CATIA V5 Design with Analysis (Tutorial 3 – Deep Fry Basket)

or Commercial Use

Infrastructure

Sketcher

Part Design (Solid-modeling)

GSD (Surface-modeling)

Assembly Design

Generative Structural Analysis

Product Engineering Optimizer

Overview of Tasks

Linked Children

Tutorial 3A - Modeling

- Build a Master Model of the basket handle
- Create the upper & the lower parts from the Master Model
- Build the mechanical features on the both parts
- Get the both parts auto-updated after modifying the outlook of the master model

Tutorial 3B - Modeling

- Build the metal arm
- Build the basket
- Add material texture onto all components
- Assemble components

Tutorial 3C – Structural analysis

- Simplify the model for analysis
- Create Meshes onto two components and create a connector between them
- Create boundary conditions & define properties
- Analyze displacements & stresses

Tutorial 3D – Structural analysis (By Nastran)

• Repeat Tutorial 3C with the use of Nastran

Tutorial 3E – Design optimization

- Create a user parameter "volume"
- Run optimization to get the minimum volume of the metal arm with the smallest part deformation

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Change the view with the mouse

- A. Panning enables you to move the model on a plane parallel to the screen. Click and hold the middle mouse button, then drag the mouse.
- **B. Rotating** enables you to rotate the model around a point. Click and hold the middle mouse button and the right button, then drag the mouse.
- **C. Zooming** enables you to increase or decrease the size of the model. Click and hold the middle button, then click ONCE and release the right button, then drag the mouse up or down.



Tutorial 3A

- Enter CATIA by double-clicking its icon on the desktop
- (If a license menu pops up), select **ED2** and close CATIA. Then reopen again
- By default, a empty "Product" file is created. But now, you don't need this, just select "**File/Close**" on the menu
- Select 'Start/Mechanical Design/Part Design" on the menu bar

- Uncheck "Enable Hybrid Design" and then click "ok"
- An empty part is now created on "Part Design" workbench. You can see a specification tree at the upper left-hand corner and xyz datum planes in the middle of the screen



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Machining	Weld Design
Digital Mockup	Mold Tooling Design
Digital Process for Mapufacturing	Structure Design
Machining Simulation	2D Lavout for 3D Design
Ergonomics Design & Analysis	Drafting
Knowledgeware	Core & Cavity Design
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Tutorial 3A

To reset the layout of workbench (optional):-

 Sometimes the workbench may not be tidy before you use; some toolbars are missing and some are at wrong positions. To reset the layout, select "View/Toolbars/Customize" and select "Toolbar/restore position" on the pop-up window; Close and exit

To rename the tree:-

- Single-click "Part1" on the tree, right-click it, and then select "**Properties**"
- Modify Part Number as "Master_handle" on the tab page "Product"
- Select "ok" to exit



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Tutorial 3A

To build 1st sketch (Cont'):-

- **Draw** an **arc** (<u>R35</u>) connecting the bigger arcs , on the positive-x side, which is NOT tangent to them
- Similarly, **Draw** another smaller **arc** (<u>R10</u>) connecting the bigger arcs on the negative-x side
- Multi-select the arc (R10) and the upper arc(R380) by pressing and holding "ctrl" key on the keyboard
- Then select "Constraints defined in dialog box" icon
- Select "**Tangency**" and "ok"
- Add another Tangency constraint between the arc R10 and the lower arc(R380) by repeating the above steps



Modeling





A- 7

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Tangent

R 10

Tangent

Tutorial 3A



Tutorial 3A

To build a solid:-

- Select "Sketch.1" on the tree / directly click on the geometry
- Click "Pad" icon
- Enter <u>20mm</u> as the length of First Limit
- Select "Mirror extent"
- Click "ok"
- A solid is created



To round the sharp edges:-

Add a "Edge Fillet" R<u>5mm</u> onto the vertical sharp edges



Non Commercial Use

Tutorial 3A

To draft both sides of the solid:-

- Click "Draft angle" icon
- Enter <u>1deg</u> as Angle
- Select a vertical face as "Face to draft" (after that, all tangent faces will be automatically selected and turn red)
- Click the selection box of "Neutral element" and then select xy plane
- Click "More"
- Check "Parting=Neutral"
- Check "Draft both sides"
- Click ok to complete







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Non Commercial Use

Tutorial 3A

To change the workbench:-

- Select 'Start/Shape/Generative Shape Design"on the menu bar
- Select "Insert/Geometrical Set" on the menu bar and click ok to complete (Now a new branch "Geometrical set" is created on the part tree, which is used to store all reference curves and surfaces)

To create a reference plane:-

- Click "plane" icon
- Select "Offset from plane" as plane type
- Select "yz plane" as Reference
- Click "Reverse Direction" in the command window
- Enter <u>38mm</u> as Offset value
- Click ok to complete





Tutorial 3A

To build 2nd sketch:-

- click "Sketch" icon and select zx plane
- Draw a horizontal axis as shown
- **Multi-select** the endpoint of the axis and plane.1 by pressing and holding "ctrl" key on the keyboard
- Then select "Constraints defined in dialog box" icon
- Select "Coincidence" and "ok"
- Draw another two arcs (R450 & R270) and add the corresponding constraints as shown
- Exit the workbench by clicking "Exit" icon
- Click on an empty space to deselect the sketch





Sketch



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Tutorial 3A

To build 3rd sketch:-

- Click "Sketch" icon and select plane.1
- Draw an arc as shown
- **Multi-select** the endpoints then the y-axis by pressing and holding "**ctrl**" key on the keyboard
- Then select "Constraints defined in dialog box"
 icon
- Select "Symmetry" and "ok"
- Add Constraint <u>R30</u> onto the arc
- Rotate the model by mouse to have an isometric view
- Multi-select the arc and the point by pressing and holding "ctrl" key on the keyboard.
- Then select "Constraints defined in dialog box"
 icon
- Select "Coincidence" and "ok"
- (Now the arc should be coincided with Sketch.2)
- Exit the workbench by clicking "Exit" icon





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Tutorial 3A

Surface

To build a Surface:-

- Click "Sweep" icon
- Select "Explicit" as Profile Type
- Select "Sketch.3" as Profile
- Select "Sketch.2" as Guided Curve
- Click ok to complete
- (On the tree, this surface is stored in "Geometrical Set.1", so it will not be mixed with solids.)





Sweep /	
Swept Surface Definition	? ×
Profile type: 🗹 ✔ 📣	
Subtype: With reference surface Profile: Sketch.3	•
Guide curve: Sketch.2	
Angle: Odeg 🔁 Law	
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Optional elements Projection of the guide curve as spine	
Spine: Default (Sketch.2)	
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Remove cutters on Preview	
Positioning parameters	-
OK Cancel Preview	v

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Tutorial 3A

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To change the workbench:-

Select 'Start/Mechanical Design/ Part Design"
 on the menu bar to go back to solid-modeling
 environment

To cut the solid with this SURFACE:-

- Click "Split" icon
- Click OK on the warning window
- Select the Yellow Surface "Sweep.1"
- Click on the arrow so that it is pointing downwards
- Click ok to complete

split É \diamond Thick Su 👉 zx plane PartBody 🔊 Pad. 1 🔂 EdaeFillet. 1 Draft.1 Solit 1 Geometrical Set.1 🖉 Plane.1 Sketch.2 Sketch.3 Sweep.1 Hide/show

To hide the surface & its curves:-

- Select the surface "Sweep.1" and click "hide/show" icon
- Hide Sketch.2 & Sketch.3 too



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Tutorial 3A

Now, we are going to create a Multisection surface for the bottom face. Before that, we need to construct three guide curves and three different sections

To change the workbench:-

Select 'Start/Shape/ Generative Shape Design" on the menu bar to go back to surface-modeling environment

To create a reference plane:-

- Click "plane" icon
- Select "Offset from plane" as plane type
- Select "xy plane" as Reference
- Click "Reverse Direction" in the command window (The arrow points to negative Z)
- Enter <u>2mm</u> as Offset value
- Click ok to complete





Tutorial 3A

To create a intersection curve:-

- Click "intersection" icon
- Right-Click on the entry box of First
 Element
- Select "Create Extract"
- Select "No propagation " for Extract
 Definition
- Select the Face $\stackrel{\frown}{\not\propto}$
- Select Plane.2 as Second Element
- Click ok to complete



To create another intersection curve on

the opposite side:-

• Repeat the above steps but select the face opposite to Face 🔀 as First Element



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Tutorial 3A

To build 4th sketch:-

- Click "Sketch" icon and select zx plane
- **Draw** a horizontal **axis** as shown
- **Multi-select** the endpoint of the axis and plane.1 by pressing and holding "**ctrl**" key on the keyboard.
- Then select "Constraints defined in dialog box" icon
- Select "Coincidence" and "ok"
- **Draw** another two **arcs** (<u>R300</u> & <u>R150</u>) and add the corresponding constraints as shown
- Exit the workbench by clicking "Exit" icon
- Click on an empty space to deselect the sketch





Sketch



Tutorial 3A

To build 5th sketch:-

- click "Sketch" icon and select plane.1
- Click "Construction/Standard element" icon so
 that the coming elements will be considered as
 construction (reference) elements
- Rotate the model by mouse to have an isometric view
- Click "Intersect 3D elements" icon
- Select the curve "Intersect.1" (A point is created)
- Similarly, click "Intersect 3D elements" icon
- Select the curve "Intersect.2" (A point is created)
- Click "Construction/Standard element" icon again to deactivate this mode.
- **Draw** an **arc** by selecting the two intersection points as the endpoints

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Sketch

Tutorial 3A

To build 5th sketch (Cont'):-

- **Multi-select** the arc and the point 💢 by pressing and holding "**ctrl**" key on the keyboard.
- Then select "Constraints defined in dialog box" icon.
- Select "Coincidence" and "ok".
- Exit the workbench by clicking "Exit" icon.



To create a reference plane:-

- Click "plane" icon
- Select "Parallel through point" as plane type
- Select "yz plane" as Reference
- Select a Endpoint → of the curve "Intersect.1"
- Click ok to complete



Tutorial 3A

To build 6th sketch:-

- click "Sketch" icon and select plane.3
- Click "Construction/Standard element" icon so that the coming elements will be considered as construction (reference) elements
- Rotate the model by mouse to have an isometric view
- click "Intersect 3D elements" icon
- Select the curve "Sketch.4" (A point is created)
- Click "Construction/Standard element" icon again to deactivate this mode.
- **Draw** an **arc** with the endpoints near the extreme points of Intersect.1 & Intersect.2
- Add three Coincidence constraints to align the arc onto the points \$\frac{1}{27}\$
- Exit the workbench by clicking "**Exit**" icon.





Tutorial 3A

To create a reference plane:-

- Click "**plane**" icon
- Select "Parallel through point" as plane type
- Select "yz plane" as Reference
- Click ok to complete



To build 7th sketch:-

- **Draw** an **arc** with the endpoints near the endpoints of Intersect.1 & Intersect.2
- Add three Coincidence Constraints to align the arc onto the points 🔀
- (Refer to the steps of building 6th sketch)



Tutorial 3A

To create a Multi-sections surface:-

- Click "Multi-sections surface" icon
- Select "Sketch.6", "Sketch.5", & "Sketch.7" in order (They will then be inserted into the entry box of **Section**)
- If any red arrows are not pointing to the same direction, click it once to reverse.
- Then click the entry box of **Guides** once
- Select "Intersect.1", "Sketch.4" & "Intersect.2"
- Click ok to complete





All red arrows should point to the same direction, otherwise the created surface will be twisted



The multi-sections face is not

big enough to cover the solid

Tutorial 3A

As seen, the multi-sections surface is not big enough to cover the whole solid...

To hide the solid:-

- Right-click on "PartBody" on the tree
- Select Hide/Show



To extend the surface:-

- Click "Extrapolate" icon
- Select "Sketch.6" as Boundary
- Select "Multi-sections surface.1" as "Extrapolated"
- Enter <u>20mm</u> as Length
- Enter Tangency as Continuity
- Select "Assembly Result"
- Click Ok to complete



Non Commercial Use

Tutorial 3A

To extend the surface on the other end:-

- Click "Extrapolate" icon
- Select "Sketch.7" as Boundary
- Select "Extrapol.1" as "Extrapolated"
- Enter <u>20mm</u> as Length
- Enter Tangency as Continuity
- Select "Assembly Result"
- Click ok to complete





To change the workbench:-

- Select 'Start/Mechanical Design/ Part Design" on the menu bar to go back to solid-modeling environment
- Unhide "PartBody"



Tutorial 3A



- Click "Split" icon.
- Click OK on the warning window.
- Select the Yellow Surface "Extrapol.2"
- Click on the arrow so that it is pointing upwards.
- Click ok to complete



To hide the surface & its curves:-

• **Hide** Everything except "Partbody", xy plane, yx plane and zx plane.



💱 EdgeFillet. 1



Tutorial 3A

Next, we are going to create a curve-based pocket on the top face:-

To change the workbench:-

Select 'Start/Shape/ Generative Shape Design" on the menu bar to go back to the surface-modeling environment

To create a reference plane (plane.5):-

- Click "plane" icon
- Select "Offset from plane" as plane type
- Select "yz plane" as Reference
- Click "Reverse Direction" in the command window (The arrow should point to negative X)
- Enter 23mm as Offset value
- Click ok to complete







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Tutorial 3A

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Non Commercial Use

To build 8th sketch:-

- Click "Sketch" icon and select plane.5
- **Draw** an **arc** (<u>R24</u>, endpoints symmetric about y-axis)
- Add a Constraint (<u>9mm</u>) to define the distance between the arc and x-axis
- Exit the workbench by clicking "Exit" icon
- Click on an empty space to deselect the sketch

To build 9th sketch:-

- Click "Sketch" icon and select zx plane
- Draw an axis
- Add a **coincidence** constraint between the axis and plane.5
- **Draw** an **arc** (<u>R28</u>, endpoints symmetric about the axis)
- Add a Constraint (*9mm*) to define the distance between the arc and x-axis
- Exit the workbench by clicking "Exit" icon
- Click on an empty space to deselect the sketch



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Tutorial 3A

To build a Surface:-

- Click "Sweep" icon
- Select "Explicit" as Profile Type
- Select "Sketch.8" as Profile
- Select "Sketch.9" as Guided Curve
- Click ok to complete

To change the workbench:-

• Select 'Start/Mechanical Design/ Part Design" on the menu bar to go back to solid-modeling environment

To cut the solid with this SURFACE:-

- Click "Split" icon.
- Click OK on the warning window.
- Select the Yellow Surface "Sweep.2"
- Click on the arrow so that it is pointing downwards.
- Click ok to complete

To hide the surface & its curves:-

 Select "Sweep.2", "Sketch.8", "Sketch.9" & "Plane.5" and click "hide/show" icon.





Sweep.2

Tutorial 3A

To add Edge Fillets:-

- Click "Edge Fillet" icon
- Enter <u>3mm</u> as Radius
- Select Tangency as Propagation
- Select the three sharp edges $\overleftrightarrow{}$
- Click ok to complete



To save the new part in a Project Folder:-

It is a good practice to store all part files of a product in one specific folder.

- Create a folder wherever you can save (by MS window technique).
- Save your current part as "master_handle_a.CATPART" into the folder.
- Add "a" after its name to remind us its version.



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Tutorial 3A

New Part

To create the upper body:-

- Select File/New on the menu bar
- Select Part as type
- Enter Upper_body as part name
- Click ok to complete
- Select Window/Tile Vertically (we can see Master_handle & Upper Body at the same time)
- Right-click "PartBody" of master handle a.CatPart;
- and then select "Copy";
- Right-click "Upper body" of the tree of Upper body . and then select "Paste Special..."
- Select "As Result with link":
- Click ok to complete.





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Tutorial 3A



To cut the solid with a plane:-

- Click "Split" icon.
- Select xy plane
- Click on the arrow so that it is pointing upwards.
- Click ok to complete

To Save the new part in a Project Folder:-

Save your current part as
 "Upper_body_a.CATPART" into the folder.



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Tutorial 3A

ew Part

To create the lower body:-

- Select File/New
- Select Part as type
- Enter Lower_body as part name
- Click ok to complete
- Select Window/Tile Vertically (we can see Master handle & Lower Body at the same time)
- Right-click "PartBody" of master handle a.CatPart
- and then select "Copy"
- Right-click "Lower body" of the tree of Lower body . and then select "Paste Special ... "
- Select "As Result with link"
- Click ok to complete





Tutorial 3A



To cut the solid with a plane:-

- Click "Split" icon.
- Select xy plane
- Click on the arrow so that it is pointing downwards.
- Click ok to complete

To save the new part in Project Folder:-

 Save your current part as "Lower_body_a.CATPART" into the folder.

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Tutorial 3A

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Building mechanical features on Upper Body:-

To add two Bodies together:-

- Right-click "Body.2" on the tree
- Select Body.2 object/add... (Body.2 will become a branch of PartBody)

To get a boundary curve from the solid:-

- Select 'Start/Shape/ Generative Shape Design" on Operations the menu bar to go back to surface-modeling environment
- Click "Boundary" icon
- Select "Point continuity" as propagation type
- Select the parting surface (both inner & outer edges will be highlighted)
- Click ok to complete
- Select "Keep only one sub-element by a Near" in the message window "Multi-result management"
- Click ok
- Select an inner face \overleftrightarrow as Reference Element
- Click ok to complete







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Tutorial 3A

To remove material from the boundary:-

- Select 'Start/Mechanical Design/ Part Design" on the menu bar to go back to solid-modeling environment
- Click "Pocket" icon
- Click OK on the warning window.
- Select the curve "Near.1"
- Select "Reverse Direction"
- Enter <u>1.5mm</u> as First Limit
- Select "Thick" option
- Enter <u>1mm</u> as thickness.1
- Enter <u>1mm</u> as thickness.2
- Click ok to complete



	Pocket Definition						
	First Limit			Second Limit			
	Туре:	Dimension	-	Type:	Dimension	•	
	Depth:	1.5mm	-	Depth:	Omm	-	
	Limit:	No selection		Limit:	No selection		
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MI	📮 Thick	Thick			Reference: No selection		
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) OK	Cancel Prev	/iew	


Tutorial 3A

To build a sketch:-

- Click "Sketch" icon and select the bottom face of Pocket.1
- **Draw** 4 **lines** (two horizontal & two vertical)
- Add a Symmetric Constraint between two horizontal lines
- Add 3 more dimensional constraints (<u>28mm</u>, <u>8mm</u> & <u>14mm</u>)
- Exit the workbench by clicking "Exit" icon

To build a solid from the open profile:-

- click "Pad" icon
- Click ok on the warning window
- Select "Thick" option
- Select "Neutral Fiber"
- Enter <u>1mm</u> as thickness.1
- Select "Merge Ends" so that the lines will be extended until they touch the solid face
- Select "Up to Next" as First Limit
- Click ok to complete



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To build another sketch:-

Click "Sketch" icon and select xy plane

Add a dimensional Constraint (20mm)

Exit the workbench by clicking "Exit" icon

Draw a circle (Dia6.0) on x-axis

Modeling

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Tutorial 3A

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To build a solid:-

- click "Pad" icon
- Select "Up to Next" as First Limit
- Select "More"
- Enter <u>5mm</u> as Second Limit
- Click ok to complete



Sketch

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Tutorial 3A



To make a pattern:-

- Multi-select Pad.2 & Hole.1
- Click "Rectangular Pattern" icon
- Click the box "Reference Element"
- Select xy plane
- Click "Reverse"
- Enter <u>2</u> as Instance
- Enter <u>62mm</u> as Spacing
- Click ok to complete



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Tutorial 3A

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To check the depth of holes:-

- Click "Sketch" icon and select zx plane
- Click "Cut Part by Sketch Plane" icon
- Visual Check whether the holes are too deep or not deep enough (for this case, they are accepted)
- Exit the workbench by clicking "Exit" icon



To make a pocket:-

- Click "Sketch" icon and select yz plane
- Draw a **circle** (<u>Dia3.0</u>) at (<u>-12.0mm</u>, 0)
- Select the circle, click "**Mirror**" icon, and then click y-axis
- Exit the workbench by clicking "Exit" icon
- Click "Pocket" icon
- Enter <u>10mm</u> as First Limit
- Check " Mirrored extent" option
- Click ok to complete

** SAVE THE FILE AGAIN **

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Tutorial 3A

Building mechanical features on Lower Body:-

To add Bodies together:-

- Right-click "Body.2" on the tree
- Select Body.2 object/add... (Body.2 will become a branch of PartBody)

To get a boundary curve from the solid:-

- Select 'Start/Shape/ Generative Shape Design" on the menu bar to go back to surface-modeling environment
- Click "Boundary" icon
- Select "Point continuity" as propagation type
- Select the parting surface (both inner & outer edges will be highlighted)
- Click ok to complete
- Select "Keep only one sub-element by a Near" in the message window "Multi-result management"
- Click ok
- Select an inner face \overleftrightarrow as Reference Element
- Click ok to complete







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Tutorial 3A

To add material from the boundary:-

- Select 'Start/Mechanical Design/ Part Design" on the menu bar to go back to solid-modeling environment
- Click "Pad" icon
- Click OK on the warning window.
- Select the curve "Near.1"
- Enter <u>1.5mm</u> as First Limit
- Select "Thick" option
- Enter <u>0mm</u> as thickness.1
- Enter <u>1mm</u> as thickness.2
- Click ok to complete

To Offset a solid face:-

- Click "Thickness" icon
- Select the parting surface
- Enter <u>-0.5mm</u> as Default thickness
- Click ok to complete



Pad	Pad Definition					? X
F	First Limit		Second Limit			
Тур	pe:	Dimension	•	Туре:	Dimension	•
Ler	ngth:	1.5mm	-	Length:	Omm	
Lim	iit:	No selection		Limit:	No selection	
P	rofile/Surf	ace		Direction	I	
Sel	lection: N	ear.1	- 🕱	🧧 Norma	l to profile	
	Thick			Reference	No selection	
F	Reverse Si	de		Thin Pad		
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R	Reverse Dir	rection		Thickness2	2: 1mm	-
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Non Commercial Use

Tutorial 3A

To build a sketch:-

- Click "Sketch" icon and select the top face of Pad.1
- Draw 4 lines (two horizontal & two vertical)
- Add a symmetric Constraint between two horizontal lines
- Add 3 more dimensional constraints (<u>28mm</u>, <u>8mm</u> & 14mm)
- Exit the workbench by clicking "Exit" icon



To build a solid from the open profile:-

- Click "Pad" icon
- Click ok on the warning window
- Select "Thick" option
- Select "Neutral Fiber"
- Enter <u>1mm</u> as thickness.1
- Click "Reverse Direction"
- Select "Merge Ends" so that the lines will be extended until they touch the solid face
- Select "Up to Next" as First Limit
- Click ok to complete





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Non Commercial Use



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Written by Dickson Sham

Tutorial 3A

To make a pad:-

- Click "Sketch" icon and select the top face of Pad.1
- **Draw 3 lines** (three horizontal lines)
- Add a symmetric Constraint between two horizontal lines
- Add 2 more dimensional constraints (<u>50mm</u>, & <u>7mm</u>)
- Exit the workbench by clicking "Exit" icon



- Click "Pad" icon
- Click ok on the warning window
- Select "Thick" option
- Select "Neutral Fiber"
- Enter 1mm as thickness.1
- Click "Reverse Direction"
- Select "Up to Next" as First Limit
- Click ok to complete

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Non Commercial Use

Preview

Cancel

OK

Tutorial 3A

To make a pocket:-

- Click "Sketch" icon and select the face of Pad.3
- Click "Cut Part by sketch plane" icon to view the sketch plane
- **Draw** a **Profile** (two vertical lines, one horizontal line & an arc)
- Add a dimensional constraint R1.5 on the arc
- Add another dimensional constraint (45mm) between the circle centre and the y-axis
- Exit the workbench by clicking "Exit" icon



- Click "Pocket" icon
- Enter <u>10mm</u> as First Limit
- Check "Mirrored Extent" option
- Click ok to complete



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Non Commercial Use

Tutorial 3A

To create a reference plane:-

- Click "plane" icon
- Select "Offset from plane" as plane type
- Select "xy plane" as Reference
- Click "Reverse Direction" in the command window (The arrow should point to negative Z)
- Enter <u>5mm</u> as Offset value
- Click ok to complete



To make a pad:-

- Click "Sketch" icon and select Plane.1
- Draw a Circle (Dia9.5mm)
- Add a dimensional constraint (20mm) between (the circle center and the y-axis
- Exit the workbench by clicking "Exit" icon
- Click "Pad" icon
- Click "Reverse Direction"
- Select "Up to Next" as First Limit Type
- Click ok to complete



Tutorial 3A

To make a pocket:-

- Click "Sketch" icon and select the top face of the cylinder
- Draw a circle
- Add a Concentric Constraint
- Add a dimensional constraint (1.5mm)
- Exit the workbench by clicking "Exit" icon



- Click "**Pocket**" icon
- Select "Up to Last" as First Limit Type
- Click "More" option
- Enter <u>-2.5mm</u> as Second Limit
- Click ok to complete

	Sketc	h Ba	sed Fea	atui	
] g		M 🖻		Po
уре			The		1
				1.5	T
			5		

			//		
Pocket Def	inition			<u>? ×</u>	
First Limi	t		Second	Limit	
Туре:	Up to last	-	Type:	Dimension	
Limit:	No selection		Depth:	-2.5mm 🚔	
Offset:	Omm	-	Limit:	No selection	
Profile/S	urface		Direction	n	
Selection:	Sketch.6		🔎 Norma	al to profile	
Thick			Reference: No selection		
Reverse	Side		- Thin Poo	:ket	
Mirrore	d extent		Thickness	1 1mm 🚍	
Reverse	Direction		Thickness	2; 1mm	
	_	<less< th=""><th>Neutra</th><th>al Fiber 🔲 Merge Ends</th></less<>	Neutra	al Fiber 🔲 Merge Ends	
		٢	ОК	Cancel Preview	

Non Commercial Use

Tutorial 3A

To make a hole:-

- Click "Hole" icon and select the top face of the cylinder
- Select "Up to Last" as Extension Type
- Enter <u>3.2mm</u> as Diameter
- Click ok to complete

Sketch-Based Features	
] I I M I O A &	finition ?X
	Extension Type Thread Definition Up To Last Diameter : 3.2mm Depth : 2.5mm Diffect : Dmm Direction Direction Reverse Normal to surface No selection
	OK Cancel Preview

To make a solid by an used sketch:-

- Click "Pad" icon
- Select "Sketch.5" as Profile (Sketch.5 has been used before and it is now hidden)
- Enter <u>1.5mm</u> as First Limit
- Check "Thick" option
- Enter <u>1.5mm</u> as Thickness.1
- Enter <u>0mm</u> as Thickness.2
- Click ok to complete



Non Commercial Use

Tutorial 3A



To hide a plane:-

- Right-click on Plane.1
- Select "Hide/Show"

SAVE THE FILE AGAIN

Tutorial 3A

Tools

A- 51

ີ

Now we have three part files:

- *Master handle* (which controls the handle outlook)
- Upper_body (which is a child of Master_handle and has its own mechanical features)
- Lower body (which is a child of Master handle and has its own mechanical features)

To modify Master Handle:-

- Single Click "Sketch.1" in Master handle
- Change R380 to R250
- Change R10 to R12
- Exit the workbench by clicking "Exit" icon

To get Upper & Lower bodies updated:-

- Activate the window of *Upper_body*
- Click "Update" icon
- After a few second, the model turns from red to blue; by then the model is updated in shape
- For *Lower body*, the steps are the same.



Version 1h- Mar08

Building the basket:-

We are going to use Surface-modeling technique to build the basket...

- Select File/New on the menu bar
- Select "Part" in "List of Type
- Click ok to complete
- Enter "Basket" as Part Name
- Click ok to complete
- Select 'Start/Shape/Generative Shape Design" on the menu bar.
- Select "Insert/Geometrical Set" on the menu bar and click ok to complete (Now a new branch "Geometrical set" is created on the part tree, which is used to store all reference curves and surfaces)

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Non Commercial Use



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Help

Tools

🔮 Geometrical Set . . .

🔍 Ordered Geometrical Set...

Window

Insert

🔒 Body



V5 for Student - [Basket]

View

Edit

🔒 🚑 X 🖬 🛍 '

File

ilane



Surfaces

1

Guide curve 1.

Modeling



l	Swept Surface Definition
	Profile type: 💉 💉 📣
	Subtype: With draft direction
	Guide curve 1: Sketch.1
	G1 Draft direction: xy plane
	Draft computation mode: 🔮 Square 🔿 Cone
	Wholly defined G1-Constant Location values
	Angle: 10deg 🚔 Law
	Angular sector: Previous 2 / 4 Next
-	
	Length type 1:
\mathcal{N}	Length 1: 60mm
$\langle $	Relimiting element 1: No selection
J	Length type 2:
	Length 2: Omm
	Relimiting element 2: No selection
	Smooth sweeping
	Angular correction: 0.5deg
	Deviation from guide(s): D.001mm
	Twisted areas management
	Remove cutters on Preview
	OK Gancel Preview
	Written by Dickson Sham

To build a sketch:-

- Click "Sketch" icon and select xy plane
- **Draw** a **centered rectangle** (center at origin, <u>170mm</u> x <u>130mm</u>)
- Exit the workbench by clicking "Exit" icon

To create a swept surface:-

- Click "Sweep" icon
- Select "Line" as Profile Type
- Select "with draft direction" as subtype
- Select "Sketch.1" as Guide Curve.1
- Select xy plane as Draft Direction
- Enter <u>10 deg</u> as Angle
- Enter <u>60mm</u> as Length.1
- Click the arrow as shown (angular sector =2)
- Click ok to complete



Tutorial 3B

To create a surface from a closed boundary:-

- Click "Fill" icon
- Select all the four edges $\overleftarrow{\times}$ of the smaller opening
- Click ok to complete (a surface will be created to fill the opening)







To Join surfaces into one:-

- Click "Join" icon
- Select surfaces "Sweep.1" & "Fill.1"

Basket 👉 xy plane

👉 yz plane

🚄 zx plane

🔁 PartBody

🗸 Geometrical Set. 1

Sketch.1

Fill.1

🔀 Join. 1

Sweep.1

after this creation

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Non Commercial Use

Click ok to complete (a new surface is created to represent both surfaces; They are hidden now)

? × Operations Join Definition Elements To Join 5weep.1 Add Mode Remove Mode Parameters Federation Sub-Elements To Remove Check tangency 📮 Check connexity 🔲 Check manifold Simplify the result Ignore erroneous elements A new surface is created; the 0.001mm -Merging distance original surfaces are hidden É 0.5deg Angular Threshold Cancel OK Preview

Version 1h- Mar08

Tutorial 3B

To add a Fillet on surface edges:-

- Click "Edge Fillet" icon
- Select all the four Vertical edges
- Enter <u>15mm</u> as Radius
- Click ok to complete



Operations

Edge Fillet Definit	ion 🤶 🗙
	2-1-4
Support:	1.1100
Extremities:	Smooth
Radius:	15mm 🚍
Object(s) to fillet:	4 elements
Propagation:	Tangency 💌
Trim ribbons	
📔 Trim support	
	More>>
	OK Scancel Preview

To add another Fillet on surface edges:-

- Click "Edge Fillet" icon
- Select an edge of the bottom face
- Enter <u>15mm</u> as Radius
- Click ok to complete



Tutorial 3B

To hide a sketch:-

type

• Right-Click on "Sketch.1"

To get a boundary from a surface:-

Select an edge of the opening

Click "Boundary" icon

Click ok to complete

Select "Hide/Show"

Boundary Operations X 88 😽 🔿 N (1 4 Select the edge

Boundary Definition ? × Propagation type: Point continuity Surface edge: EdgeFillet.2\Edge.5 Limit1: No selection Limit2: No selection OK © Cancel

Select "Point continuity" as propagation



Tutorial 3B

To create a swept surface:-

- Click "Sweep" icon
- Select "Circle" as Profile Type
- Select "Center & Radius" as Subtype
- Select the curve "Boundary.1" as Center Curve
- Enter <u>1.5mm</u> as Radius
- Click ok to complete

To hide a curve:-

- Right-click on "Boundary.1"
- Select "Hide/Show"

Surfaces	Swept Surface Definition
J < 🕿 🕫 🖾	Profile type: 🎻 📢 🎻
pe Center	GCR Subtype: Center and radius Center curve: Boundary.1 Radius: 1.5mm
Basket	Optional elements
- ∠ xy plane - ∠ yz plane	Spine: Default (Boundary.1)
<pre>> y c plane > PartBody Geometrical Set.1 > Sketch.1 > Sketch.1 > Fill.1 > Join.1 > EdgeFillet.1 > EdgeFillet.2 > Radius=15mm > EdgeFillet.2 > Sweep.2</pre>	Relimiter 1: No selection Relimiter 2: No selection Smooth sweeping
™ Badius=1.5mm	

A- 57 Non Commercial Use

To create a reference plane:-

- Click "plane" icon
- Select "Offset from plane" as plane type
- Select "yz plane" as Reference
- Click "Reverse Direction" in the command window (The arrow should point to negative X)
- Enter 150mm as Offset value
- Click ok to complete

To make a point on a new sketch:-

- Click "Sketch" icon and select plane.1
- **Draw** a **point** (x=20, y= 40)
- Exit the workbench by clicking "Exit" icon

Profile





Modeling

Tutorial 3B

A- 59

To make another point on a new sketch:-

- Click on empty space to deselect "Sketch.2"
- Click "Sketch" icon and select plane.1
- **Draw** a **point** \checkmark (33mm above the previous point)
- Exit the workbench by clicking "Exit" icon

To project a point onto a surface:-

- Click "Projection" icon
- Select "Along a direction" as Projection Type
- Select the point "Sketch.2" as Projected
- Select the surface "Edgefillet.2" as Support
- Select Plane.1 as Direction
- Click ok to complete



Version 1h- Mar08

Tutorial 3B

To project another point onto a surface:-

- Click "Projection" icon again
- Select "Along a direction" as Projection Type
- Select the point "Sketch.3" as Projected
- Select the surface "Edgefillet.2" as Support
- Select Plane.1 as Direction
- Click ok to complete



To make a mirror copy:-

- Click "Symmetry" icon
- Select the point "Project.1" as Element
- Select zx plane as Reference
- Click ok to complete
- Click "Symmetry" icon again
- Select the point "Project.2" as Element
- Select zx plane as Reference
- Click ok to complete



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Tutorial 3B

To hide all reference elements:-

- Multi-select Plane.1, Sketch.2 & Sketch.3
- Right-click on anyone
- Select "Hide/Show"

To add Material Texture:-

- Download a texture from a shared library at http://www.planit3d.com/source/texture_files/metal/metal.html
- Save the texture file into the project folder
- Click "Apply Material" icon
- Click B&W Tiling
- Select the surface "EdgeFillet.2" on the tree
- Click ok to complete











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To view the material texture:-

• Click "Shading with material" icon

To modify the texture:-

- Double-click "B&W" Tiling (under Edgefillet.2) on the tree
- Select the tab page "Texture"
- Select "Image" as Type
- Click "..." icon of Image Name
- Select the downloaded texture file
- Select "Cubical Mapping"
- Enter <u>30mm</u> as Material Size
- Click ok to complete





Properties	×
Current selection : B&W Tiling	
Rendering Inheritance Feature Properties Analysis Dra	1
Material size: 30 mm	
Lighting Texture	
Type Image	

Tutorial 3B



Don't

Building the metal arm:-

We are going to learn how to build 3D curves to represent the metal arm...

- Select "File/New" on the menu bar
- Select "Part" in "List of Type
- Click ok to complete
- Enter "Metal_arm" as Part Name
- Click ok to complete
- Select 'Start/Shape/Generative Shape Design"on the menu bar.
- Select "Insert/Geometrical Set" on the menu bar and click ok to complete





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Modeling





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Non Commercial Use

Tutorial 3B

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Non Commercial Use

To create a connecting 3D curve:-

- Click "Connect Curve" icon
- Select the endpoint of Sketch.1 💢
- Select "Tangency" as Continuity of First Curve
- Select the endpoint of Sketch.2
- Select "Tangency" as Continuity of Second Curve
- Click "Preview" to have a preview
- Click "Reverse Direction" if the curve is flipped
- Click ok to complete

To create a reference plane:-

- Click "plane" icon
- Select "Parallel through point" as plane type
- Select "xy plane" as Reference
- Select the endpoint of Sketch.1 💥
- Click ok to complete



Version 1h- Mar08

Tutorial 3B



Tutorial 3B



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Tutorial 3B

Non Commercial Use

To create a round corner between two lines:-

- Click "Corner" icon
- Select "Line.1" as Element 1
- Select "Trim element 1"
- Select "Sketch.2" as Element 2
- Select "Trim element 2"
- Enter <u>5mm</u> as Radius
- Click ok to complete

- Similarly, click "Corner" icon again
- Select "Corner.1" as Element 1
- Select "Trim element 1"
- Select "Symmetry.2" as Element 2
- Select "Trim element 2"
- Enter <u>5mm</u> as Radius
- Click ok to complete



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Tutorial 3B

To group all lines & curves:-

- Click "Join" icon
- Select all lines & curves in the screen, which includes: "Sketch.1", "Connect.1", "Corner.2", "Symmetry.3"& "Symmetry.1"
- Click ok to complete





Draw a circle here Corner.2 Symmetry.3 Symmetry.1 Connect.1 Sketch.1

To make a sketch:-

- Click "Sketch" icon and select zx plane
- Draw a circle (<u>Dia 3.0</u>)
- Add a coincidence constraints between the circle center and the line
- Exit the workbench by clicking "Exit" icon



Tutorial 3B

To make a solid:-

- Select 'Start/Mechanical Design/Part Design" on the menu bar to go back to solidmodeling environment
- Click "Rib" icon
- Click ok on the warning window
- Select "Sketch.3" as Profile
- Select "Join.1" as Center Curve
- Click ok to complete



To add material texture:-

- Click "Apply Material" icon
- Select "Iron" in the tab page "Metal"
- Select "PartBody" on the tree
- Click ok to complete

To save the new part in Project Folder:-

Save your current part as "Metal arm a.CATPART" into the folder.



Apply Material

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Non Commercial Use

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- 🖻

Bronze

Eroded metal 1

Magnesium

Steel
Tutorial 3B

Assemble components together...

In the folder, you should have five part files;

- Master_handle_a.CATPART
- Upper_body_a.CATPART
- Lower_body_a.CATPART
- Basket_a.CATPART
- Metal_arm_a.CATPART



To go to a new Workbench:-

- Select 'Start/Mechanical Design/Assembly Design" on the menu bar.
- You may need to reset the layout of the toolbars if the workbench isn't tidy.

To rename the tree:-

- Single-click "Product1" on the tree, right-click it, and then select "**Properties**".
- Modify Part Number as "Basket_assm" on the tab page "Product".
- Select "ok" to exit .

<mark>] <u>S</u>tart <u>F</u>ile <u>E</u>dit <u>V</u>iew Insert</mark>	<u>T</u> ools <u>A</u> nalyze <u>W</u> indow <u>H</u> elp
Infrastructure	▶
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<u>,≓S</u> hape	Assembly Design
Analysis & Simulation	Sketcher
AEC Plant	Product Functional Tolerancing & Annotation
Machining	Weld Design
Digital Mockup	Mold Tooling Design
Eguipment & Systems	Shuth we Decise
Digital Process for Manufacturing	
Machining Simulation	2D Layout for 3D Design
Ergonomics Design & Analysis	Parafting
	Core & Cavity Decign



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Tutorial 3B



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Tutorial 3B

To move a part by "Compass":-

- Click and hold the **RED** dot of the compass
- Drag it onto the part that you want to move
- The compass will then turn into green and its axis labels will be v-u-w
- Drag along the green lines/arcs of the compass to move the part to a desired position
- Repeat the steps so that all parts are NEARLY at desired positions
- Now the parts are separated. It is easier for us to select part features later

To reset "Compass" as original:-

- Click and hold the red dot of the compass.
- Drag it onto the coordinate system at lower right-hand corner of the window and then release.
- 7







Tutorial 3B

To assemble parts by adding constraints:-

(1) Fix "Basket" in space

- Click "**Fix**" icon
- Select "Basket" on tree; Now the part "Basket" is fixed in position.

(2) Link "Metal Arm" to "Basket"

- Click "Coincidence Constraint" icon
- Check "Do not prompt in future" and click "close" to close the message box.
- Select zx plane of Metal Arm
- Select zx plane of Basket
- Click ok to complete

(If you want to delete a constraint, just click the constraint either on the model or on the tree, and then press "Delete" key on keyboard.)



Coincidence constraint



Tutorial 3B

(Cont')

- Click "Offset Constraint" icon
- Select the point of Metal Arm 💢
- Select the point of Basket
- Enter <u>2mm</u> as Value
- Click ok to complete
- Click "Update" Icon position.



s to update the

- **UNHIDE** the curve "Join.1" of Metal Arm
- Similarly, Click "Offset Constraint" icon again
- Select the point of Metal Arm
- Select the point of Basket
- Enter <u>2mm</u> as Value
- Click ok to complete
- Click "Update" Icon position.



to update the

• *HIDE* the curve "Join.1" of Metal Arm again

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Tutorial 3B



Tutorial 3B



Click "Update" Icon position.



to update the

Tutorial 3B

To hide all constraints:-

Just single-click "Constraints" on the tree and right-click to show the contextual menu; then select "Hide/Show"

To hide all datum planes:-

- Select "Edit/Search..." on the menu bar and then click "Load all type" icon Hide/Show
- Select "Plane" as Type
- Click "Search & Select" icon
- Click "Hide/Show" icon

To Save all files:-

- Select "File/Save all"
- Click OK to close this message box (because you have to define the file location of the new Product file)
- Click "Save As..." icon
- Enter "Basket assm a.CATProduct" as filename and save it in your project folder.

****CLOSE ALL FILES****

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END of Tutorial 3B

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Tutorial 3C

We are going to estimate the deflection of the basket under the maximum load by Finite Element Analysis...

Assumptions:

- Linear Behavior of the material
- Displacements will be small such that a linear solution is valid
- The spot weld joint between the basket and the metal arm will not break under the load
- Loading rate should be sufficiently low
- Load is uniformly distributed on the bottom faces of the basket
- The deformation of the basket is much lower than that of the metal arm





Written by Dickson Sham

On Boundary

4.51e+007 3.61e+007 2.71e+007 1.81e+007 9.1e+006 9.06e+004

Tutorial 3C

File/Open/ Basket_assm_a.CATProduct

To go to a new Workbench:-

- Select 'Start/Analysis/Advanced Meshing Tools" on the menu bar.
- Select Static Analysis and then click ok

(1) To simplify the model for analysis:-

- Click "+" next to "Link Manager.1" on the tree
- Click "+" next to "Link.1" on the tree
- Hide Upper_body.1 & Lower_body.1
- Hide the surface "Sweep.2" under Basket.1
- Hide "PartBody" under Metal_arm.1
- Show the curve "Join.1" under Metal arm.1



Tutorial 3C

Now, we should see the below elements only:

- One surface
- Six points (4 points on surface, 2 points on curve)
- One Curve (grouped)



(2a) To create a 2D mesh:-

- Click "Surface Mesher" icon
- Select the surface
- Select "quadrangle" as Shape
- Select "Linear" as Type
- Enter 5mm as Mesh Size
- Leave all the rest default options unchecked
- Enter <u>Omm</u> as Constraint sag
- Click ok to complete .

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Meshin M

Mesh

Mesh size

Tutorial 3C

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(Cont'):-

- Click "Project External Points" icon (so that nodes are created on the positions of spot welding)
- Select the four points on the surface
- If you select a wrong element accidentally, and then remove it from the list click
- Enter 0.01mm as Tolerance
- Select "Project on geometry"
- Click ok to complete (Now 4 red dots appears on the surface)



- Select the surface on which 4 red dots have been just created
- Enter 5mm as Mesh Size
- Click ok to complete (Meshing this surface before other surfaces will result in an uniform mesh distribution on this surface)



Geometry Selector

Written by Dickson Sham

Analysis by CATIA

- 🗆 ×

Written by Dickson Sham



Version 1h- Mar08



Tutorial 3C

Loads

œ

👼 Properties

Materials

E Restra Loads.

Sensors.1

(4) To Create a Force:-

- Click "Distributed Force" icon
- Select the bottom faces (9 faces) of the surface as shown
- Enter -15N as Z Force
- Click ok to complete



(5) To Create a User Material:-

- Click "User Material" icon
- Select "Iron" from the catalog
- Click ok to complete (it will be created on the tree)
- Double-click it on the tree to view its properties
- Leave everything unchanged
- Click ok to quit



Properties

.1	C	iurrent selection : Iron
1		Feature Properties Rendering Inheritance Analysis
lateria		Material Isotropic Material
e		Structural Properties
ints.1		Young Modulus 1.2e+011N_m2
1		Poisson Ratio 0.291
Case		
s.1		Density 7870kg_m3

Thermal Expansion 1.21e-005_Kdeg

Yield Strength 3.1e+008N_m2

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Non Commercial	Use

Written by Dickson Sham

Dra

Tutorial 3C

(6a) To define Properties of Surface Mesh.1:-

- Click "2D Property" icon
- Select the surface
- Select "User-defined material" option (because the original material is used for rendering only, not the actual material of the basket) 2D Prope
- Click the entry box "No selection" once
- Select "User Material.1" on the tree
- Enter 1mm as Thickness
- Click ok to complete

(6b) To define Properties of 1D Mesh.1:-

- Click "1D Property" icon
- Select the curve
- Select "User-defined material" option .
- Click the entry box "No selection" once
- Select "User Material, 1" on the tree
- Select "Cylindrical Beam" as Type
- Click the icon \sum
- Enter 1.5mm as radius
- Click ok to complete

2D p	property 1D property
not	ger ⊠ Groߣ € Ø
2D Property Name 2D Property.1 Supports 1 Face Material User Material.1	
User-defined material Thickness 1mm Data Mapping	1D Property C Name 1D Property.1
OK Cancel	Material User Material.1

Supports 1 Edge
Material User Material.1
User-defined material
Type Cylindrical beam 🔽 🗶
Orientation geometry No selection
Offset None
Released DOF None
Variable beam factors
OK Cancel

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Tutorial 3C

(7) To build Connections between meshes:-

- (We have four pairs of points and we are going to build a connection for EVERY PAIR independently)
- Click "General Analysis Connection" icon
- Select *point 1* as First Element
- Click the box "No selection" of Second Element
- Select *point 2* as Second Element
- Click ok to complete
- Repeat the above steps for the remaining three pairs





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Tutorial 3C

(8) To define Properties of connections:-

- Click "Rigid Connection Property" icon
- Select 'General Analysis Connection.1" on tree
- Click ok to complete
- Repeat the above steps for 'General Analysis Connection.2"
 'General Analysis Connection.3"
 'General Analysis Connection.4"

** SAVE ALL FILES**

- Select "File/Save all"
- Click OK to close this message box (because you have to define the file location of the new Product file)
- Click "Save As..." icon
- Enter "Analysis_a.CATAnalysis" as filename and save it in your project folder.



What we have already done ...

- 1. Simplify the model (Hide unnecessary parts & features)
- 2. Create Meshes (2D & 1D)
- 3. Create a constraint (Clamp)
- 4. Create a force (distributed force)
- 5. Create a User material
- 6. Define Properties of Meshes
- 7. Create connectors
- 8. Define Properties of Connectors

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Tutorial 3C

C....凶

(9) To start Computation:-

- Click "Compute" icon
- Select "All"
- Click ok

(10) To View the result:-

- Click "Von Mises stress" icon
- Click "Shading with material"
- (We can see the stress distribution on the basket. The stress value is not the same as the real case because we simplify the metal net as a metal sheet)
- Click "Animate" icon to see the variations in stress with different degrees of displacement



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Tutorial 3C

(Cont'):-

- Click "Displacement" icon
- (We can see the maximum displacement of the basket is about 96.7mm!)

The Displacement is too big to accept. Now we are going to shorten the length of the metal arm and make it thicker...

To edit the part "Metal Arm":-

- Right-click "Metal arm" on tree
- Select Metal_arm.object/Open in new window



(Con't):-

Analysis by CATIA



To increase the diameter of Metal Arm (1D- Mesh):-

Double-click "Sketch, 1" on tree

- Select "Window/Analysis1" on the menu bar to go back to the analysis workbench (The metal arm in the assembly is also updated)
- Double-Click "1D Property.1" icon on tree
- Click the icon \checkmark
- Change Radius from 1.5 to 1.75
- Click ok to complete



Non Commercial Use

Tutorial 3C

To Compute Analysis again:-

• Click "Compute" icon



- Click ok
- Click "Displacement" icon to view the update displacement
- (The maximum displacement is now decreased to 27.8mm after the modification of the metal arm)

** CLOSE ALL FILES WITHOUT SAVING**



END of Tutorial 3C

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Written by Dickson Sham

Tutorial 3D

We are going to estimate the deflection of the basket again by another FEA tool, MSC PATRAN...

Assumptions: (Same as Tutorial 3C)

- Linear Behavior of the material
- Displacements will be small such that a linear solution Is valid
- The spot weld joint between the basket and the metal arm will not break under the load
- Loading rate should be sufficiently low
- Load is uniformly distributed on the bottom faces of the basket
- The deformation of the basket is much lower than that of the metal arm



A- 95 Non Commercial Use

Written by Dickson Sham

Analysis by Patran

Tutorial 3D

(1) To prepare the 3D model for analysis:-

- Enter CATIA
- File/Open/ Basket_assm_a.CATProduct

Now, we should see the below elements only:

- One surface
- Six points (4 points on surface, 2 points on curve)
- One Curve (grouped)
- Hidden elements are:-
 - Upper_body.1 & Lower_body.1
 - "Sweep.2" under Basket.1
 - "PartBody" under Metal_arm.1



Tutorial 3D

Export the file in an IGES file:-

- File/Save as
- Select "igs" as File Type
- Enter "Basket_assm_a.igs" as File Name
- (Keep the file folder unchanged)
- Click "Save" to complete

File/Close/Basket_assm_a.Catproduct

Check and Re-save the IGES again:-

- File/Open/Basket_assm_a.igs
- (From the file, we can see that all elements are stored in the same level of the tree, and the product structure has been eliminated)
- Select "File/Save as" on the menu bar
- Select "igs" as File Type
- Select the file "Basket_assm_a.igs"
- Click "Save" and then "yes" to overwrite the file

File name:	Basket_assm_a.igs	-	Save
Save as type:	igs	•	Cancel
Save as new docu	iment		1.



Tutorial 3D

Template Database Name

C:WSC.SoftwareWSC.Patran\2005/template.db

Close CATIA

Enter MSC Patran

(1) Double-Click "Patran 2005 r2" icon on the desktop

(2) To Create a NEW Database:

- File/New
- Select Your project folder
- Enter "basket.db" as File Name
- Click OK
- Select "Based on Model" as Tolerance
- Select "MSC Nastran" as Analysis Code
- Select "Structural" as Analysis Type
- Click OK

(3) Import the model geometry:-

- File/Import
- Select "IGES" as Source
- Click "IGES Options"
- Click "Model Unit" and select "millimeters"
- Click Ok then YES to accept

Modify Preferen	ces			C Default
Look jn: 🛅 Proj	ect_basket		-	Approximate Maximum Model Dimension: 10.0
File name: Ba Files of type: Da	sket tabase Files (* db)	OK Cancel		Analysis Code: MSC.Nastran ▼ Analysis Type: Structural ▼
				OK Reset
ſ	Object: Mod	el ▼	<u> </u>	Model Unit Override
	Source: IGES			1000.0 (Millimeters)
	Current Group			Poort

Model Preference for:

and on Mode

OK

Basket.db

-Tolerance

Non Commercial Use

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default_group

IGES Options..

Written by Dickson Sham

Cancel

Tutorial 3D

- Select your project folder
- Select the file "Basket_assm_a.igs"
- Click APPLY
- (all elements in IGES have been imported; the white "+" is the model origin)

PATRAN Geometry	
Geometry Types	Quantity Created
Point	6
Curve	13
Surface	17

- To correct surface gaps, Click the icon "Geometry"
- Select "Edit/Surface/Sew"
- Select all surfaces
- Click APPLY
- Click "Yes for all"





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REMINDER

•The unit of the imported model is mm. To have the unit consistency in Patran, remember to use the SI(mm) units as shown

Quantity	SI	SI(mm)
Length	m	mm
Force	N	N
Mass	kg	$tonne~(10^3 kg)$
Time	s	S
Stress	$Pa ({ m N}/m^2)$	$MPa ({ m N}/mm^2)$
Energy	J	$mJ~(10^{-3}{ m J})$
Density	kg/m^3	$tonne/mm^3$

Tutorial 3D



Written by Dickson Sham

Tutorial 3D

(4a) To associate points to a curve:-

- Click "Geometry" icon on the top menu
- Select Action/Associate
- Select *Object/Point*
- Select Method/Curve
- Click the Entry Box of *Point List* Once, then . select a point on the curve1
- Click the Entry Box of *Curve List*, then select the curve1
- (If Auto-execute is checked, it is not necessary to click Apply)
- Click the box of *Point List* again
- Select another point on the curve13
- Click the Entry Box of Curve List, then select the curve13
- (If Auto-execute is checked, it is not necessary to click Apply)

Irve:- op menu	Geometry Elemen	its Loads/BCs	Action: Object: Method:	Associate Point Curve
		•	Point List	ecute
<i>t</i> Once, then	/		Curve List	
s <i>t</i> , then select			Curve 1	-Apply-
s not			12	
n ve13		curve1	Action: Object:	Associate
s <i>t</i> , then select			Method:	Curve 🔻
s not			Point List	
			Curve List	
A- 101	1			-Apply-
Non Comme	ercial Use			Written by Dickson Sham

Tutorial 3D

Analysis by Patran

Action: Associate Object: Point 💌 Method: Surface Auto Execute Point List Point 66 68 65 67 Surface List Surface 2 -Apply-

(4b) To associate points to a surface:-

- (Keep Action/Associate)
- Select Object/Point
- Select Method/Surface
- Click the Entry Box of *Point List* Once, then select a point on the surface
- Click the box again
- Press and hold "SHIFT" key
- Select another point on the surface
- (Repeat the steps until all 4 points are selected)
- Click the Entry Box of Surface List, then select the Surface (on which the 4 points are located)
- (If Auto-execute is checked, it is not necessary to click Apply)

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Tutorial 3D Action: Create (5a) To create a 1D Mesh:-Object: Mesh 餾 Click "Elements" icon on the top menu Geometry Elements Loads/BCs Ν. Type: Curve Select Action/Create Output ID List Select Object/Mesh Node 1 11 Element Select Type/Curve Select Bar2 as Topology Topology Bar2 Node Coordinate Frames... Click the Entry Box of Curve List Once, then Curve List select all the curves on screen (Do not select Curve 1:13 any surface edges) Global Edge Length Automatic Calculation **Deselect Automatic Calculation** 5.0 Value Enter 5 for the Global Edge Length Prop. Name: - None -Prop. Type: - N/A -Click **Apply** Select Existing Prop... Create New Property... Click "Node size" icon to make the nodes bigger ĩL, Apply-(If we look closer to the nodes, we can see that a node is created on the location of the associated point) A-103

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Tutorial 3D





Tutorial 3D

(Cont'):-

- Select "Define Terms.."
- Select "Create Independent" and then another node of the curve
- Select the corresponding node on the surface as Dependent Node
- (If Auto-execute is checked, it is not necessary to click Apply)
- Click "Cancel" 🔀 on the submenu
- Repeat the steps until all four MPCs are created
 - MPC.1
 - MPC.2
 - MPC.3
 - MPC.4



A I		Define Terms
	1	Dependent Terms (No Max)
Action:	Create 🔻	Nodes (1*)
Object:	MPC 🔻	233
Type:	Rigid (Fixed)	
A polyoio F		
Code:	references. : MSC Nastran	_ Independent Terms (1)
Type:	Structural	Nodes (1)
		8
место Та		
14		
	Define Terms	Create Dependent C Modify
	-Apply-	V Auto Execute
	γ	Node List
		Node 233
		Apply Clear Can
	\sim	
		MPC.4
	<	
	\mathbf{i}	
		MBCO
	WPC.3	
		1 + 1 + 1 + 1 + 1 + 1 + 1 + 1 + 1 + 1 +
		$(\rightarrow \checkmark \land \land$

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Tutorial 3D

To show the meshes only:-

- Click "Smooth Shaded" icon to switch to the shading mode
- Click "Plot/Erase" icon
- Select "Erase" under Geometry
- Click ok
- (Previously, the geometry and the meshes were overlapped together, but now only meshes are shown on the screen)



Viewport Dis

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Analysis by Patran

Selected Entities	
Plot	Erase
Coord	Frames
Posted Entities Geometry Plot	Erase
FEM Plot	Erase
All Plot	Erase
	ок


Tutorial 3D



Tutorial 3D

(Cont') :-

- Click "Select Application Region" icon
- Select FEM under Geometry Filter
- Click "Front View" icon
- Click "Polygon pick" icon

Click "Add" icon

Finally Click Apply

• Click at **1**, **2**, **3**

Click ok

• then Double-Click at **4** to select all nodes within the region

Application Region

Add

OK

Node 41:65

Remove

Select Nodes



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Tutorial 3D

(9) To create a Distributed Load :-

- Select Action/Create
- Select Object/Total Load
- Select Type/Element Uniform
- Click the entry box of New Set Name
- Enter "force" in the box
- Select **2D** as Target Element Type
- Click "Input Data..."
- Enter <0 0 -15> under Surf. Load (Patran will distributed the 15N load evenly over the area of the Application Region)
- Click ok

Action: Create Object: Total Load Type: Element Uniform	Load/BC Set Scale Factor 1. Surf Load <f1 f2="" f3=""> K 0 0 -15 ></f1>
Current Load Case: Default Type: Static	Edge Load <f1 f2="" f3=""></f1>
Existing Sets	Spatial Fields
New Set Name	FEM Dependent Data Analysis Coordinate Frame Coord 0 OK Reset
force	

Tutorial 3D

Plot/Erase

×

鹗

 \mathbf{O}

(Cont') :-

- Click "Plot/Erase" icon
- Select "Plot" under Geometry
- Click ok
- Click "Select Application Region" icon
- Select "Geometry" under Geometry Filter
- Click "Surface or Face" icon for Picking
 Filter
- Press and Hold "Shift" key on the keyboard
- Multi-select all bottom faces (9 faces)
- Click "Add" icon
- Click OK
- Click Apply







5.00

15.00

Tutorial 3D

(10)	10) To create Material Properties :-					
•	Click "Materials" icon on the top menu	nts Loads/BCs Materials				
•	Select Action/Create Select Object/Isotropic Select Method/Manual Input	Action: Create Object: Isotropic Method: Manual Input Property Name Value				
•	Enter "Iron" for Material Name Click on "Input Properties" Enter <u>1.2E5</u> and <u>0.29</u> for Elastic Modulus(N_mm2) and Poisson ratio respectively	Existing Materials Elastic Modulus = 1.2E5 Poisson Ratio = 0.29 Shear Modulus = Image: Control of the second				
•	Click ok Click Apply	Fitter Material Name Iron OK Clear				

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Tutorial 3D

(11) To create Element Properties :-

(FOR 1D MESH)

- Click "Properties" icon on the top menu
- Select Action/Create
- Select Object/1D
- Select Type/Beam
- Enter "metal_arm" as Property Set Name
- Click "Input Properties" icon
- Click the icon next to "Mat Prop Name" and then select Iron
- Click "Create Sections Beam Library"

Loads/BCs Materials	Properties			
Action: Create Object: 1D Type: Beam				
Prop. Sets By Name	Input Properties General Section Beam (CBAR) Property Name	Value	Value Type	
Filter *	[Section Name]	Da:	Properties V	T 1
Property Set Name	Material Name	m:lron	Mat Prop Name	N K
metal_arm	Bar Orientation		Vector	I
Ontions:	[Offset @ Node 1]		Vector	III
General Section	[Offset @ Node 2]		Vector	**
Standard Formulation	[Pinned DOFs @ Node 1]		String 🔻	
	[Pinned DOFs @ Node 2]		String 🔻	
Input Properties	Area		Real Scalar	
		Create Sections	-	

Tutorial 3D

- Enter "Cross_sect" as New Section Name
- Click the arrow icon \bigstar
- Select the cross-section with a solid circle
- Enter <u>1.5</u> as R
- Click ok
- Enter <1 0 0> as Bar Orientation
- Click the entry box of "Select Members" and then select the curve on the screen
- Click Add
- Click Apply
- (Optional) Display the cross-section by selecting...
 - Display/Load/BC/Elem. Props...using Beam Display/3D: Full-Span +Offsets.



Beam Library	
Action: Create Cbject: Standard Shape Method: NASTRAN Standard	
Existing Sections	
* Filter	
New Section Name	
	Spatial Scalar Fields
Calculate/Display	Write to Report File

Input Properties		
General Section Beam (CBAR)		
Property Name	Value	Value Type
[Section Name]	cross_sect	Dimensions 💌 🔳
Material Name	m:Iron	Mat Prop Name
Bar Orientation	<1 0 0>	Vector 🔻

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Tutorial 3D

(Cont') :-

(FOR 2D MESH)

- Select Action/Create
- Select Object/2D
- Select Type/Shell
- Enter "basket" as Property Set Name
- Click "Input Properties" icon
- Click the icon next to "Mat Prop Name" and then select Iron (it has been selected by default)
- Enter <u>1</u> as Thickness and Click OK $\stackrel{\checkmark}{\asymp}$
- Click the entry box of "Select Members" and then select all surfaces on the screen
- Click Add
- Click Apply

l I			
Action: Create	Properties		
Object: 2D 🔻			
Type: Shell	-		
Pron Sets By Nor			
	Input Properties Stan. Homogeneous Plate(CO	QUAD4)	
	Property Name	Value	Value Type
	Material Name	m:iron	Mat Prop Name
Filter	[Material Orientation]		
Property Set Nam	Thickness	1	Real Scalar 🔻 🚽
basket	[Nonstructural Mass]		Real Scalar
	[Plate Offset]		Real Scalar
Options:	[Fiber Dist. 1]		 Real Scalar
Homogeneous	IFiber Dist. 21		Real Scalar 3
Standard Formulat			

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Tutorial 3D



- Click "Load Case" icon on the top menu
- Select Action/Modify
- Select Load case **Default** from "Select Load Case to Modify"
- Check to see that the Total Load and the fixed constraint are assigned to the default load case
- Click Cancel





Tutorial 3D

(13) -	To Run the Analysis:-		
•	Click "Analysis" icon on the top menu	Load Ca Fields	sis
•	Select Action/Analyze	Action: Analyze 🔻	
•	Select Object/Entire Model	Object: Entire Model 🔻	Translation Parameters
•	Select Method/Full Run	Method: Full Run 🔻	OP2 VXDB Print Punch
			MASTER Only MASTER/DBALL
•	Click "Translation Parameters"	Code: MSC.Nastran	XDB Buffer Size: 1024 💌
•	Select XDB and Print	Type: Structural	
•	Click OK	Available Jobs	MSC.Nastran Solution Type
•	Click "Solution Type"		Solution Type:
•	Select "Linear Static" as Solution Type	T F	C LINEAR STATIC
•	Click Ok	Job Name	C NONLINEAR STATIC C NORMAL MODES
		basket	
•	Click Apply	Job Description MSC.Nastran job created on 15- Dec-06 at 17:47:22	C COMPLEX EIGENVALUE C FREQUENCY RESPONSE C TRANSIENT RESPONSE C NONLINEAR TRANSIENT
		Translation Parameters	C IMPLICIT NONLINEAR
	٨	Solution Type Direct Text Input	Select ASET/QSET
	A-	δ	

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(14) To Read the

٠

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Analysis by Patran

	l uto	orial	3D	Action:	Access Results
o Read the results:-		11		Object: Method:	Attach XDB Result Entities
Select Action/Access Re Select Object/Attach XDI Select Method/Result En	sults B tities	Select File	Fields Analysis	Code: Type:	MSC.Nastran Structural Jobs
Click " Select Results Fil Select the file "basket.xd Click Apply	e…" b" and Click OK	 ■ assm.xdb ■ basket.xdb ■ truss.xdb ■ truss.xdb ■ file name: basket.x Files of type: Files (*.x) 	adb 0 adb) v Car	K Job Nam basket Job Desc MSC.Na Dec-06	ription stran job created on 15- at 17:47:22
Click " Results " icon on the Select <i>Action/Create</i> Select <i>Object/Quick Plot</i>	ne top menu elds	Analy: Analy: Obje	sis Results		Select Results File
Nar08	A- Non Corr	119 Select Defa	t Result Cases ault, A1:Static Subcase USE		Written by Dickson Sham

(Con't):-

Select "Displacement, Translational" for Fringe Result

~_Y X

- Select "Displacement, Translational" for Deformation Result
- Click Apply

MSC.Patran 2005 15-Dec-06 19:04:48 9.16+001 Fringe: Default, A1:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED) 8.50+001 Deform: Default, A1:Static Subcase, Displacements, Translational, 7.85+001 7.19+001 6.54+001 5.89+001 5.23+001 4.58+001 3.92+001 3.27+001 2.62+001 1.96+00 1.31+001

The maximum **Displacement = 98mm**

Result by CATIA = 97mm

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Analysis by Patran

1		Select Fringe Result		
		Bar Stresses, Minimum Combined Constraint Forces, Translational Displacements, Translational Stress Tensor,		
		Quantity: Magnitude 🔻		
sult				
		Select Deformation Result		
		Constraint Forces, Translational		
		Displacements, Translational		
	9.81+001			
ED)	9.16+001			
	7.95+001			
	7.00+001			
	6 54+001			
	5 89+001	🔽 Animate		
	5.23+001			
	4.58+001			
	3.92+001	Apply		
	3.27+001			
	2.62+001			
	1.96+001			
	1.31+001			
	6.54+000			
	0.			
default_	_Fringe :			
Max 9.81	+001 @Nd 1845			
Min 0.@	PNd 42			
detault_	_Deformation :			
Max 9.81	+UUI @Nd 1845			

END of Tutorial 3D

Tutorial 3E

We know we can get a stronger metal arm by shortening its length and/or increasing its diameter. But what are their best values so that we can use the minimal material to support the load? (Less Volume Less Cost)

** File/Open... Analysis_a.CATAnalysis**

To Define a User Parameter "Volume":-

- Click "Formula" icon
- Select "Volume" as Type
- Select "Single Value"
- Click "New Parameter of Type"
- Rename "Volume.1" to "Metal_arm_volume"
- Click "Add Formula"





Tutorial 3E

(Con't):-

- Select "Part Measures" on the list of "Dictionary"
- **Double Click** "smartvolume(elem:solid,...)" under the list of "Members of Part Measures"
- (now Metal_arm_volume=smartvolume())
- Then click the space between two blankets





- Maximize the product tree and maximize the part tree of Metal Arm
- Click "Partbody" under *Metal_arm* once
- **Double-click** "Metal_arm/Partbody" on the list of "Member of all"
- Click ok to complete

Formula Editor : Metal_arm	_volume			? ×
				K 90
Metal_arm_volume			=	
smartVolume(Metal_arm\PartB	iody)			
Dictionary Parameters Design Table Operators Pointer on value functions Point Constructors Law Line Constructors	Members of Parameters All Renamed parameters String Boolean CstAttr_Mode Length Curve Constraint	Members of All Metal_arm\PartBody\Rib Metal_arm\PartBody\Rib Metal_arm\PartBody\Ski Metal_arm\PartBody\Ski Metal_arm\PartBody\Ski Metal_arm\PartBody\Ski Metal_arm\PartBody\Ski	0.2\ThickThin1 0.2\ThickThin2 0.2\Activity etch.3 etch.3\Coincidence.18 etch.3\Radius.19 0.2	×
			🌖 ок	Gancel

Tutorial 3E

(Con't):-

- (The system measured the volume of "Metal_arm" and return the value as 3.652e-006m^3)
- Click ok to complete

C	Double click on a parameter to edit it						
	Parameter	Value	Formula	Active			
	`Iron\Iron.1.1\Thermal Expansion`	1.21e-005					
8	`Iron\Iron.1.1\Yield Strength`	3.1e+008N					
	`Finite Element Model.1\Distributed Force.1\Force Vector.1\	ON					
	`Finite Element Model.1\Distributed Force.1\Force Vector.1\	ON					
	`Finite Element Model.1\Distributed Force.1\Force Vector.1\	-15N					
	`Finite Element Model.1\Energy\Energy`	0.4473	Valuated by :Energy				
	Metal_arm_volume	3.652e-006m3	= smartVolume(Metal_arm\P	yes	-		
E	Edit name or value of the current parameter						

To minimize the tree:-

• Click "+" next to "Link Manger.1"

To Display the User-defined Parameter "Volume" on the tree:-

- Select Tools/Options... on the menu bar
- Select "Analysis & Simulation" on the left list
- Select the tab page "General"
- Select "Show parameters"
- Select "Show relations"



(Con't):-

- Select "Parameters and Measures" on the left list
- Select the tab page "Knowledge"
- Select "With Value"
- Click ok to complete (Now we can see the user-defined parameter "volume" with its value on the tree)

binks Manager. 1 🖪 Analysis Connection Manager.1 Finite Element Model.1 Nodes and Elements Options 🕅 Surface Mesh.1 Knowledge Units Knowle Options 1D Mesh.1 1 Rigid Connection Mesh.1 🐖 General 1 🗊 Rigid Connection Mesh.2 🖬 With value 10 Rigid Connection Mesh.3 Display With formula 🔊 Rigid Connection Mesh.4 Compatibility Properties.1 🚔 Materials.1 Surrounded by the -14 🥝 User Material.1 C Devices and Virtu Relations update in part context 🔥 Static Case Creation of synchr 🕒 Restraints. 1 Infrastructure Loads.1 Creation of relation 🙀 Static Case Solution. 1 Mechanical Design 💑 Translational displacement vector.1 -K. Von Mises Stress (nodal values).1 - Stress principa epsor symbol.1 🗜 🔃 Sensors. 1 Parameters Metal_arm_volume=3.652e-006 🖁 🖥 Relations

To Create a measurement senor:-

- Right-click "Sensors" on the tree
- Select "Create Local Sensor"
- Select "Displacement Magnitude" on the list
- Click ok to complete



Design Optimization

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(Cont'):-

- Double-Click "Displacement Magnitude.1" on the tree
- Unhide the whole assembly
- Select the edge 🛠 as Supports (Test point)
- Select "Maximum" as Post-Treatment
- Select "Create Parameters"
- Click ok to complete



Design Optimization

Local Sensor	_ 🗆 🗙
Name Displacement Magnitude.1	
Supports 1 Edge	
Solution	
Solution Static Case Solution.1	
Values	
Position: Node (from solver)	<u>~</u>
Value type: Real	_
Complex part:	~
🖾 Do not combine	
Filters	
Show filters for: All	•
Axis system: None (Cartesian)	
Component: All	-
Layer:	-
🥥 Lamina: 📔 🛁 🔿 Ply id:	_
Prac-freatment Maximum	•
Create Parameters	
	Cancel



Now we have a sensor to measure the maximum displacement of the basket

Produck

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To Create a case of Optimization:-

- Select "Start/Knowledgeware/Product Engineering Optimizer" on the menu bar
- Click "Optimization" icon
 - (1) To define Free Parameters:-
 - (For this case, we choose
 - Diameter of the metal wire
 - Length of a portion of the metal arm)
 - Click "Edit List" icon
 - Maximize the tree so that we can see the Geometrical set.1 of Metal Arm
 - Click "Sketch.1" once (only related parameters are shown on the list)
 - Click on the dimension"100" or click "Offset.10" on the list (They are the same)
 - Click the arrow icon
 - Click ok to complete

Design Optimization

Tutorial 3E

- (Con't) :-
 - (Now a parameter is added on the list of Free Parameters)
 - Single Click on this parameter
 - Click "Edit ranges and step"
 - Select the box next to Inf. Range
 - Enter <u>40mm</u> as Inf. Range (lower limit)
 - Select the box next to Sup. Range
 - Enter <u>100mm</u> as Sup. Range (upper limit)
 - Select the box next to Step
 - Enter <u>0.5mm</u> as Step
 - Click ok to complete

Op	timization									? ×
	Problem	Constraint	s Com	putation:	s results	1				
	Optimization	type: Maxi	mization							-
	Optimized	parameter								
	?			0					Select.	
	Target valu	e: Maximum	-							
	Free Para	meters								
	Name		Value	Inf. I	Range	Sup. Ra	nge	Step		
	`Metal_arr	n\Geome	100mm					Auto.		
										- 11
2	/	Set R	ange			(Edit	ranges	and ste	ep
	۲ ∕ ۲									
A	X	40-10	umm	201	rithm					Ţ
			/	i	Modify	the rand	ies an	d the s	ten	2
	2	\checkmark			r Iodii y	che rung	jes ur		p	
	Ĩ				📔 Inf.	Range	40mr	n		-
					🔽 Sun	. Range	100m	100		
			allena.		- Jop					
	,atti			990m-	📕 Ste	P	0.5			ŧ
		4					0	OK I		
	00000	NG YOMONO, YOM	кыхмананан	ananan				OK I	- C	ancel

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(Con't):-

- Click "+" next to "Link Manger.1" to minimize the tree
- Click "Edit List" icon again
- Maximize the tree so that we can see the Properties.1 of Finite Element Model.1
- Click "1D Property.1" once (only related parameters are shown on the list)
- Click the one and the only parameter on the left list, which is the radius of the 1D mesh
- Click the arrow icon
- Click ok to complete

Design Optimization

Non Commercial Use

Tutorial 3E

- (Con't) :-
 - (Now the 2nd parameter is added on the list of Free Parameters)
 - Single Click on this parameter
 - Click "Edit ranges and step"
 - Select the box next to Inf. Range
 - Enter <u>1.5mm</u> as Inf. Range (lower limit)
 - Select the box next to Sup. Range
 - Enter <u>2mm</u> as Sup. Range (upper limit)
 - Select the box next to Step
 - Enter <u>0.1mm</u> as Step
 - Click ok to complete

timization	1						Ŷ
Problem	Constraint	s Comp	utations results	1			
ptimization	type: Maxi	imization					-
Optimized	parameter						
?			0				5elect
arget valu	e: Maximum	H					
Free Para	, meters —						
Name		Value	Inf. Range	Sup. Ran	ige S	tep	
`Metal_arı	m\Geome	100mm	40mm	100mm	0	.5mm	
Finite Elei	ment Mo	1.5mm	Umm	1.798e+	з Р	uto.	
Edit list					Edit r	anges (and step
	Set	Range					
	1.5	2.0mr	n Modify	the rang	es and	l the s	tep 🤶
		/					
			🔎 Inf	f. Range	1.5mm	h	
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- (2) To define Constraints:-
- Select the tab page "Constraint"
- Click "New..." icon
- Select "Displacement Magnitude" under Sensor.1 on the tree
- Then add "<=20mm" after the words
- Click ok to complete

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		ancel

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- (3) Define computation method:-
 - Select the tab page "Problem"
 - Select " Only Constraints" as **Optimization Type**
 - Select "Simulated Annealing Algorithm" as Algorithm type"
 - Set Termination Criteria: _
 - Enter 100 as Maximum number of а. updates
 - b. Enter <u>20</u> as Consecutive updates without improvements
 - Enter 3 as Maximum Time (minutes) C. (If any of these is fulfilled, the computation will stop)
 - Select "Save Optimization data" (so that we can see all raw data after computation)

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To RUN Optimization:-

- Click "Run Optimization" icon
- Enter "Data" as file name (all the raw data will be stored in this excel file)
- Click "Save" to start computation

(The computer starts to search all possible values of the two free parameters so that the constraint can be met)

(After 3 minutes)

To review the results:-

- Select the tab page "Computation Results"
- On the list, all attempts failed until #19 (i.e. Result - Target = 0)

	Optimization	? ×
	Problem Constraints Computations results	
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(Cont'):-

- Select "Show Curve" icon
- On the list, the four attempts failed but the rest can meet the requirement
 (i.e. Result Target = 0)

To Further optimize the parameters (for the minimum volume of metal arm):-

- Select the tab page "**Problem**" again
- Select "Minimization" as Optimization type
- Click "Select" icon

requirement. But which one can give the minimum volume?

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Choose the pa

Filter On Analy Filter Name : Filter Type :

Parameters

(Cont') :-

- Select "Metal arm volume" on the list
- Click ok to complete
- Select "Gradient Algorithm with ٠ **Constraints**" as Algorithm type"
- Set Termination Criteria: ٠
- Enter 100 as Maximum number of а. updates
- Enter 20 as Consecutive updates b. without improvements
- Enter 3 as Maximum Time (minutes) C. (If any of these is fulfilled, the computation will stop)

To RUN Optimization again:-

- Click "Run Optimization" icon
- Select Yes to overwrite the data file

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_	Edit list	dit ranges and step
•	Available algorithms Algorithm type : Gradient Algorithm With Constraint(s)	•
	Selected algorithm settings	
	Running Criteria Convergence speed : Fast	
	Termination criteria Maximum number of updates 100 Consecutive updates without improvements 20 Maximum time (minutes) 3	
	Optimization data Save optimization data	
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To review the result:-

- Select the tab page "Computation Results"
- Select "Select parameters" icon, then add "metal_arm_volume" onto the list "parameters to be displayed"
- Select "Show Curve..." icon

From the curve, the best values are: Wire Radius = 1.9mm Arm Length = 40mm

Its volume is the smallest but it is still so strong that the deflection is lower than the required limit.

In this revised optimization, all cases can meet the requirement (i.e. deflection of basket is smaller than 20mm)

To Save all files:-

• Select "File/Save all"

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END of Tutorial 3E

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