

### Helpful Hints and Tips: Adding Contact in a Hand Meshed Model



ESRD, Inc. 2018





© 2018 ESRD, Inc. All Rights Reserved. StressCheck® is a registered trademark of ESRD, Inc.





- Model Overview
- Problem Outline
- Setting up SFAT problem
- Exporting SCW from SFAT
- Deleting elements
- Creating new circles with temporary "gap" parameter
- Creating nodes, elements, contact zones
- Reassigning bushing material and contact constraints
- Set "gap" to 0 and solve

#### Model Overview

- Single protruding fastener, double-shear lap joint
- Aluminum plates, titanium fastener, and steel bushing in top plate only
- All neat fit parts, no gaps between plates
- SCW or SCP model exported from SFAT





#### **Problem Outline**

- By default, SFAT currently assumes a neat fit bushing and it is modeled as though it is bonded to the hole bore in the plate (no contact)
  - SFAT is primarily used to compute joint stiffness, but here we are interested in max. S1 at hole bore
- In this problem, we will add contact between the bushing OD and hole bore









- General instructions for setting up an SFAT problem can be found in the SFAT User's Guide
- This particular problem uses the following configuration:
  - Joint shear type: double, fastener type: protruding, plate width = 0.568" & edge distance = 0.378"
  - Bolt shank diameter = 0.210" (neat fit), fastener nut/head dimensions left at default values, with custom fastener material properties (E = 16.50 Msi and v = 0.31)
  - Plate 1, 2, & 3 are identical with aluminum material properties (E = 10 Msi, v = 0.33), thickness = 0.21"
  - No gaps between plates
  - Plate 1 bushing thickness = 0.03" with custom material properties (E = 28.50 Msi, v = 0.29)
  - No bushing for plates 2, 3
  - No washers
  - Joint shear load = 1000 lb, no fastener pre-load
- On the SFAT Input Summary page, click on the drop-down arrow next to the Solve button and select "Build mesh only"

Solve 💌
Multi-run
Single run
Build mesh only

 Once the mesh builder is completed, click the Export button on the Results View page & save the file as "SFATExample.scw" and open it in StressCheck

#### **Delete bushing contact zones**



- In order to delete the bushing elements, the contact zones on the yellow bushing elements must first be deleted to remove element dependencies:
  - Display elements and contact zones only
  - Turn off wetted faces
  - Shrink elements



### **Delete bushing contact zones**



- **STRESSCHECK**
- In order to delete the bushing elements, the contact zones on the yellow bushing elements must first be deleted to remove element dependencies:
  - Display elements and contact zones only
  - Turn off wetted faces
  - Shrink elements
  - In the Mesh tab, set the AOM to Select > Contact Zone > Faces
  - Hold SHIFT while left-clicking to select each contact zone (3 total) on bushing elements



#### 7/25/2018

#### **Delete bushing contact zones**

 In order to delete the bushing elements, the contact zones on the yellow bushing elements must first be deleted to remove element dependencies:

ESRD. Inc.

- Display elements and contact zones only
- Turn off wetted faces
- Shrink elements
- In the Mesh tab, set the AOM to Select > Contact Zone > Faces
- Hold SHIFT while left-clicking to select each contact zone (3 total) on bushing elements
- Press Delete and ignore the warning about set TOP\_LOW being deleted





8

STRESSCHECK

#### **Delete bushing elements**

• Open "SFATExample.scw" in a new StressCheck instance and set display:

+ 🗆 📙 🔶 轴 🖽 🕀 🎫 🥅 🔻

「コマダ 田 口 回 図 回 日 役 😔 🍭

- Display elements only
- Keep wetted faces off
- Unshrink elements
- To delete all the yellow bushing elements, go to the Mesh tab and set the AOM to Select > Any Element > Selection:
  - Hold CTRL + SHIFT and left-click on any one of the yellow elements (this will select the entire set of bushing elements, and the Sets Browser window will pop up)
  - Press Delete and ignore the warning about set TOP\_BUSHING being deleted







### Create "gap" parameter



- Open the Model Info window and go to the Parameters tab:
  - Create a new parameter called "gap" and set it to something small (0.005)
    - This is a temporary parameter that will help us to create new bearing elements so that contact can be applied to both the inner and outer surfaces

StressCheck Model Information ×												
	Model Inf	o Parameters Rules							_			
	Name	Description	Expression	Value	Limit	Class		Sort				
	delta	Joint Pre-load displacement		0.0000e+000 ≑		General	•					
	gap	Temporary gap parameter		5.0000e-003 ≑		General	•					
V Joint Shear				1.0000e+003 ≑		General	•					
	Accept Delete Auto Step: 0.2											
-	Input	Settings										

#### • Generate the bushing wireframe:

- Turn on display systems, curves, and elements + 
   Lo 
   B 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C 
   C
- In the Geometry tab, set the AOM to Create > Circle > Local, and make sure the Curve/Surface toggle is set to Curves
- Turn on the Radius checkbox and enter: 0.21/2+gap
  - We know the pin diameter is 0.21" from our SFAT problem, but if we weren't sure, we could measure it by computing the distance between 2 nodes by setting the AOM to Check > Node > Distance in the Mesh tab and then selecting two nodes in sequence



STRESSCHE

 With the AOM set to Create > Circle > Locate and Radius: 0.21/2+gap, left-click on one of the local systems centered on the fastener shank at the **lower** face of the top plate to create a circle representing the bushing ID





STRESSCHECK

 With the AOM set to Create > Circle > Locate and Radius: 0.21/2+gap, left-click on one of the local systems centered on the fastener shank at the upper face of the top plate to create a circle representing the bushing ID





STRESSCHE

- Repeat the previous two steps with Radius: 0.21/2+0.03-gap to create circles representing the bushing OD
  - The pin diameter is 0.21", and the bushing thickness is 0.03"



#### Create nodes on the new circles

#### Create nodes for hand-meshed elements:

- Turn on display nodes
- In the Mesh tab, set the AOM to Create > Node > Offset
- Turn on the Repeat checkbox and enter # = 8
- Enter Off1: 0 + 45, and Off2: 0 + 0
- Left-click once on each of the 4 circle curves we just created
  - This will create 8 nodes uniformly spaced apart by 45 degrees on each circle curve





#### Create nodes on the new circles



• Here, the mesh is transparent and all the new nodes are highlighted in red for clarity





- Manually mesh bushing with hexahedra:
  - In the Mesh tab, set the AOM to Create > Hexahedron > Selection
  - With elements and nodes displayed, turn on wireframe mode

,∎ < 😵 🖽 🗅 🖿 🖻 🐼 🖿 🖿 😔 😂 🗄 + 🗖 ⊢ 🗢 🖼 🕀 🚥 📾 🗸





- Manually mesh bushing with hexahedra:
  - In the Mesh tab, set the AOM to Create > Hexahedron > Selection
  - With elements and nodes displayed, turn on wireframe mode

,∎ < 😵 🖽 🗅 🖿 🖻 🐼 🖿 🖿 😔 😂 🗄 + 🗖 ⊢ 🗢 🖼 🕀 🚥 📾 🗸





- Manually mesh bushing with hexahedra:
  - In the Mesh tab, set the AOM to Create > Hexahedron > Selection
  - With elements and nodes displayed, turn on wireframe mode

,∎ ⊲ 😵 🖽 🗅 🖿 🞯 🐼 🖿 🖿 🚱 😔 🗧 + 💶 ⊢ 🔿 🏎 📾 🗣 🚥 📾 ▼





- Manually mesh bushing with hexahedra:
  - In the Mesh tab, set the AOM to Create > Hexahedron > Selection
  - With elements and nodes displayed, turn on wireframe mode

,∎ ⊲ 😵 🖽 🗅 🖿 🖻 🗃 📾 📾 😔 🍩 🗄 + 🗖 ⊢ 🗢 🖼 🕀 🚥 📾 ▼





Re-apply the bushing and plate contact zones:

- Display elements & contact zones, turn on shaded mode, make sure wetted faces is still turned off, and shrink the elements 💷 🕸 🖻 🖻 🖻 🖻 📾 😂 🍩
- We need to create 5 contact zones, each spanning multiple element faces:
  - 1) Inner faces of the bushing (bore)
  - 2) Outer faces of the bushing (shank)
  - 3) Faces of the top plate hole (bore)
  - 4) Bottom faces of the bushing & top plate
  - 5) Top faces of the bushing & top plate



- In the Mesh tab, set the AOM to Create > Contact Zone > Face Surface
- Left-click on an inner element face of the bushing to select all faces on the bore as shown below, and click Accept to create the contact zone

eometry M	esh Se	ction Prop.	Thickness	Material	Loa
reate	$\sim$	Contact Zor	ne v F	ace Surface	• ~
Screen:					
Tol.:	0.000	000e+000			
Data Inde	×				
Face(s) 53/3	3, 54/3, 5	5/3, 56/3,			
Accept	R	eplace	Delete	DeL	.ast





- STRESSCHECK
- Left-click on an outer element face of the bushing to select all faces on the shank as shown below, and click Accept to create the contact zone

StressChee	ck Input				x
Geometry	Mesh	Section Prop.	Thickness	Material	Loa 🔹 🕨
Create		✓ Contact Zor	ne v Fa	ce Surface	e ~
Scre	en.				
Tol	.: 0	.000000e+000			
Data [	la dau				
Eace(e) F	3/3 5//	3 55/3 56/3			
Acce	pt	Replace	Delete	DeL	.ast
Merg	e.	Сору	Assoc.	Dese	lect







 Left-click on an element face of the top plate hole to select all faces on the hole bore as shown below, and click Accept to create the contact zone

StressChee	ck Input				
Geometry	Mesh	Section Prop.	Thickness	Material	Loa
Create		✓ Contact Zor	ne v Fa	ace Surfac	e v
Scre	en:				
		00000000			
	.: U	.000000e+000			
Data	ndex				
Face(s) 5	53/3, 54/	3, 55/3, 56/3,			
Acce	pt	Replace	Delete	Del	.ast
Merg	je	Сору	Assoc.	Dese	elect





faces of bushi	ng elements as w
bushing, then	press Accept to c
<ul> <li>TIP: If you a deselect ind</li> </ul>	accidentally select u dividual objects whe
StressCheck Input ×	
Geometry Mesh Section Prop. Thickness Material Loa	
Create ~ Contact Zone ~ Faces ~	

 Set AOM to Create > Contact Zone > Faces and hold SHIFT while left-clicking to select the top faces of bushing elements as well as the top faces of the innermost ring of plate elements at the bushing, then press Accept to create the contact zone

 TIP: If you accidentally select undesired faces, you can hold CTRL and left-click (or left-click and drag) to deselect individual objects when multiple objects are currently selected











- Set AOM to Create > Contact Zone > Faces and hold SHIFT while left-clicking to select the bottom faces of bushing elements as well as the bottom element faces of the top plate as shown below, then press Accept to create the contact zone
  - TIP: If you accidentally select undesired faces, you can hold CTRL and left-click (or left-click and drag) to deselect individual objects when multiple objects are currently selected



#### Assign material properties to bushing elements

#### • Re-assign bushing materials:

- Turn off Display Contact Zones
- In the Material tab > Assign sub-tab, set the AOM to Select > Any Element > Selection, choose "topBushing" from the ID input field drop-down arrow, and choose "Fastener" in the Color drop-down menu
- Hold SHIFT and left click on each of the 4 bushing elements, then click Accept

Geometry	Mesh	Sectio	n Prop.	Thickne	ss N	laterial	Loa 1
Select		~ Any	/ Element	$\sim$	Sele	ction	``
Object	Material I	D Sys	tem All/S	Set			
Element	bot Plate		0, bot	_plate, El	asticit	у	~
D:	topBush	ning	$\sim$	Set:	New	set	~
Scale	:		•	]	<u> </u>	ulti-laye	r
System	Global		$\sim$	Color.		Fasten	er v
-							
Type:	Homog	eneous	~				
Type:	Homog	eneous	~				
Define	Homog	Fitting	~				





ESRD, Inc.

STRESSCHECK

#### Delete errant contact constraint records

STRESS CHECK

- Delete constraint records assigned to the following link sets:
  - FAST\_TOP\_MID, FAST\_TOP\_UP, TOP\_MID\_HZ

tressCheck Input x						
Section Prop. Thickness Material Load Constraint So • •						
Select V Any Curve V General V						
Object Method Name Data						
Face ,Symmetry ,CONST ,botSym ~						
ID: Set: New set ~	🐼 Edit Const	raints			×	
Scale:	Constraint ID	Show All	~			
Direction: XYZ ~	Delete	Purge ID P	urge All	< Pre	evious Next >	StressCheck
System: Global V	Object	Method	ID	Data	^	
	Contact	Spring	CONST	fast bot mid		
_	Contact	Spring	CONST	fast_mid		Delete 3 selected records?
Data Type: 🔄 Fixed	Contact	Spring	CONST	fast_top_mid		
	Contact	Spring	CONST	fast_top_up		
	Face	Symmetry	CONST	midSym		
	Face	Symmetry	CONST	mid_Uz_0		
⊠ X:	Contact	Spring	CONST	SET61		Yes No
	Face	Symmetry	CONST	topBush_Sym		
	Face	Symmetry	CONST	topSym		
	Contact	Spring	CONST	top_mid_hz		
⊻Z:						

Deselect



- Assign contact between bushing, fastener and plates
  - Turn on Display Contact Zones and turn off Display Elements (leave Shrink Elements on)
  - In the Constraints tab, set the AOM to Select > Contact Zone > Contact, enter ID: CONST, make sure Set is set to "New set" and enter a Contact Constant of 1e6
  - Left-click to select a pair of contact zones and press Accept

StressCheck Input		
Section Prop. I hickness Material Load Constraint So • • • Select		
Object Method Name Data Contact Spring CONST bot_mid_hz ~		
□ID: CONST Set: New set ✓		
Direction: Nom./Tan. V System: V		
Control Constants   10000-000		
Contact Constant: 1.0000e+000 Centerate		
Accent Replace Delete Pume	Z	Z
Edit Deselect		

STRESSCHECK

Repeat for all unassigned contact zone pairs!







STRESSCHECK

Repeat for all unassigned contact zone pairs!







ESRD, Inc.

STRESSCHECK

Repeat for all unassigned contact zone pairs!







#### **Create Solution Record**

- Since we used "Build-Only", there is no Solution ID:
  - In Solution ID tab, enter Solution ID: SOL
  - Select Constraint Record CONST and select Load Record LOAD

	Calution ID	<b>D1 1 1</b>		<b>.</b>	<b>T</b> 4
Constraint	Solution ID	p-Discretization	h-	Discretization	li 1
Define	$\sim$	Name	$\sim$	Selection	~
Active, So	olution, Cons	traint, Load			
					~
$\checkmark$	Solution ID	SOL			
Constrain	t Records				
CONST					
(	Constraint ID	CONST			
Load Red	cords				
LOAD					
	Load ID	: LOAD			
	Load ID	LOAD			
Solutions	Load ID	LOAD			
Solutions	Load ID	LOAD			



#### **Create Solution Record**

- Since we used "Build-Only", there is no Solution ID:
  - In Solution ID tab, enter Solution ID: SOL
  - Select Constraint Record CONST and select Load Record LOAD
  - Press Accept to create the Solution Record

Constraint	Solution ID	p-Discretiza	tion h-l	Discretizati	on Ta
Define	~ 1	Name	$\sim$	Selection	~
Active, So	lution, Constr	aint, Load			
YES, S	OL, CONST,	LOAD			~
$\checkmark$	Solution ID:	SOL			
Constraint	Records				
CONST					
(	Constraint ID:	CONST			
Load Rec	ords				
LOAD					
	Load ID:	LOAD			,
Solutions	Configuratio	ns			



# Change gap parameter



Our temporary gap parameter is still set to 0.005"

StressChe	eck Model Information			_	_	_	×
		-					
Name	Description	Expression	Value	Limit	Class	S	ort
delta	Joint Pre-load displacement		0.0000e+000 韋		General	-	
gap	Temporary gap parameter		5.0000e-003 ≑		General	-	
V	Joint Shear		1.0000e+003 🌻		General	-	
Accept	Delete A	Auto Step: 0.2					
· · · [							
Input	Settings						

## Change gap parameter



- Set the gap back to "0.0"
  - Display elements only
  - Open the Model Info window and go to the Parameters tab
  - Change the parameter "gap" to a value of 0
  - Press Accept



Solve

ESRD, Inc.

#### 37

Solver a linear p-extension: Solver a linear p-extension:

- Open the Solver interface
- In the Linear tab, set the p-limits from 6 to 8 •
- Go to the SOLVE! Tab, and press Solve

StressCheck Solver ×	StressCheck Linear Solver	StressCheck Linear Solver
Margin Check         Buckling         Crack Path         SOLVE !           Linear         Nonlinear         Coldworking         Modal         Measurement	Linear         Nonlinear         Coldworking         Modal         Measurement           Margin Check         Buckling         Crack Path         SOLVE !	Linear         Nonlinear         Coldworking         Modal         Measurement           Margin Check         Buckling         Crack Path         SOLVE
Extension: Upward-p v p-limits: 6 to 8	Execute:       Initialize          Run Mode:       Automatic          Method:       Direct          Converge:       None          Display:       Results	Execute:       Restart       Status: COMPLETE         Run Mode:       Automatic       Max: 3, Iter: 40         Method:       Direct       Max: Contact Pressure Error: 0.45%         Converge:       None       Display:       Results
	Scratch directory: C:\Users\ALEDBE~1\AppData\Local\Tem Solve Check Solve Settings File	Scratch directory:       C:\Users\ALEDBE~1\AppData\Local\Tem         Solve       Check         Solve       Settings         File



Measurement

SOLVE !