CSE.ILLINOIS.EDU

İ ABAQUS

Contents

- 1. Exercise 5.1: Plastic Connecting Lug
 - 1. Import model
 - 2. Modify Properties
 - 3. Error Check
 - 4. Rerun & Postprocess
- 2. Credits

Exercise 5.1: Plastic Connecting Lug

We try to add some plasticity into the linear connecting lug model

Import the example cae file for linear elastic Connecting Lug model

In the main menu, select Plug-ins→Abaqus→Getting Started, select Connecting lug example (Run, you may see some error message, it is Ok).



In the Models tree, you have two models, one is explicit, the other is standard, we copy the standard model, name it Plasticity. (Note: Right click on standard and copy model)

Modify material property and Apply Large Load

The previous model is linear elastic, we need to change the material property. We will first try perfect plasticity (see below). Expand the model Plasticity → Materials, edit the Steel material. In the Edit Material dialog, select Mechanical→Plasticity→Plastic



From the stress-strain curve, the yield stress is 380E6 and plastic strain is 0.0(perfect plasticity) Expand the Step tree, double click on the second step: LugLoad, the Edit Step dialog appears. Here we go to Incremenation tab, change the Initial increment size to 0.2 (Note: For a nonlinear process, you always need small increment to get converge.) To make the plasticity occur, we need a load larger than 380E6, expand LugLoad tree, expand Loads, Pressure load (Created). In the Load Manager, Edit Load, change the Magnitude to 10E7.

Error check and Correction

Now, you can run the job, close Plasticity tree and expand Analysis. double click on the Jobs, create a new job named PlasticLug. Select Model Plasticity. In the description, write down "Perfect Plasticity". Submit the job.

🐥 Create Job 🛛 🔀
Name: PlasticLug
Source: Model 💌
Plasticity
explicit
standard
Continue Cancel

Unfortunately, we have some errors, open the monitor,

we can see that in the Errors Tab: "Time increment required is less than the minimum specified". Enter the Visualization module, in the Main menu, select Tool→Job Diagnostics; here you can see the details of each step and increment. Expand to the 1st Step, 1st increment, 1st iteration. You can see the residual

Job History	Summary Warnings Residuals Contact Elements	
Iteration 2 Iteration 3 Iteration 4 Iteration 5 Increment 10 Iteration 1	Summary Large strain increments: 34 Large strain increments	
 Attempt 2 Iteration 1 Iteration 2 Iteration 3 Iteration 4 Iteration 5 Iteration 6 Iteration 7 Iteration 8 Increment 11 Attempt 1 Attempt 1 Attempt 3 Iteration 1 Iteration 2 Iteration 3 Iteration 4 	Details Element LUG-1.58 LUG-1.59 LUG-1.61 LUG-1.62 LUG-1.63 LUG-1.64 LUG-1.91 LUG-1.92 LUG-1.93 LUG-1.94 LUG-1.95 Highlight selections in viewport	
	Dismiss	

load of 60 kN exceeds the limit load of the lug, and the lug collapses when the material yields at all the integration points through its thickness. The lug then has no stiffness to resist further deformation because of the perfectly plastic post-yield behavior of the steel.

Now we add a hardening process in the material property (Fig). Open the Steel material, in the Edit Material dialog, edit the Plastic Material Behaviors. Add one more line. E=580E6, v=0.35. Save and run again.



Rerun and Postprocess

This time, we success! Try to visualize the Von-Mises stress in deformed shape, and draw stress-strain curve (XY data, you may need to toggle on strain in the field output and rerun it to get strain data).

- 1. Draw the Von Mises contour.
- 2. Query tools

From the main menu, select Tool > Query or use the V icon, Select Probe values in Visualization Module Queries . Select Element, output position: Integration Pt, choose elements from viewport. 3.Draw stress-strain curve This time use two field outputs draw one XYplot. We will plot stress-strain curve at one integration point at the specific element shown below.



Use Display Group tool visualize only this element. Create XY Data from ODB field output. Extract Von-Mises stress and Ell as stress/strain value. In the Element/Nodes tab, toggle on Edit Selection, Method: Pick from viewport.

XY Data from ODB Field Ou Steps/Frames	tput	
Note: XY Data will be extract Variables Elements/Nodes Selection Method Pick from viewport Element labels	Edit Selection Add Selection Delete Selection	Active Steps/Frames
Element sets Internal sets		
	Highlight items in viewport	
Save	Plot	Dismiss

Select the element, click Done, click Save. Create XY Data by Operate on XY data.

🐥 Create XY Data 🛛 🔯
Source
ODB history output
ODB field output
Thickness
Free body
Operate on XY data
O ASCII file
Keyboard
Path
Continue Cancel

ample: maxEnvelope("XVData-2", "XVData-4") * 2.5 + "XVData-5"		
mbine ("E:E11 PI: LUG-1 E: 97 IP: 1", "S:Mises PI: LUG-1 E: 97 IP: 1")		
(Y Data	×	Operators A - XYData float, or intege
		X - XYData
ame Description 511 05 110 1 56 Cover Sink Data Sc511 at and instance 100 1 along at 07 intervalian point 1		I - integer
ETL PE LUG-LE S From Field Data: EELL at part instance LUG-Leternent 97 integration point 1 ETL PE LUG-LE (Ecom Field Data: EELL at part instance LUG-Leternent 97 integration point 2		E - float
E11 PE LUG-1 E (From Field Data: E/E11 at part instance LUG-1 element 97 integration point 2		/
E11 PL LUG-1 E: From Field Data: E-E11 at part instance LUG-1 element 97 integration point 5		1/A
E11 PE LUG-1 E-1 From Field Data: E-E11 at part instance LUG-1 element 97 integration point 5		abs(A)
F11 PE LUG-1 E: (From Field Data: E:E11 at part instance LUG-1 element 97 integration point 5		acos(A)
F11 PE LUG-1 E: (From Field Data: E:E11 at part instance LUG-1 element 97 integration point 7		append((X,X,))
F11 PE LUG-1 E (From Field Data: E/E11 at part instance LUG-1 element 97 integration point 8		asin(A)
Mises PELLIG-1 From Field Data: SMises at nart instance LUG-1 element 97 integration point 1		atan(A)
Mises Pl: LUG-1 From Field Data: S: Mises at part instance LUG-1 element 97 integration point 2		avg((A,A,))
Mises PI: LUG-1 From Field Data: S:Mises at part instance LUG-1 element 97 integration point 3		butterworthFilter(X,F)
Mises PI: LUG-1 From Field Data: SMises at part instance LUG-1 element 97 integration point 4		chebyshev1Filter(X,F,F)
Mises PI: LUG-1 From Field Data: S: Mises at part instance LUG-1 element 97 integration point 5		chebyshev2Filter(X,F,F)
Mises PI: LUG-1 From Field Data: S: Mises at part instance LUG-1 element 97 integration point 6		combine(X,X)
Mises PI: LUG-1 From Field Data: S: Mises at part instance LUG-1 element 97 integration point 7		cos(A)
Mises PE LUG-1 From Field Data: S: Mises at part instance LUG-1 element 97 integration point 8		cosn(A)
		currentMin(X)
		currentRna(X)
		currentAvg(X)
		decimateFilter(X,I)
		degreeToRadian(A)
		differentiate(X)
		exp(A)
		fit(X)
and the standard		integrate(X)
add to Expression D Skip Checks		interpolate(X)

In the Operate an XY Data box, choose operator combine, and select the stress/strain at the same integration point, plot the figure.

Credits

Neal Davis, Ruizhi Li, and Binyue Hou developed these materials for <u>Computational</u> <u>Science and Engineering</u> at the University of Illinois at Urbana–Champaign.

This content is available under a <u>Creative Commons Attribution-</u> <u>NonCommercial 4.0 Unported License</u>.

