Blog

BLOG

Differences between ABAQUS Standard & ABAQUS Explicit

Posted by Experts Of CAE Assistant Group

21
FEB

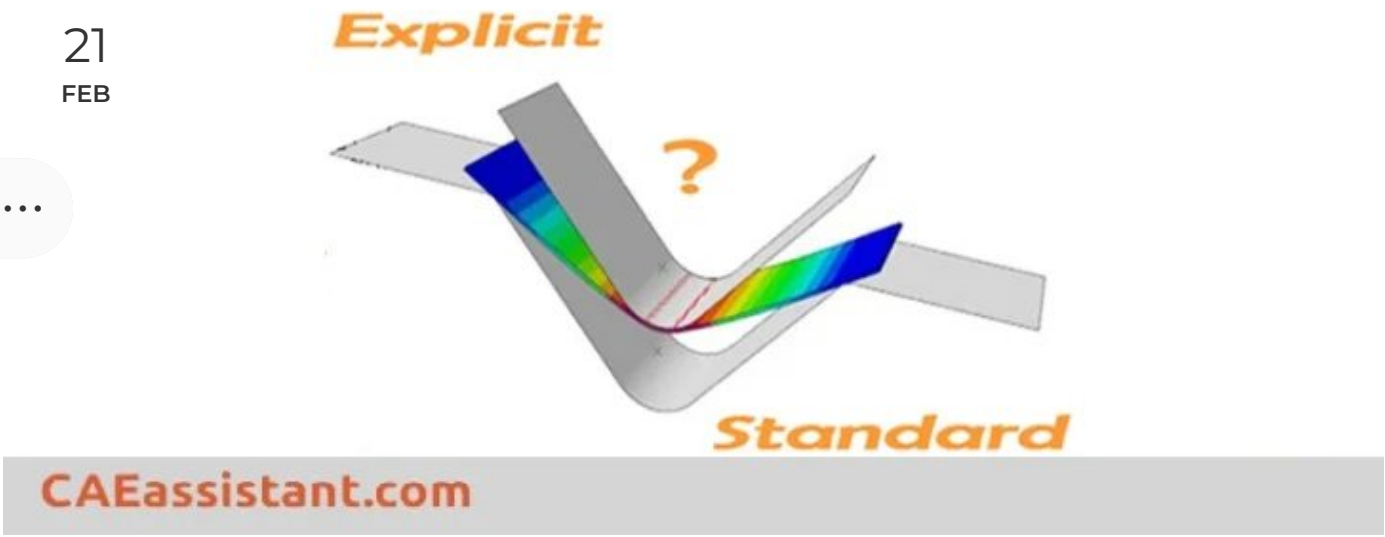
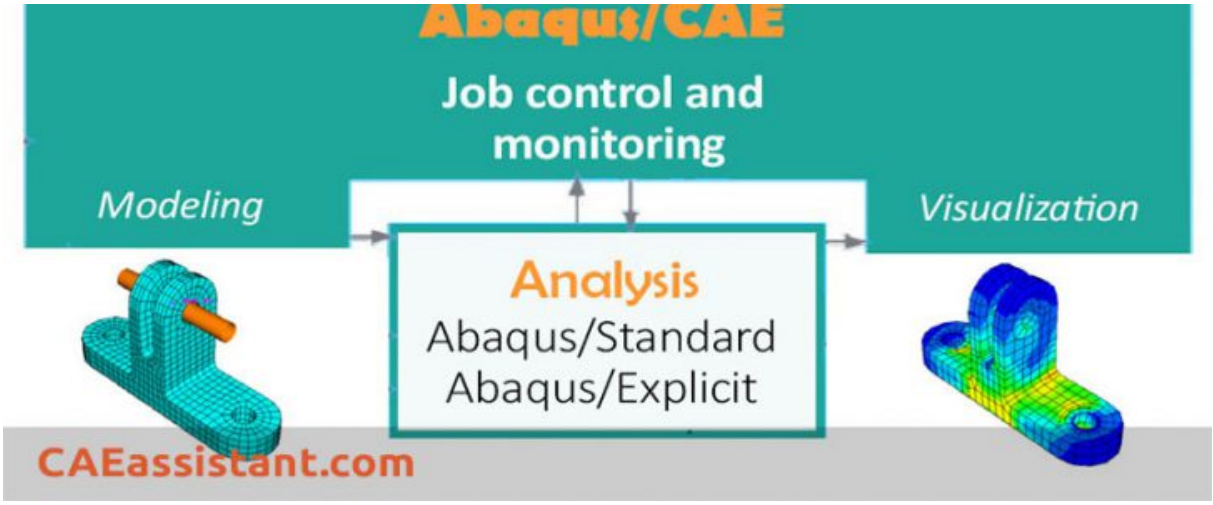


Table of Contents

Abaqus Explicit or Abaqus Implicit?

Have you ever wondered, "**Abaqus Standard or Abaqus Explicit?**" It's a common question when you're gearing up for your analysis. Aside from the specialized CFD solver (**Abaqus/CFD**) designed for fluid problems, Abaqus revolves around its two core analysis modules: Abaqus Implicit and Abaqus Explicit. In this post, we're about to unveil the key differences between these two solvers and help you pick the one that's just right for your analysis needs. Stick around with **CAE Assistant**!



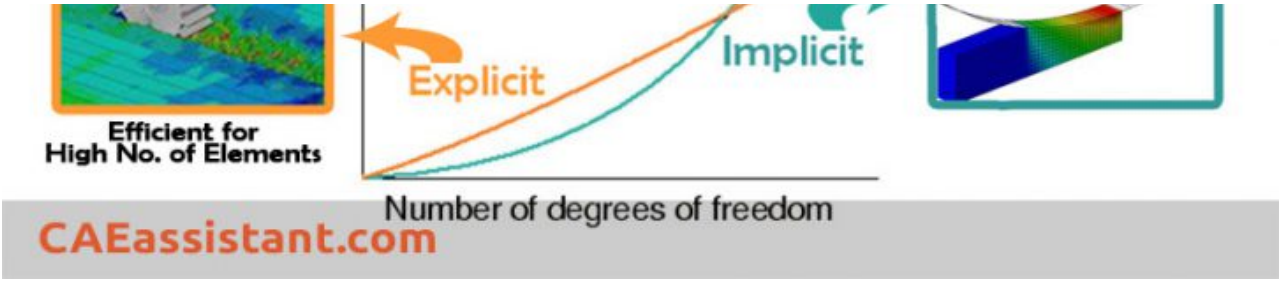
Abaqus implicit and Explicit Solvers | Abaqus standard vs implicit!

First, I have to say that there are no differences between Abaqus Standard and Implicit. In fact, these are two names for one solver. The main discussion is about Abaqus Implicit and Abaqus Explicit solvers.

These solvers are based on two approaches in FEM analysis, namely **implicit** (for Abaqus/Standard) and **explicit**. The distinction between the two different numerical approaches makes it possible to understand which solver to use.

Read More: **Debugging of ABAQUS errors**

In the case of the implicit method, equilibrium is enforced between externally applied load and internally generated reaction forces at every solution step (**Newton Raphson** method).
In the case of the Abaqus explicit method, there is no enforcement of equilibrium. But this does not mean that explicit is not accurate. You can minimize its deviation from equilibrium to almost zero by increasing the number of solution steps, i.e. reducing the time step size.
We can list the main differences below:
Abaqus Implicit is unconditionally stable.
Implicit schema is *incremental* as well as *iterative*. However, **explicit** schema is only *incremental*.
In terms of cost per Increment, it is costly for implicit and cheaper for explicit. Disk space and memory usage are typically much smaller than that for implicit. The explicit method shows great cost savings over the implicit method as the model size increases:



Therefore, **Abaqus/Standard**(or Abaqus implicit) must iterate to determine the solution to a nonlinear problem but **Abaqus/Explicit** determines the solution without iterating by *explicitly* advancing the kinematic state from the previous increment. Read More: Quasi Static

Abaqus Fundamentals Matt Veidth

Free Abaqus Course

Lifetime Access All Levels 27 Lessons 0 Quizzes 1131 Students

✅ Abaqus Free Course for Beginners, Intermediate | ✓ UMAT Subroutine Writing |✓ Python Scripting...

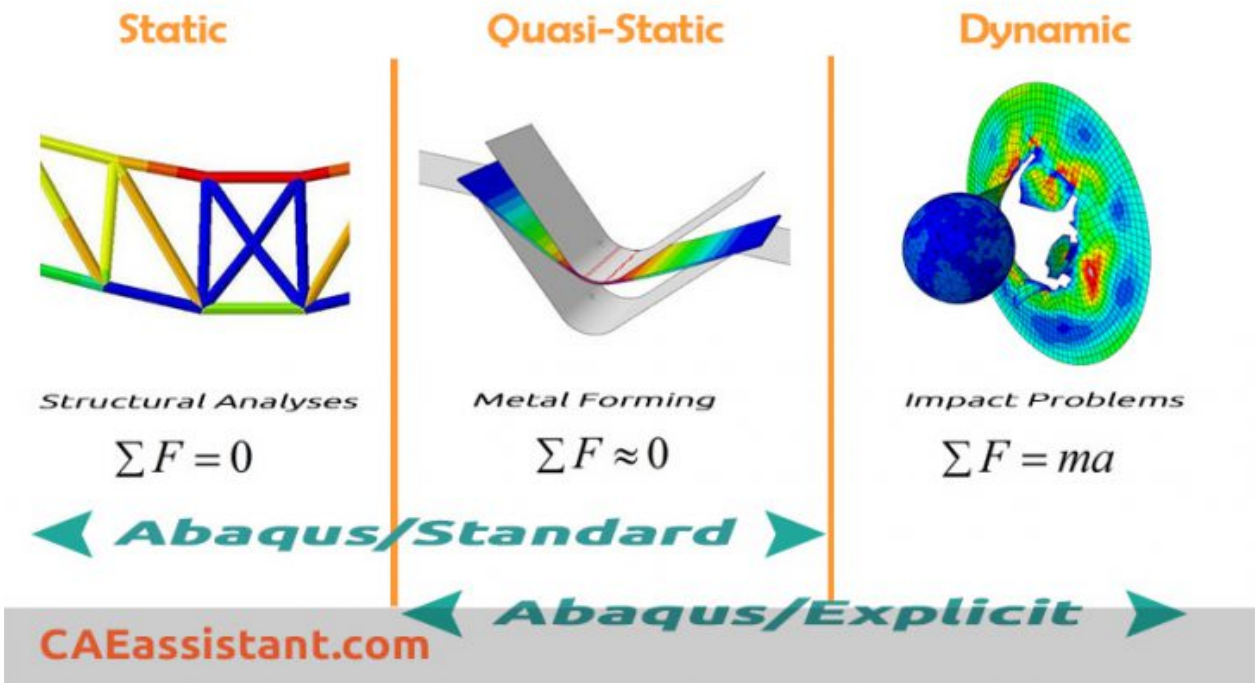
Free

View More

Explicit or Standard, Which one should I use?

For many analyses, it is clear whether Abaqus/Standard or Abaqus/Explicit should be used. For example, Abaqus/Standard is more efficient for solving smooth nonlinear problems; on the other hand, Abaqus/Explicit is the clear choice for high-speed dynamic analyses such as crash analysis or drop test. There are, however, certain problems that can be simulated well with either program.

Typically, these are problems that usually Abaqus/Standard can solve but may have difficulty converging because of contact or material complexities, resulting in a large number of iterations. For example, in problems where very complex contact conditions or very large deformations are present. Such analyses are expensive in Abaqus/Standard because each iteration requires solving a large set of linear equations.



In the video below, prepared by the CAE Assistant team, see the complete comparison between the Abaqus standard solver and the Abaqus explicit solver:

In this post, we have tried to explain 'Abaqus standard and Abaqus explicit' thoroughly so that you can choose the right solver for your needs. Also we clarified there is no difference between Abaqus standard vs implict. they typically refer to the same solver.

additionally, It would be useful to see **Abaqus Documentation** to understand how it would be hard to start an Abaqus simulation without any **Abaqus tutorial**. Also, please share your views with the **CAE Assistant** experts in the comment section. We really appreciate your feedback, as it helps us improve our tutorials and fulfill all your CAE needs without requiring additional tutorials.

Which of the Explicit or Standard solvers is more suitable for my problem?

What are the main differences between Explicit and Implicit Solvers in Abaqus?

What is the difference between the solving strategy of Abaqus/Standard and Abaqus/Explicit?

- What methods are used to analyze problems in Implicit and Explicit Solver?
- How can we solve problems that involve several analysis stages?

ABAQUS | Abaqus_Explicit | Abaqus_Standard | Explicit | Implicit | Solver

Newer Older

2 THOUGHTS ON "DIFFERENCES BETWEEN ABAQUS STANDARD & ABAQUS EXPLICIT"

Turkce says:

February 10, 2021 at 8:44 am

Good write-up. I definitely appreciate this site. Keep it up! Elle Tanny Kirtley

LOG IN TO REPLY

Sara.nik-4868 says:

November 12, 2022 at 11:25 am

Hello, thank you for this article. I needed to choose one of these two solver to solve my problem. This article helped me choose the most suitable solver. thank you

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

- About Us
- Portfolio
- Contact Us
- Privacy Policy
- Terms & Conditions
- Cookie Policy
- Join Us
- FAQs