

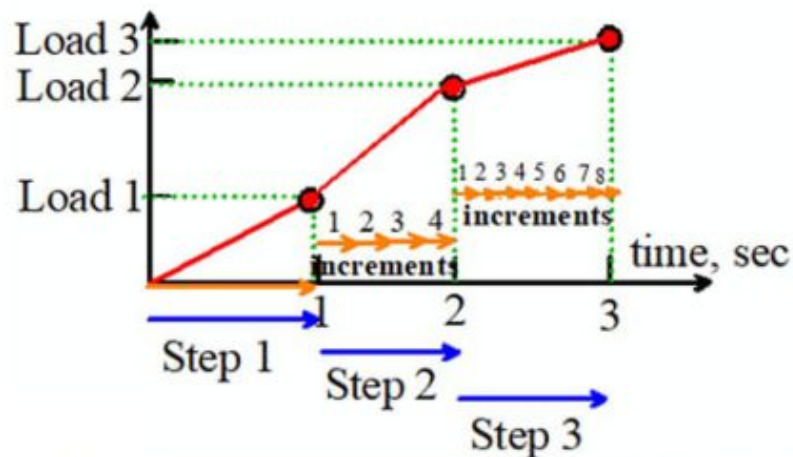
Blog

BLOG, UNCATEGORIZED

Abaqus Step time | Increment meaning | Iteration vs increment

Posted by Experts Of CAE Assistant Group

... 03
FEB



CAEassistant.com

Table of Contents



1. All you need to know about : Step in Abaqus, Step increments, Iteration, Attempts
2. 1. Abaqus step time | Step in Abaqus
 - 2.1.1.1. Why do we use several Abaqus step time?
3. 2. Abaqus increments | Increment meaning
4. 3. Frame
5. 4. Iteration
 - 5.1. 4.1. Iteration vs increment
6. 5. Attempts
7. 6. How to learn the basic concepts of Abaqus?

All you need to know about : Step in Abaqus, Step increments, Iteration, Attempts

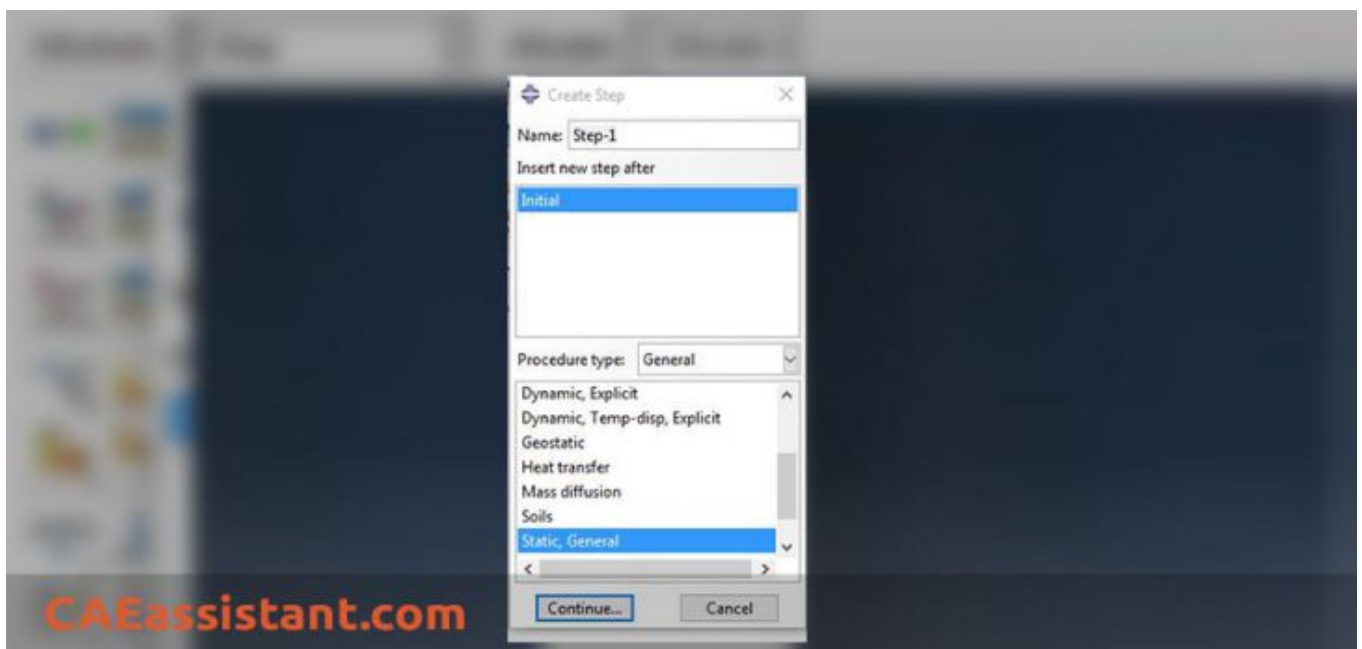
Are you new to Abaqus? You've probably heard the terms "Abaqus step time," "Abaqus increments," "step increment," and the subsidiary terms "frame" and "attempt." These terms can be confusing, but don't worry! In this article, we'll explain what each term means and how they relate to each other. We'll also compare iterations and increments, so you can choose the right one for your analysis. **So come with me** and learn about the basics of step in Abaqus, increments, iterations, and attempts. By the end of this article, you'll be well on your way to becoming an Abaqus pro!



1. Abaqus step time | Step in Abaqus

A basic concept in Abaqus is the division of the problem history into steps. Abaqus step time 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 in Abaqus, 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.:



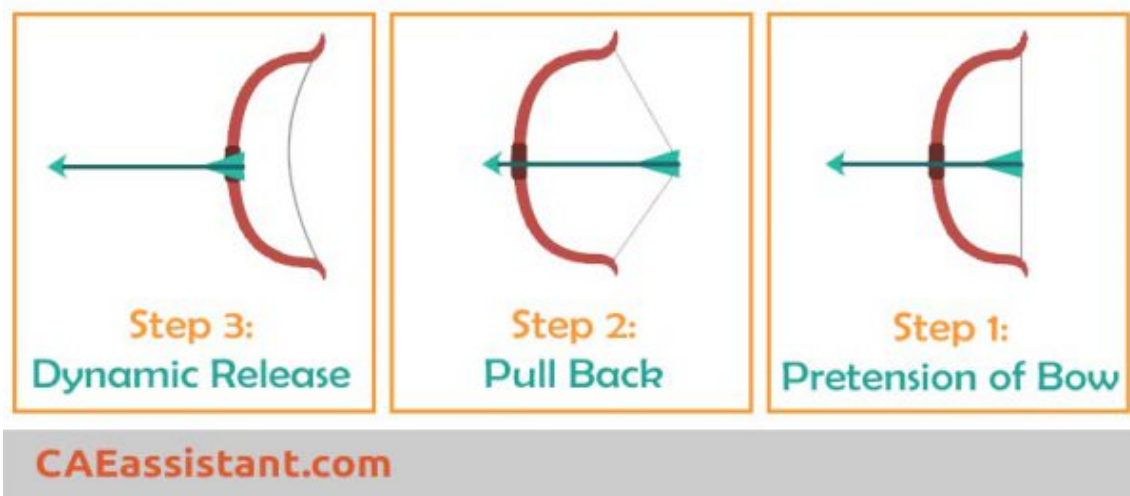
The rest of the step definition consists of load, boundary (in the **Load** module) and output request (in the **Step** module) specifications.

Curious about the differences between Abaqus Standard and Abaqus Explicit? Just simply click on [ABAQUS Standard](#) or [ABAQUS explicit](#) which have been expertly crafted by CAE assistants. Enjoy your exploration!

1.1. Why do we use several Abaqus step time?

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 a step in Abaqus 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:

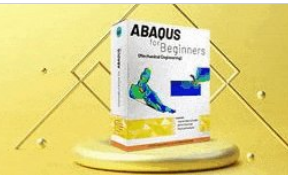


The sink is formed from sheet steel using a punch, a die, and a blank holder. This forming simulation will consist of a number of steps:

- 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

Abaqus for beginners

Mechanical Engineering



Have you started simulation in Abaqus recently? I recommend you download these CAE assistant tutorials! [FREE ABAQUS TRAINING](#).

Now that you are familiar with the Abaqus step, let's delve into the concept of Abaqus increments, which falls within the subgroup of time-related aspects in an Abaqus step time. In the following section, we will take a thorough look at Abaqus increments.

2. Abaqus increments | Increment meaning

An increment is part of a step in Abaqus (Abaqus step time). 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.

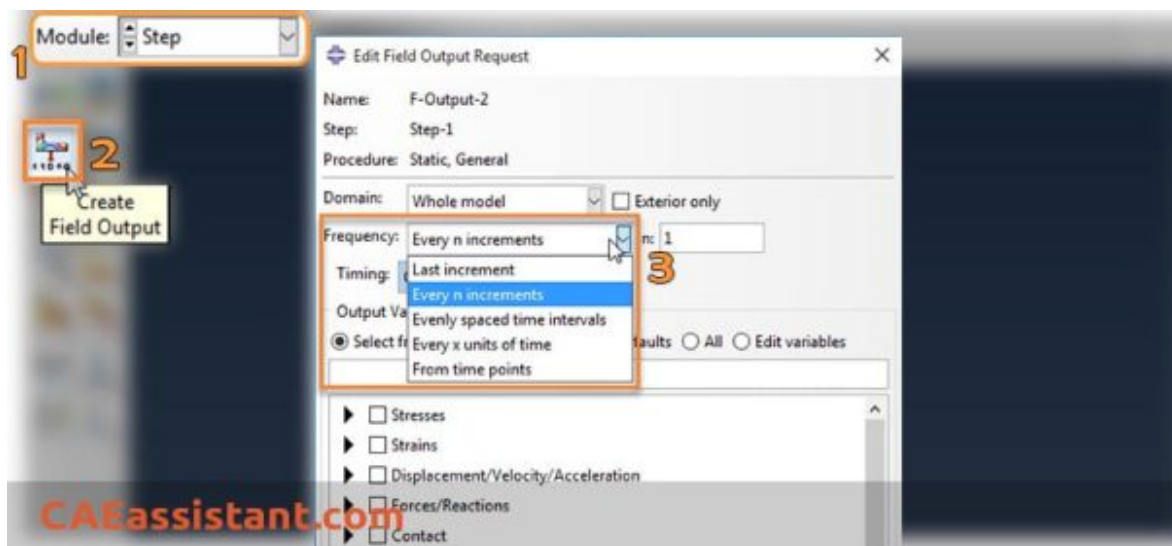
When you know what Abaqus increments are, you can assume that a *solution step* (we use the term often when talking about numerical algorithms) is almost the same as that of an increment in Abaqus. To clarify with an example, if we apply a load in 1000 solution steps for a time period of 5 s, then the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

This was all you need to know about meaning of increment (increment meaning). But now, Imagine you're conducting an analysis. How would you prefer to generate a report of the results? For enhanced accuracy, at which specific Abaqus increment do you intend to extract the results? How can you communicate this requirement to Abaqus? Let's explore these aspects in the next section.

Read More: [Units in Abaqus | Abaqus units](#)

3. Frame

At the end of each Abaqus increment, the structure is in (approximate) equilibrium, and results are available for writing to the output database or results in files. The Abaqus 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:



Now that you have learned the Abaqus step, which can encompass multiple increments, and you are acquainted with the frame concept that allows us to receive the results of our desired Abaqus increments, the question arises: How does ABAQUS ensure equilibrium is reached during the solution process? Let's proceed to the next section to address this inquiry.

4. Iteration

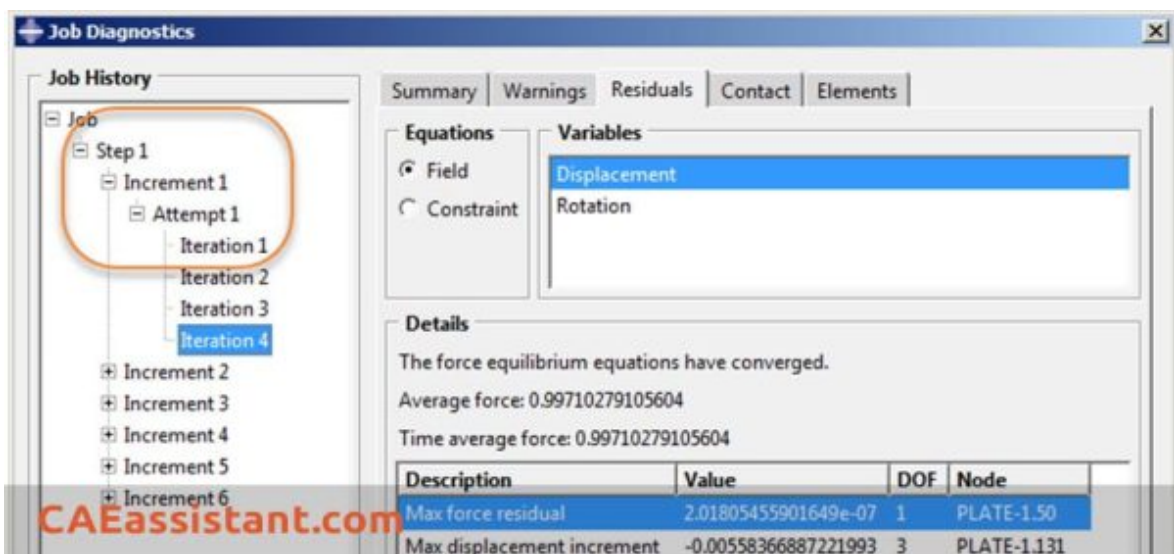
As you would have observed so far, we keep moving deeper after every definition. I mean, Step in Abaqus is the general and logical categorization of the problem at hand. "Increment" in one step deeper and is a subset of a "Step". All of these terms are applicable both in explicit and implicit methods.

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.



4.1. Iteration vs increment

The Abaqus/Standard solver uses the Newton method to solve nonlinear equilibrium equations. Many problems involve time-dependent responses, so the solution process typically consists of a series of increments along with trial-and-error iterations to reach equilibrium within each increment.

Sometimes, to achieve sufficient accuracy, these increments need to be small. The choice of increment size is often related to computational efficiency. If the number of increments is high, more trial-and-error iterations are required. On the other hand, the Newton method has a limited radius of convergence, and larger increments may lead to failure to converge to the necessary solution due to the initial state being far from the equilibrium state.

As we compared iteration vs increment You might be wondering: how many times is this trial-and-error process repeated? When does Abaqus realize it needs to adjust the increment size and give it another shot? These are the kind of questions that underscore the importance of the “Attempt” concept. We’ll delve into this concept in the next section.

5. Attempts

As explained before, Abaqus/Standard chooses 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 the 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.

All the concepts that we covered are interconnected in a chain. For instance, understanding the meaning of “increment” (increment meaning), helps easier understand the learning concepts related to “attempts”. So, why not take a step back and revisit these key concepts?

Also, you’re welcome to leave your comments at the end of this article. Your insights are valuable in helping us make things even better!

Step	Increment	Att	Severe Disc It	Equl	Total	Total	Step LPF	Time/LPF Inc
1	12	1	0				589	2.31743e-05
1	13	1U	0	4	4	0.937589	0.937589	3.47614e-05
1	13	2	0					1e-05
1	14	1U	0				599	1.5e-05
1	14	2U	0				599	1e-05

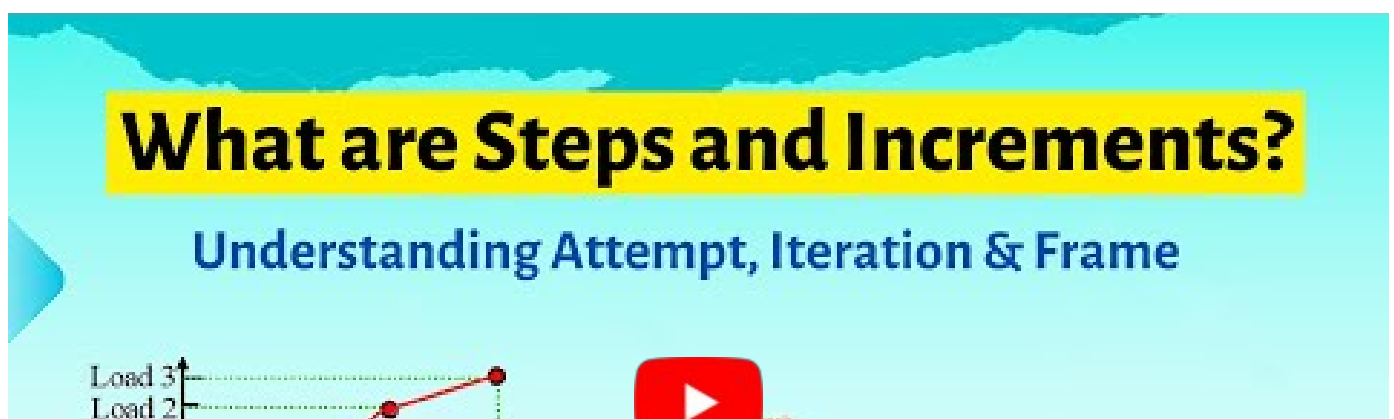
Log ! Errors ! Warnings Output Data File Message File Status File

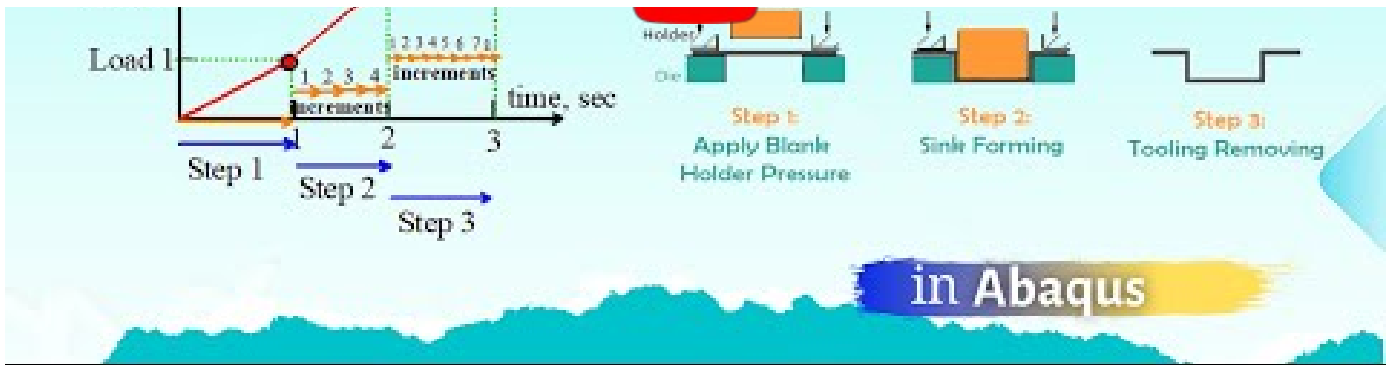
Time increment required is less than the minimum specified
Abaqus/Standard Analysis exited with an error - Please see the message file for possible error messages if the file exists.

Search Text
Text to find: Match case

CAEassistant.com

In the below video, I have summarized all these useful points about Step, Increment(step increment), and Iteration vs increment in Abaqus. you can learn easily in a few minutes:

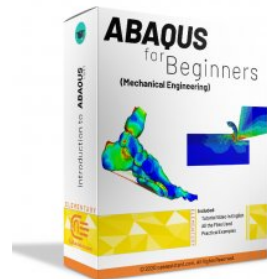
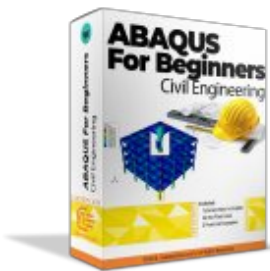




It would be useful to see [Abaqus Documentation](#) to understand how it would be hard to start an Abaqus simulation without any [Abaqus tutorial](#).

6. How to learn the basic concepts of Abaqus?

Abaqus offers multiple concepts to kickstart an analysis, and the ones covered in this article (Abaqus step, Abaqus increments, and more) are just a few examples. If you're aiming to establish a solid foundational understanding of Abaqus, you can easily access the free "[Abaqus course for beginner| FEM simulation tutorial](#)." The CAE Assistant team has assembled comprehensive resources to help you begin with Abaqus, including hands-on tutorials and even guidance on coding without encountering any bugs.



Get this post as a PDF file: [caeassistant.com-step increment-iteration-abaqus](https://caeassistant.com-step-increment-iteration-abaqus)

- ✦ 1. What is an iteration in ABAQUS?
- ✦ 2. What is a step in Abaqus?
- ✦ 3. What is an increment in Abaqus?
- ✦ 4. What is a frame in Abaqus?

+ 5. What is an attempt in Abaqus?

• Attempts

• Increment

• Iteration



< NEWER



OLDER >

3 THOUGHTS ON “ABAQUS STEP TIME | INCREMENT MEANING | ITERATION VS INCREMENT”



Japan Portal says:

February 11, 2021 at 9:22 pm

I really enjoy this template youve got going on on your web page. What is the name of the theme by the way? I was thinking of using this style for the web site I am going to build for my class room project.

[LOG IN TO REPLY](#)



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.

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,
Sternngasse 3/2/6 1010,
Vienna, Austria (Sales
Representative)



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

- [About Us](#)
- [Portfolio](#)
- [Contact Us](#)
- [Privacy Policy](#)
- [Terms & Conditions](#)
- [Cookie Policy](#)
- [Join Us](#)
- [FAQs](#)



CAE Assistant ©All Rights Reserved

