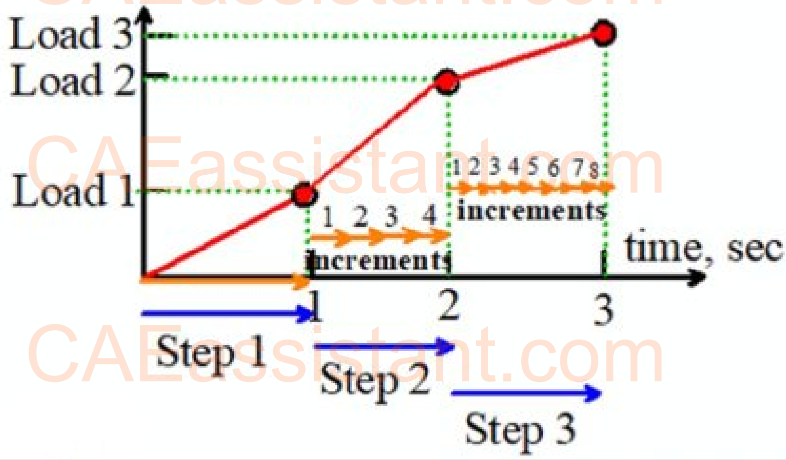
CAE Assistant
Your Assistant in CAE Challenges

BLOG, UNCATEGORIZED

Step, Increment, Iteration and Attempt concepts in Abaqus

Posted by Experts Of CAE Assistant Group

03
FEB

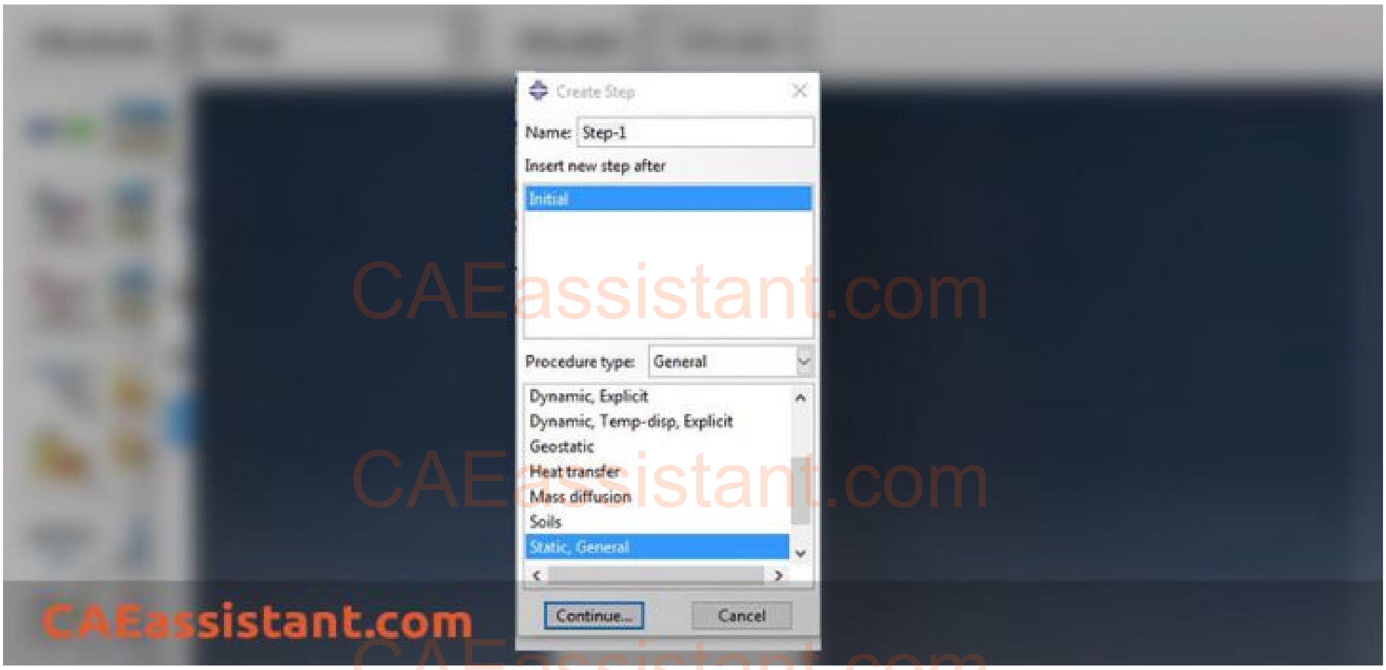


The terms step, increment, and iteration and two subsidiary terms—frame and attempt can be conceptually confusing to an Abaqus beginner. It is important that you clearly understand the difference between an analysis step, a load increment, and an iteration. Learn quickly about the Abaqus step and increment iteration in this post.

1. Step (Abaqus step)

A basic concept in Abaqus is the division of the problem history into steps. Abaqus step is any convenient phase of the loading history—a thermal transient, a static, a dynamic transient, etc. In its simplest form, a step can be just a static analysis of a load change from one magnitude to another.

For each step, the user chooses an analysis procedure. This choice defines the type of analysis to perform during the step: static stress analysis, dynamic stress analysis, eigenvalue buckling, transient heat transfer analysis, etc.:

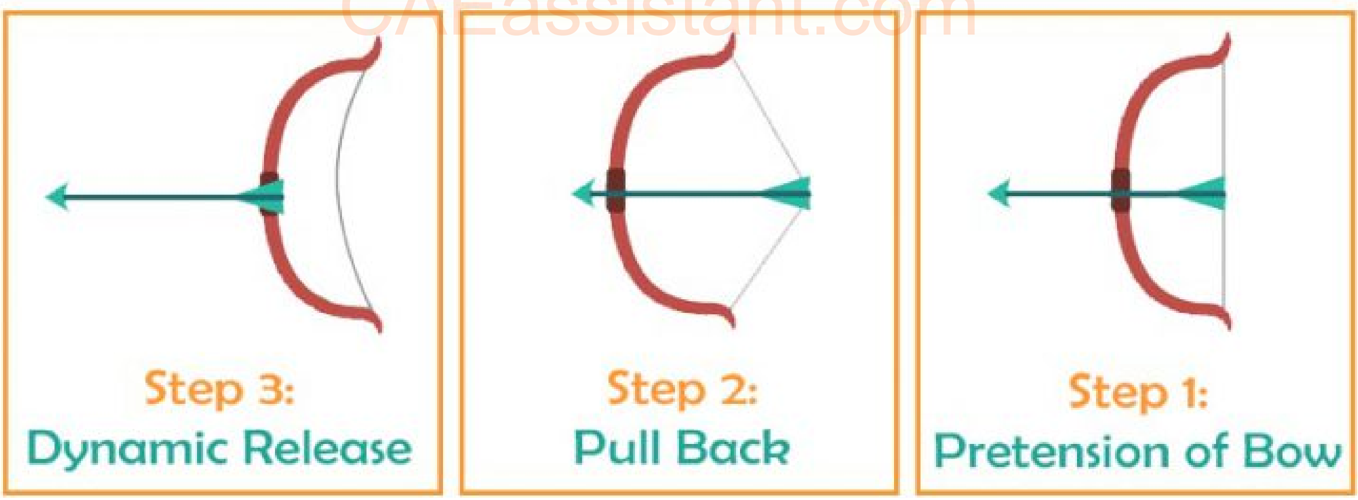


The rest of the step definition consists of load, boundary (in the **Load** module) and output request (in the **Step** module) specifications.

Why do we use several steps?

Any real problem will usually consist of more than one step during a simulation. This might consist of a loading step or a step where we apply boundary conditions.

Sometimes, we use the steps to define different phases of loading. An example would be the simulation of a simple bow and arrow:



- Step 1: stringing the bow to pretension the bowstring.
- Step 2: pulling back the string with an arrow, thus storing more strain energy in the system.
- Step 3: releasing the bowstring. The strain energy stored in the system by pulling back the bowstring in Step 2 imparts kinetic energy to the arrow and causes it to leave the bow.

Sometimes when complex load history is present, we need to define multiple steps to ease the process of solving for Abaqus. As an example, consider a schematic representation of the steps in the manufacture of a stainless steel sink:

The sink is formed from sheet steel using a punch, a die, and a blank holder. This forming simulation will consist of a number of steps:

- Step 1: the application of blank holder pressure
- Step 2: simulating the punching operation
- Step 3: the removal of the tooling, allowing the sink to spring back into its final shape

Have you started simulation in Abaqus recently? I recommend you download these free tutorials here! FREE ABAQUS TUTORIAL

2. Increment

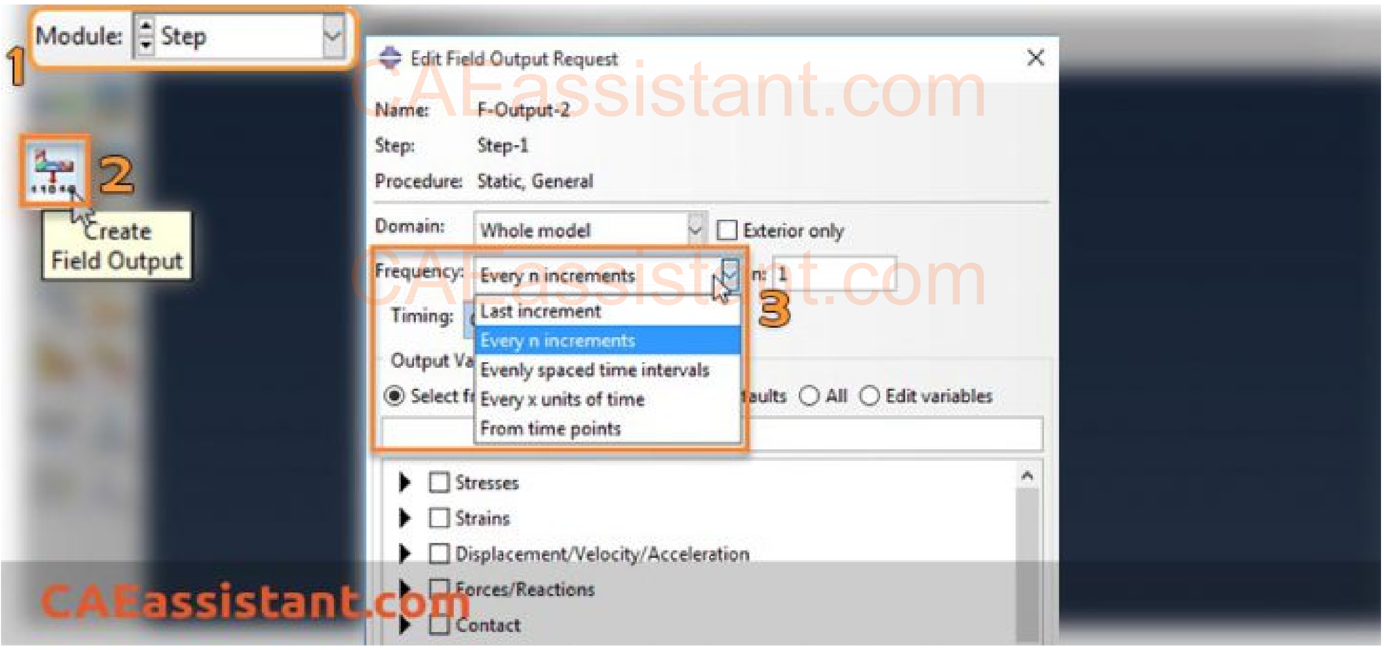
An increment is part of a step (Abaqus step). In nonlinear analyses, the total load applied in a step is broken into smaller increments so that the nonlinear solution path can be followed. Therefore, “Increment” is the part of the total load that is applied gradually.

In Abaqus/Standard you suggest the size of the first increment, and Abaqus/Standard automatically chooses the size of the subsequent increments. In Abaqus/Explicit the default time incrementation is fully automatic and does not require user intervention (however you can change it). Because the explicit method is conditionally stable, there is a stability limit for the time increment. It mainly depends on the length of the smallest element.

When you know what an increment is, you can assume that a *solution step* (we use the term often when talking about numerical algorithms) is almost the same as that of an increment in Abaqus. To clarify with an example, if we apply a load in 1000 solution steps for a time period of 5 s, then the time step size or increment is 0.005 s.

3. Frame

At the end of each increment, the structure is in (approximate) equilibrium and results are available for writing to the output database or results in files. The increments at which you select results to be written to the output database file are called “frames”. The max no. of frames in an analysis can be no. of increments but due to saving storage memory, you can request fewer frames:



4. Iteration

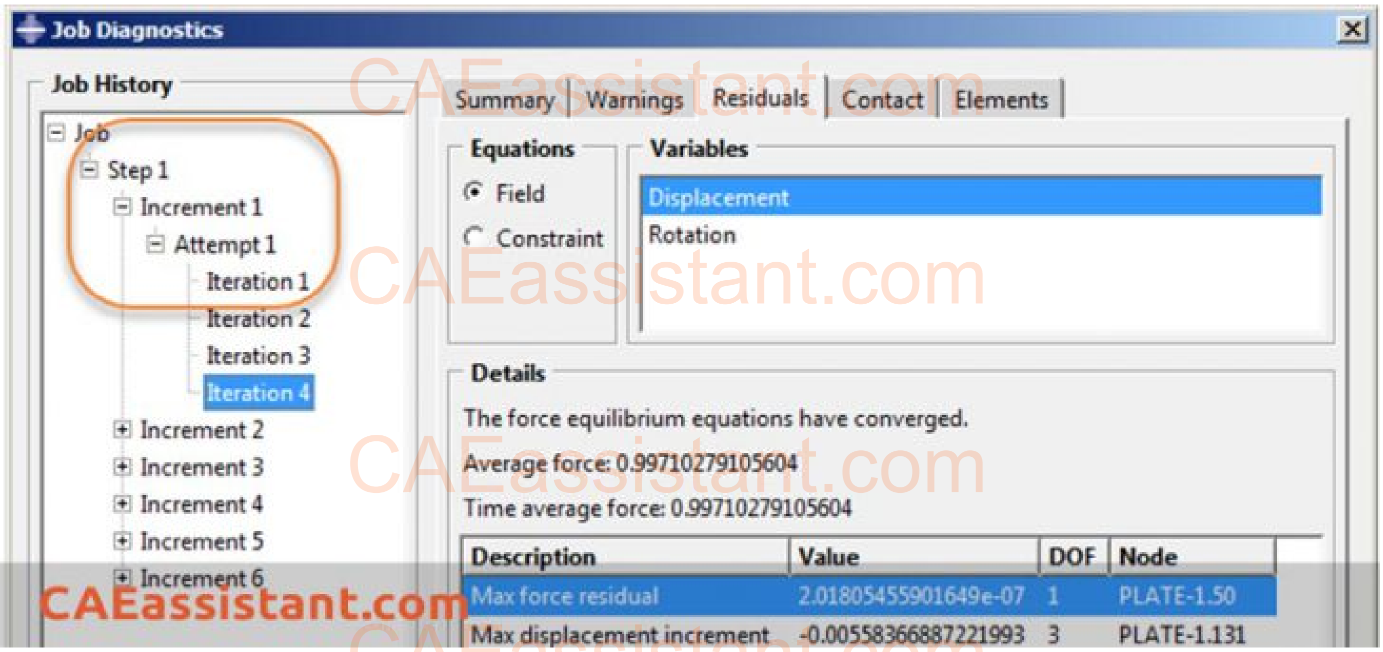
As you would have observed so far, we keep moving deeper after every definition. I mean, “Step” is the general and logical categorization of the problem at hand. “Increment” in one step deeper and is a subset of a “Step”. All of these terms are applicable both in explicit and implicit methods.

Then comes “Iteration” where we go further into a “Step”. This term is meaningful just when we are using an implicit solver (Abaqus/standard).

As you know, in the case of an implicit solver, we seek the equilibrium after at every increment by checking the difference between externally applied force and internal reaction force. This difference is called *residual*.

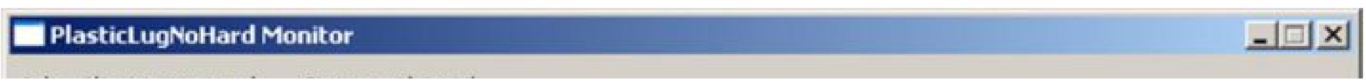
Abaqus uses a tolerance value (a value close to zero) defined in its solver settings. Till the difference between the forces come under this tolerance value, the solver carries several “iterations”. Therefore, Abaqus carry out iterations within an increment in order to attain equilibrium.

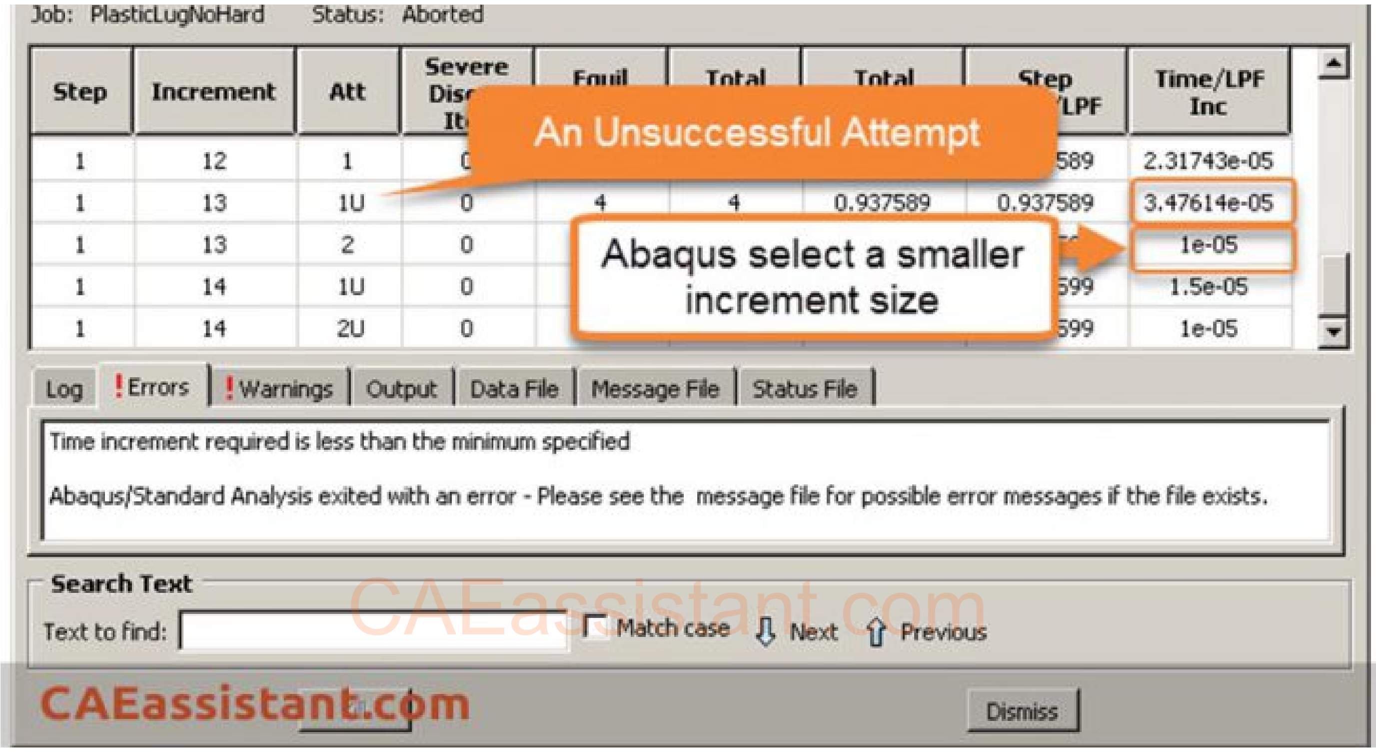
There are no iterations in an ABAQUS/Explicit analysis.



5. Attempts

As explained before, Abaqus/Standard choose the size of the increments automatically (except for the first increment). Every increment size selection in Abaqus is called an “attempt”. If Abaqus is unable to find a solution with selected increment size after iterating several times, it makes a cutback in the increment size and begins a new attempt. There are no attempts in an Abaqus/Explicit analysis.





It would be useful to see **Abaqus Documentation** to understand how it would be hard to start an Abaqus simulation without any **Abaqus tutorial**.

Attempts Increment Iteration

NEWER OLDER

2 THOUGHTS ON "STEP, INCREMENT, ITERATION AND ATTEMPT CONCEPTS IN ABAQUS"

Hossemeddinesaidi says:

July 29, 2022 at 10:53 pm

thank's very helpfull

LOG IN TO REPLY

Young_man70 says:

August 28, 2022 at 3:11 pm

thanks, very useful article.

LOG IN TO REPLY

LEAVE A REPLY

You must be logged in to post a comment.

ADDRESSES

Carrer de Jaume II ,46015,Valencia ,Spain

REON INTERNATIONAL GROUP LTD, 21 Hill Street, Unit 5, Haverfordwest, Dyfed, United Kingdom, SA61 1QQ (Sales Representative)

Enviroflex GmbH, Sterngasse 3/2/6 1010, Vienna, Austria (Sales Representative)

CAE Assistant
Your Assistant in CAE Challenges

With our assistance, consider your simulation project is done; we brought together a set of services and tutorial material to meet all your needs in CAE.

LINKS

- Contact Us
- Privacy Policy
- Terms & Conditions
- Cookie Policy
- Join Us
- FAQs
- Site Map

VISA MasterCard PayPal AMERICAN EXPRESS VISA Electron Maestro