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Mastercam® X4 Training Tutorials - Solids Applications
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Software: Mastercam X4 Solids
Authors: Mariana Lendel

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Sincerely,
Mariana Lendel
HOW TO USE THIS BOOK

This book consist of six projects which are used to illustrate Solid commands. Each tutorial has easy to follow procedures with screen shots and menus from Mastercam.

This material discusses Mastercam features such as Extrude, Revolve, Sweep, and Boolean Remove & Add. You will create Solid Geometry from Surfaces, and you will be introduced to Loft Commands, Shell, Chamfer and Fillet features.

LEGEND:

⊙ Step to follow to complete the tutorial

✎ Additional explanation for the current step

☐ Callouts that give direction on how to complete the task

☐ Callouts that describe the parameters used in the current step

**Bold** text (usually) represents Mastercam terminology
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## Getting Started

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## General Notes

### Tutorial Quiz Answers

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TUTORIAL # 3
2D WIREFRAME & SOLID GEOMETRY USING EXTRUDE (CREATE BODY), SWEEP (CUT BODY), EXTRUDE (ADD BOSS), MODIFY A FACE DRAFT ANGLE, CHAMFER FOUR EDGES AT 45° ANGLE, FILLET THE EDGES OF THE TOP FACE AND HOLLOW OUT THE SOLID USING SHELL COMMAND.
Objectives:

The Student will design 2-D wireframe in Top Construction Plane using:

- Create rectangle knowing the width and the height of the rectangles and a point placement.
- Create arc knowing the center point, the radius, initial angle and final angle.
- Create line polar knowing the starting point, the angle and the length of the line.
- Create line knowing the endpoints of the line.

The Student will design 2-D wireframe in Front Construction Plane using:

- Create arc knowing the start point, the radius, initial angle and final angle.

The Student will create the solid on Level 2 using:

- Level command to change the main level.
- Create a solid by extruding a closed contour.
- Cut the solid using sweep-cut body command.
- Add a boss using extrude command with trim to a face option.
- Change one face draft angle using draft face command.
- Chamfer at 45 °, four edges knowing the distance.
- Fillet the top face of the solid.
- Hollow the solid at a given thickness.
- Make the wireframe invisible to generate the solid.
FACE A HAS A DRAFT ANGLE OF 20 DEGREES.
FACE B, C AND D HAVE A DRAFT ANGLE OF 10 DEGREES.

TITLE: TUTORIAL #3

MATERIAL: ALUMINUM 6061

DATE: NOV 23, 2001 eMastercam.com
WIREFRAME CREATION
Useful tools to start the geometry creation

Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the solid geometry. See Getting started page A-4.

Before starting the geometry make sure that the Grid is enabled. It will show you at each moment where is the part origin. See Getting started page A-5 for details.

Select the Solids manager to the left of the screen. If it is hidden press Alt + O to display it.
STEP 1: TOP CONSTRUCTION PLANE. CREATE TWO RECTANGLES KNOWING THE POINT PLACEMENT, THE WIDTH AND THE HEIGHT OF THE RECTANGLES

Step Preview:

Sub Step Preview 1.1

Create

Rectangular Shapes

Enter the **Width** and the **Height** as shown in the following screenshot.

Select the lower left radio button as the anchor.
[Select position of base point]: Select the Origin (in the center of the grid).

Select the Apply button to continue using the same command.

Change the depth of construction plane to Z 1.35 by entering the value in the Status Bar.

Sub Step Preview 1.2

Enter the Width and the Height as shown in the following screenshot.

Select the lower left radio button as the anchor.

[Select position of base point]: Select the Fast Point icon.

Enter the values: 0.5, 0.375 [Enter]

Select the OK button to exit the command.
STEP 2: CHANGE THE CONSTRUCTION PLANE TO THE RIGHT SIDE AND THE CONSTRUCTION DEPTH TO Z 3.5

- Change the **Graphic View** to **Isometric**

- Using the **Fit** icon to fit the drawing to the screen.

- Select the **drop down arrow** as shown to the right and select **Set planes** to **RIGHT**.

- Change the depth of plane to **Z 3.5** by entering the value in the **Status Bar** (Enter)

*The drawing should look as shown to the right*


*Step Preview:*
Create

- Arc
- Arc Polar

Enter the Radius **4.0**

- To set the other parameters of the arc use **Tab** key. Note that the diameter value is automatically changed by the system based on the radius.

Enter the Start Angle **50** (Tab)

Enter the End Angle **130** (Enter)

[Enter the center point]: Select the **Fast Point** icon

Enter the value **1.5, -3.0**. (Enter)

Select the **OK** button to exit the command.

Using the **Fit** icon to fit the drawing to the screen.

The drawing should look as shown in the picture to the right.

**STEP 4: CREATE TWO LINES KNOWING THE START POINT, ANGLE AND LENGTH**

*Step Preview:*
Create

- Line
- Line Endpoint

Enter the line Length: 2.0

- Enter the Angle: 90 (Enter)

- [Specify the first endpoint]: Select the Endpoint A

  Make sure that the Endpoint of the entity is selected. You should see the cursor icon for Endpoint before you click to select the point.

- Select the Apply button to continue using the same command.

- Click on the Length and Angle icons to maintain the values for the second line.

  Note that the values are highlighted in red.

- [Specify an endpoint]: Select the Endpoint B

- Select the OK button to exit the command.

STEP 5: CREATE LINE ENDPOINTS

Step Preview:
Create

Line

[Specify the first endpoint]: Select the Endpoint A

[Specify the second endpoint]: Select the Endpoint B

Select the OK button to exit the command.

The drawing should look as shown in the following picture.

**STEP 6: CHANGE THE CONSTRUCTION PLANE TO FRONT AND THE CONSTRUCTION DEPTH TO Z 1.5**

Select the drop down arrow and Set planes to FRONT.

Change the depth of the plane to Z -1.5 by entering the value in the Status Bar.

Step Preview:

Create

Arc

Arc Polar Endpoints

Make sure that Start point icon is enabled in the Ribbon bar.

Enter the Radius 8.0

To set the other parameters of the arc use Tab key. Note that the diameter value is automatically changed by the system based on the radius.

Enter the Start Angle 75 (Tab)

Enter the End Angle 105 (Enter)

[Enter the start point]: Select the Midpoint of the arc as shown in the following picture.

Make sure that Midpoint cursor icon is on.

Select the OK button twice to exit the command.

Using the Fit icon to fit the drawing to the screen.
The drawing should look as shown in the following picture.

**STEP 8: SAVE THE FILE**

File
- Save As
  - Enter the name in the **File Name**: “Wireframe_3”
  - Select the **Save** icon.

**SOLID CREATION**

- Click on the line at the top of the Solid toolbar as shown.
- Move the bar inside the graphic area.
STEP 9: CHANGE THE MAIN LEVEL TO LEVEL 2

Levels allow you to efficiently organize the file. A Mastercam file can contain separate levels for wireframe, surfaces, solids, and toolpaths. By organizing your drawing into levels, you can more easily control which areas of the drawing are visible at any time. It also allows you to easily select entities that have to be changed. You are always drawing on the main level. There are 255 levels in Mastercam.

Select **Level** in the **Status Bar**.

Change the **Main Level Number**: to 2 and give the **Name** “Solids”.

Select the **OK** button to exit.

STEP 10: CREATE A SOLID USING EXTRUDE COMMAND

*Step Preview:*

![Step Preview Diagram]
Solids Extrude – Draft: Tilts the walls of the extruded solid by the defined angle.

Solids

- Extrude
  - Select the chain as shown in the picture.

- The arrow should point upwards (as in the picture above), if not enable Reverse direction.

- Select the OK button to exit the Chaining dialog box.

- Select Create Body
- Select Extend by specified distance
- Enter the Distance: 1.5
- Enable Draft and disable Outward
- Enter Angle: 10

- Select the OK button.
The drawing should look as shown in the following picture.

**STEP 11: CUT THE SOLID USING SWEEP COMMAND**

*Step Preview:*

**Solids Sweep:** The solids sweep function lets you sweep closed, chains of curves to create cuts on an existing body or bosses. Mastercam sweeps the selected entities by translating and rotating the shape along the entire distance of a single chain of curves, called the path chain.
Solids

Sweep

- [Select chains to be swept]: Select the chain as shown in the following picture.

- Select the OK button to exit Chaining.

- [Select chain for sweep path]: Select Entity A

- Select the Cut Body option.

- Select the OK button.
The drawing should look as shown in the following picture. You can press Alt + S keys simultaneous at any time to see the part shaded.

Press Alt + S keys simultaneously to return to the wireframe display.

**STEP 12: ADD A BOSS USING EXTRUDE COMMAND**

*Step Preview:*
Solids

Select the chain as shown in the picture to the right

Select the OK button to exit Chaining dialog box.

The arrow should point inwards (as in the above picture), if not enable reverse direction (See directions below.)

Reverse direction.

- Choose Add Boss.
- Disable Draft
- Select Extend by specified distance
- Distance: 1.0
- Select Trim to selected Face(s)
- Select the OK button.

[Select faces to extrude to]: Select the top face as shown in the following picture.
Select End Selection button.

The drawing should look as shown in the following pictures

STEP 13: CHANGE “FACE A” ANGLE USING DRAFT FACES COMMAND

Step Preview:

Select this Face

[Select faces to draft]: Select the face as shown in the picture to the right.
Enable Draft to Face

Draft Angle: 20

Select the OK button to exit Draft Face Parameters dialog box.

Enable the Select from back button in the Ribbon Bar.

Select planar face specifying draft plane: Select the bottom face as shown in the picture below.

The arrow should point upwards (as in the picture to the right), if not

Select Reverse It button in the Draft Directions

Select the OK button to continue.
The drawing should look as shown in the following pictures

**STEP 14: CHAMFER, AT 45 DEGREES THE FOUR EDGES OF THE PART KNOWING ONE DISTANCE**

*Step Preview:*

Enable the `Select edge` button and disable `Select face and solid` buttons in the Ribbon Bar.

[Select entities to chamfer]: Select the four edges: Entity A, Entity B, Entity C & Entity D
Select End Selection button.

Enter the Distance: 0.1

Select the OK button to exit Chamfer Parameters.

The drawing should look as shown in the following pictures

STEP 15: FILLET THE TOP FACE WITH A 0.25 RADIUS

Step Preview:

Enable only Select edge and make sure that Select face and solid buttons are disable in the Ribbon Bar.
Enable **Toggle Verify Selection** button.

The **Toggle Verify Selection** option detects multiple solutions (based on the cursor position) when you make a selection. It cycles through the possible solutions. When the intended selection is highlighted in the graphics window, choose **OK** button to select it.

[Select entities to fillet]: Select the edge shown to the right

Select **Forward button** till the system highlights the desired edge.

Continue to select the other edges of the top face that are highlighted in the picture to the right.

Select **End Selection** button.

Enter the **Radius**: 0.25

Select the **Mitered corners** option

Select the **OK** button.

Mitered corners option allows you to miter three or more filleted edges that meet at a vertex. Mastercam extends each fillet to the extent of the edge.

The part should look as in the following pictures.
STEP 16: HOLLOW OUT THE SOLID AT A THICKNESS OF 0.0625

**Step Preview:**

**Solids Shell:** Lets you hollow solid bodies by removing material and optionally leaving selected faces open. The remaining faces are thickened by a specified amount.

**Solids**

- **Shell**
  
  - Make sure that only **Select face** button is enabled.

- Enable **Select from back** button.

- [Select body or face to be left open]: Select the bottom face as shown in the picture to the right.

- Select **End Selection** button.
Select **Inward** for the shell direction.

**Shell Thickness**: 0.0625

Select the **OK** button to exit Shell Solid dialog box.

*The final part should look as in the following pictures*

![Series of pictures showing the final part]

**STEP 17: HIDE THE WIREFRAME GEOMETRY USED TO GENERATE THE SOLID AND DISPLAY ONLY THE SOLID USING LEVEL MANAGER OPTIONS**

Select **Level** from the Status Bar.

Make **Level 1** invisible by clicking on the check mark next to **Level 1** in the **Visible** column as shown.

By doing this you will keep only the solid on the screen. To make the wireframe visible again select the box next to **Level 1** in the **Visible** column.

Select the **OK** button to exit.
Select the Repaint button.

The final part should look as shown in the following picture.

STEP 18: SAVE THE FILE

File
- Save As
  - Save this file under “Solid_3”
  - Select Save icon.
The draft surface height is 2.000 " and has a draft angle of 5 deg.
The top fillet radius is .100
The second fillet radius is .250
TUTORIAL 3 QUIZ

What does a solid extrude draft allow the user to do?

How does the solids sweep operation work?

What does a solid sweep allow the user to achieve?

What does a Solids Shell operation allow the user to do?