Blog

BLOG

Speeding up quasi static analysis in Abaqus | Increasing Abaqus load rate

Posted by Writers Of CAE Assistant Group

03
JUL

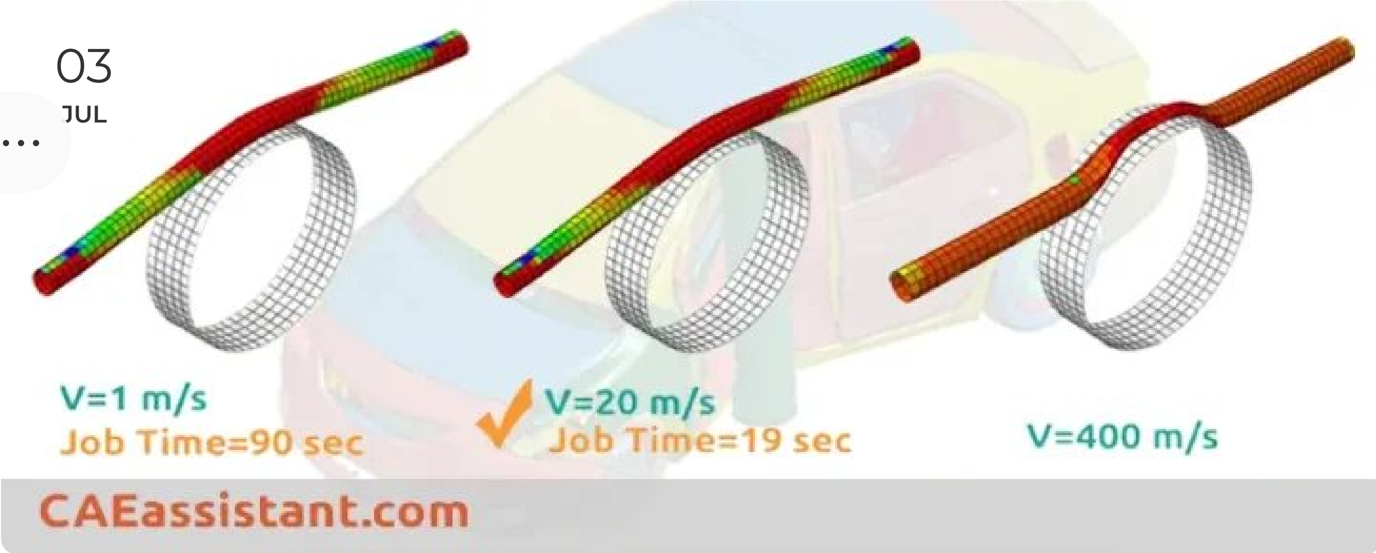


Table of Contents

1. Increasing load rate in an Abaqus quasi static analysis
2. Quasi static analysis in Abaqus/Explicit Problems
2.1. 1. Increasing Abaqus Load Rates
2.2. 2. Mass scaling
3. Finding out Abaqus load rate is appropriate or not
3.1. 1) Running several simulations with different load rates
3.2. 2) Using natural frequency to check the Abaqus load rate
4. Example (Door Beam Intrusion Test)
4.1. Free Abaqus Course
5. Limitations
6. Using Smooth Step amplitude curve
7. Quiz Time!
8. Practice Time!

Increasing load rate in an Abaqus quasi static analysis

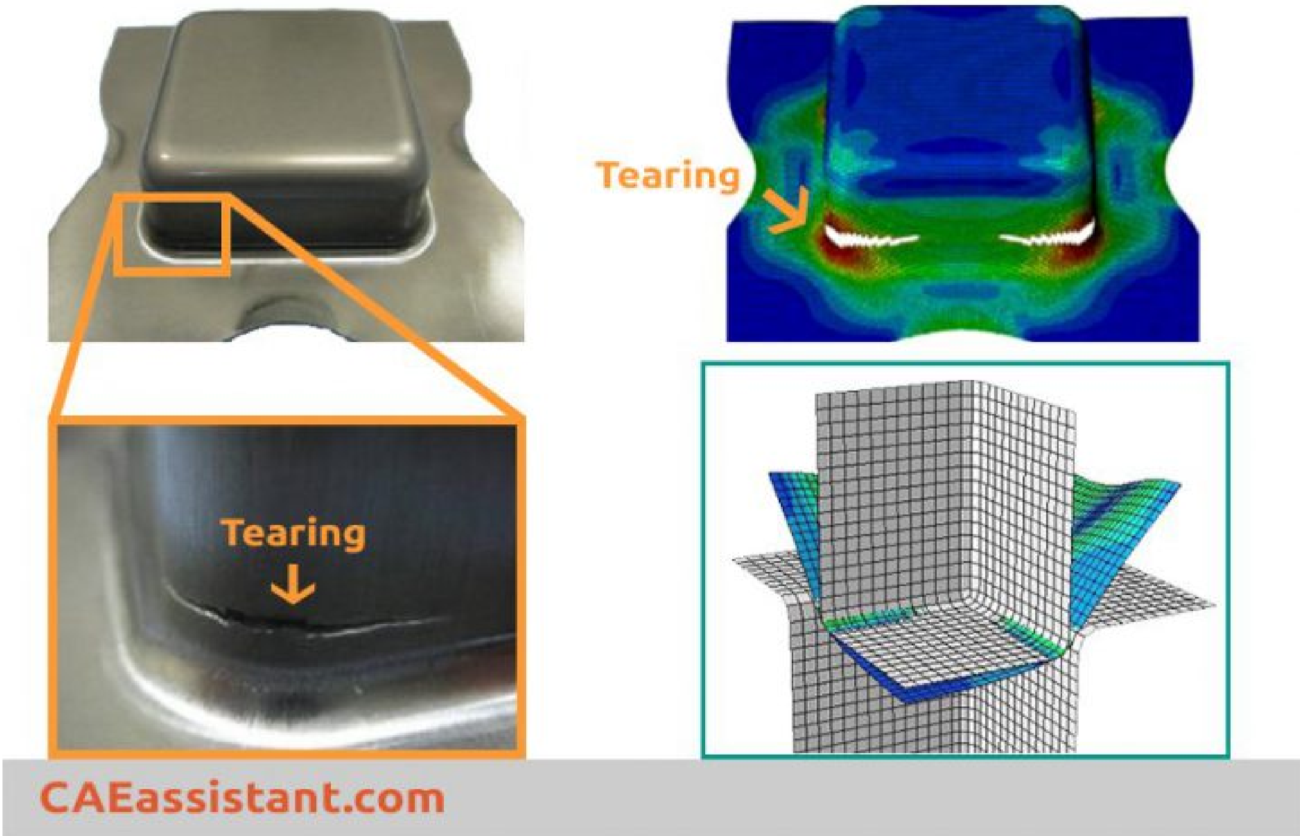
A lot of Abaqus users face a common problem: their analyses, especially the **quasi static analysis**, take a really long time to finish. This can be quite annoying. Fortunately, there are two methods to speed up these Abaqus quasi static analysis: one is using "Abaqus mass scaling," and the other is increasing the "**Abaqus load rate**." In this post, we'll concentrate on the latter method, which involves making your analyses faster by adjusting the load rate. We'll also guide you on how to determine which load rate is suitable for your quasi static analysis in Abaqus.

Quasi static problems are one of those that usually would be solved with Abaqus/Standard but may have difficulty converging because of contact or material complexities, resulting in a large number of iterations. Challenging nonlinear quasi-static problems often involve:

* Very complex contact conditions, which Abaqus/Standard may fail to converge due to contact issues.

** Very large deformations that can lead to severe mesh distortion.

For example, typically, in metal forming analysis, we face such difficulties:



Example: Simulation of tearing in a deep drawing process
It is really hard to model such a problem with Abaqus/Standard.

Stay with us, The **CAE Assistant group**, which always trying to be your complete assistant in CAE challenges. In addition, you can see this Q&A to have a sense of how we decide

whether an analysis is quasi-static (static) or not:
How can I know if my simulation is quasi-static or not?

Quasi static analysis in Abaqus/Explicit Problems

Abaqus/Explicit is more efficient for modeling highly nonlinear static (quasi-static analysis) problems. This is especially true for three-dimensional problems involving contact and very large deformations like metal forming.

Application of Abaqus/Explicit to model quasi-static events requires special consideration. It is computationally impractical to model the process in its natural time period. Literally, millions of time increments would be required. Therefore, we **artificially increase the speed of the process** in the simulation to obtain an economical solution.

If you would like to know more about Abaqus/Standard and Abaqus/Explicit, we've already covered the differences between them in this article titled,
Abaqus/Standard or Abaqus/Explicit?

Two approaches to obtaining economical Abaqus quasi-static analysis solutions are:

1. Increasing Abaqus Load Rates

We can artificially reduce the time scale of the process by increasing Abaqus loading rate. Increased load rates reduce the time scale of the simulation, so fewer increments are needed to complete the job.

Increasing load rates by a factor of **f**, increases the analysis speed by a factor of **f**.

2. Mass scaling

It increases the size of the stable time increment, so fewer increments are needed to complete the job. Artificially increasing the material density (mass scaling) by a factor of f^2 increases the analysis speed by a factor of f.

In this article, our focus is on **increasing Abaqus load rates**.

To reduce the number of require increments in an quasi static analysis in Abaqus/Explicit, we can speed up the simulation compared to the time of the actual process—that is, we can artificially reduce the time period of the event or, equally, increase the rate of loading. This will introduce possible errors. If the loading rate is increased too much, the increased inertia forces will change the predicted response. In an extreme case, the problem will exhibit a wave propagation response. The only way to avoid this error is to choose a load rate that is not too large.

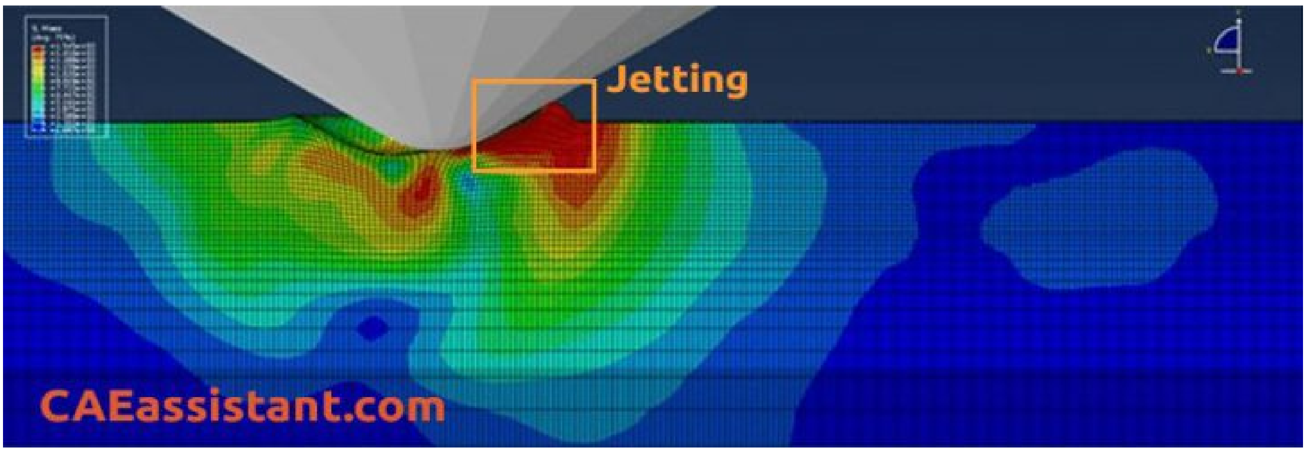
Finding out Abaqus load rate is appropriate or not

1) Running several simulations with different load rates

1. Run a series of simulations in the order from the fastest load rate to the slowest. As you know, the analysis time is greater for slower Abaqus load rates.
2. Examine the results (deformed shapes, stresses, strains and energies) to get an understanding of the effects of varying the model when changing the load rate Abaqus:

»Excessive tool speeds in sheet metal forming tend to promote unrealistic localized stretching.

»Excessive tool speeds in bulk forming simulations cause jetting (hydrodynamic-type response).



»Excessive loading rates can cause highly localized deformation near the applied load.

»Excessive loading rates in an Abaqus quasi-static collapse analysis can result in a steep initial slope of the load versus displacement curve due to increased (non-structural) resistance to initial deformation. Sometimes, localized buckling may occur near the applied load.

2) Using natural frequency to check the Abaqus load rate

The dominant response of a quasi-static analysis will be the first structural mode. Therefore, we use the frequency of this mode to estimate the proper load rate Abaqus:

1. Estimate the first natural frequency (**f**) of the model. In simple models, we may find this frequency by available analytical relations. For models that are more complex, first, run a **Frequency** analysis in Abaqus.
2. Calculate the corresponding time period (**T**) using the first natural frequency of the model:

$$T=1/f$$

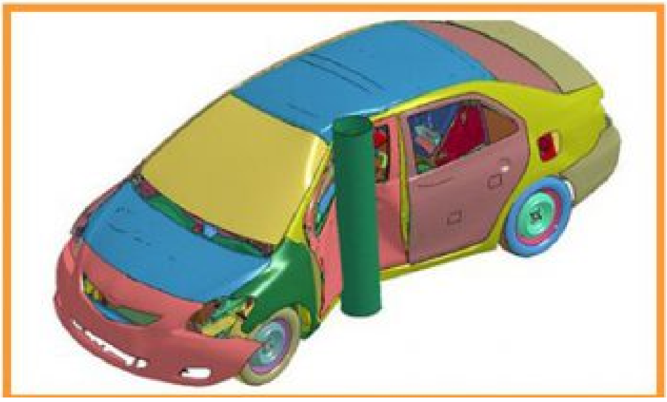
3. Run the **Explicit** analysis (step time=**T**) and estimate the global deflection (**D**) in the impact direction of the model during this time (**T**).
4. Calculate the impact velocity (**V**):

$$V=D/T$$

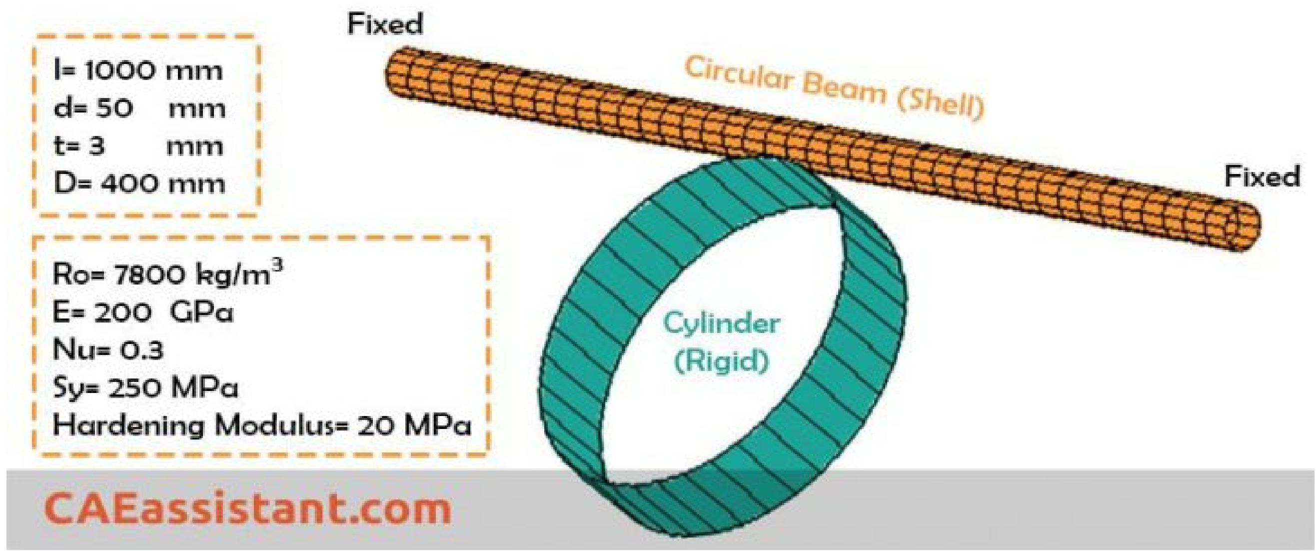
5. A general recommendation is to limit the impact velocity to less than **1%** of the wave speed of the material. Typical wave speed in metals is **5000 m/sec**.

Example (Door Beam Intrusion Test)

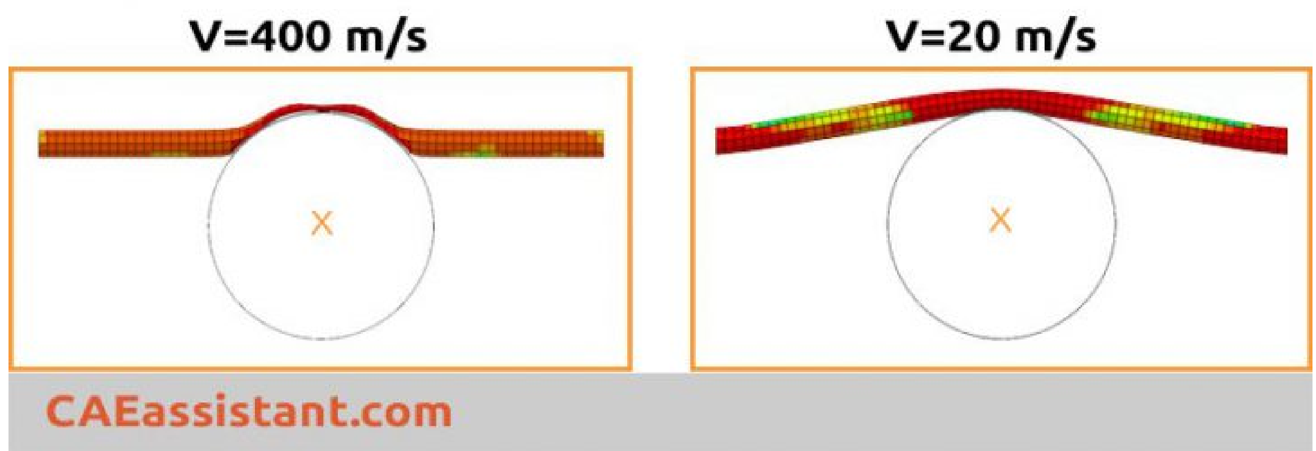
To illustrate the problem of determining the proper loading rate, consider the deformation of a side intrusion beam in a car door. This test falls into the category of Abaqus quasi-static analysis



We model the test as the circular beam (length of ***l***, diameter of ***d*** and thickness of ***t***) is fixed at each end, and a rigid cylinder (diameter of ***D***) deforms the beam.



Here, we check the velocity of 20 m/s and 400 m/s for a cylinder to see which one can be applicable to our problem.



- The frequency of the first mode is approximately 250 Hz: **f=**250
- This rate corresponds to a period of 4 milliseconds: **T=**1/250=0.004 s
- Using a velocity of 20 m/sec, the analysis shows cylinder will be pushed into the beam **0.1 m** in **4 milliseconds**: **D=0.1 m**
- The impact velocity is:

$$V=D/T= 0.08/0.004= 20 \text{ m/s}$$

- Recalling the wave speed in metals is about 5000 m/sec, so the impact velocity 25 m/sec is about 0.5% of the wave speed (less than 1%).

If we check the velocity of 400 m/s it will result in about 4% of wave speed (unacceptable).

Limitations

i. As the speed of the process is increased, a state of static equilibrium evolves into a state of dynamic equilibrium and inertia forces become more dominant. We should try to model the process in the shortest time period (largest load rate Abaqus) in which inertia forces are still insignificant.

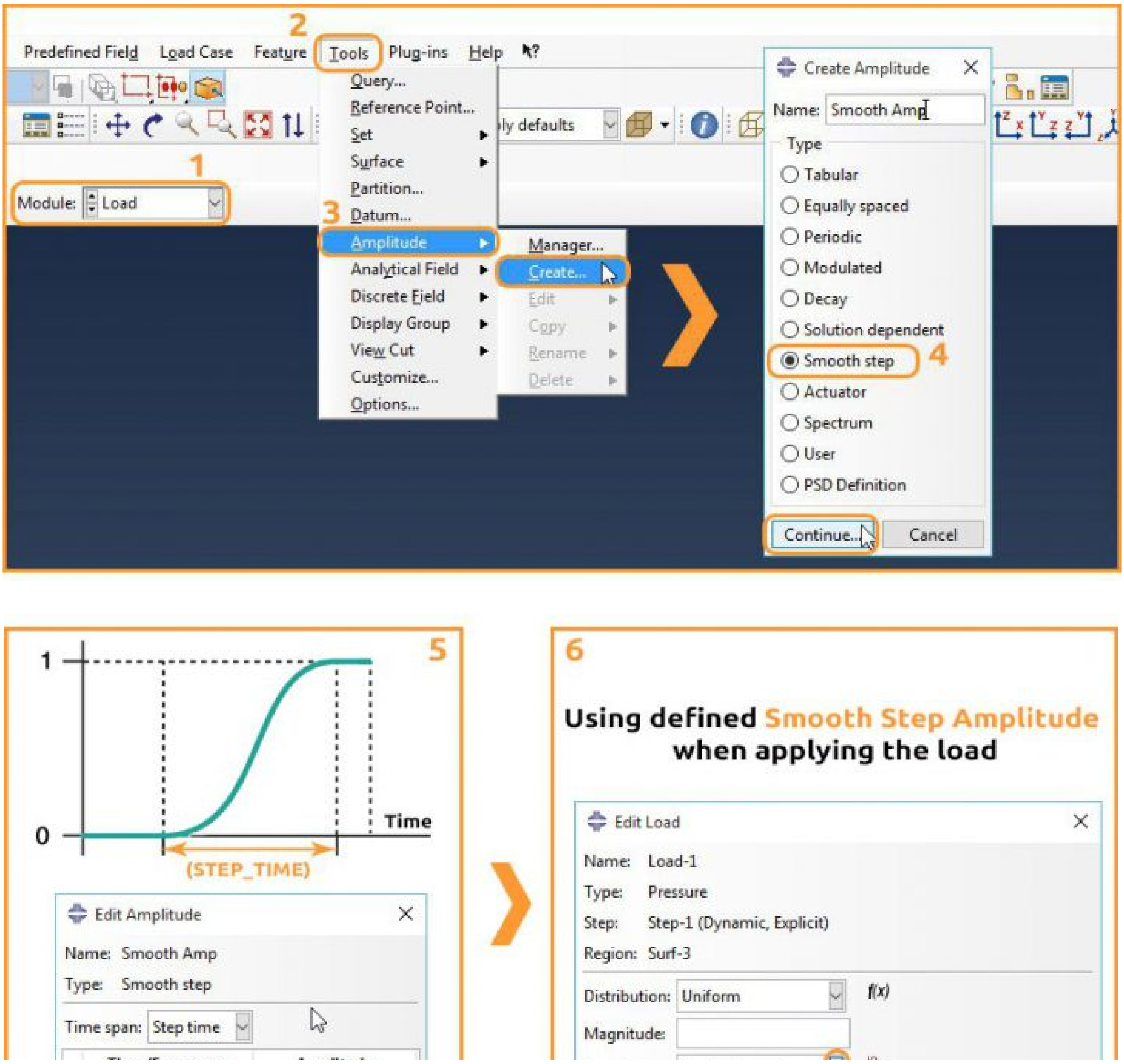
ii. Some aspects of the problem other than inertia forces—for example, material behavior—may also be rate-dependent. In this case, the actual time period of the event being modeled cannot be changed. The mass scaling approach gets attractive in such problems.

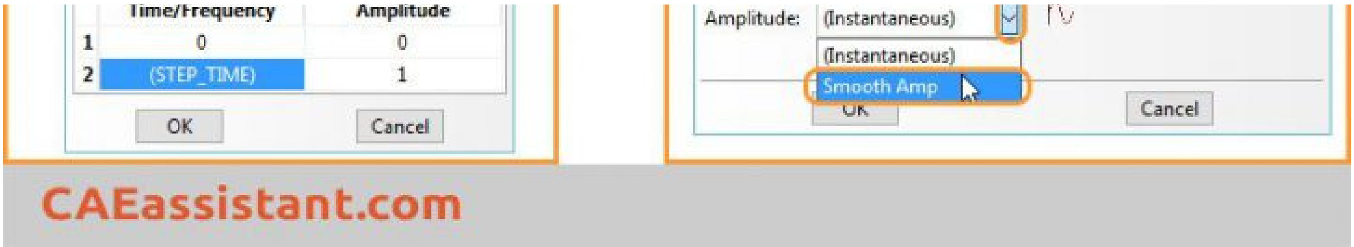
Using Smooth Step amplitude curve

We could obtain a more accurate Abaqus quasi-static solution by applying loads gradually.

By default, Abaqus/Explicit loads are applied immediately and remain constant throughout the step. Instantaneous loading may induce the propagation of a stress wave through the model, producing undesired results. For instance, constant velocity boundary conditions result in a sudden impact load onto a deformable body.

We can ramp up (or down) the loading gradually from (to) zero by defining a smooth step amplitude in an Abaqus quasi-static solution:





It would be useful to see **Abaqus Documentation** to understand how it would be hard to start an Abaqus simulation without any **Abaqus tutorial**. If you want to get complete information about load rate and mass scaling methods, watch the below demo video of the **Abaqus course for beginners** package:

In this post, we completely covered speeding up quasi-static analysis in Abaqus by increasing the load rate. You can now choose the appropriate Abaqus load rate and examine it using the methods we discussed, such as 'Running several simulations' and 'Using natural frequency.' Additionally, by studying the example provided, you can gain a deeper understanding of Abaqus quasi static problems.

It's your turn to dive into the article and join our quiz! Don't forget to check out the 'Practice Time' section; you're supposed to conduct the intrusion test. Please share your experiences, questions, or any comments you might have with us, The CAE Assistant, your assistant in CAE challenges."

Quiz Time!

1. Abaqus/standard is not appropriate for metal forming simulation at all. (True/False)

2. Stretching is one of the bulk metal forming processes in which Abaqus/explicit is more efficient to simulate. (True/False)

3. We artificially increase the time scale of the process by increasing the Abaqus loading rate. (True/False)

4. Jetting is a hydrodynamic-type response when tool speed in bulk forming simulations is excessive happens. (True/False)

5. Mass scaling by a factor of **f** decreases the computational cost by a factor of $\sqrt{f}$. (True/False)

6. Increasing load rate by a factor of **f** decreases the computational cost by a factor of **f.** (True/False)

7. By default, Abaqus/Explicit apply loads gradually throughout the step. (True/False)

Practice Time!

Try to model intrusion tests, one of the Abaqus quasi static problems. you can model based on the information provided about geometry, material, etc. First, conduct a **Frequency** analysis to find the basic frequency (first mode) of the beam (with given BC). Then, run three **Dynamic, Explicit** analyses (as shown in the poster of the article) and compare results.

ABAQUS · Abaqus_Explicit · Increased_Load_Rate · Mass_Scaling · Quasi-Static

2 THOUGHTS ON “SPEEDING UP QUASI STATIC ANALYSIS IN ABAQUS | INCREASING ABAQUS LOAD RATE”

Margy Jacobo says:

December 8, 2020 at 2:37 am

finest post

LOG IN TO REPLY

Facebook says:

February 10, 2021 at 4:52 am

Some genuinely prize articles on this site, saved to favorites . Kerrill Bondie Bearce

LOG IN TO REPLY

LEAVE A REPLY

You must be logged in to post a comment.

Representative)