Exercise 3

In this exercise we will create mechanism connections for an assembly that is designed using the Standard Skeleton model. We will also see how to simulate a mechanism in the sketcher.

Set the working directory to CRANK_SLIDER folder and open the assembly **CRANK_SLIDER.ASM**

It will appear as shown below.



All the components have been placed in the assembly by using the Default constraint.

Open the skeleton model in a new window.

Select the "Sketch 1" and pick $\overset{\checkmark}{=}$ and sketch will appear as shown below.





Select the "60" and "130" dimension values and pick **Lock** in the short-cut menu.

The locked dimensions will appear in different color as shown below.



Now take the mouse pointer over the left line as shown below.



Drag the line either to left or right and rotate the mouse pointer around the end point of line. Notice that sketched entities will simulate the motion of mechanism as shown below.



Pick 🔊 to undo all drag section operations.

Pick \checkmark to apply the changes and exit sketcher.

Locking of dimensions has no effect on the parts that are dependent on the sketch. We did so just to simulate the mechanism motion in the sketcher.

➡ Creating Mechanism Connections

Now we will change the assembly constraints to mechanism connections. After this we will be able to simulate the mechanism.

Before proceeding hide the 01__ASM_ALL_DTM_PLN layer to unclutter the screen.

BASE.PRT is the ground body in the mechanism so we do not need to change the assembly constraints as it will not move with respect to the assembly.

Select the DRIVING_ARM.PRT and pick 🧉.

Right-click the Default constraint in the Placement slide-up panel and pick Delete

Select the Pin connection from the list.

Select the **A_4** axis from the component as shown below.



Notice that the selected axis belongs to external copy geometry feature. Although we can select an axis from the part geometry but it is not as stable reference as the axis from external copy geometry. So it is preferable to select the references copied from skeleton.

Select the **A_4** axis from the skeleton model as Assembly reference.



Now system will activate the Translation constraint so select the **FRONT** datum plane as component reference and **SKL_FRONT** as assembly reference as shown below.

Connection_1 (Pin) Axis alignment	🗹 Constraint Enabled	
➡ Translation	Constraint Type	
DRIVING_ARM:FRONT:F3(D	Coincident 👻	
CRANK_SLIDER_SKEL:SKL_FI	Offset	
Rotation Axis	0.00 ¥ Flip	
New Set	Status Connection Definition Complete.	

Pick **v** to apply the changes and exit the dashboard.

Select the SLIDING_ARM.PRT and pick

Right-click the Default constraint in the Placement slide-up panel and pick Delete

Select the Cylinder connection from the list.

Select the **A_5** axis from the component as shown below.



Select the **A_5** axis from the DRIVING_ARM as Assembly reference.



Pick on the New Set (text icon) to create a new connection.

Connection_2 (Cylinder)	🗹 Constraint Enabled	
Axis alignment	Constraint Type	
DRIVING_ARM:A_5(AXIS):F5	Coincident 💌	
• Translation Axis	Offset	
Rotation Axis	0.00 v Flip	
New Set		

Select the **Bearing** connection from the list.

Placement	Move	Option	s	Flexibility	Properties
 Connection Connection Point alignment 	n_2 (Cylinder) n_18 (Bearing) Inment		Constrain	traint Enabled	
Select c	omponent item	1	Coi	ncident 💌	
Select a	ssembly item				

Select the end vertex of the datum curve from the SLIDING_ARM as shown below.



Select the horizontal curve form the skeleton model as Assembly reference. The desired curve is shown in the figure below.



Pick \checkmark to apply the changes and exit the dashboard.

Action 2 Moving the Bodies

Now we will use the drag functionality to move bodies. Before that hide the 07___ASM_ALL_SKELETONS and 03___PRT_ALL_CURVES layers.



Pick on any entity belonging to SWIVEL_ARM as shown below.



Now move the mouse to drag the selected body. After moving towards left it should appear as shown below.



If the mechanism moves correctly it means you have successfully completed the exercise.

Exercise 4

In this exercise we will learn how to use the motion skeleton to incorporate motion at the top level of a design project.

We will create a four bar mechanism driven by the skeleton as shown below.



Set the working directory to FOUR_BAR folder and open the assembly FOUR_BAR.ASM

➡ Creating Motion Skeleton

Pick the Create a component icon El Create

Component Create dialog box will appear. Here change the Type to **Skeleton Model** and Subtype to **Motion** as shown below.

Type Part Subassembly Skeleton Mo Bulk Item Envelope	y odel	Sub-type O Standard Motion Body
Name:	МОТ	10N_SKEL_0001
Common name:		

Change the Name to **MOTION_SKEL** and pick

Pick OK tab and Create Options dialog box will appear. Pick Browse... tab and select the TEMPLATE_ASS.ASM as shown below.

creation op	tions
Creation Method	
 Copy from existing 	
O Locate default datums	
O Empty	
O Create features	
template_ass.asm	Browse.
	11

TEMPLATE_ASS.ASM is the template file that contains the default datum features, layers and accuracy settings.

Pick OK and newly created motion skeleton model will appear as the first feature in the model tree as shown below.



Notice that motion skeleton model is a subassembly.

Motion skeleton models are subassemblies that contain conceptual design of a mechanism and body skeletons.

You should notice that motion skeleton has a unique icon in the model tree and is different from the Standard skeleton.



Icon for Standard Skeleton

Icon for Motion Skeleton

➡ Creating Conceptual Sketch for Mechanism

Open the skeleton model in new window and notice that it has the default datum planes and coordinate system.

Now we will create the conceptual mechanism by creating a sketch feature.

Pick $\widehat{\mathbb{N}}$ and select the sketching references as shown below

	Sketch	×
Placement	Properties	
Sketch Plan	e	
Plane	ASM_FRONT:F3(DATUM PLANE)	Use Previous
Sketch Orie Sketch vie	entation Flip	
Reference	ASM_RIGHT:F1(DATUM PLAN	IE)
Orientatio	n Right 🔻	
		Sketch Cancel

Now sketch the section as shown below.





➡ Creating Body Skeletons

In the following section, we will create body skeletons.

First of all we will create the ground body of the mechanism.

Pick the Create a component icon Greate

In Component Create dialog box, change the Type to Skeleton Model and Sub-type to Body.

	Create	Component X
Type O Part O Subassembl O Skeleton Mo O Bulk Item O Envelope	y odel	Sub-type O Standard O Motion O Body
Name: Common name:	BOD	Y_SKEL_0001
Common name:		<u>O</u> K <u>C</u> ancel



In the Create Options dialog box, pick Browse... tab and select the TEMPLATE_SKL.PRT.

Creation Opti	ions	×
Creation Method		
Copy from existing		
O Locate default datums		
O Empty		
O Create features		
Copy From		
template_skl.prt	Browse.	

Pick OK and BODY DEFINITION dialog box will appear.

Select the highlighted entity of the sketch shown in the figure below.

BODY DEFINITION	x
References Properties	
Chains	
1 One-by-One Chain	
	Details More
ОК	Cancel

Pick OK to complete the definition of the body.

The first body is always categorized as Ground Body. There are no connection definitions for the ground body.

Now we will create the second body skeleton.

Pick the Create a component icon Greate

In Component Create dialog box, change the Type to Skeleton Model and Sub-type to Body

OK

Change the Name to **BODY_SKEL_2** and pick

In the Create Options dialog box, pick Browse... tab and select the TEMPLATE_SKL.PRT as shown below.

Pick OK and BODY DEFINITION dialog box will appear .

Notice that now the BODY DEFINITION dialog box is different than the dialog box for ground skeleton.

Select the highlighted entity of the sketch shown in the figure below.

	BODY DEFINITION	×
	References Properties Chains	
	1 One-by-One Chain	Details More
	✓ Use connections in placeme Reference Conne	ent definition
		Þ
7	Update	ent dialog
	ОК	Cancel

Pick **Update** and system will apply the appropriate connection for this body skeleton as shown below.

Chains	perties	
1 One-by-One Cl	nain	Details More
✓ Use connection Reference	ons in placeme	ent definition
End:Curve:F5	Pin	ASM_FRON
4		Þ
1	Update	

System automatically creates the required axes for the connections in model tree.

Pick OK to complete the definition of the body.

Now we will create the third body skeleton.

Pick the Create a component icon Greate

In Component Create dialog box, change the Type to Skeleton Model and Sub-type to Body

Change the Name to **BODY_SKEL_3** and pick

In the Create Options dialog box, pick Browse... tab and select the TEMPLATE_SKL.PRT

Pick OK and BODY DEFINITION dialog box will appear.

Select the highlighted entity of the sketch shown in the figure below.

Top-Down Design - A Practical Approach

BODY DEFINITIO	N X
References Properties Chains	
1 One-by-One Chain	Details More
Use connections in place	ment definition
Keference Conne	Reference (
۲. Update	
Invoke component place	ment dialog
0	K Cancel

Pick Update and system will apply appropriate constraints as shown below.

Chains	Properties		
1 One-by-O	ne Chain	Details.	
		More	
End:Curver	o Pin	ASM_FRONT:F3(D	
4	Update		

Pick OK

OK to complete the definition of the body.

Now we will create the fourth body skeleton.

Pick the Create a component icon ^{III} Create

In Component Create dialog box, change the Type to Skeleton Model and Sub-type to Body

OK

Change the Name to **BODY_SKEL_4** and pick

In the Create Options dialog box, pick Browse... tab and select the TEMPLATE_SKL.PRT

Pick OK and BODY DEFINITION dialog box will appear.

Select the highlighted entity of the sketch shown in the figure below.

	BODY DEFINITION		×
	References Chains	Properties	
	1 One-by-One	e Chain	Details More
	Use conne	ections in placem	ent definition
	Reference	Conne	Kelerence
	•	Undate	F
/	Invoke cor	mponent placem	ent dialog
		OK	Cancel

Pick **Update** and system will apply the appropriate constraints for this body skeleton as shown below.

Top-Down Design - A Practical Approach

Chains	operties		
1 One-by-One C	hain	Details	
		More	
End:Curve:F5	Pin	ASM_FRONT:F3(D	
Reference	Conne	Reference plane	
End:Curve:F5	Pin	ASM_FRONT:F3(D	
End.Cuive.r.)	FILL	ASIM_FROMTES(D	
4			
	Update		

Pick or to complete the definition of the body.

Notice that system automatically creates the required axes for the connections in a separate group as shown below.



🔿 Moving Body Skeletons

Now we will use the drag functionality to move assembly entities (body skeletons) to see if the mechanism works properly or not.



Now move the mouse to drag the selected body. You will notice that the other bodies also move according to the mechanism connections as shown below.



Left-click when you want to finish the dragging.

Look at the message area to see how the mouse buttons will act in the drag mode.

➡ Creating Body Parts

Now we will create body parts by referencing the body skeletons.

Switch to the FOUR_BAR.ASM window.

Pick the Create a component icon	4	Create
----------------------------------	---	--------

Component Create dialog box will appear. Enter *LINK1* as the Name.

	Create (Component X
Type Image: Part Image: Subassemb Image: Skeleton M Image:	ly odel	Sub-type Solid Sheetmetal Intersect
Name: Common name:	Link1	
	1	<u>O</u> K <u>C</u> ancel

Pick OK and Create Options dialog box will appear. Pick Browse... tab and select the TEMPLATE_PART.PRT as template.

Check Attach Component to Body option to attach the skeleton body with this part.

reation Method Copy from existing Locate default datums Empty Create features opy From emplate_part.prt Bro acement Leave component unplaced Attach component to body	
Copy from existing Locate default datums Empty Create features opy From emplate_part.prt Bro acement Leave component unplaced Attach component to body	
) Locate default datums) Empty) Create features opy From emplate_part.prt Bro acement] Leave component unplaced] Attach component to body	
) Empty) Create features opy From emplate_part.prt Bro acement] Leave component unplaced] Attach component to body	
Create features opy From emplate_part.prt Bro acement Leave component unplaced Attach component to body	
opy From emplate_part.prt Bro acement] Leave component unplaced] Attach component to body	
emplate_part.prt Bro acement Cacement Cacement unplaced Cattach component to body	
acement Leave component unplaced Attach component to body	rowse
Ceave component unplaced	
Attach component to body	
Attach component to body	

When "Attach Component to Body" is checked, system copies the geometry of the selected Body Skeleton to this part.

Select the BODY_SKEL_	1 component in the model tree.	The dialog box will appear	[.] as shown
below.			

Opy from existing	
O Locate default datums	
O Empty	
 Create features 	
Copy From	
template_part.prt	Browse
Placement	
Leave component unplaced	
Attach component to body	
PODV SKEL 1 DPT	A
BODT_SKEL_T.PKT	

Pick or and new component will appear in the model tree.

System automatically creates the placement constraints and copies the geometry from the body skeleton to this part.

Now we will create the part for second body of mechanism.

Pick the Create a component icon 🖳 Create

Component Create dialog box will appear. Enter *LINK2* as the Name.

Pick ok and Create Options dialog box will appear. Pick Browse... tab and select the TEMPLATE PART.PRT as template.

Check Attach Component to Body option to attach the skeleton body with this part.

Select the **BODY_SKEL_2** component in the model tree.

Browse

Pick OK and new component will appear in the model tree.

You can create LINK3.PRT and LINK4.PRT by attaching them to BODY_SKEL_3 and BODY_SKEL_4 respectively on your own. But it is not necessary for the purpose of this exercise.

➡ Creating Geometry for Parts

Now we will create the solid geometry in individual parts.

🔿 First Part

Open the LINK1.PRT in a new window and notice that it contains an External Merge feature as shown below.



The merge feature references the geometry in the BODY_SKEL_1.PRT as shown below.



Merge feature copies the complete geometry of a source part to target part associatively.

Pick to invoke Extrude tool.

Select the sketching references as sown below.

		Sketch			x
Placement					
Sketch Plan	e				
Plane	FRONT:F3(DATUM PLAN	E)	Use Pre	evious
Sketch Orie	ntation				
Sketch vie	ew direction	Flip			
Reference	RIGHT	TF1(DATUM P	LANE	:)	
Orientatio	n Right	•			
				Sketch	Cancel

Sketch as shown below.



After completing the sketch, change the depth option to Symmetric ($^{-\Box}$) and enter **25** as depth value.

		H H	20		
Pick 🗹 icon to compl Again pick 👉 to invo	lete the feat ke Extrude	ure. tool.			
Select the sketching re	eterences as	S SOWN DEIOW.		×	
	Placement Sketch Plane	ane FRONT:F3(DATUM PLANE)	Use Previous		

RIGHT:F1(DATUM PLANE)

Sketch

Cancel

-

Sketch view direction Flip

Reference

Orientation Right

Sketch as shown below.

- 8.00

After completing the sketch, change the depth option to Symmetric (⁻) and enter **15** as depth value.





Copy this extrude feature using Paste Special option so that part appear as shown below.



Again pick *to invoke Extrude tool.*

Select the sketching references as sown below.

	Sketch)
Placement			
Sketch Plan	e		
Plane	FRONT:F3(DATUM PLANE)	Use Previous	
Sketch Orie	ntation		
		18112 11	
Reference	RIGHT:F1(DATUM PLA	NE)	
Reference Orientatio	RIGHT:F1(DATUM PLA	NE)	

Sketch two circles aligned to the end points of the datum curve as shown below.





Specify the **Through All** ($\exists b$) depth option for both directions.

Pick *icon to complete the feature. It will appear as shown below.*



Select the sketching references as sown below.

		Sketch			×
Placement					
Sketch Plane					
Plane R	IGHT:F1(D	ATUM PLAN	E)	Use Pre	evious
Sketch Orient	ation				
Sketch view	direction	Flip			
Reference	TOP:F2	2(DATUM PL	ANE)		
Orientation	Тор	-			
			c	ketch	Cancal

Sketch as shown below.



After completing the sketch pick icon to create the feature as cut.

Specify the **Through All** ($\exists b$) depth option for both directions.

Pick icon to complete the feature. It will appear as shown below.



🔿 Second Part

Now we will create the geometry in second body.

Open the LINK2.PRT in a new window and notice that it contains an External Merge feature.

The merge feature references the geometry in the BODY_SKEL_2.PRT as shown below.



First we will create a datum plane that passes through the shown datum curve.

Pick \square on the Model tab and select the references as shown below.

FRONT	Datum Plane	х
	Placement Display Properties References	
ТОР	Curve:F5(EXTERNAL MERGE) Through FRONT:F3(DATUM PLANE) Normal	
	Offset Translation	
	OK Cance	el

Notice that datum plane is normal to the FRONT datum plane.

Pick to apply the changes and exit dialog box.

Pick ^{to Revolve} to invoke Revolve tool.

Select the sketching references as sown below.

	Sketch		X
Placement			
Sketch Plan	ie		
Plane	FRONT:F3(DATUM PLANE)	Use Previous	
Sketch Orie	entation		
Sketch vi	ew direction Flip		
		-	
Referenc	DIM1:F6(DATUM PLANE	:)	
Referenc Orientatio	n Right v	-)	

Here DTM1 is the newly created datum plane.

Select the datum curve and its end points as reference then sketch as shown below.



Notice that centerline is aligned to the datum curve.

After completing the sketch make sure that rotation angle is set to **360**.



Select the following edge for applying round.



Pick Round to access the Round tool.

Enter 3 as the radius value

Pick **v** icon to complete the feature.

Now pick *to invoke Extrude tool.*

Select the sketching references as sown below.

		Sketch			×
Placement					
Sketch Plane					
Plane)TM1:F6(D/	ATUM PLAN	E)	Use Pre	evious
Sketch Orien	tation				
Sketch viev	w direction	Flip			
Reference	FRONT	T:F3(DATUM	PLAN	E)	
Orientation	Left	-			
				8 0.0	1 1 1 2 2 2

Sketch as shown below.

	V V	16.00
		10.00
	H	
5.00		

After completing the sketch pick icon to create the feature as cut.

Specify the **Through All** ($\exists b$) depth option for both directions.

Pick \checkmark icon to complete the feature. It will appear as shown below.



Again pick *to invoke Extrude tool.*

Select the sketching references as sown below.

	Sketch	3
Placemen	t	
Sketch Pl	ane	
Plane	FRONT:F3(DATUM PLANE)	Use Previous
Sketch O	rientation	
Sketch	view direction Flip	
Referen	ce DTM1:F6(DATUM PLAN	E)
Orientat	tion Right 💌	

Select the end of datum curve as reference and sketch as shown below.



Notice center of arc is aligned to the end point of datum curve.

After completing the sketch, change the depth option to Symmetric ($-\Box$) and enter **4.8** as depth value.

Pick *icon to complete the feature. It will appear as shown below.*

Again pick 🗰 to invoke Extrude tool.

Select the sketching references as sown below.

	Sketch)
Placement			
Sketch Pla	ine		
Plane	FRONT:F3(DATUM PLANE)	Use Pre	evious
Sketch Or	ientation		
Sketch v	view direction Flip		
Referen		E)	
		10	
Orientati	on Right		

Sketch two circles aligned to the end points of the datum curve as shown below.



After completing the sketch pick icon to create the feature as cut.

Specify the **Through All** $(\exists b \)$ depth option for both directions.

Pick **v** icon to complete the feature. It will appear as shown below.



In the same way you can create geometry for other components.

Anticipation of the section of the s

Now we will use the drag functionality to move bodies.

Pick the Drag Packaged Component icon

Pick on any entity belonging to LINK1 as shown below.

Drag	x		
Snapshots	_		k
			Surf:F7
	_	_	

Now move the mouse to drag the selected body. After moving towards left it should appear as shown below.

Drag × १८९९ हि [™] у ▶ Snapshots	
Close	

If the mechanism moves correctly it means you have successfully completed the exercise. You can complete other parts yourself.