Project 2 of MAEG 5130

Linear Isotropic and Linear kinematic Plasticity with Abaqus CAE

Deadline: March/22/2023

This project aims to show the varying responses of the materials with linear isotropic and linear kinematic plasticity under cyclic loading. For this project, students need to obtain the stress-strain curves from the models with linear isotropic and linear kinematic hardening in Abaqus CAE (student version), respectively. In particular, one cubic element is subject to a tension and compression cyclic loading along one direction expressed as

$$\varepsilon(t)=0.006\sin(0.25t)$$

where ε represents the uniaxial strain applied, and $t \leqslant 30$. Speaking of Abaqus implementation, the actual displacement might be adjusted based on the size of the constructed element to get the desired stress-strain response, and periodic amplitude is a highly-recommended choice.

The material properties (mm-ton-sec unit system) used in the simulation are given below

Density = 7.85×10^{-9}

Young's modulus = 200000

Poisson's ratio = 0.27

For yield behaviour in linear isotropic plasticity and linear kinematic plasticity, please use the

following data:

Yield stress	Plastic strain
500	0
700	0.01

It suggests that simulation time is set to 30 and Abaqus step is Dynamic, Explicit (another step analysis is acceptable as long as it works). Having finished the simulation, students should submit a PDF report, including the loading history and the true stress-strain response of the models with these two hardening laws. The conclusions should also be stated in the report after the comparison between these stress-strain curves.

Reference information that may be useful:

1. The definition of isotropic and kinematic hardening is shown in Sections 5.5.1 and 5.5.2 of the textbook "Nonlinear finite elements for continua and structures, 2nd edition". The online version may be found on the CUHK library website. Equations in these two sections are not required for this project and will be discussed in the course lectures later.

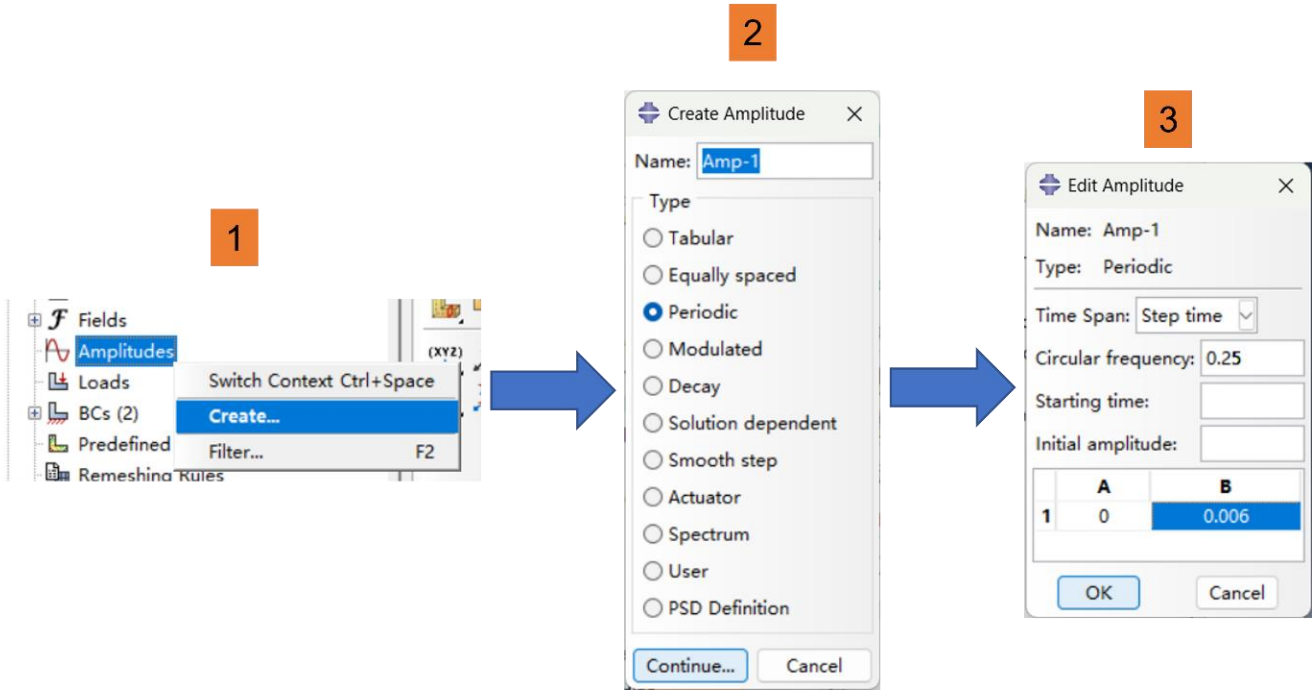
2. Abaqus modelling for isotropic and kinematic hardening in section 23.2.1 of the following link:

http://130.149.89.49:2080/v6.14/books/usb/pt05ch23s02abm17.html

3. Periodic amplitude:

https://abaqus-docs.mit.edu/2017/English/SIMACAECAERefMap/simacae-t-ampperiodic.htm

The following figure shows the setup process for the periodic amplitude.

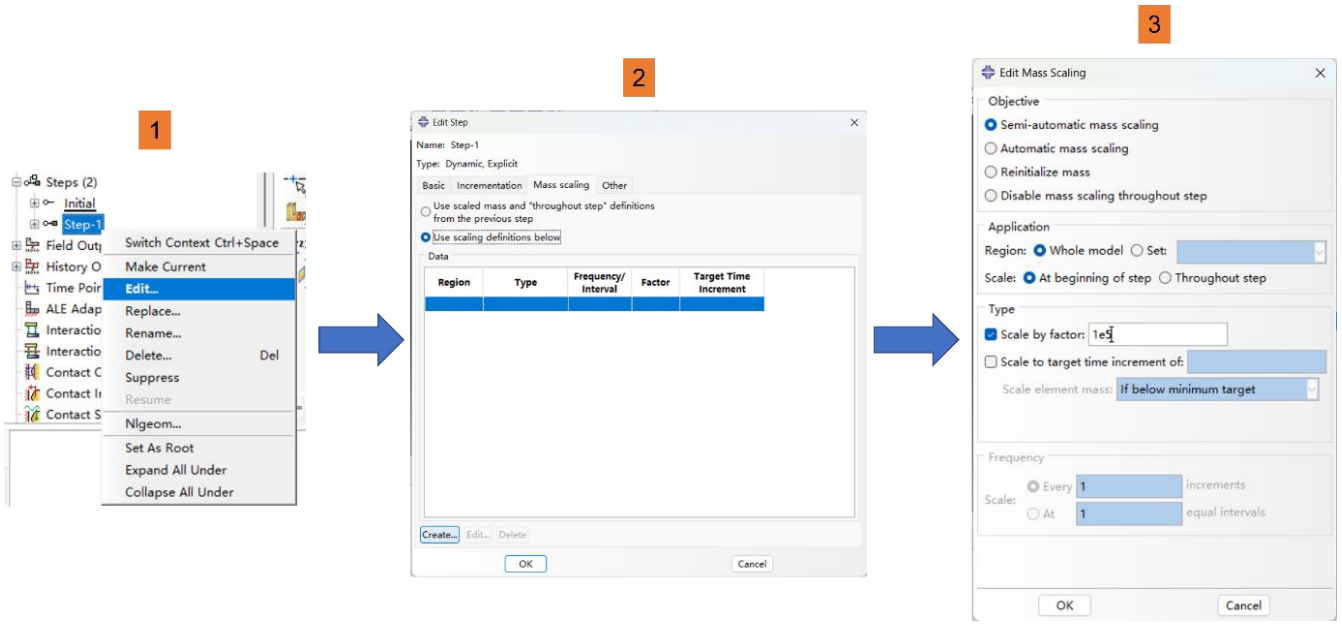


4. The other general operation will be demonstrated in the "Software Resource and Example.pdf" and "ABAQUS_CAE_Tutorial.pdf" files.

5. The mass scaling is helpful to save the calculation time:

https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usb/pt04ch11s07aus63.html

The following figure shows a proper setup:


1
Steps (2)
Initial
Step-1
Field Outp
History O
Time Poir
ALE Adap
Interactio
Interactio
Contact C
Contact I
Contact S
Switch Context Ctrl+Space
Make Current
Edit...
Replace...
Rename...
Delete... Del
Suppress
Resume
Nlgeom...
Set As Root
Expand All Under
Collapse All Under
2
Edit Step
Name: Step-1
Type: Dynamic, Explicit
Basic
Incrementation
Mass scaling
Other
Use scaled mass and "throughout step" definitions from the previous step
Use scaling definitions below
Data
Region
Type
Frequency/ Interval
Factor
Target Time Increment
Create...
Edit...
Delete
OK
Cancel
3
Edit Mass Scaling
Objective
Semi-automatic mass scaling
Automatic mass scaling
Reinitialize mass
Disable mass scaling throughout step
Application
Region: Whole model Set:
Scale: At beginning of step Throughout step
Type
Scale by factor: 1e5
Scale to target time increment of:
Scale element mass: If below minimum target
Frequency
Scale: Every 1 increments
At 1 equal intervals
OK
Cancel