University of Nottingham Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques



Worked Example 2 using the ABAQUS FE Software Material Properties, Boundary Conditions, Loads, Solving, Viewing Results



Options for accessing Abaqus:

Abaqus student edition is available from: <u>https://academy.3ds.com/en/software/abaqus-</u> <u>student-edition</u> (you will need to create an account – analysis is limited to structural models of up to 1000 nodes) or you can access via the <u>Remote Desktop Web Client</u>. Steps:

1. Open the cae file from the previous exercise (bracket.cae)



You can specify the properties of a part or part region by creating a section and assigning it to the part. When you create or edit a material, profile, or section, you must enter data in the appropriate editor. For example, when you create a material, you must enter data in the material editor.



2. Select the Property module

3. Select Create Material



escription:		
Material B	ehaviors	
	Elasticity Plasticity Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites Damage for Elastomers Deformation Plasticity Damping Expansion Brittle Cracking Eos	Elastic Hyperelastic Hyperfoam Low Density Foam Hypoelastic Porous Elastic Viscoelastic
	<u>V</u> iscosity <u>S</u> uper Elasticity	

4. In the Edit Material window, call the material Aluminium and then select Mechanical -> Elasticity -> Elastic

5. Enter the values shown below to define the Young's Modulus and Poisson's ratio, leave all other options as-is (Isotropic for e.g.) then click OK

≑ Edit Material		×
Name: Aluminium		
Description:		1
Material Behaviors		
Elastic		
<u>G</u> eneral <u>M</u> echanica	I <u>T</u> hermal <u>E</u> lectrical/Magnetic <u>O</u> t	her 🔗
Elastic		
Type: Isotropic	\searrow	▼ Suboptions
Use temperature-de	ependent data	
Number of field variab	les: 0	
Moduli time scale (for	viscoelasticity): Long-term	
No tension		
Data		
Young's Modulus	Poisson's Ratio	
1 70e3	0.3	
O	C	Cancel

The material definition will appear under the Materials section of the Model tree:

Model	Results	Material Lib	orary				1
😫 Mod	el Databas	e	× ‡	Ê.	202	٠ <mark>``@</mark> `	
⊟ <u>#</u> ∦ M ⊟ <u>M</u> o	odels (1) odel-1						
	Parts Parts Mater Alumi	(1) ials (1) nium	>				
	 ☆ Calibr ☆ Section 	ations ons es					Ð

6. Next, we need to create a section and assign the material definition to it. Select Create Section

III All Resources	😼 Engineering	g Desktop \times				
 Abaqus/CAE 2020 - Model File Model Viewpor File Model Viewpor Model Results Materia Model Database Model Database Model-1 Parts (1) Medel-1 Parts (1) Calibrations Sections Profiles Assembly Galibrations Sections Profiles Assembly Galibrations Sections Field Output Research History Output Time Points Hate Adaptive Nations Contact Control Contact Control Contact Stabilization Contact Stabilization	el Database: C:\Users\ t <u>View Material</u> Proper Library Library C C C C C C C C C C C C C C C C C C C	Lepzcjb\OneDrive - TI Section Profile Image: state stat	he University of <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u> <u>Composite</u>	Nottingham\Teac Assign Special Model: Model: Model:	hing\ABAQU	
The model data	base "C:\Users	s\epzcjb\OneDr	ive - The U	Jniversity o	f Notting	gh
→→	to search		0	Ħ 📄		

7. Leave the default options in the Create Section window (Name, Solid, Homogeneous) and click Continue...

Category	Туре
Solid	Homogeneous
🔾 Shell	Generalized plane strain
Beam	Eulerian
O Other	Composite

8. Next, we assign the material to the section, by default, the Aluminium material definition is chosen, leave the defaults and click OK



The section definition will appear under the Sections section of the Model tree:



9. To assign the section to the part, click Assign Section

🗰 All Resources 🛛 🗔 Engineering D	esktop X
+ Abaqus/CAE 2020 - Model Database: C:\Users\epz	jb\OneDrive - The University of Nottingham\Teaching\A
<u>File Model Viewport View Material Se</u>	ction <u>P</u> rofile <u>C</u> omposite <u>A</u> ssign Specia <u>l</u> Feat <u>u</u>
i 🗋 🚰 🔜 🖶 불 불	🕂 C 🔍 🔩 🚺 🚹 🛔 📙 🗛 💷
i 🎨 Property d	efaults 🖂 🖽 👻 🕴 🚺
Model Results Material Library	Module: Property V Model: Model-1
Se Model Database	
□ 🏦 Models (1)	÷ =
⊟ <u>Model-1</u>	
Parts (1)	
	Assign
Calibrations	The second secon
🖃 🤹 Sections (1)	
Section-1	
a assembly	
⊞ o唱 Steps (1)	
Field Output Requests	Bt. Da
History Output Requests	
ALE Adaptive Mesh Constraints	
1 Interactions	+ /
묩 Interaction Properties	
Contact Controls	
Contact Stabilizations	
Constraints	🔓 🖳
Connector Sections	(xy2) 4
	(++) A.
Loads	⇔, <u>↓</u> ,
BCs	
E Predefined Fields	
Remeshing Rules	
All Resources All Resources Constraints Contact Stabilizations Contact Stabilizations Constraints Contact Stabilizations Contact Stabiliza	
Annotations	
□ ‡‡ Analysis	Y
- A L L L L D	•
Co-executions	
The second secon	z 🗕 🗖 x
The model database "C:\Users\ep	zcjb\OneDrive - The University of Not
>>>	
F Type here to search	O Ħ 📄 🖭

10. When prompted to select the regions to be assigned a section, click on the bracket part in the viewport. This will also create a set containing the geometry, leave the default name (Set-1). Click Done.



11. The Edit Section Assignment window will appear, click OK to apply the Section-1 definition previously created with the Aluminium material definition

negion		
Region:	Set-3	
Section		
Section:	Section-1	~ \$
Note: L	ist contains only section pplicable to the selected of the sel	ons ed regions.
107		
Туре:	Solid, Homogeneous	



The part will turn green to show it has a section definition applied

12. Select the Assembly module



13. Next, add an instance of the bracket to the assembly, select Create Instance



14. The Create Instance window will open and by default the Bracket part will be selected. Click OK to add the instance and close the window.



An ABAQUS/CAE model uses the following two types of steps:

The initial step

ABAQUS/CAE creates a special initial step at the beginning of the model's step sequence and names it Initial. ABAQUS/CAE creates only one initial step for your model, and it cannot be renamed, edited, replaced, copied, or deleted.

The initial step allows you to define boundary conditions, predefined fields, and interactions that are applicable at the very beginning of the analysis. For example, if a boundary condition or interaction is applied throughout the analysis, it is usually convenient to apply such conditions in the initial step. Likewise, when the first analysis step is a linear perturbation step, conditions applied in the initial step form part of the base state for the perturbation.

Analysis steps

The initial step is followed by one or more analysis steps. Each analysis step is associated with a specific procedure that defines the type of analysis to be performed during the step, such as a static stress analysis or a transient heat transfer analysis. You can change the analysis procedure from step to step in any meaningful way, so you have great flexibility in performing analyses. Since the state of the model (stresses, strains, temperatures, etc.) is updated throughout all general analysis steps, the effects of previous history are always included in the response for each new analysis step.

There is no limit to the number of analysis steps you can define, but there are restrictions on the step sequence.

15. Select the Step module



16. Click Create Step



17. The Create Step window will appear where the Name of the step, position of the step and step type can be defined. Leave the defaults and create a Static, General step called Step-1, it will appear after the Initial step. Click Continue...

💠 Create Step	×
Name: Step-1	
Insert new step after	
Initial	
	-
Procedure type: General	
Procedure type: General Dynamic, Temp-disp, Explicit	^
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic	^
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer	^
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion	^
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Soils	^
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Soils Static, General	^
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Soils Static, General Static, Riks	
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Soils Static, General Static, Riks <	
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Soils Static, General Static, Riks	>

18. The Edit Step window will appear, where further options regarding the step can be defined, leave the default options and click OK

🖨 Edit Step					×
Name: Step-1					
Type: Static, General					
Basic Incrementation	Other				
Description:					
Time period: 1					
NIgeom: Off (This On of lar	setting con rge displace	trols the inclusi ments and affe	on of nonlinear cts subsequent :	effects steps.)	
Automatic stabilization:	None			\sim	
Include adiabatic heat	ting effects				
OI	K			Cancel	
0.0					

III Resources	Genering D	esktop $ imes$	
💠 Abaqus/CAE 2020 - Mod	el Database: C:\Users\epz	cjb\OneDrive - The	University of Nottingham\Teaching
Eile Model Viewpo Eile Model Viewpo	rt <u>V</u> iew <u>S</u> tep <u>O</u> utpu	it Oth <u>e</u> r <u>T</u> ools	Plug-ins <u>H</u> elp \?
i 🗋 🗃 🖩 🖶 🇯		। 🕂 🔿 🔍 🖓	< 1 : 目 昌 :
	Assembly	defaults 🗸 🛱	- 6
		Madula: A Sta	n
Model Results		Module: 🚽 Ste	p Model: Model-
🚝 Model Database	A A A A	۰ 📭 🕶	
🗏 🏥 Models (1)	1		
□ Model-1		110+0	
Parts (1)			
La Materials (1)		-+	
Calibrations		K 🛄	
🗏 🤹 Sections (1)		b	
Section-1		(XYZ)	
Profiles		+ 1	
Assembly		G	
Buschet 1	1)		
Bracket-1	onstraints		
E Position Co)		
🕀 👉 Sets (1)			
🛛 💐 Surfaces			
Connector	Assignments		
🗄 🛱 Engineering	Features		
□ o ^t ta Steps (2)			
H 0+1 Stop_1			
Elen conout R	equests (1)		
History Output	t Requests (1)		
Time Points			
📙 ALE Adaptive N	Mesh Constraints		
Interactions			
H Interaction Pro	perties		
Contact Control	ols		
A Contact Initial	zations		
Constraints	200013		
E Connector Sec	tions		Y
⊕ <i>F</i> Fields			A
Amplitudes			I
Loads			
BCs	lds.		
Remeshing Ru	les		
Doptimization T	asks .		
The model data	abase "C:\Users\ej	pzcjb\OneDriv	e - The University of No
>>>			
Type here	to search		O 🛱 🥫 🖭

Step-1 will appear under the Steps section of the model tree

19. Select the Load module, this will be used to apply the boundary conditions and loading to the part.



20. Click Create Boundary Condition



21. To create a fixed boundary condition (Encastre) on the rear face of the bracket, keep the default options in the Create Boundary Conditions window. Click Continue...

Name: BC-1		
Procedure: Static, Genera Category	I Types for Selected St	tep
 Mechanical Electrical/Magnetic Other 	Symmetry/Antisymr Displacement/Rotati Velocity/Angular vel Connector displacer Connector velocity	netry/Encastre ion ocity nent
Continue		Cancel



22. Rotate the bracket in the viewport and select the rear face, click Done

23. Select Encastre (this constrains all six degrees of freedom) in the Edit Boundary Condition window and click OK



The constraints will be shown on the model



24. Click Create Load



25. In the Create Load window, chose Mechanical, Pressure, click Continue...

Name: Load-1		
Step: Step-1	~	
Procedure: Static, Genera	ł	
Category	Types for Selected Step	
Mechanical	Concentrated force	^
() Thermal	Moment	
() Acoustic	Pressure	
O Fluid	Shell edge load	
O Electrical/Magnetic	Pipe pressure	
O Mass diffusion	Body force	
() Other	Line load	
	Gravity	
	Bolt load	v

26. Rotate the view and select the top face of the protrusion, as shown, click Done



27. In the Edit Load window, specify the Magnitude as 100 (this value is MPa as the geometry is in mm and the Properties in MPa) and click OK

ype: Pre tep: Ste egion: Sur	ssure p-1 (Static, General) f-2	
)istribution:	Uniform 🖌	f(x)
Magnitude:	100	
Amplitude:	(Ramp)	A

The details of the load will be added (as arrows) to the part in the viewport



28. Select the Job module





29. Click Create Job and click Continue... in the Create Job window

30. Click OK to select the default options in the Edit Job window

Edit Job Name: Job-1 Model: Mode	I-1	c/Standard			×
Description:	ici. Abaqu	s/ standard			
Submission	General	Memory	Parallelization	Precision	
C Recover	Explicit)	_	Hos	t name:	
 Backgrou Submit Tim Immedia 	e tely		Typ	e	
O Wart:	hrs. m	in. ġ			
	ОК			Cancel	

Job-1 will appear under the Jobs section of the Analysis Tree

	- 🔚 Predefined Fields	
	Remeshing Rules	
	Optimization Tasks	
	🔓 Sketches	
	Annotations	
	🗄 🔹 Analysis	
6	📮 🛃 Jobs (1)	
٢	Job-1	
	Reg Adaptivity Processes	
	Co-executions	
	Optimization Processes	4
	< >>	
	The model database "C:\Users\ The job "Job-1" has been crea The job "Job-1" has been crea	epz ted ted
	⊕ Type here to search	

31. Open the Job Manager



32. In the Job Manager window, select Job-1 and click Submit

Name	Model	Туре	Status	Write Input
Job-1	Model-1	Full Analysis	None	Data Check
			\subset	Submit
				Continue
				Monitor
				Results
				Kill

You may receive a warning that Job files for Job-1 already exist, someone may have used the default name before, click OK to overwrite – this shows the importance of naming if you're running important analyses!

The following messages should appear in the message window:

The job input file "Job-1.inp" has been submitted for analysis.

Job Job-1: Analysis Input File Processor completed successfully.

Job Job-1: Abaqus/Standard completed successfully.

Job Job-1 completed successfully.

The Job should take around 30s to run through the Virtual Desktop

33. When the job has Completed, click Results in the Job Manager to start the Post-Processor

Name	Model	Туре	Status	Write Input
Job-1	Model-1	Full Analysis	Completed	Data Check
				Submit
				Continue
				Monitor
			\subset	Results
				Kill
Create	Edit	Rename	Delete	Dismiss

The window will change as below

III All Resources	Conjineering Desktop X	🔊 💉 🕲 … Chris Bennett
🗢 Abaqus/CAE 2020 - Model D	tabase: C/Users/epzg8/OneDrive - The University of Nottingham/\Teaching\ABAQU5\bracket.cae [Viewport: 1]	- a ×
Eile Model Viewport	jew Besult Blot Animate Report Ωptions Iools Plug-ins Help №?	_ 8 ×
1 🗋 🖬 🖶 🛔 📳		
	🔯 Visualization defaults 🗹 🗊 📲 🖗 🕅 👘 Primary 🔄 S 🔄 Mises 🔄 🖏 🗌 🖕 🖉 🖓 🖓 🖓 🤹 🦉 🖉 🖉 🖉 🕐 🕐 🖓 👘 🕅	
Model Results	Module 2 Vaualization 🗠 Model: 🖟 c./vernp/16b-1.odb 🗹	H4 41 IF H4 🗟 🔞
Session Data		*
Couput Outwhere (1) Couput Outwhere (1) Sectures (7) Sectures (7) XVota Avota Prote Prote Prote Prote Prote Tempory Coupty Outwork (1) Tempory Tempor	Image: Step-1 in Step-1 in Step Time = 1.000	
		js simulia
The job 'Job-1' The job input fi Job Job-1: Analy Job Job-1: Abaqu	aw been created in "Job-1 ing" has been submitted for analysis is Ingut File Processor completed successfully. "Standard completed successfully.	· · ·
P Type here to	earch O 🛱 👼 🕾 🌩	へ 🕳 🔛 (4)) 🖋 ENG 🔐 14-39 💭

34. Click Plot Contours on Deformed Shape to view the von Mises stresses on the exaggerated deformed geometry



You should see the stress contour results below, note the Deformation Scale Factor is 7.428 meaning the deformation results are exaggerated by more than 7 times.

