Multiple Bolt Preload Setup in Mentat



• Objectives:

- This workshop illustrates Efficient and user friendly method of defining common behaviour multiple bolts using the latest Bolts feature introduced in Mentat 2018
- This new feature is useful in assemblies with a large number of fasteners with preload or, springs. Multiple bolts can quickly be generated using a single action.
- Using this method, after selecting the elements of bolt, Mentat will automatically create and position the control node. The axial direction of the bolt is also automatically determined. User has to define the preload applied on control node in the first degree of freedom as this new technique is based on "Cross-Section" option. The control node of the Cross-Section has only one degree of freedom.
- During post processing, the bolt force is plotted in the bolt axial direction. Global variables for control node displacement, as well as axial and shear force are also available.
- With this new bolt modeling technique, for Large Strain analysis, the limitation of small rotations that existed in previous releases has now been removed.

• Software Version :

Marc and Mentat 2018

• Files Required:

- bolted_plates.mud

Suggested Exercise Steps:

- 1. Open the file bolted_plates.mud in Mentat
- 2. Check the contact bodies
- 3. Populate the contact table
- 4. Create control node for bolt preload using Bolt feature
- 5. Define Boundary Condition, Clamp the lower plate
- 6. Define point load on the top plate
- 7. Apply Bolt preload on all 3 bolts of magnitude 2000N on the control nodes
- 8. Apply locking constraint on control nodes
- 9. Create 3 loadcases; Bolt preload, bolt locking & External loading. Choose appropriate Loads & Boundary conditions for each of these loadcases.

10.Create & run the job

Step 1. Open An Existing Database

Open an existing database.

- a. Select File > Open.
- b. Select bolt_preload.mud.
- c. Click Open.

M I	File Select View	/ Tools	Window	Help							
	Mod New Open Description Save Save and Exit Save As	el Ctrl+s a Ctrl+s Ctrl+s Ctrl+s	Shift+N D S Shift+S		larc Mentat ok in:	Oper Uter	C:\Training	- G	00	6	x)
	Restore Resu Open Default Open Import Export Current Directory Edit File Exit	Its	> >	File	Desktop Document e name: es of type: Absolute Pat	ts bolt_ All M th Cu	preload.mud odel Files (*.mud *.mfd) rrrent Directory			•	Oper C Cancel

Step 2. Check Contact Bodies

Identify contact bodies

- a. Expand Contact bodies then Meshed (Deformable).
- b. Click Toggle Identification of Contact Bodies.
- c. Click Fit.







Step 3. Populate contact table

Create contact tables:

- a. Right-click on the blank space in the Model Navigator and select: Contact > New Contact Table.
- b. Click Populate / Manipulate
- c. Enable Contact Distance **Check (Meshed Bodies** Only).
- d. Select Automatic as Contac Distance.
- e. Enable Skip Self-Contact Entries.
- Click **Execute** button f.
- Click OK g.

	P	-	
Materials •		M Popu	late / Manipulate Contact Table
Contact		Name	ctable1
Contact			Settings
ToolBox Net	w Contact Body	c) 🔽 😋	ontact Distance Check (Meshed Bodies Only)
Links 🕨 Det	tect Meshed Bodies	í í	
t Initial Conditions New	w Contact Interaction		Contact Distance Automatic (d)
Bour we condition	w Contact Table a	ſ	Contact Table Entries
Net	w Contact Area		Remove If No Contact Detected
Net	w Exclude Segments		Contact Interaction
×			Create Default
Entry Matrix 💌		Conta	act Type Touching
ries	-	Name	Automatic
Second		ļ	
/Type 1 2 3 4 5			Populate / Manipulate
ned (Deformable)		Method	All Body Pairs 🔻
ned (Deformable)			e Skip Self-Contact Entries
ned (Deformable)			
ped (Deformable)			Events ()
ed (Deformable)			Execute
Full Default Contact Touching Glued			ок д
к			



Step 4. Specify friction coefficient

Specify friction coefficient

- a. Expand Contact Interactions and double click interact1.
- b. Click Friction.
- c. Enter 0.3 for Friction Coefficient.
- d. Click OK twice.

Model List
🖃 🐋 bolt_preload
🕀 🖮 Mesh (43847)
🕀 🚟 Tables (2)
🕀 👼 Geometric Properties (1)
🕀 🙀 Materials (1)
🕀 🕂 Contact Bodies (5)
🚍 🙀 Contact Interactions (1)
🖹 🙀 Meshed (Deformable) vs. Meshed (D
interact1(a)
E Gontact Tables (1)
ctable1
have made and have a second

📹 Con	tact Interaction	Properties	×								
Name	nteract1										
Туре	Meshed (Deform	nable)									
	Meshed (Deform	nable)									
Current	: Job <i>job 1</i>										
		Contact Detec	tion								
Conta	ct Tolerance	Default	•								
Conta	ct Tolerance Bias	Factor Default	•								
Segme	Segments At Sharp Corners 📃 Tangential Contact Tolerance Extension										
<u></u>		Contact									
At Sta	art Of Analysis	Stress-Free Proje	ection Onto Contact Surface								
Conta	ct Type Touc	hina 🔻	Augmentation								
		Touching Cont	act								
			i								
Sh	ow Properties	Structural 💌									
	Friction (h	Wear	Separation								
	U										
Res	=t		OK (1								
📹 Cor	ntact Interactio	on Friction Param	eters X								
Name	interact1										
Туре	Meshed (Defo	rmable)									
	Meshed (Defo	rmable)									
Friction	n Coefficient	0.3 C	Table								
Friction	n Stress Limit	1e+20	Table								
🔳 Ani	sotropic Friction										

(d)

OK

Step 5. Make the plates invisible for bolt creation

Display only bolts and hide the plates

- a. Right click on **Contact Bodies** and click **Visibility.**
- b. Disable upper_plate and lower_plate contact bodies.
- c. Click Apply Changes.
- d. Click OK.



	📹 Contact Body Visibi	lity	×	
			🔲 Auto-Ap	ply
	Visibility	Туре		
Ц	📕 📃 upper_plate	Meshe	d (Deformable)	
	📃 📃 lower_plate	Meshee	d (Deformable)	
	middle_bolt	Meshee	d (Deformable)	
	Ieft_bolt	Meshee	d (Deformable)	
	✓ right_bolt	Meshee	d (Deformable)	
	Change To Visible	All	List	
	Change To Invisible	All	List	
	Discard Changes A	oply Changes	с) ок (d

Step 6. Bolt creation

V

a	Geometry & Mesh Tab	bles & Coord. Syst.	Geometric Properties	Material Properties	Contact	Toolbox	Links Ir	nitial Cond	itions Boundary Condit	ions Mesh Adaptivity	Loadcases	Jobs	Results
n Menu	Transformations ▼ C Cavities ▼ M Bolts ▼ C	Cross Sections 🔻 Matching Boundaries Chains 💌	Sink Point Groups Node Properties Streamline Regions 	Cracks Crack Initiators Delamination	▼ Wel	d Paths 🔻 d Fillers 👻	Coils Circuit	ts 👻	Design Variables 🔻 Design Constraints 🔻				
Mail	New Bolt	ral		Fracture Mecha	nics V	Velding	Electroma	agnetics	Design				
	Create Multiple Bo	olts a											
	Show Menu												
	Edit								🖌 Create Mu	Itiple Bolts			x
	Identify					100				Rolt Nam			
	Properties					n.		\checkmark	Passanama	DOILINAII	-		— []
С	reate bolts					5		V	Start Index	1			
a.	Select Toolb	oox > Bolts	>			_		– (e)		Avial Direct	ion		
	Create Multi	iple Bolts.				\sim			A shares the	Axial Direct	h		
b.	Verify Autom	natic is sele	cted					۲	Automatic		\mathbf{U}		
	as Axial Dire	ection.								ation Control Node	On Bolt Ax	dis —	
C.	To specify th	ne location o	f						Default		. on boic n		•
	control node	on bolt axis	, set						Offeet (Erac			\bigcirc	
	Offset as "1	".				+			Unset (Frac	uon of boit axis)	1	\bigcirc	
d.	Click Create	Multiple Bo	olts.						·	- Cross-Section L	ocation —		;

- e. Click All Visible Elements.
- f. Select Toggle Nodes Display.

(d)

Offset (Fraction) From Center Of Gravity 0

Create Multiple Bolts

OK

Н

Step 6. Bolt creation (Contd.)

Note that control nodes are created for each bolt in the normal direction, at distance of 1, as specified in previous step.



Note -The control node has only 1 dof in the direction of the cross section normal vector

Step 7. Apply boundary conditions

ramp external load

Clear

1

time

Specify bolt preload

- a. Right-click on the Model Navigator and select:
 Boundary Conditions >
 New Boundary
 Condition > Structural >
 Point Load.
- b. Enter boundary condition Name: bolt_preload.
- c. Enable Force X
- d. Enter 2000 for Force X.
- e. Click table
- f. Select
 ramp_bolt_preload.
- g. In the miscellaneous menu, change the node picking option from Standard Node Picking to Bolt Control Node Picking
- h. Click on any single element on each bolt to select the control node of it's corresponding bolt.
- i. Click **Toggle Boundary Conditions Display** to turn display on.



specifying a force on the first degree of freedom of control node. Notice that this force works in the direction of the cross section normal vector.

OK

Step 7. Apply boundary conditions (Contd.)

۲

٠

۲

۲

Name Type

Method

Links

Initial Conditions

Mesh Adaptivity

Boundary Conditions

Specify Bolt lock boundary condition

- a. Right-click on the blank space in the Model Navigator and select: Boundary Conditions > New Boundary Condition > Structural > Fixed **Displacement.**
- b. Enter boundary condition Name: bolt lock.
- Enable **Displacement X** C.
- Click Add. d.
- e. Select all the control nodes of the bolt.
- Repeat step a. f.



Fixed Displacement a New Boundary Condition Structural Fixed Acceleration State Variable Point Load X M Boundary Condition Properties Edge Load (b bolt_lock Face Load fixed_displacement Properties Global Load Entered Values Gravity Load Reference Position Position At Activation Of BC Centrifugal Load Time Dependence Tables -Edge Foundation Displacement X 0 Table Face Foundation Displacement Y Fluid Drag Displacement Z Rotation X Cavity Pressure Load Rotation Y Cavity Mass Load Rotation Z DOF-Set Nodes Fixed Harmonic Displacement Rem 0 Fixed Harmonic Acceleration Rem 3 Harmonic Point Load Rem 0 Rem 0 Harmonic Edge Load Rem 0 Harmonic Face Load Rem 0 Harmonic Global Load Rem 0 Rem 0 Rem 0

OK

Step 7. Apply boundary conditions (Cont.)

Model

÷...

Ė٠

÷

÷ 1000

Ē. 2

÷.

List

🖬 bolt_preload

С

Mesh (43850)

Materials (1)

Contact Bodies (5)

Geometric Properties (1)

🖻 🖳 Meshed (Deformable) (5)

🗹 🎄 upper_plate

🗹 🎎 lower_plate

🗹 🎄 middle_bolt

🗹 🏯 right bol

🗹 🎎 left bolt

Tables (2)

Specify fixed boundary conditions

- a. Enter fixed for Name.
- b. Enable Displacement X, Displacement Y, Displacement Z.
- c. Enable visibility of every contact bodies.
- d. In the **miscellaneous** menu, change the node picking option from Bolt Control Node Picking to Standard Node Picking.
- Click Nodes: Add. e.
- a



8

+

🖌 Standard Node Picking

Properties

Position At Activation Of BC

Entities

Add

Add

Add

Add

Add

Add

Add

Add

Add

e

Rem 0

Rem 0

Rem

Rem 0

Rem 0

Rem 0

Rem 0

OK

Table

Table

Table

m 0

290 Rem

0

Bolt Control Node Picking

 \mathbf{C}

d

X

Name

Type

h

Method

Reference Position

Time Dependence

Displacement X 0

Displacement Y 0

Displacement Z 0

Rotation X Rotation Y

Rotation Z

티 🔼

Boundary Condition Properties

a

Entered Values

Tables

Model Section Vertices

Nodes

Points

Curves

Surfaces

Solid Vertices

Solid Edges

Solid Faces

Solids

Clear

fixed displacement

fixed

Step 7. Apply boundary conditions (Cont.)

Fixed Displacement Initial Conditions Specify point load on top plate Fixed Acceleration a. Right-click on the Model Navigator Point Load a Boundary Conditions New Boundary Condition Structural ۲ and select: Boundary Conditions Edge Load State Variable Mesh Adaptivity > New Boundary Condition > Face Load General Structural > Point Load. Global Load Loadcases b. Enter external load for Name. Jobs Enable Force Z. C. Results х Enter 50 for Force Z. Boundary Condition Properties d. Click Table. Name external_load b e. х Currently Defined Tables (All Types) Type point load Select ramp external load. f. # V Independent Variable Type(Properties Name Click Nodes: Add g Method Entered Values ramp bolt preload 1 time Select right lower edge nodes of h. f amp_external_load Follower Force 1 time top plate. Click **OK** Force X Clear OK Force Y MSC Software d Force Z 50 Table ramp_external_loa С upper_plate Moment X Moment Y lower_plate Moment Z middle_bolt left_bolt Entities Model Section Vertices 0 Add lem right_bolt e Nodes Add 58 Rem Points 0 Add Rem h Solid Vertices Add 0 Rem 🗶 🗹 📼 👁 Clear OK i

Bolt Modeling – 3 Steps



Step 7. Define Loadcases

Create First Loadcase to preload the bolt:

- a. Right-click a blank place in the Model Navigator. Select
 Loadcases > New Loadcase > Static.
- b. Enter lc1_bolt_preload for Name.
- c. Specify Number of Steps as 10.
- d. Click Loads
- e. Under Applied Loads, disable bolt_lock and external_load.
- f. Click OK.
- g. Click Contact.
- h. Click Contact Table.
- i. Select ctable1.
- j. Click OK.

<u>redhad</u>

Mesh Adaptivity

Loadcases

Jobs

 k. Right click on Ic1_bolt_preload in the Model Navigator and select
 Copy to create a new loadcase.

New Loadcase



Step 7. Define Loadcase (Cont.)

Define Second Loadcase to the bolt:

- a. Double click on Icase2 loadcase.
- b. Enter lc2 bolt lock Name.
- c. Specify Number of Step
- d. Click Loads.
- e. Verify fixed and bolt is selected.
- Click OK. f.
- g. Right click on Ic1_bolt_preload in the Navigator and select Cor create a new loadcase.

🖻 🐺 Loadcases (2) 🖻 🐺 Structural Stati

🦏 Sets (7)

Boundary Condition

	M Select Loads	X
adcase to lock	Applied	Loads
	✓ bolt_preload	point_load
n Icase2	e bolt_lock	fixed_displacement
	▼ fixed	fixed_displacement
1+ lock for	external_load	point_load
er of Steps as 1.		
	Clear	OK
and bolt lock	tar)-onon	STS Con Contraction
	🖻 🔫 Loadcases (2)	
	🖻 🐺 Structural St	tatic (2)
	💌 🚺 lc1_bolt	_preload
	Tcase2	(a)
and in the Medel	🗄 🥰 Sets (7)	-
	hand	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
select Copy to		
badcase.		
indary Conditions (4)		
dcases (2)		
Structural Static (2)		
tc1_bolt_preload	New Loadcase : Static	
te lc2 bolt lock		
s (7)	Change Type	
m	Rename	
	Сору 🕝	
	Delete	
	Properties	

f

	ame pe loads lGaps Contact lGlobal Reme VCCT lCrack Initiat Design Cons Supe	Ic2_bolt_loc Structural static d shing Crack Propa ors	ex b		🔲 Inertia	a Relief	
Tyr	pe Loads Gaps Contact Global Reme VCCT Crack Initiat Design Cons Supe	Structural static d shing Crack Propa ors	agation		🗌 Inertia	a Relief	
	l Loads Gaps Contact Global Reme VCCT Crack Initiat Design Cons Supe	static d shing Crack Propa ors	agation		🔲 Inertia	a Relief	
	I Loads Gaps Contact Global Reme VCCT Crack Initiat Design Cons Supe	d shing Crack Propa ors	agation		🗌 Inertia	a Relief	
	Gaps Contact Global Reme VCCT Crack Initiat Design Cons	shing Crack Propa ors	agation				
	Contact Global Reme VCCT Crack Initiat Design Cons	shing Crack Propa ors	agation				
] Global Reme VCCT] Crack Initiat] Design Cons Supe	shing Crack Propa ors	agation				
	VCCT Crack Initiat Design Cons Supe	Crack Propa ors	agation				
	Crack Initiat Design Cons	ors					
	Design Cons	te nin te					
	Supe						
	oupe	rplasticity C	Control				
	S	olution Cont	trol				
	Con	vergence Te	esting				
	Numerical Preferences						
To	tal Loadcase	Time	1		🔲 Termir	nation Criteria	
				ng Proo	cedure		\bigcirc
F	Fixed 💿	Constant 1	Time Step	1		# Steps 1	C
A	Adaptive 🔘	Multi-Crite	ria			Param	eters
	0	Arc Length	n			Param	eters
	0	Temperatu	ire			Param	eters
	Tim	ne Step Cut	Back				
	Automatic	Time Step C	ut Back				
#	# Cut Backs A	llowed	10				
	Remesh Du	iring Cut Ba	ck Process				
	Lo	adcase Res	ults				
	Deactivation	n / NC Machi	ining				
	Input File Te	ext 🛛	Include File				
	Title						

Step 7. Define Loadcase (Cont.)

Define Third Loadcase to keep the bolt locked and apply point load :

- a. Double click on Icase3 loadcase.
- b. Enter
 lc3_external_loading for
 Name.
- c. Specify Number of Steps as 10.
- d. Click Loads.
- e. Select fixed, bolt_lock and external_load.
- f. Click OK.
- g. Click OK.

h Anna the arrive the	and the second								
E Loadcases (3)									
🖃 🐺 Structural Static (3)									
lc1 bolt	lc1 bolt preload								
trase3 a									
🕀 🧮 Sets (7)									
Lan	man								
Colored Lands	X`								
M Select Loads									
Applied	Loads								
✓ bolt_preload	point_load								
bolt_lock	fixed_displacement								
e fixed	fixed displacement								
i inted									
v external_load	point_load								
v external_load	point_load								
v external_load	point_load								
external_load Gradually Released Loads	point_load								

🔞 Loadcase F	properties		~			×
Name	lc3_exter	nal_loading	(b)			
Туре	Structura	1/	0			
	static					
Loads	(d)			🔲 Inertia	a Relief	
Gaps	\smile					
Contact						
🔲 Global Rem	eshing					
VCCT	Crack Pro	pagation				
🔲 Crack Initia	itors					
Design Con	straints					
Sup	erplasticity	Control				
	Solution Co	ntrol				
Cor	nvergence	Testing				
Nun	nerical Pref	erences				
Total Loadcase	Time	1		Termi	nation Criteria	
	_	Steppi	ng Pro	cedure		
Fixed (Constan	t Time Step	0.1		# Steps 10	\mathbf{C}
Adaptive) Multi-Cri	teria			Param	eters
) Arc Leng	ith			Param	eters
) Tempera	iture			Param	eters
Ті	me Step Cu	ut Back				
Automatic	Time Step	Cut Back	_			
# Cut Backs /	Allowed	10				
Remesh D	uring Cut E	Back Process				
L	oadcase Re	esults				
Deactivatio	n / NC Mad	thining				
🔲 Input File T	ext	Include File	2			
Title						
Reset						OK

Step 8. Create a Job

Create a Job

- a. Right-click a blank place in the Model Navigator.
 Select Jobs > New Job > Structural.
- b. Select lc1_bolt_preload, lc2_bolt_lock and lc3_external_loading.
- c. Click Analysis Options.
- d. Click Large Strain.
- e. Click OK.
- f. Click Contact Control.







Define Contact Settings

- a. Select **Segment To Segment** as **Method.**
- b. Select Version 2 as Default Settings.
- c. Verify Finite Sliding is selected as Model.
- d. Select Coulomb Bilinear (Displacement) as Type.
- e. Click Initial Contact.
- f. Click Contact Table.
- g. Select ctable1.
- h. Click OK.
- i. Click OK.

🛃 Currently	Defined C
	ctable1 g
Clear	ОК
	Currently Clear

📹 Conta	ct Contro	bl			x				
Name	job 1								
Туре	Structur	a/							
Method			Segment To	Segmen	a) -				
Default Se	ttings		Version 2	(b)	-				
Contact D	etection I	Method	Standard	Ŭ	-				
·			ding	<u> </u>	ĺ				
Model		Finite S	liding	c	-				
Threshol	d	Automa	atic 🔹 💌						
		Frie	ction						
Туре		Coulomb) Bilinear (Disp	lacement	d				
🗌 Initial	Contact		e						
	Advanced Contact Control								
		C	ж (ј)						

Step 8. Create a Job (Cont.)

Select initial loads	🔞 Select Initial Loads 🛛 📃	x		
	Boundary Conditions Clear			
	bolt_preload point_load			
b. Select Fixed.	bolt_lock fixed_displacement			
c. Click OK.	fixed fixed_displacement			
d Click Job Posults	external_load point_load			
e. Select Cauchy Stress and	Taikel Conditions			
Equivalent Von Mises Stress to	Clear	Job Properties		×
request these outputs.		Name job 1		
Click Run.		Type Structural		
		Linear Elastic Analysis	Londeneon	
Job Results		Selected Clear	Loducases	
Name job1	ок С	lc1 bolt preload	Structural static	Up Down
Post File		lc2_bolt_lock	Structural static	Up Down
Binary V Status File	ng Adams MNE	lc3_external_loading	Structural static	Up Down
Default Style Increment Frequency 1 Flowlines Force	Balance			
Selected Element Quantities	Available Element Tensors			
Clear	Filter Apply Clear			
Quantity Layers	Stress	Available		
Cauchy Stress Default Cir	Global Stress			
	Cauchy Stress			
e	Available Element Scalars			
	Filter Apply Clear			
	Equivalent Von Mises Stress			
	Mean Normal Stress Equivalent Cauchy Stress			
Element Results All Dointe	Total Strain Energy Density		Design	Analysis Options
Selected Nodal Quantities		Contact Control	Global-Local	Job Parameters
Default		Mesh Adaptivity	Steady State Rolling	Analysis Dimension
		Active Cracks	Map Temperature	3-D 💌
		Crack Initiators	Model Sections	Material Data File
		Deactivation DMIG	Out	Check
Contact Glue Forces		🗆 Input File Text 🛛 In	dude File	
Iterative Results Off	-	Title		Run
ок (f		Reset		ОК

Step 9. Run the Job

Run the Analysis

a. Click Save Model.

- b. Click Submit (1).
- c. Click Monitor.
- d. When status shows Complete, Click Open Post File (Model Plot Results Menu).

┥ Ru	n Job							×
Name	job1							
Туре	Structura	a/						
Us	er Subro	utine Fi	le					
So	lver/Para	llelizatio	on					
Symmetric Solution No DL			DМ	DM				
Pardiso Direct Sparse Solver			1 Assembly/Recovery Thi					
1 Solver Thread								
Title	St	yle ·	Table-Dr	iven		-	Sa	ave Mode
Submit (1) (b) Advanced Job Submission								
Update Mor		Monit	or)) Kill			
Status				Complete				
Current Increment (Cycle) 21 (2)								
Singularity Ratio					0			
Convergence Ratio				0.02124				
Analysis Time				3				
Wall Time				154				
Total								
Cycle	es	28		Cut Backs 0				
Sepa	rations	0		Reme	Remeshes 0			
Exit Number 3004		3004	Exit Message					
Edit	Outpu	ıt File	Log Fi	e Status File Any File				
Open Post File (Model Plot Results Mer d								
Reset					ОК			

Step 10. Create Equivalent Von Mises Stress Contour Plot

