

# FFlex Mesher Suspension Tutorial (Durability)





#### Copyright © 2020 FunctionBay, Inc. All rights reserved.

User and training documentation from FunctionBay, Inc. is subjected to the copyright laws of the Republic of Korea and other countries and is provided under a license agreement that restricts copying, disclosure, and use of such documentation. FunctionBay, Inc. hereby grants to the licensed user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the FunctionBay, Inc. copyright notice and any other proprietary notice provided by FunctionBay, Inc. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of FunctionBay, Inc. and no authorization is granted to make copies for such purpose.

Information described herein is furnished for general information only, is subjected to change without notice, and should not be construed as a warranty or commitment by FunctionBay, Inc. FunctionBay, Inc. assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the Republic of Korea and other countries. UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

#### Registered Trademarks of FunctionBay, Inc. or Subsidiary

RecurDyn is a registered trademark of FunctionBay, Inc.

RecurDyn/Professional, RecurDyn/ProcessNet, RecurDyn/Acoustics, RecurDyn/AutoDesign, RecurDyn/Bearing, RecurDyn/Belt, RecurDyn/Chain, RecurDyn/CoLink, RecurDyn/Control, RecurDyn/Crank, RecurDyn/Durability, RecurDyn/EHD, RecurDyn/Engine, RecurDyn/eTemplate, RecurDyn/FFlex, RecurDyn/Gear, RecurDyn/DriveTrain, RecurDyn/HAT, RecurDyn/Linear, RecurDyn/Mesher, RecurDyn/MTT2D, RecurDyn/MTT3D, RecurDyn/Particleworks I/F, RecurDyn/Piston, RecurDyn/R2R2D, RecurDyn/RFlex, RecurDyn/RFlexGen, RecurDyn/SPI, RecurDyn/Spring, RecurDyn/TimingChain, RecurDyn/Tire, RecurDyn/Track\_HM, RecurDyn/Track\_LM, RecurDyn/TSG, RecurDyn/Valve

are trademarks of FunctionBay, Inc.

#### **Edition Note**

This document describes the release information of **RecurDyn V9R4**.

# **Table of Contents**

Introduction	4
Task Objectives	4
Requirements	5
Tasks	5
stimated Time to Complete	5
Calling the Inital Model	6
Task Objective	6
Estimated Time to Complete	6
Calling an Rdyn Model	7
Running an Initial Simulation with the Suspension Model	8
Viewing the Results	8
Creating an FFlex Body	9
Task Objective	9
Estimated Time to Complete	9
Creating a UCA Body Mesh	10
Creating an LCA Body Mesh	15
Conducting Durability Analysis	21
Task Objective	21
Estimated Time to Complete	21
Conducting a Durability Analysis on the UCA_FE Body	22
Conducting a Durability Analysis on an LCA_FE Body	31
Analyzing and Reviewing Results	36
Task Objective	36
Estimated Time to Complete	36
Analyzing Durability Results	37



## Introduction

Fatigue and durability analyses are designed to determine how long a flexible body or specific area of a flexible body modeled in RecurDyn can stably endure various dynamic loads. Such analyses can also determine how stable a body is. The focus on time distinguishes these forms of analysis from other analysis methods, such as those used to determine maximum stress and maximum deformation rates.

This tutorial teaches you how to use **RecuDyn/Durability** to determine the fatigue life and fatigue damage in a dynamic system composed of flexible bodies. This tutorial also briefly describes functions and theoretical backgrounds used in RecurDyn/Durability. Thus, even users who do not have a thorough grounding in the theoretical background of durability analysis can learn to use RecurDyn/Durability simply by completing this tutorial.

This tutorial uses the MBD model, which features a simplified system intended for experimental purposes. By following the steps in this tutorial, you will learn how to replace specific sections of the flexible bodies in the model using RecurDyn/Mesher and verify the properties and durability results of these flexible bodies. Later in this tutorial, you will learn how to produce durability results for a specific section of the model.

### **Task Objectives**

This tutorial covers the following:

- Creating a flexible body using RecurDyn Mesher
- Requirements for durability analysis
- Methods of conducting durability analysis
- Obtaining durability analysis results
- Analyzing durability analysis results

### Requirements

This tutorial is intended for intermediate users who have read and understood the basic tutorial as well as the FFlex and RFlex tutorials available from RecurDyn. If you have not completed these tutorials, then you are advised to complete them before proceeding with this tutorial. In addition, this tutorial requires a basic understanding of dynamics and the finite element method.

### Tasks

This tutorial consists of the tasks outlined in the following table. This table also shows the time required to complete each task.

Procedures	Time (minutes)
Calling an Rdyn model	10
Creating an FFlex body using Mesher	15
Replacing an RFlex body	20
Creating a patch set to verify the fatigue results	5
Setting fatigue preferences	5
Performing a fatigue evaluation	5
Verifying the fatigue results	5
Total	65



This tutorial takes approximately 65 minutes to complete.



# **Calling the Inital Model**

### **Task Objective**

This chapter teaches you how to open an initial model, simulate it, and observe the behavior of the suspension model.



10 minutes

### **Calling a Rdyn Model**

### To open RecurDyn and call the initial model:



- 2. When the **Start RecurDyn** dialog window appears, close it.
- 3. From the File menu, click Open.
- 4. From the **Durability** tutorial path, select the **RD\_Durability\_Start.rdyn** file. (**The file location:** <Install Dir>\Help\Tutorial\PostAnalysis\Durability\FFlexMesherSuspension, ask your instructor for the location of the directory if you cannot find it).
- 5. Click **Open**.

The model shown below opens.



This model is designed to test the durability of the parts in a suspension system. A suspension system absorbs shocks from street surfaces in order to prevent them from being transferred to the vehicle or passengers. In this model, the tire rests on the shaker so that any excitations pass directly from the shaker to the tire and the suspension system.

The suspension system in this model consists of one tire and three subsystems. The three subsystems include two shock absorbers with built-in coil springs and a suspension assembly. The suspension assembly consists of a mount, a knuckle, an LCA (lower connecting arm), and a UCA (upper connecting arm). In this tutorial, you will test the durability of the LCA and the UCA.

Dynamic/Kinematic Analysis General Parameter Initial Condition

Plot Multiplier Step Factor

Frequency Response Analysis

Hide RecurDyn during Simulation

Output File Name

Static Analysis

State Matrix

Display Animation
Gravity
X 0. Y

1.2

200.

Y 0. Z -9806.65 Gravity Newton - Kilogram - Millimeter - Second

OK

Simulate

End Time

Include

Unit

Step

×

Pv

Pv

Pv

Cancel

#### To save the initial model:

1. From the **File** menu, click **Save As**.

(Save this model in the different path because it is impossible to simulate directly in the tutorial path.)

### **Running an Initial Simulation with the Suspension Model**

In this task, you will run an initial simulation on the model to understand how it operates.

#### To run an initial simulation:



1. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**.

The **Dynamic/Kinematic Analysis** dialog window appears.

- 2. Define the **End Time** and **Step** values as follows:
  - End Time: 1.2
  - Step: 200
  - Plot Multiplier Step Factor: 1
- 3. Click Simulate.

### **Viewing the Results**

#### To view the results:

#### From the Animation Control group in the Analysis tab, click Play/Pause.

The excitation is transferred to the tire through the shaker. The animation shows any possible vibrations or seismic excitation caused by driving the vehicle. At this point, you can observe the behavior of certain devices, including the shock absorbers.



# **Creating an FFlex Body**

This chapter teaches you how to conduct a durability analysis on a flexible body. This procedure maintains some elements of the previous model, such as its joint and force, but replaces certain rigid bodies with flexible bodies to increase the efficiency of the model.

### **Task Objective**

This task teaches you how to replace an existing rigid body with a flexible body using the Mesher provided in RecurDyn FFlex (Full Flex).



15 minutes

### **Creating a UCA Body Mesh**

1. In **Assembly** mode, double-click the **Suspension\_Assy** created as a Subsystem to enter the **Subsystem** mode.





2. In **Subsystem** mode, select the **UCA** body and Right-click the selected body, and select **Mesh** from the context menu, as shown in the figure to the right.

The UCA body appears as shown in the figure below.



3. In **Mesh** mode, click **Assist** from the **Mesher** group in the **Mesher** tab.

The **Assist Modeling** dialog window appears.

Assist

**Tip:** The **Geometries** in the **Preserve Constraint** are applied to the **Assist Modeling** dialog automatically after the user selects the **Target Body**. However, following behind steps are recommended to the user to produce accurate results, especially in the tutorial.

**Tip:** The FDR checkboxes specify whether or not to create an FDR (Force Distributed Rigid) element after the Mesh operation. The Sel. checkboxes specify whether or not to maintain the joint or force previously created. In this tutorial, these functions are all necessary. Select all of the checkboxes.

4. In the Add/Remove column of the dialog window, click Gr to assign the geometries for the areas where the Slave Nodes will be created. Referring to the following page, select the geometries where the slave nodes will be created for the three FDRs. (Tip: When you click the Gr button, a marker appears on the screen indicating where the master node will be generated. This helps you locate the areas to assign slave nodes.)

Assist Modeling ☑ Preserve Constrai	int			
Name	FDR	Geometry	Add/Remove	Sel.
Spherical_UCA@	~	C1_ImportedSolid1.Face4	Gr	
Bushing_UCA1	~	C1_ImportedSolid1.Fac	Gr	
Bushing_UCA2	~	C1_ImportedSolid1.Face2	Gr	<b>~</b>
✓ Pre-Create Patch(	Line) Se	et & Preserve Contact		
Name	Set	Geometry	Add/Remove	Geo
		OK Cance	4	

**Tip:** Due to the nature of the FDR, the master node is automatically generated at the point where the joint and the force are created. You must define the slave nodes directly.

5. After all the geometries are assigned, right-click the model, and click **Finish Operation**.

100	angene addite tot a	cel7
	Finish Operation	
3	Translate	T
ф	Rotate	R
2	Zoom	z
寄	View Center	c
30	Select Zoom	5
10	Fit	
	View Control	
	Rendering Mode	,
	Select Box	
	Select List	

6. For the other two areas, select the **Surfaces** where the **Slave Nodes** will be generated, as shown in figures below.





7. Once you have selected all the geometry items, click **OK**.

Assist Modeling				
Target Body		C1_ImportedSolid1		В
Preserve Constra	int			
Name	FDR	Geometry	Add/Remove	Sel.
Spherical_UCA@	~	C1_ImportedSolid1.Fac	Gr	
Bushing_UCA2	~	C1_ImportedSolid1.Face2	Gr	
Bushing_UCA1	~	C1_ImportedSolid1.Fac	Gr	
Pre-Create Patch	Line) Se	et & Preserve Contact	_	
Pre-Create Patch	Line) Set	et & Preserve Contact Geometry	Add/Remove	Geo
Pre-Create Patch Name Name	Line) Set	et & Preserve Contact Geometry	Add/Remove	Geo

8. From the **Mesher** group in the **Mesh** tab, click **Adv.Mesh**.



Advanced Mesh							
Target Geometry	C1_ImportedSolid1 Gr	Target Face & Type Info	ormation ——				
Mesh Type	Solid4(Tetra4) 🔻	Formal Type	All		•		
Property	PSolid1 P	Name		Tj	/pe		
Mesh Option		Face18		(	Cyl		
Avg. Element Size	7.96284294128418	Face2		(	Cyl		✓
Avg. Element Size		Face12		4-S	ided		
Min Element Size	2.80114722251892	Face9		4-S	ided		
		Face10		(	Cyl		
Structured Out	put Simple Pattern 🔻	Face21		4-S	ided		
🗸 Include Assist I	Modeling Preview	4 Sided More Type -					,
	Lindate Preview	Name	1st Edge (Red	I)	3rd Edge (Blue	)	
	opullerrenew	Face13	Edge17	Gr	Edge36	Gr	
		Face1	Edge27	Gr	Edge23	Gr	
		Face16	Edge28	Gr	Edge41	Gr	
		Face5	Edge20	Gr	Edge19	Gr	
		Face6	Edge17	Gr	Edge2	Gr	
Revert	Mesh Cancel						

The Advanced Mesh dialog window appears.

- 9. In the **Advanced Mesh** dialog window, perform the following:
  - In the **Mesh Type** dropdown menu, select **Solid4 (Tetra4)**.
  - Select the Include Assist Modeling checkbox. (Tip: You must select this to ensure that the FDRs are automatically generated according to the conditions defined in Assist Modeling.)
  - Click Mesh.
  - Once Meshing is complete, click **Close**.

The mesh model of the UCA body appears, as shown below.



**Tip:** RecurDyn offers two auto-meshers, Mesh and Advanced Mesh. The Mesh auto-mesher applies tessellation to the geometry in the GUI before generating the mesh. The Advanced Mesh auto-mesher skips the tessellation stage altogether and generates the mesh directly. By doing so, Advanced Mesh minimizes the distortion of the CAD geometry. However, Advanced Mesh also increases the Mesh failure rate.

- 10. Perform either of the following steps to return to the parent mode:
  - Right-click the Working window, and from the menu that appears, click Exit.
  - From the **Exit** group in the **Mesher** tab, click **Exit**.

L.

Note that the existing UCA Rigid Body (UCA) disappears and it is replaced by the UCA Flexible body (UCA\_FE), as shown in the figure below. Also, the joint and the force previously created are maintained as they were in the GUI. At this moment, to see the icons, click **Icon Control** on the toolbar and select **All Icons**.



- 11. To check if the modeling is correctly configured for the **UCA Flexible body**, return to the **Assembly** mode and run **Dynamic Analysis** using the previous conditions. The analysis will be completed quickly.
- 12. From the FFlex group in the Flexible tab, click Contour to view the results.



### **Creating an LCA Body Mesh**

1. In **Assembly** mode, double-click the **Suspension\_Assy** created as a subsystem to enter the **Subsystem** mode.

In **Subsystem** mode, select the body and the **LCA** body. Right-click the selected body, and select **Mesh** from the context menu, as shown in the figure to the right.

2. The LCA body appears as shown in the figure below.





3. In **Mesh** mode, click **Assist** from the **Mesher** group in the **Mesher** tab.

The **Assist Modeling** dialog window appears.

4. In the Add/Remove column of the dialog window, click Gr to assign the geometries for the areas where the Slave Nodes will be created. Referring to the following page, select the geometries where the slave nodes will be created for three FDRs. (Tip: When you click the Gr button, a marker appears on the screen indicating where the master node will be generated. This helps you locate the areas to assign slave nodes.)

Name       FDR       Geometry       Add/Remove       Sel.         Spherical1_Shoc       ✓       Unite2.Face33       Gr       ✓         Spherical1_Shoc       ✓       Unite2.Face1       Gr       ✓         Spherical_LCA@       ✓       Unite2.Face24       Gr       ✓         Bushing_LCA1@       ✓       Unite2.Face28       Gr       ✓         Bushing_LCA2@       ✓       Unite2.Face29       Gr       ✓         Pre-Create Patch(Line) Set & Preserve Contact       Name       Set       Geometry       Add/Remove       Geo	Assist Modeling ✓ Preserve Constrai	nt			
Spherical1_Shoc       V       Unite2.Face33       Gr       V         Spherical1_Shoc       V       Unite2.Face1       Gr       V         Spherical_LCA@       V       Unite2.Face24       Gr       V         Bushing_LCA1@       V       Unite2.Face28       Gr       V         Bushing_LCA2@       V       Unite2.Face29       Gr       V         V       Pre-Create Patch(Line) Set & Preserve Contact         Name       Set       Geometry       Add/Remove       Geo	Name	FDR	Geometry	Add/Remove	Sel.
Spherical_Shoc       ✓       Unite2.Face1       Gr       ✓         Spherical_LCA@       ✓       Unite2.Face24       Gr       ✓         Bushing_LCA1@       ✓       Unite2.Face28       Gr       ✓         Bushing_LCA2@       ✓       Unite2.Face29       Gr       ✓         ✓       Pre-Create Patch(Line) Set & Preserve Contact         Name       Set       Geometry       Add/Remove       Geo	Spherical1_Shoc	~	Unite2.Face33	Gr	<b>V</b>
Spherical_LCA@       ✓       Unite2.Face24       Gr       ✓         Bushing_LCA1@       ✓       Unite2.Face28       Gr       ✓         Bushing_LCA2@       ✓       Unite2.Face29       Gr       ✓         ✓       Pre-Create Patch(Line) Set & Preserve Contact         Name       Set       Geometry       Add/Remove       Geo	Spherical1_Shoc	~	Unite2.Face1	Gr	<b>I</b>
Bushing_LCA1@     ✓     Unite2.Face28     Gr     ✓       Bushing_LCA2@     ✓     Unite2.Face29     Gr     ✓	Spherical_LCA@	~	Unite2.Face24	Gr	<b>V</b>
Bushing_LCA2@     ✓     Unite2.Face29     Gr     ✓       ✓     Pre-Create Patch(Line) Set & Preserve Contact       Name     Set     Geometry     Add/Remove     Geo	Bushing_LCA1@	~	Unite2.Face28	Gr	<b>I</b>
✓ Pre-Create Patch(Line) Set & Preserve Contact       Name     Set       Geometry     Add/Remove	Bushing_LCA2@	7	Unite2.Face29	Gr	<b>I</b>
	Pre-Create Patch(	Line) Se	et & Preserve Contact Geometry	Add/Remove	Geo

**Tip:** Due to the nature of the FDR, the master node is automatically generated at the point where the joint and the force are created. You must define the slave nodes directly.

5. After all the geometries are assigned, right-click the model, and click **Finish Operation**.



6. For the other four areas, select the **Surfaces** where the **Slave Nodes** will be created, as shown in the figures below:



7. Once you have selected all the geometry items, click the **OK** button.

Assist Modeling				
Target Body		Unite2		В
Preserve Constrai	nt			
Name	FDR	Geometry	Add/Remove	Sel.
Spherical1_Shoc	<b>~</b>	Unite2.Face1	Gr	
Spherical1_Shoc	<ul> <li>Image: A start of the start of</li></ul>	Unite2.Face33	Gr	<ul> <li>Image: A set of the set of the</li></ul>
Spherical_LCA@	<ul> <li>Image: A start of the start of</li></ul>	Unite2.Face24	Gr	V
Bushing_LCA1@	~	Unite2.Face28	Gr	V
Bushing_LCA2@	<b>~</b>	Unite2.Face29	Gr	
Name	Set	Geometry	Add/Remove	Geo
		OK Cance	H.	



8. From the **Mesher** group in the **Mesher** tab, click **Geo. Refine**. (**Tip:** The geometry refinement function is only effective when using the mesh method. That is, as with the UCA body, if the advanced mesh method is used, then it has no effect at all.)

The Geometry Refinement dialog window appears.

arget Geometry	Unite2
Geometry Plane Tolerance	0.1
Effective Min Value	0.00664473536791677
Angle Tolerance (deg.)	3.
Effective Min Value	0.5
Max Facet Size	10.
Remove Small Feature	1.04912181587635e-002
Effective Ratio	0.01049121815876 Select
Remove Edge	Select Clear
Preview Line	Shade Update

- 9. In the Geometry Refinement dialog window, change the **Plane Tolerance** to **0.1**.
- **10.** In the Geometry Refinement dialog window, check out the **Angle Tolerance**
- 11. Select the **Preview** checkbox.

The tessellated shape appears, as shown in the figure to the right.

12. Click the **OK** button to close the dialog window.



**Tip:** RecurDyn features two Auto-Meshers, Mesh and Advanced Mesh. The Mesh auto-mesher applies tessellation on the geometry in the GUI before generating the mesh. The Advanced Mesh auto-mesher skips the tessellation stage altogether and generates the mesh directly. The LCA body has a lot of curved surfaces, so you should use the Mesh auto-mesher (which applies tessellation) to reduce the number of nodes. The geometry refinement function used for the LCA body applies the tessellation and has nothing to do with the Advanced Mesh method.



### 13. Click AutoMesh.

The **Mesh** dialog window appears.

- 14. In the **Mesh dialog** window, perform the following:
  - Select Solid4 (Tetra4) as the Mesh Type, as shown in the figure to the right.
  - In the Mesh Option group, set the Max Element Size to 4 and the Min Element Size to 2.
  - Select the Include Assist Modeling checkbox.
  - Click the **Mesh** button.

Ҟ Mesh	3
Target Geometry	Unite2 Gr
Use Current Shell Mesh Inf	0.
Mesh Type	Solid4(Tetra4)
Property	PSolid1 P
Mesh Option	
Max Element Size	4
Min Element Size	2
Chordal Error Ratio	Relative 🔻 0.1
Structured Output	Simple Pattern 💌
Close Gaps	
Weight on Quadrangles	ow , , , , , High
Additio	nal Options
✓ Include Assist Modeling	
Auto Flex Merge	
Create Beam Element with	Pre-Stress
Revert	Mesh Cancel

The mesh model and FDR of the LCA body are created, as shown in the figure below.



- 15. Perform either of the following steps to return to the parent mode:
  - Right-click the Working window, and from the menu that appears, click **Exit**.
  - From the **Mesher** group in the **Mesher** tab, click **Exit**.

Note that the existing **LCA Rigid Body (LCA)** disappears and it is replaced by the **LCA Flexible body (LCA\_FE)**, as shown in the figure below. Also note that the joint and the force previously created are maintained from the GUI. At this point, to see the icons, click **Icon Control** on the toolbar and select the **All Icons** checkbox.

Å,



16. To check if the modeling is correctly configured for the created LCA Flexible body, Exit to the Assembly mode and run the Dynamic Analysis under the previous conditions. The analysis will take a little longer than in the previous step, but it will not take too long to produce the results. From the FFlex group in the Flexible tab, click Contour to produce the results shown below.





# **Conducting Durability Analysis**

### **Task Objective**

This chapter teaches you how to analyze the durability of an FFlex body.



40 minutes

### Conducting a Durability Analysis on the UCA\_FE Body

### To create a patch set:

- 1. In **Assembly** mode, double-click the **Suspension\_Assy** to enter **Subsystem** mode.
- 2. Double-click the **UCA\_FE** body to proceed to the **Body Edit mode**.
- 3. From the **Set** group in the **FFlex Edit** tab, click **Patch Set**.
- 4. Click the **Add/Remove** button, as shown below. Click and drag the mouse button to select the entire body.

General External Patch Se	t	
Color	Automatic	
Add/Remove		
Add/Remove (Continu	ous) Tolerance (Degree)	45
	Check Reverse D	irection
Add/Remove (Select Fr	ont)	
Add (Node Set)		
Preview Normal		
Normal Adjust		
Automatic	Auto Adjust	Switch
Manual	Select Target	Switch
No. of Patches 0		
	General External Patch So Color Add/Remove Add/Remove (Continu Add/Remove (Select Fr Add (Node Set) Preview Normal Normal Adjust Automatic Manual No. of Patches	General       External Patch Set         Color       Automatic         Add/Remove       Add/Remove         Add/Remove (Continuous)       Tolerance (Degree)         Check Reverse D       Check Reverse D         Add (Node Set)       Check Reverse D         Preview Normal       Normal Adjust         Manual       Select Target         No. of Patches       0

- 5. Right click the body and click **Finish Operation** in the context menu.
- 6. In the Patch Set dialog window, click **OK**.
- 7. Confirm that the patch set is created and click **Exit** from the **Exit** group in the **FFlex Edit** tab to return to the parent mode.





8. To set the patch set for the **LCA\_FE** body, repeat steps 2 to 7, as shown in the figure below.

	Patch Set [ Current Unit : N/kg/mm/s/deg ]
	General External Patch Set
	Color Automatic •
	Add/Remove
	Add/Remove (Continuous) Tolerance (Degree) 45
	Check Reverse Direction
	Add/Remove (Select Front)
	Add (Node Set)
	Preview Normal     Normal Adjust
	Automatic Auto Adjust Switch
<b>O</b>	Manual Select Target Switch

When you have completed the above procedures, the patch sets have been created for each of the two Flexible Bodies.



9. Right-click the Working window, and from the menu that appears, click **Exit** to return to the **Assembly** mode.

#### To retrieve the animation file:



From the **Animation Control** group in the **Analysis** tab, click **Reload the last animation file** icon.

Note that all of animation-related buttons are activated.

**Tip:** After you create the patch sets for the UCA\_Fe and LCA\_FE bodies, it may look as if the previous analysis results are unavailable. However, those procedures have no effect on any of the dynamic analysis results. Therefore, there is no need to perform dynamic analysis again. You can simply retrieve the animation file, or the RAD file, from the previously analyzed results.

#### To set the analysis preferences:



- 1. From the **Durability** group in the **Post Analysis** tab, click **Preference**.
- 2. In the Preferences dialog window, on the **Material** tab, specify the path of the **Material Library file** for the Fatigue Analysis.

(C:\Users\<Your Windows Login ID>\Documents\RecurDyn\<RecurDyn Version>\Durability or the equivalent path).

3. On the **Fatigue Influencing Factors** tab, in the Fatigue Factors group, set the **Notch Factor Amp (Kf, Kt)** to **2**, as shown below.

Convergence Control	Rainflow Cou	unting
Material	Fatigue Influencing Fa	ctors
Fatigue Factors		
Notch Factor Amp (Kf, Kt)	2.	Pv
Surface Factor (ms)	Polished 🔻 1.0	
Size Factor (md)	1.	Pv
Load Factor (mt)	1.	Pv
Other Factor (mo)	1.	Pv
Overall Scale Factor		
Scale Factor (fs)	1.	Pv

The **Notch Factor** increases the analytically derived stress value to account for stress concentrations due to the cracks, holes, and notches (V grooves) caused by the design and processing of the structure. Therefore, the larger the notch factor value is, the more severe the durability analysis will be.

**Tip:** For reference, the values included in the Fatigue Influencing Factors represent the physical conditions of actual test samples in an actual durability analysis experiment, so they all have default values of 1. Modifying each factor varies the S-N curve accordingly: making it possible to simulate more severe conditions in the durability analysis. If the values other than the notch factor are less than 1 and the notch factor value is greater than 1, then the conditions simulated in the durability analysis will be more severe.

4. In the **Preferences** dialog window, do not change the Convergence Control and **Rainflow Counting** values, and click **OK**.

#### To conduct the fatigue evaluation:



1. From the **Durability** group in the **Post Analysis** tab, click **Fatigue Evaluation**.

The Fatigue Evaluation dialog window appears.

- 2. In the **Fatigue Evaluation** dialog window, perform the following:
  - Set the **Axial Mode** to **Uni-Axia**l.
  - Set the **Searching Increment** to **5 Deg**.

Axial Mode		🖲 Uni-Axia	al I	OB	8i-Axial
Stress - Based	1	⊖ Strai	n - Based		O Safety Factor
Life Criterion			Manson-Col	ffin	
Mean Stress Effect			Goodman		
BWI Weld			class B		
Num of Std.Deviatio	ins		2.		
Searching Incremen	t		5 Deg		
Material					
Material < mm-N >			[Steel] MAN	TEN [ Samp	ole.xml] S-
Element / Patch Set					E
Time History			History_1		SE
Occurrence			1.		F
Pre-Stress File					
Recalculate Reco	very Dat	ta			
Fatigue Results	6	ace Node ID	Damage (	Max.)	Life ( Min.)
				_	

3. In the Material group, click "...".

The Material Manager dialog window appears.

- 4. In the **Material Manager** dialog window, perform the following:
  - Select and right-click [Steel] 1020, and click Make Active in the context menu, as shown in the figure to the right.
  - Click **OK**.

Material Manag	jer			
Select Sample.xm		nl	▼ New Lit	orary
			Unit mm-N	•
Name		Reference	Description	Yi 📥
[Steel] MANTEN		SAE J1099, FEB. 1975	Entry 61, data ref 3	=
[Stainless Steel]	304/160	SAE J1099, FEB. 1975	Entry 36, data ref 1	
[Stainless Steel]	304/327	SAE J1099, FEB. 1975	Entry 37, data ref 1	
[Stainless Steel]	310/145	SAE J1099, FEB. 1975	Entry 38, data ref 1	
[Steel] AM350		SAE J1099, FEB. 1975	Entry 5, data ref 1/[	
[Steel] GAINEX		SAE J1099, FEB. 1975	Entry 7, data ref 7/[	
[Steel] H11/660		SAE J1099, FEB. 1975	Entry 8, data ref 6/[	
[Steel] RQC100		SAE J1099, FEB. 1975	Entry 12, data ref 1	
[Steel] 10B62		SAE J1099, FEB. 1975	Entry 13, data ref 7;	
[Steel] 1005/90		SAE J1099, FEB. 1975	Entry 17, data ref 7	
[Steel] 1005/327		SAE J1099, FEB. 1975	Entry 16, data ref 7	
[Steel] 1015/80		SAE J1099, FEB. 1975	Entry 18, data ref 4	
[Steel] 1045/225		SAE J1099, FEB. 1975	Entry 21, data ref 7	
[Steel] 1045/390		SAE J1099, FEB. 1975	Entry 23, data ref 7	
[Steel] 1045/410		SAE J1099, FEB. 1975	Entry 22, data ref 7	
[Steel] 1045/450		SAE J1099, FEB. 1975	Entry 24, data ref 7	
[Steel] 1045/500		SAE J1099, FEB. 1975	Entry 25, data ref 7	
[Steel] 1045/595		SAE J1099, FEB. 1975	Entry 26, data ref 7	
[Steel] 1020		Make Active	) data ref 1	
[Steel] 1040		Edit	), data ref 1	
ICteell 11//		CUIL	I data ref 1	
		New Material		
		Out		
		Cut		
		Сору		
		Paste		
		Delete		

- 5. In the Element/Patch Set group, click EL.
- 6. Select the set **Patch Set** for the **UCA\_FE** body.

Since the UCA\_FE body is included in the Suspension\_Assy subsystem, you must click the UCA\_FE body button while pressing the Shift Key. As a result, UCA\_FE.SetPatch1@Suspension\_Assy will appear on the Element/Patch Set item.

- 7. Click the **SEL** button in the **Time History** text box.
- 8. A Time History set is defined already, in the **Time History** dialog window. To change the range of time, click **R**.
- 9. In the **Time Range** dialog window, click **All**.
- 10. Click Close in the Time Range dialog window.
- 11. Click OK





- 12. In the **Occurrence** text box, type **1000000**.
- 13. Click Calculation.

A Progress Bar appears to display the fatigue analysis progress. Upon completion of the analysis, the results will appear in the results group, as shown in the figure to the right.

Axial Mode Life Criteria ————	Uni-Axial	0	Bi-Axial
Stress - Based	⊖ Strain	- Based	○ Safety Factor
Life Criterion		Manson-Coffin	-
Mean Stress Effect		Goodman	•
BWI Weld		class B	~
Num of Std.Deviations		2.	
Searching Increment		5 Deg	-
Material			
Material < mm-N >		[Steel] 1020 [ Sample.	xml ] S-N
Element / Patch Set		UCA_FE.SetPatch1@S	Suspension_Assy EL
Time History		History_1	SEL
Occurrence		1000000	Pv
Pre-Stress File			
Recalculate Recover	y Data		
Fatigue Results		1	1
Time Range History_1	Face Node ID 1280,1281,462	Damage ( Max.) 3.3033604699511	Life ( Min.) 302722.033849

- 14. In the Fatigue Evaluation dialog window, click **Fatigue Tools**.
  - Click the **Rainflow Counting** button in the Fatigue Tools dialog.
  - As shown below, the Rainflow Counting results in the Excel are based on the Stress Time History applied to the patch zone where the damage is largest. The results are displayed in the numbers of cycles according to the stress amplitude and the mean stress.

Fatigue Tools
Time History History_1 💌
Patch Info.
Max Damage     Ouser-defined     Sel
Node IDs 136,137,364
Rainflow Counting Plot History
Close



Click the **Plot History** in this dialog.



• As shown below, it is possible to check the Stress Time History on the patch which has the maximum damage results among the defined patches.

#### To verify the contour results:

1. From the **Durability** group in the **Post Analysis** tab, click **Contour**.

The Durability Contour dialog window appears.

- 2. In the **Durability Contour** dialog window, perform the following:
  - In the **Contour** Option group, select **Life**.
  - Click **Calculation**.

Contour

- Select Enable Log Scale.
- Click **Contour View** to view the results.

lime History	History_1
/iew Type	Contour 🗸
Contour Option	Style Option
⊖ Damage	Color Option Edit
Band Option	Colors Spectrum 🔻
Legend Type Display	Style Stepped 🔻
	Text Color 🔹
Location Bottom 🔻	
Show Text Legend	
Band Level(10~50) 10	
Min Mary Online	
- Min/Max Option	
Display	Mesh Lines
Display	Mesh Lines
Display  Calculation	Mesh Lines       Vector Color       Vector Size Factor
Calculation Min 6.7176e+05 6.7176e+05	Mesh Lines
Calculation           Min         6.7176e+05         6.7176e+05           Max         3.3167e+09         3.3167e+09	Mesh Lines       Vector Color       Vector Size Factor       3.89       Probe Option       O Node
Min/Max Option           Display           Calculation           Min           6.7176e+05           6.7176e+05           Max           3.3167e+09           Show Min/Max	Mesh Lines
Min/Max Option Display ▼ Calculation Min 6.7176e+05 6.7176e+05 Max 3.3167e+09 3.3167e+09 Show Min/Max ✓Enable Log Scale	Mesh Lines

**3.** To make the results more visible, click **Edit** in the **Style Options** group of the Durability Contour dialog window, and change the colors as follows.

Select Band Color	
First Band Color (Min) Last Band Color (Max)	· ·
ОК	Cancel



4. Click **Contour View** again to highlight the least durable sections in red, as shown below. This makes it easier to identify the areas with a relatively short fatigue life. (**Tip:** At this point, if you would like a more detailed Contour Plot, then select **Wireframe** in the toolbar before viewing the results.)



5. Select **Contour & Damage** in the **Veiw** Type on options in order to check the damage direction on each patch.

Contour	•
Contour	
Damage Vector	
Contour & Damage Vector	

As shown below, if the View button is pushed, the damage direction will appear.



### Conducting a Durability Analysis on an LCA\_FE Body

#### To set the analysis preference:



- 1. From the **Durability** group in the **Post Analysis** tab, click **Preference**.
  - The Preference dialog window appears.
- 2. In the Preference dialog window, perform the following:
  - On the Fatigue Influencing Factors tab, in the Fatigue Factors group, change the Notch Factor Amp (Kf, Kt) from 2 to 1.

Convergence Control	Rainflow Coun	iting
Material	Fatigue Influencing Fact	ors
Fatigue Factors		
Notch Factor Amp (Kf,Kt)	1.	Pv
Surface Factor (ms)	Polished 🔻 1.0	
Size Factor (md)	1.	Pv
Load Factor (mt)	1.	Pv
Other Factor (mo)	1.	Pv
Overall Scale Factor		
Scale Factor (fs)	1.	Pv

On the Rainflow Counting tab, in the Rainflow Counting Data Control, change the Number of Ranges to
 6. (Tip: At this point, if you change the number of ranges when running Rainflow Counting in the Fatigue Evaluation dialog window and set the number of cycles on mean stress and stress amplitude, then the number of ranges for the mean stress and stress amplitude in the Excel chart will be the value you have set.)

	Fatigu	e Influencing Fa	ctors
Convergence Con	trol	Rainflow Cou	nting
Rainflow Counting Da	ta Control ——		
Peak - Valley	0.		Pv
Number of Ranges	6.		Pv

#### To use a user-defined S-N curve:



- 1. From the **Durability** group in the **Post Analysis** tab, click **Fatigue**.
  - The Fatigue Evaluation dialog window appears.
- 2. In the Fatigue Evaluation dialog window, perform the following.
  - Change the **Axial Mode** to **Uni-Axial**.
  - In the Life Criteria group, select User-Defined.
  - Set the **Searching Increment** item to **10 Deg**.
  - Set the **Occurrence** to **1000000**.

Axial Mode	🖲 Uni-Axia	O Bi-Axial
Life Criteria		
Stress - Based	⊖ Strair	n - Based O Safety Factor
Life Criterion		User-Defined 💌
Mean Stress Effect		Goodman
BWI Weld		class B 💌
Num of Std.Deviations		2.
Searching Increment		10 Deg 💌
Material		
S-N Curve < mm-N >		User-Defined S-N
Element / Patch Set		LCA_FE.SetPatch1@Suspension_Assy EL
Time History		History_1 SEL
Occurrence		1000000. Pv
Fatigue Results		- - -
Time Range	Face Node ID	Damage ( Max.) Life ( Min.)
Fatigue Tools	mport	Calculation OK Cancel

- In the Material group, click "...".
   The User-Defined dialog window appears.
- 4. In the **User-Defined** dialog window, perform the following:
  - Click the Add Row button 9 times, and enter the values shown in the figure to the right.
  - Click Draw to show the S-N curve for the given values.
  - Click Close.
- 5. In the **Element/Patch Set** group, click **EL**.
- 6. Select **Patch Set** for the **LCA\_FE** body.

User-D	efined						
		Unit	t mm-N 🔻				
No	Cycle to Failure	Stress Amplitude	Add Row				
1	1	820					
2	10	630	Insert Row				
3	100	470					
4	1000	360	Delete Row				
5	10000	270					
6	100000	200	Draw				
7	1000000	150	Class				
8	1000000	120	Clear				
9	10000000	90	Import				
	Import						
			Export				
L			Export				
Interpo	lationType	Linear	· · · · · · · · · · · · · · · · · · ·				
Mater	ial for Stress Based Life Crite	rion					
Yield	Stress	317.2	Pv				
Ultim	Ultimate Strength 565.4 Pv						
Fatigue Strength Coefficient 917. Pv							
Close							

Return to the Fatigue Evaluation dialog window and note that the Element/Patch Set item maintains the patch data of the UCA\_FE body set in the previous step. For this tutorial, however, hold down the **Shift Key** and select the patch data that was defined as the LCA\_FE body, as shown in the figure below.



### 7. Click Calculation.

The Fatigue Analysis processing status appears in a Progress Bar, as shown below.



When the Fatigue Analysis is complete, the Fatigue Evaluation dialog window displays the **Maximum Damage** and **Minimum Life**.

Axial Mode Life Criteria ———	Uni-Axial	0	Bi-Axial	
Stress - Based	⊖ Strain	- Based	○ Safety Factor	
Life Criterion		User-Defined		
Mean Stress Effect		Goodman		
BWI Weld		class B		
Num of Std.Deviations		2.		
Searching Increment		10 Deg 👻		
S-N Curve < mm-N >		User-Defined	S-N	
Element / Patch Set		LCA_FE.SetPatch1@Suspension_Assy EL		
Time History		History_1 SEL		
Occurrence		1000000. Pv		
Pre-Stress File				
Recalculate Reco	very Data			
Fatigue Results	Free Node ID	Demons (March		
History 1	71524,77590,65551	1.	1.	

8. In the **Fatigue Evaluation** dialog window, click **Fatigue Tool**.

As shown below, the **Rainflow Counting** results displayed in the Excel file are based on the Stress Time History applied to the patch zone where the damage is the largest. Unlike the **Rainflow Counting** result for the UCA\_FE body, there are 6 ranges in the 3D chart because you changed the number of ranges in the **Rainflow Counting** data set to 6 in the Preference dialog window.



- 9. Open the **Contour** dialog window, and select **Life** in the **Contour Option** group, as you did when verifying the durability analysis results for the UCA\_FE body.
- **10.** Click **Calculation** and note that the Min/Max values are calculated in the dialog window. Then, click **Contour View** to display the results, as shown below.

Durability Contour	×	
Time History	History_1	
View Type	Contour	
Contour Option	Style Option	
🔿 Damage 🛞 Life	Color Option Edit	
Band Option	Colors Spectrum •	
Legend Type Display 💌	Style Stepped •	
Location Bottom *	Text Color	
Show Text Learnd		
Band Level(10-50) 10		
Min/Max Option		JAN STAN
Display 💌	Mesh Lines	
Calculation	Vector Color	
	O Node	
Max 100 100	Patrin     Clear	
Show Min/Max		
C. and the state	Show Probe Results	
Expert	View OK	1.00e+01 1.50e+01 2.51e+01 3.90e+01 6.31e+01 1.00e+02
10		



# **Analyzing and Reviewing Results**

### **Task Objective**

This chapter teaches you how to analyze and review the durability results of the two models.



5 minutes

### **Analyzing Durability Results**

#### Analyzing the results for the UCA\_FE body

Before conducting the durability analysis, if you compared the Von-Mises stresses for the UCA\_FE and LCA\_FE bodies after the final step in Chapter 3, then the stress generated in the UCA\_FE body would be relatively low. Therefore, you would anticipate that its durability would be very high as well. However, In actuality, the rainflow counting result derived from the stress time history used in the fatigue analysis, shown in the figure below, reveals that the largest stress range (where the stress range is twice the stress amplitude) repeatedly added on the weakest point (the patch zone) is about 3 Mpa and the mean stress is less than 1 Mpa.



 For reference, the relationship between the stress amplitude, stress range, mean stress, and cycle are shown in the stress time history below.



- As you can see from the simulation results, the load was applied to the UCA\_FE body repeatedly in the same direction and our durability analysis used the Uni-Axial for the axial type. Since the loads were added a million times during the simulation (occurrence = 1,000,000), this stress life criteria assumes a highly repetitive life. Therefore, the mean stress effect was applied here.
- The durability analysis also used 1020 series steel (available in the Durability Material Library), which is largely used in the vehicle structures.
- The results indicate that the repetition number for the most damaged section, which is the section with the shortest fatigue life, is about 5e+06. In other words, under the load conditions of the simulation, Fatigue Evaluation is the value obtained by multiplying Occurrence condition (1e + 06) and fatigue life (5e + 06).

Axial Mode	Uni-Axial	(	Bi-Axial	
Life Criteria				
Stress - Based	⊖ Strain	- Based	◯ Safety Factor	
Life Criterion		Manson-Coffin		
Mean Stress Effect		Goodman		
BWI Weld		class B	Y	
Num of Std.Deviations		2.		
Searching Increment		5 Deg		
Material				
Material < mm-N >		[Steel] 1020 [ Samp	le.xml] S-N	
Element / Patch Set		UCA_FE.SetPatch1@Suspension_Assy EL		
Time History		History_1	SEL	
Occurrence		1000000.	Pv	
Pre-Stress File				
Recalculate Recovery	/ Data			
Fatigue Results				
Time Range	Face Node ID	Damage ( Max.)	Life ( Min.)	
History_1	1280,1281,462	3.3033604680415.	302722.034024	

- Assuming that the fatigue life result shown above has an infinite life, then you can apply the safety factor to the life criteria to derive another fatigue analysis result for the UCA\_FE Body design.
- To do so, select Safety Factor for Life Criteria only, and leave the searching increment, element/patch set, and materials the same as those applied to the UCA\_FE body. This will produce the results shown below. (Tip: At this point, change the notch factor to 2 in the Preference dialog window.)

⊖ Strain	- Based Goodman	Safety Factors	or •
	Goodman		-
	Goodman		
			~
	class B		~
	2.		
	5 Deg		-
	[Steel] 1020 [ Sa	ample.xml]	Η
	UCA_FE.SetPat	ch1@Suspension_Assy	EL
	History_1		SEL
	1000000.		
Face Node ID		Safety Factor (Min.)	
1280,1	281,462	32.462583820492	
	Face N 1280,1	2.           5 Deg           [Steel] 1020 [Si           UCA_FE.SetPat           History_1           1000000.	Z.         S Deg         [Steel] 1020 [Sample.xml]         UCA_FE.SetPatch1@Suspension_Assy         History_1         1000000.         Face Node ID         Safety Factor (Min.)         1280,1281,462         32.462583820492         Calculation       OK

• Select this option in the **Contour Plot** to produce the following results.

Durability Contour					
Time History	History_1				
View Type	Contour			2	
Contour Option	Style Option			$\geq$	
Safety Factor	Color Option Edit			$>_{\circ}$	
Band Option	Colors Spectrum 💌				
Legend Type Display 💌	Style Stepped 💌	X VX BI			
Pattern	Text Color				
Location					
Show Text Legend					
Band Level(10~50) 10					
- Min/Max Option	Mesh Lines				
Display	Vector Color				
Calculation	Vector Size Factor 3.89				
Min 105.8 105.8	Probe Option				
Max 552.57 552.57	O Node Select				
Show Min/Max	Patch     Clear		115		
Enable Log Scale	Show Probe Results				
Front	View OK				
- Capini			<u>V</u>		
1.06e - 02 62	Serete 1.384+62 1. 148+02 1. 05++V	2 1476+02 2.895+02 3.376+02 3.976+02 4.69	-02 5.55-02		

 The conditions required in the Safety Factor calculation are the stress amplitude and the mean stress from the Rainflow Counting results. Therefore, the occurrence data related to the number of cycles is unnecessary. The durability analysis results shown above reveal that the UCA\_FE body is very stable.

#### Analyzing the results for the LCA\_FE body

- Unlike the UCA\_FE body, you must enter the formulas for the durability analysis of the LCA\_FE body instead of using the ones provided by RecurDyn/durability, which create specific S-N Curves. This means that you can directly input data obtained from external experiments and/or related resources (stress amplitude vs. cycle) when setting the materials for the fatigue analysis by selecting User-Defined S-N Curve.
- As you can see in the rainflow counting results graph, the stress range and the mean stress are approximately 100 times larger than the results obtained for the UCA\_FE body.
- You can conclude from the fatigue analysis results that the LCA body can survive only up to 1 million times under the given load conditions. Therefore, a design change is essential in order to make the product more durability.
- If you were to determine the stability of the LCA\_FE Body based on the Von-Misess stress only, then the maximum stress would be about 139 Mpa, as shown below, which is far less than the yielded stress (262 Mpa). Therefore, this design could be considered stable. As far as durability is concerned, however, it is necessary to alter the design.



Fatigue Evaluation				
Axial Mode	Uni-Axial	(	Bi-Axial	
– Life Criteria –––––				
Stress - Based	🔘 Strain	- Based	◯ Safety Factor	
Life Criterion		User-Defined	•	
Mean Stress Effect		Goodman	•	
BWI Weld		class B	•	
Num of Std.Deviations	;	2.		
Searching Increment		10 Deg	•	
- Material				
S-N Curve < mm-N >		User-Defined	S-N	
Element / Patch Set		LCA_FE.SetPatch1@	Suspension_Assy EL	
Time History		History_1 SEL		
Occurrence		1000000. Pv		
– Fatigue Results –––––				
Time Range	Face Node ID	Damage ( Max.)	Life ( Min.)	
History_1	71524,77590,65551	1.	1.	
Fatigue Tools	Import	Calculation	OK Cancel	

Thanks for participating in this tutorial