Managing External References

External References (ER) are essential for a design project to ensure that components interface and fit together according to design intent. They make sure that models update automatically upon regeneration.

Generally, external references should be created only between a skeleton and a given model. External references between any other models are difficult to manage and should be avoided. Furthermore it is recommended that a model should copy references only from a skeleton part and skeleton should never copy references from any other model except another skeleton.

The following figure shows how difficult it becomes to manage all the relations when you create the references between parts. So this approach should never be used.



Wrong way of creating references

The following figure shows the recommended way of distributing references. In this case the project is properly structured and easy to manage. We always know where to look for controlling the interface and fit of different parts



Recommended way of creating references

Special attention should be given while creating external references as if not managed properly external references may become bottleneck.

Exercise 1

In this exercise we will see the preferred and wrong way (that is often tempting) of creating external references

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

This assembly consists of two parts as shown below.



The assembly of both parts is shown below.



If you look at the SOCKET_FRONT you will notice that it contains three bosses as shown below.



Now we will create three holes in the SOCKET_COVER, aligned to the axis of bosses in SOCKET_FRONT

➡ Wrong Way of Creating References

First of all we will show the wrong way to create the holes.

The most tempted way to create the holes is to create a cut feature in assembly mode and use the datum axis in the SOCKET_FRONT part as reference.

So we will create an extruded cut feature while in assembly mode.

Select the SOCKET_COVER.PRT in the assembly window and pick Activate



Pick to invoke Extrude tool.

Select highlighted surface as the Sketch Plane



System will select the orientation reference automatically.

Select the three axis in the SOCKET_FRONT as reference as shown below.

	References – 🗆 🗙
	A_4(AXIS):F14(EXTRUDE_5):SOCKET_FRONT A_6(AXIS):F14(EXTRUDE_5):SOCKET_FRONT A_5(AXIS):F14(EXTRUDE_5):SOCKET_FRONT
	Replace Delete Solve
A_\$6 A_\$4	Reference status Unsolved sketch
	Close

Now sketch three circles, their centers should be aligned to the referenced axis, as shown below.



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

Specify the depth option to **Through All** icon in the Depth options list

Pick the $\frac{1}{2}$ icon to reverse the depth direction.

Pick **v** icon or middle-click to complete the feature.

Select the SOCKET.ASM in the assembly window and pick Activate



The assembly will appear as shown below.



Look in the model tree and notice that a new feature appears in the SOCKET_FRONT.PRT as shown below.



Right-click the SOCKET_COVER.PRT in the model tree and pick **Information > Reference Viewer** to open the Reference Viewer.

Pick ^{III} to set the model as current object and system will show the dialog box as shown below.

				Reference Vie	ewer	- 🗆 X
<u>F</u> ile <u>V</u> iev	w <u>A</u> ctions					
Reference	Filters Paths		Current Object: 🗍 SOCI	KET_COVER.PRT	Use Previous 💌	
Show:	 References Dependencies 		New Path: 🗾 SOCKET_	COVER		
Scope	All	*	SOCKET_FRONT.	PRT		
Status	All	*				
Strong/Weal	k All	*				
₹ ₹ ₹ From Inte From Ex From Su	Placement Relation System Components in path dependent Children ccluded Children uppressed Children		SOCKET_SKEL.P	RT 📎	SOCKET_COVER.PRT	No Children found
Filter By:	Parent References	-	Dereste	~	Current Object	Children
		1,201	Parents	~	current object	children X
				Close]	

It can be seen that SOCKET_COVER has two parts as parents i.e. SOCKET_FRONT and SOCKET_SKEL. Furthermore current object is referencing three features in SOCKET_FRONT

Now pick on ^(S) in front of SOCKET_COVER.PRT to see the details. It can be seen that two features are referencing from other parts. If you place mouse pointer over "CG_CURVES", system will highlight the path toward SOCKET_SKEL.PRT. If you place mouse pointer over the "Extrude 6" feature, system will highlight the path to SOCKET_FRONT as shown in figure below.

Top-Down Design - A Practical Approach

Current Object: SOCKET_COVER.PRT Use Previous	s 🕶
New Path: SOCKET_COVER	RT No Children found
Parents X Current Object	Children ×

The highlighting of path towards SOCKET_FRONT implies that it contains the parents of this feature. Keep in mind that "Extrude 6" is the feature that we just created.

So we can say that one component of the assembly is referencing another component. This practice is called "Part to Part" referencing. Although it is a quick way to maintain fit between parts but it can become very difficult to manage especially when number of parts increases. You cannot keep track of all the references and relations between components. So it must be avoided.

External references should only be created to the skeleton model, and not to any other model.

Pick Close to close the Reference Viewer.

Breaking Part to Part References

Now we will show you how to break the external references between parts and instead control their geometry from skeleton.

First we will create the datum points in the skeleton model and then use them as references to control the location of bosses and holes in individual parts.

Open the skeleton SOCKET_SKEL.PRT in a separate window.

We will create a sketch containing geometry points. So pick will on the Model tab.

Select the sketching references as shown below

	Sketch			х
Placement	Properties			
Sketch Pla	ne			
Plane	SKL_FRONT:F3(DATUM P	LANE)	Use Previous	
Sketch Ori Sketch v	entation ew direction Flip			
Referenc	e SKL_RIGHT:F1(DATI	UM PLANE)		
Orientatio	n Right 💌			
		Sk	cetch Can	cel

Pick Geometry Point icon × Point



Place three datum points and dimension them as shown below.



At the moment, don't worry about the exact dimensions. We will redefine this feature in the assembly mode and set the dimensions.

Pick \checkmark to complete the section.

Now we will redefine the publish geometry feature and add these points. But first we should reorder this datum point feature to appear before the publish geometry feature.

So drag the "Datum Point" before the publish geometry feature as shown below.



Select the publish geometry feature and pick $\overset{\checkmark}{=}$

Activate the References collector and pick the three datum points, that we created above.

	Publish Geom	etry	
References	Properties		
Surface Sets			
Click here	to add item		
			Details
Chain			
1 One-by-Or	e Chain	*	
2 One-by-Or	ie Chain		Details
3 One-by-Or	ie Chain		botanom
References			
PNT0:F11(SK PNT1:F11(SK PNT2:F11(SK	ETCH_7) ETCH_7) ETCH_7)		Details
Annotations			
0 items selecte	ed		Edit
0 items selecte	ed		Edit

Pick \checkmark to apply the changes and exit the dialog box.

We assume that locations specified for the bosses in SOCKET_FRONT represent design intent. So we will modify dimensions of the points in the skeleton so that they confirm to the design intent.

Switch to the assembly model then select newly created sketch feature and pick $\overset{\checkmark}{=}$ as shown below.



Now change the dimensions as shown below.



Notice that now the points lie exactly over the centers of the bosses. Also observe that we are not using any reference from any other part. We are just visually aligning the points to the existing axis. We can achieve this intent by other methods e.g. measuring the distances of the

axis from some references and then modifying the dimensions. But this technique seems more robust for this case.

Pick \checkmark to complete the section.

Unhide the 07___PRT_COPY_GEOM layer in the assembly model.

Now open SOCKET_FRONT.PRT in separate window and regenerate the part by picking ²².

Notice that the datum points that we just added in the publish geometry appears in this part as shown below. (Make sure that layer 07___PRT_COPY_GEOM is not hidden)



Select the "Extrude 5" feature in the model tree and pick

k 🍯

In the sketcher mode, delete all the existing references as shown below.

	Reference	s -	. 🗆)
RIGHT:F1(DATUM PL Curve:F5(COPY GE0 Curve:F5(COPY GE0 Curve:F5(COPY GE0	ANE) OMETRY) OMETRY) OMETRY)		
k k ⊻ sec	Select:	Use Edge/Offset	v
Replace Delete	<u>S</u> olve		
Reference status Fully Placed			
			<u>C</u> lose

Pick the datum points as new references as shown below.

PNT0:F5(COPY	GEOMET	RY)		٦
PNT2:F5(COPY	GEOMET	RY)		
k k x	sec	Select:	Use Edge/Offset	r.
Replace De	lete	Solve]	
Reference status				
Unsolved sketc	:h			

Now pick --- Coincident and align the centre of circles to these new references. You should delete any other constraints if they conflict with new constraints. Now the sketch will appear as shown below.



Pick \checkmark to complete the section.

Pick ✓ to apply the changes and exit the dialog box.

Now open the SOCKET_COVER.PRT in separate window and regenerate the part by picking ¹²/₂.

Notice that the datum points that we just added in the publish geometry appears in this part as shown below. (Make sure that layer 07___PRT_COPY_GEOM is not hidden)



Select the "Extrude 6" feature in the model tree and pick ${}^{\checkmark}$

In the sketcher mode, delete all the existing references as shown below.

	Reference	IS	- 🗆 X
A_5(AXIS):F14(EXTRL A_6(AXIS):F14(EXTRL A_4(AXIS):F14(EXTRL	JDE_5):SO(JDE_5):SO(JDE_5):SO(CKET_FRONT CKET_FRONT CKET_FRONT	
k k sec	Select:	Use Edge/Of	fset 💌
Replace Delete	Solve		
Reference status			
			Close

Pick the datum points as new references as shown below.

A	eference		
PNT0:F5(COPY GEOME PNT2:F5(COPY GEOME PNT1:F5(COPY GEOME	ETRY) ETRY) ETRY)		
k k sec	Select:	Use Edge/Offset	•
Replace Delete	Solve]	
Reference status Unsolved sketch			
		Clo	ose

Now align the centre of circles to these new references. You should delete any other constraints if they conflict with new constraints. Now the sketch will appear as shown below.



Pick \checkmark to complete the section.

Pick ✓ to apply the changes and exit the dialog box.

Switch to the assembly model

Right-click SOCKET_COVER.PRT in the model tree and pick **Information > Reference Viewer** to open the Reference Viewer.

Pick 🧾 to set the model as current object and system will show the dialog box as shown below.

		Reference Viewer	- 🗆 X
<u>F</u> ile <u>V</u> ie	w <u>A</u> ctions		
Reference	e Filters Paths	Current Object: SOCKET_COVER.PRT Use Previous 💌 🛄	
Show:	 References Dependencies 	New Path: SOCKET_COVER	
Scope	All		
Status	All	•	
Strong/Wea	ak All	×	
¥ ¥ ¥ ¥ From II ¥ From E	Regular Placement Relation System Components in path dependent Children xcluded Children	SOCKET_COVER.PRT	hildren foun
Filter	Model Tree		
Filter By:	Parent References	Parents X Current Object	Children X
		Close	

It can be seen that SOCKET_COVER has only skeleton part as parent. Furthermore only the Copy Geometry is the feature with external references.

External references should only be created to the skeleton model, and not to any other model.

Pick Close to close the Reference Viewer.