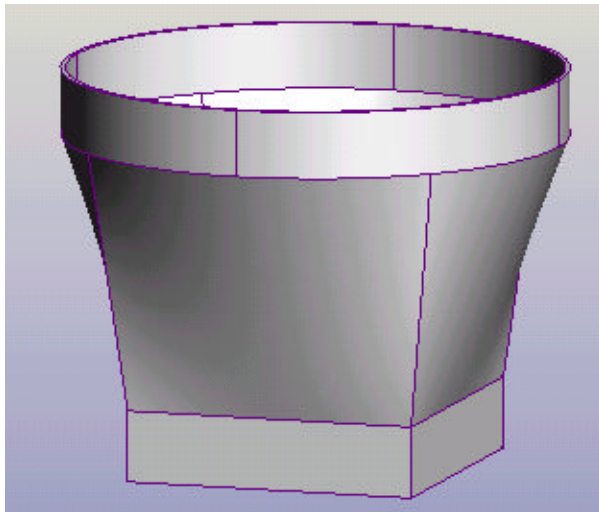


Offset the Cylinder & Update

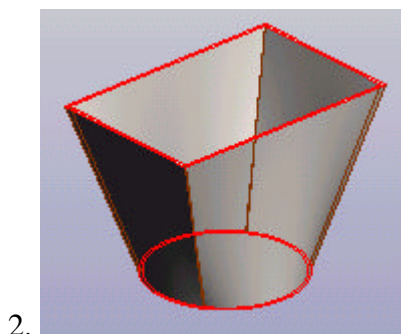


# Transition Pieces

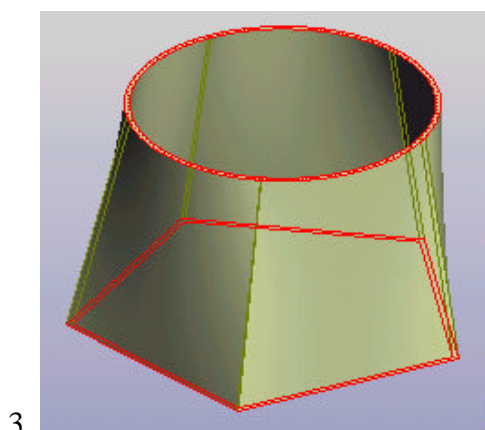
1. Use Features > Loft through Profiles to show Transition Pieces when developing using Triangulation.



Square to Round



Round to Rectangular



Pentagon to Round

## About Creating and Modifying Templates

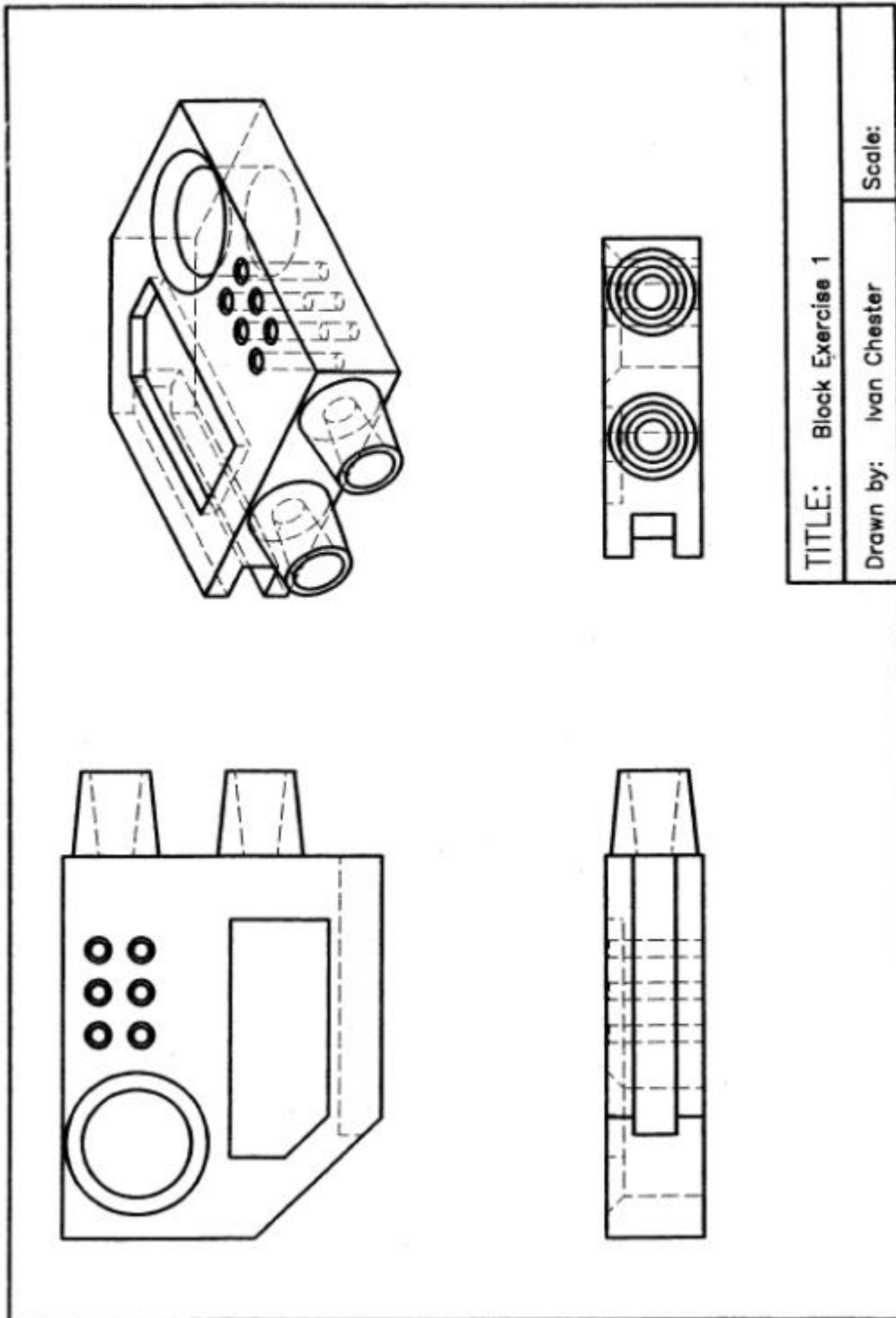
You can create new templates by modifying the templates already provided.

***To create a new template, perform the following steps:***

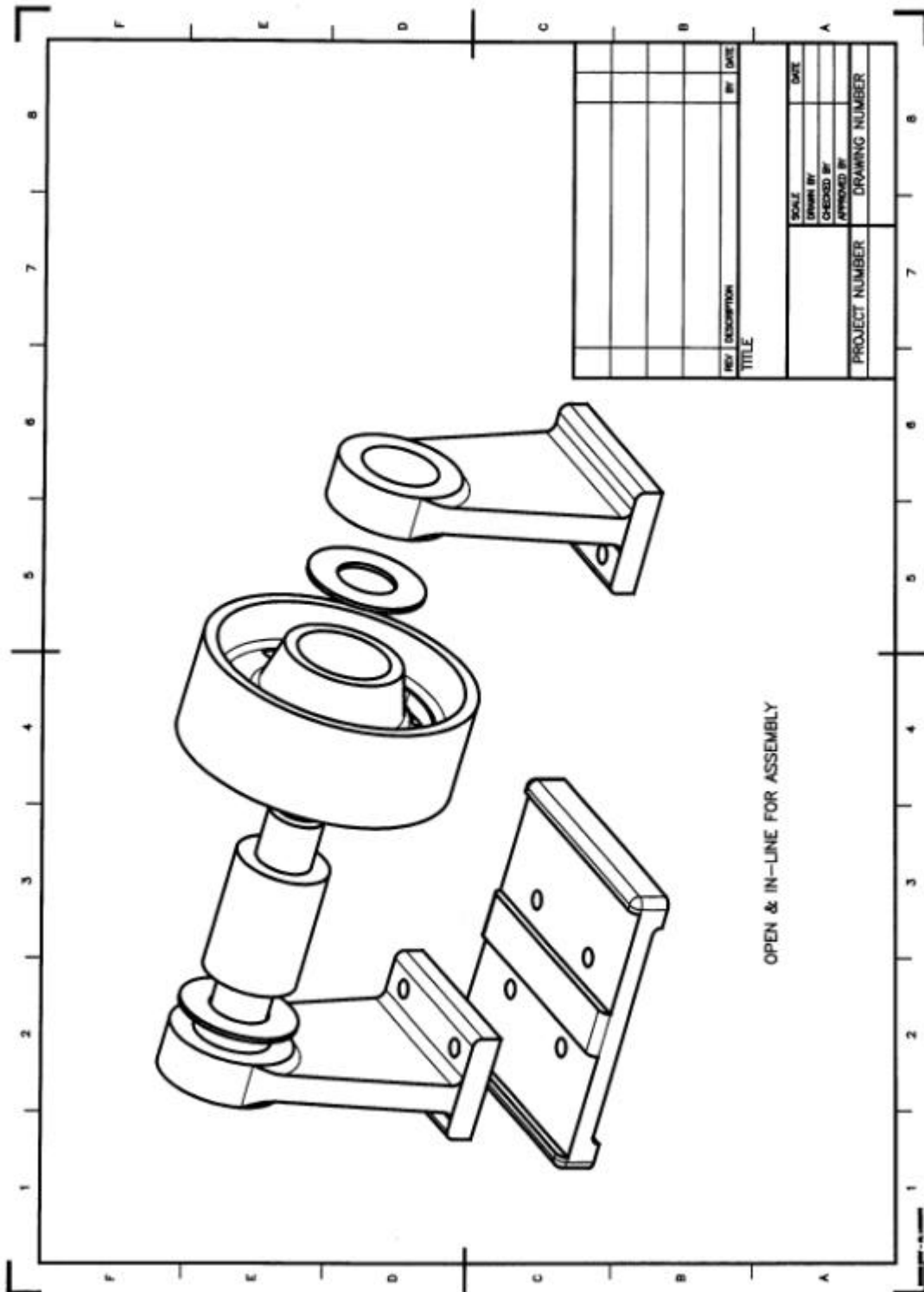
1. Create a bitmap to display on the button. The bitmap should be no larger than 84 pixels wide by 60 pixels high.
2. Save the bitmap with the BMP extension, in the application subfolder `\Program \Drawing Formats`.
3. Determine which of the six current templates most nearly matches the layout you want. You are going to backup and modify that file.
4. Click **File> Open> Drawing** and open the template file.
5. Click **File> Save Copy As** and use a name for future reference to indicate it is the backup of the original you are modifying.
6. Edit the source template as desired, then save and close the file. In a template, you can create the same visual elements as a format, in fact, templates can use format files, which will be automatically used by each drawing created. To have an existing format used automatically, use it as the format for the template. To include view layouts sketch rectangles and label them with the orientation. The properties of each view can be defined by typing Visual Basic commands as notes inside the rectangles and labels. Examples and hints are included in the templates to help you.
7. Using a text editor, navigate to the application subfolder `\Program\Drawing Formats`, and open the file `Templates.lst`.
8. The file is organized as four-line blocks of listings, each corresponding a template file. Locate the block for the file you are modifying. Overwrite the existing data as follows.
  1. Do not modify the first line, the name of the template file. It is used to link to the selection button.
  2. On the second line, type the name of the bitmap file that you want to appear on the selection button.
  3. On the third line, type the description that you want to appear in the dialog box when the cursor pauses over the selection button. Enclose the text in double quotation marks.
  4. On the fourth line, type one of the following:  
`NoDesign` (to open an empty drawing with no views)  
`NewDesign` (to open an empty drawing with empty views)  
`SelectDesign` (to prompt the user to select a saved design, currently open in a window, to display in the drawing).
9. Save and close the file.

### Notes

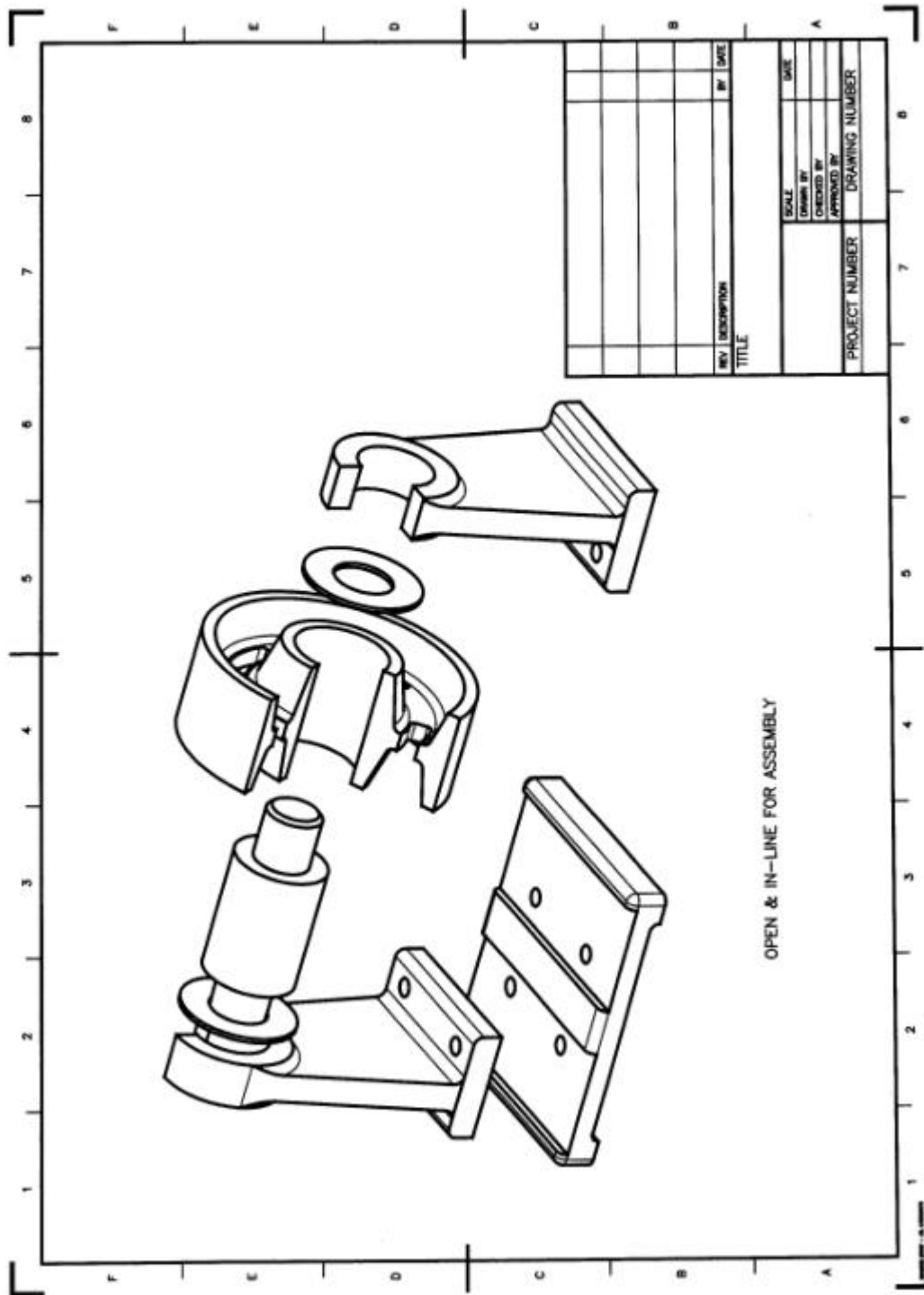
If you are using Windows 2000, you must have administrator privileges to modify or create a file in a folder used by an application, including template files.



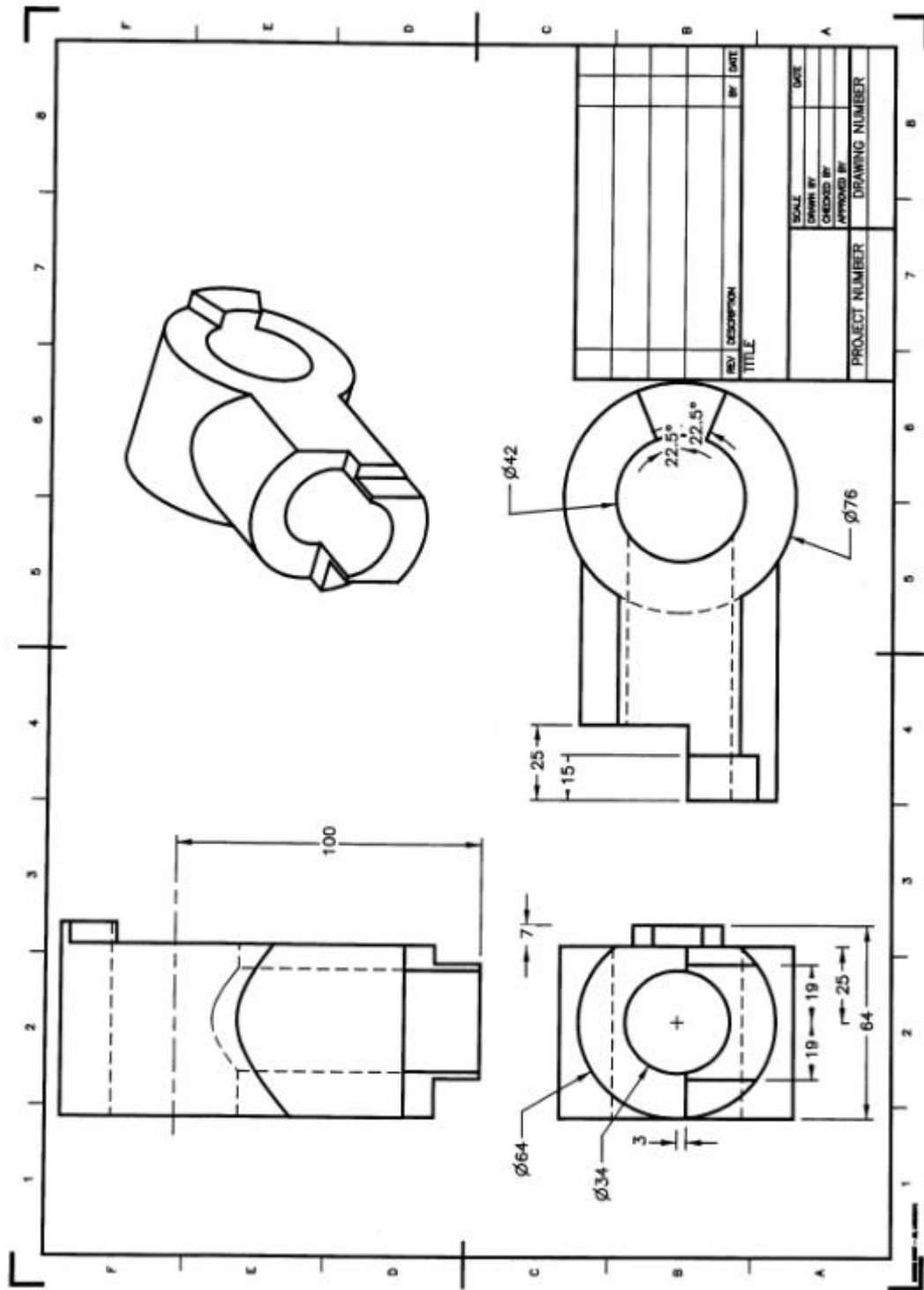
TITLE: Block Exercise 1	
Drawn by: Ivan Chester	Scale:



A Pro/DESKTOP sample file shown as an Open-In-Line Pictorial



A Pro/DESKTOP sample file shown as an Open-In-Line Pictorial with partial sectioning



Graphics 2000 Book 2- A. Boundy/I. Hass

