

camInstructor

Objectives

You will create the geometry for Solids-Lesson-11, and then generate the solid from the geometry. This Lesson covers the following topics:

Create a 2-dimensional drawing by:

Creating lines Trimming geometry Mirroring geometry

Create a solid model: Extrude and Create a solid body Extrude and Cut a solid body

SOLIDS-LESSON-11 DRAWING



Solids-Lesson-11- The Process

Geometry and Solid Creation

- **TASK 1:** Setting the environment
- **TASK 2:** Create contour geometry
- TASK 3: Mirror geometry
- TASK 4: Extrude solid
- TASK 5: Extrude base solid
- **TASK 6:** Create the solid cut geometry
- **TASK 7:** Cut the solid holes
- **TASK 8:** Save the drawing

Geometry Creation

TASK 1: SETTING THE ENVIRONMENT

- Before starting the geometry creation you should set up the grid, toolbars and machine type as outlined in the Setting up the Environment section at the beginning of this text:
- 1. Set up the Grid. This will help identify the location of the origin.
- 2. Customize the toolbars to machine a 2D part.

TASK 2: CREATE CONTOUR GEOMETRY - X0 Y0 LOWER LEFT CORNER

- In this task you will create the geometry for the contour.
- 1. Select the Front Plane



3. On the graphics screen you are prompted "**Specify the first endpoint**" and the Line ribbon bar appears. Click on the **Multi-line** icon if it is not already depressed.

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4. To satisfy this first prompt click on the **FastPoint** Icon on the Auto Cursor ribbon bar and input **0,0** then hit the **enter key**.

0,0	-9

5. The prompt will change to **"Specify the second endpoint"**. Click on the **Fastpoint** icon again, input a value of **0,0.5** and hit the **enter key**.

0,0.5

6. Select the Screen Fit icon found at the top of the screen to fit the part to the screen

-9



Select the **Un Zoom 50%** icon found at the top of the screen
 The line just created should appear as below:

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9. Select the Fastpoint icon, enter the value 0.115,0.5 and then press the enter key.
 10. Select the Fastpoint icon, enter the value 0.115,0.975 and then press the enter key.
 11. Select the Fastpoint icon, enter the value 0,0.975 and then press the enter key.
 12. Select the Fastpoint icon, enter the value 0,1.255 and then press the enter key.
 13. Select the Fastpoint icon, enter the value 0.455,1.255 and then press the enter key.
 14. Select the Fastpoint icon, enter the value 0.455,1.530 and then press the enter key.
 15. Select the Fastpoint icon, enter the value 0.295,1.530 and then press the enter key.
 16. Select the Fastpoint icon, enter the value 0.295,1.890 and then press the enter key.
 17. Select the Fastpoint icon, enter the value 0.455,1.340 and then press the enter key.
 18. Select the Fastpoint icon, enter the value 0.905,1.340 and then press the enter key.
 19. Select the Fastpoint icon, enter the value 0.945,0 and then press the enter key.
 21. Select the Fastpoint icon, enter the value 0.945,0 and then press the enter key.
 22. Select the Fastpoint icon, enter the value 0.945,0 and then press the enter key.
 23. Click on the OK icon

24. The chain should appear as shown in the image below:



25. Select Create>Line>Endpoint...

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26. On the graphics screen you are prompted "**Specify the first endpoint**" and the Line ribbon bar appears. Select the end point of the second line as shown in the left image below:



27. The prompt will now read **"Specify the second endpoint"**. To satisfy the prompt, select the horizontal line icon on the ribbon bar and select any point to the right of the chain already created, as shown in the left image above.

28. Click on the OK icon . 29. Select Edit>Trim/Break>Trim/Break/Extend...

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	Trim / Break	Þ	8 4	Trim / Break / Extend
*	Join entities		8 #	Trim Many
~*	Modify Spline		*	Break Two Pieces
8,	Convert NURBS		*	Break at Intersection
診	Simplify		**	Break Many Pieces
				Break Drafting into Lines
Ŵ	Set Normal		٥	Break Circles
Ŵ	Change Normal		Q	Close arc

30. The **Trim/Extend** ribbon bar will appear with the prompt "**Select the entity to trim/extend**". Select the horizontal line first, then select the vertical line second, as shown in the image below:



31. Click on the **OK** icon 32. The geometry should now appear as in the image below:



TASK 3: MIRROR THE GEOMETRY



2. The prompt "Mirror: select entities to mirror" appears in the graphics screen. To satisfy the prompt, click on the All button in the selection toolbar. The Select All window will appear, select Wireframe as shown below:

Select	All 🔀
	All Entities
	×form Result
	×form Group
	Group Manager
	Entities
	Color
	Level
	Width
	Style
	Point
	Miscellaneous
J	* Ø
	Wireframe Points Lines Arcs Splines

- 3. Click on the **OK** icon
- 4. Click on the End Selection icon
- 5. The **Mirror** window will now appear. Make sure **Copy** is selected and then click on the **Select Line** icon to select geometry for the mirroring axis:



- 6. Select the vertical line shown in the right image above.
- 7. Click on the **OK** icon **I** in the **Mirror** window.
- 8. In the main screen, select the Clear Colours icon



9. The image below displays the mirrored geometry:



TASK 4: EXTRUDE SOLID

1.	Select Solid-Extru	ue.
Solid	ds Xform Machine Type To	
	Extrude 🗲 🗕	
3	Revolve	
C	Sweep	
4	Loft	
	Fillet 🕨 🕨	
	Chamfer 🕨 🕨	
	Shell	
4	Trim	
-	Thicken	
	Remove Faces	
	Draft Faces	
P		

- On the screen you will now see the Chaining dialog box, with the Chain Button selected as shown by the arrow. In the graphics screen a prompt to "Select chain(s)" to be extruded is displayed.
- 2. Select the chain as shown in the **Graphics Screen** diagram:



- 4. On the screen the **Extrude Chain** window will be displayed. Make sure the following items are checked and values are entered;
 - 1. Create Body is selected.
 - 2. Extend by specified distance is selected and the value 1.90" is entered.

3. Note the **arrow** direction. If the arrow is not pointing in the proper direction, select the **Reverse direction** check box or use the **Re-Select** option.

trude Chain	
Extrude Thin Wall	
Name Extru	de
Extrusion Operation Create Body Cut Body Add Boss Combine Ops	Draft Draft ✓ Outward Angle: 5.0
Extrusion Distance/Di	rection
Extend by specified Distance:	1 distance
Extend through all Extend to point Vector: Re-Select Trim to selected Fa Reverse direction Both directions	0, 0, -1 ce(s)



- 5. Click on the **OK** icon to complete this feature.
- 6. To change from a **wireframe image** to a **solid image** select the **Alt and S** or the **Shading** icon. Your screen should look like the screenshots below:



TASK 5: EXTRUDE BASE SOLID

1. Select Solid>Extrude...

Solic	s Xform Machine Type To						
	Extrude 🗲						
3	Revolve						
C	Sweep						
4	Loft						
	Fillet 🕨 🕨						
	Chamfer 🕨 🕨						
	Shell						
4	Trim						
-	Thicken						
	Remove Faces						
	Draft Faces						

- On the screen you will now see the Chaining dialog box, with the Chain Button selected as shown by the arrow. In the graphics screen a prompt to "Select chain(s)" to be extruded is displayed.
- 2. Select the chain as shown in the Graphics Screen diagram:



- 4. On the screen the **Extrude Chain** window will be displayed. Make sure the following items are checked and values are entered;
 - 1. Create Body is selected.
 - 2. Extend by specified distance is selected and the value 2.965" is entered.

3. Note the **arrow** direction. If the arrow is not pointing in the proper direction, select the **Reverse direction** check box or use the **Re-Select** option.



- 7. The prompt "Select target body for Boolean operation".
- 8. Select both of the solids as shown in the image below:



TASK 6: CREATING THE SOLID CUT GEOMETRY

- 6 6 ନ୍ତି 6÷ 1. Select the Right plane icon 2. Select Create>Arc>Circle Center Point... Create Solids Xform Machine Type Toolpaths Screen Art Settings Point i 🔁 🔎 🗞 🌾 🕄 📢 ۲ Line ۲ 门 Circle Edge Point... Arc Fillet 📑 Circle Center Point... < 5 Chamfer Arc Polar... Spline Arc Polar Endpoints... Arc Endpoints... Curve Arc 3 Points... Surface Drafting Arc Tangent....
- 3. The **Circle Center Point** ribbon bar appears and you are prompted to "**Enter the center point**".

X 0.2525 V X 3.8775 V Z 0.0 V V X 2 AL.	any In	00 2 00
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- 4. To satisfy this first prompt click on the FastPoint Icon on the Auto Cursor ribbon bar
- 5. In the space input the **-0.955,0.738** values for the center of the circle and hit **enter**. **Note** that there is a comma between the X and Y values, and you do not need to input the Z value for this example.

-0.955,0.738				
		$\overline{}$	7	

6. You have now input the coordinates for the center of this circle and you now need to supply the value for the diameter. Click in the space for diameter (shown below) and enter a value of **0.3125**.

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	7. 3. 3 .	Click on Select th Select C	the OK ne Top p reate>A	ico olar	on to complete this feature.
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		Fillet		•	🧭 Circle Center Point 🗲 🗕
		Chamfer		•	(R) Arc Polar
		Spline		•	🟫 Arc Polar Endpoints
		Curve		. ▶	🛟 Arc Endpoints
		Surface		•	🛟 Arc 3 Points
		Drafting			Arc Tangent

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10. The **Circle Center Point** ribbon bar appears and you are prompted to **Enter the center point**.

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- 11. To satisfy this first prompt click on the FastPoint Icon on the Auto Cursor ribbon bar
- 12. In the space input the **0.945,-0.955** values for the center of the circle and hit **enter**. **Note** that there is a comma between the X and Y values, and you do not need to input the Z value for this example.

0.945,-0.955	T	- 2	j
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13. You have now input the coordinates for the center of this circle and you now need to supply the value for the diameter. Click in the space for diameter (shown below) and enter a value of **0.3125**.

	0.15625	~	\bigcirc	0.3125	~	←
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14. While still in the Circle Center Point command, select the FastPoint Icon on the Auto



15. In the space input the **0.945,-2.400** values for the center of the circle and hit **enter**. **Note** that there is a comma between the X and Y values, and you do not need to input the Z value for this example.

```
0.945,-2.400
```

- 16. Click in the space for diameter and enter a value of 0.3125.
- 17. Click on the **OK** icon to complete this feature.
- 18. Select the Isometric view to see the placement of the circles created





TASK 7: CUT THE SOLID HOLES

1. Select Solid>Extrude...

s Xform Machine Type To			
Extrude 🗲 🗕			
Revolve			
Sweep			
Loft			
Fillet 🕨 🕨			
Chamfer 🕨 🕨			
Shell			
Trim			
Thicken			
Remove Faces			
Draft Faces			

2. On the screen you will now see the **Chaining** dialog box, with the **Chain Button** selected as shown by the arrow. In the graphics screen a prompt to "**Select chain(s)**" to be **extruded** is displayed. Select the chains, the three circles as shown in the **Graphics Screen** diagram:



- 4. On the screen the **Extrude Chain** window will be displayed. Make sure the following items are checked and values are entered;
 - 1. Cut Body is selected.
 - 2. Select Extend through all.

3. Note the **arrow** directions. If the arrows are not pointing in the proper direction, select the **Reverse direction** check box or use the **Re-Select** option.

	Extrude	Chain	Window
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Extrude Chain				
Extrude Thin Wall				
Name Extrude Cut				
Extrusion Operation □ Draft ○ Create Body □ Draft 1 ⊙ Cut Body ○ dut Body ○ Add Boss □ Combine Ops				
Extrusion Distance/Direction C Extend by specified distance Distance: 2965				
2 Extend through all Extend to point				
Vector: 1, 0, 0				
Trim to selected Face(s) Reverse direction Both directions Split draft				
× × ?				

Arrow Direction Screen

- 5. Click on the **OK** icon to complete this feature.
- 6. To change from a **wireframe image** to a **solid image** select the **Alt and S** or the **Shading** icon. Your screen should look like the screenshots below:



TASK 8: SAVE THE DRAWING

- 1. Select File.
- 2. Select Save as...
- In the File name box, type Solids-Lesson-11.
 Save to an appropriate location.
- 5. Select the green check mark button is ave the file and complete this function.

File name:	SOLIDS-LESSON-11.MCX
Save as type:	Mastercam X Files (*.MCX)



