

# Blog

BLOG

## Abaqus convergence issues in simple terms | FEA convergence problem

Posted by  Experts Of CAE Assistant Group



Engineer students and whoever is working with finite element software, especially the ABAQUS, heard the word “Abaqus Convergence problem”. Here, we will explain what that all mean and answer common questions such as “what is the FEA convergence problem?”, “how to identify the ABAQUS convergence issues?”. Also Abaqus contact convergence is discussed overall. Good news, you can start CAE Assistant free Abaqus course and get the Free Abaqus convergence issues PDF at the end of this blog.

### Table of Contents



- 1. 1. Abaqus convergence issues | Introduction
- 2. 2. FEA convergence problem
- 3. 3. Symptoms for the ABAQUS convergence issues
- 4. 4. Why we face with ABAQUS convergence issues? | Abaqus convergence problem
  - 4.1. Free Abaqus Course
- 5. 5. Identifying ABAQUS convergence issues
  - 5.1. 5.1. One of the best ways is to simulate a simpler model:
  - 5.2. 5.2. Set increment values
  - 5.3. 5.3. See the reasons for FEA convergence issues

# 1. Abaqus convergence issues | Introduction

Engineers using the ABAQUS software have two primary methods for solving finite element analysis (FEA) problems. The first method is called “Implicit,” used in ABAQUS/Standard, and the second is called “Explicit,” used in ABAQUS/Explicit.

The key difference between these two methods lies in how they approach problem-solving. In the Implicit method, equations are formulated in matrix form and solved using numerical techniques. On the other hand, in the Explicit method, such equations are not constructed in matrix form, and the problem is solved in a time-stepping and numerical manner. In the **Implicit method**, FEA convergence problems can **sometimes arise**, whereas in the **Explicit method**, convergence problems are **less likely**, although element distortion issues for specific elements may occur.

In the following section, we will explore a fundamental aspect of FEA convergence issues to help you grasp the concept. Additionally, consider starting our free ABAQUS course to enhance your understanding of FEA analysis and reduce the likelihood of encountering convergence issues. Let's delve into it.

☆☆☆ **Free Abaqus Course** | 10 hours Video 👤+1000 Students ☺ Lifetime Access

✓ Module by Module Training Analyses Tutorial

✓ Standard/Explicit

✓ Subroutines (UMAT) Training Examples

✓ Python Scripting Lesson &

[Start Free Abaqus Course](#)

## 2. FEA convergence problem

The finite element method uses this main equation to solve a problem:

The “K” is the stiffness matrix, the “U” is called the displacement matrix, and the “F” is the force matrix (see figure 1 as an example). When this equation is solved correctly, we will have accurate results, and here we use the term “the problem is convergence”. If by any means the equation cannot be solved or has some issues which give us inaccurate results, we use

the term “there are some FEA convergence issues or convergence problems.”

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} k_{x1x1} & k_{x1y1} & k_{x1x2} & k_{x1y2} & k_{x1x3} & k_{x1y3} \\ k_{y1x1} & k_{y1y1} & k_{y1x2} & k_{y1y2} & k_{y1x3} & k_{y1y3} \\ k_{x2x1} & k_{x2y1} & k_{x2x2} & k_{x2y2} & k_{x2x3} & k_{x2y3} \\ k_{y2x1} & k_{y2y1} & k_{y2x2} & k_{y2y2} & k_{y2x3} & k_{y2y3} \\ k_{x3x1} & k_{x3y1} & k_{x3x2} & k_{x3y2} & k_{x3x3} & k_{x3y3} \\ k_{y3x1} & k_{y3y1} & k_{y3x2} & k_{y3y2} & k_{y3x3} & k_{y3y3} \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix}$$

**f = Ku**

Figure-1 an example of a stiffness matrix with the dimension 6×6

### 3. Symptoms for the ABAQUS convergence issues

Now, how do we know there is a FEA convergence issue? You could see the solving procedure when running a job by clicking on the “monitor.” As you see (figure2), there are “Error” and “Warning” tabs. You can see some messages that inform you about your issues. Also, you could find more information in “.msg” (Standard solver), “.dat,” and “. sta” (Explicit solver) files. These convergence problems are more in nonlinear models than the linear ones. Some examples of these symptoms are shown in figure 1.

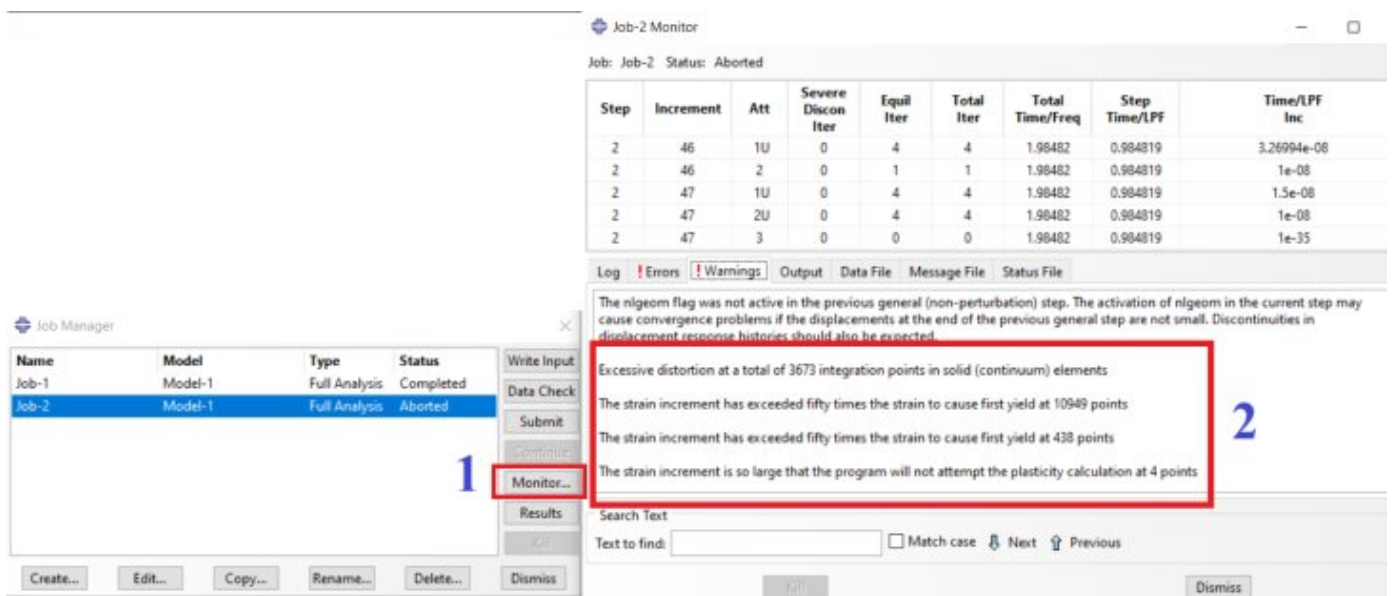


Figure-2: some warnings in the monitor window / Abaqus convergence problem

## 4. Why we face with ABAQUS convergence issues? | Abaqus convergence problem

This section will explain the reasons for the ABAQUS convergence issues or Abaqus convergence problems. It may be more than we explain here, but we discussed the most important and common ones.

Incomplete and defective modeling in FE is the most common reason for convergence issues; for example:

- Defining inappropriate constraints that cause conflicts in boundary conditions or contact conditions.
- Using wrong elements.
- Defining inadequate material in the property module.
- Defining inappropriate boundary conditions or contacts that may lead to Abaqus contact convergence issues.
- Modeling an unstable physical system.
- Setting an improper increment size can lead you further away from resolving FEA convergence problems.

Now, we intend to explain one or two of these conflicts that lead to Abaqus convergence issues. See figure 3 for an example. Imagine you have a box, and an amount of pressure is applied to it. You want to calculate the stress; if you do not define the boundary condition, the box moves, and the software cannot calculate the stress, and you will see an error. Note that this only happens when you analyze a static problem and use the Static step. Another example is if you constrain a surface and apply pressure on it simultaneously (see figure 4), the software will show an error; because it is not possible to have the surface fixed and apply pressure at the same time (inappropriate boundary conditions).

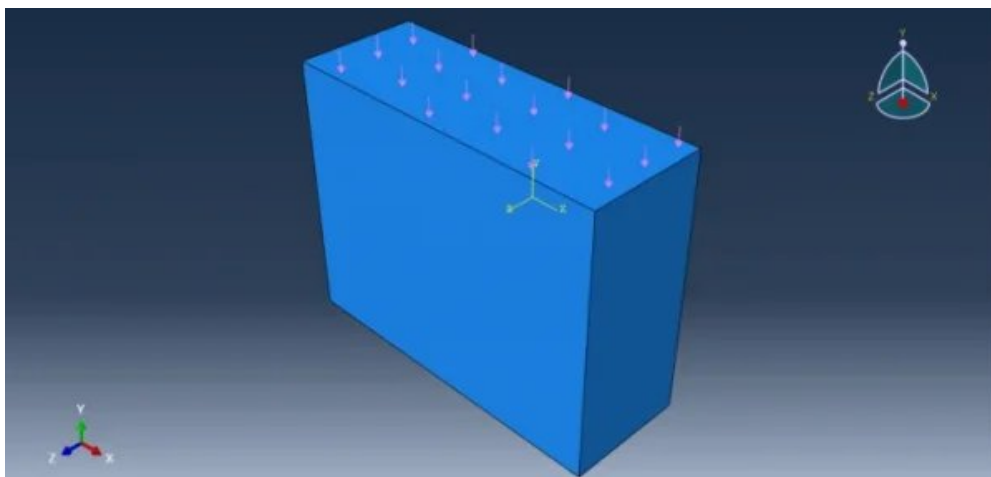


Figure-3: pressurized box without boundary conditions

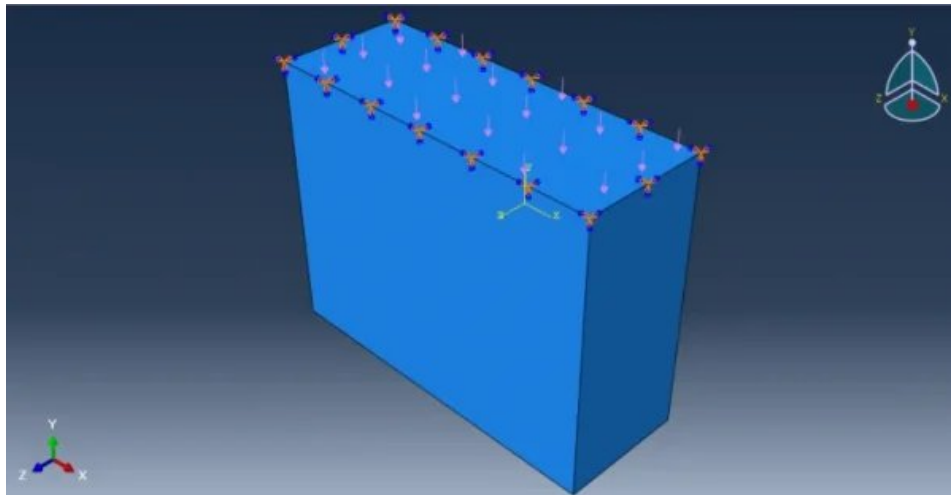
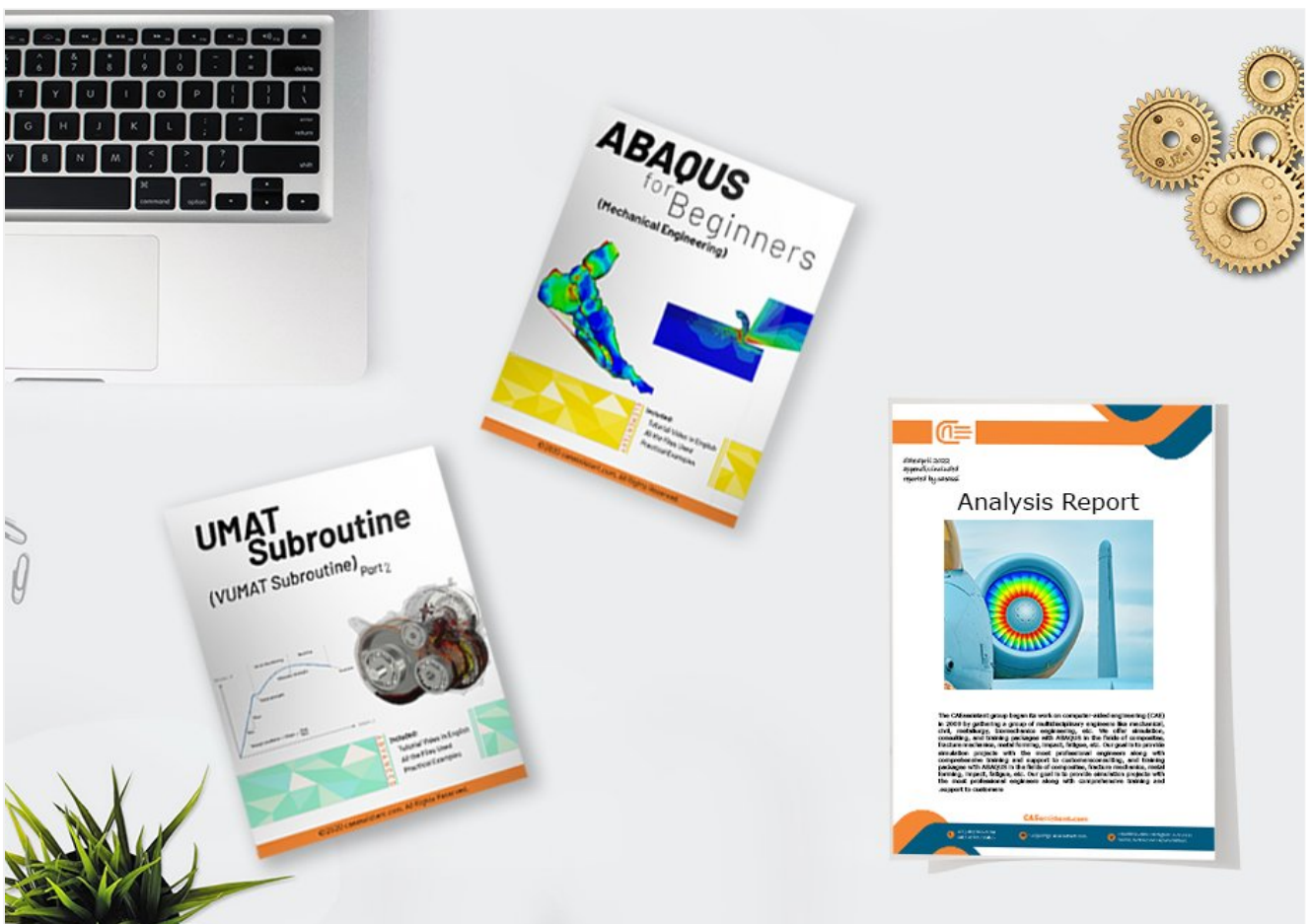


Figure-4 inappropriate boundary conditions



Abaqus Fundamentals

Matt Veidth

## Free Abaqus Course

🕒 Lifetime Access 📊 All Levels 📖 27 Lessons 🧩 0 Quizzes 👥 1147 Students

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



Free

[View More](#)

## 5. Identifying ABAQUS convergence issues

Here, we intend to present some recommendations to identify which symptoms and reasons are causing the issues, the “Errors” and the “Warnings.” The general way is to list the top potential reasons and then check them to see the changes in the software. In the end, start to fix them one at a time. Now, here are some of our recommendations:

### 5.1. One of the best ways is to simulate a simpler model:

- If possible, make a 2D model or a linear one to have fewer details and elements.
- If possible, do not enter Plasticity or Nonlinear geometry to understand the model's behavior.
- In a model with several pieces, insert one piece at a time to minimize the number of FEA convergence problem sources.

### 5.2. Set increment values

Set appropriate increment values that include initial increment, minimum increment, maximum increment size, and the maximum number of increments. To learn more about increments, [click here](#).

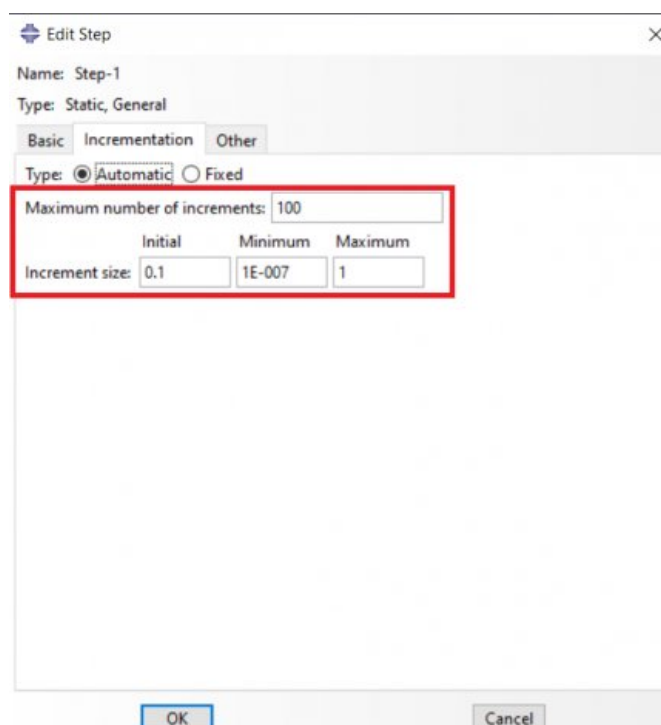


Figure-5 Static general step window and Incrementation tab

When the ABAQUS standard solver starts to run a Job, it splits the step in the maximum number of increments you specified (system default is 100). According to the specified increment size (Figure 5), the solver starts to run the Job. In the case of increments in the ABAQUS Standard solver, you typically encounter three errors that often indicate an ABAQUS convergence problem:

**“Too many attempts made for this increment”**; means the solver attempted several times to calculate the equations for this increment, but the FEA convergence conditions were not satisfied; so, you either look into the “Increment size” again and modify it or look somewhere else to overcome this Error.

**“Too many increments needed to complete the step”**; means the solver needs more increments; therefore, you have to increase the “Maximum number of increments” until you overcome this convergence problem.

**“Time increment required is less than the minimum specified”**; in this Error, you have to decrease the “Minimum” in “Increment size” to satisfy the convergence conditions.

---

Read More: [Abaqus tutorial PDF & video](#)

---

### 5.3. See the reasons for FEA convergence issues

See the reasons for convergence issues reported in the .dat, .sta, .msg, and .odb files.

You can add more information in the message files. For example, use the keyword command `“*PRINT, CONTACT=YES”` in the input file of the model (-.inp) to get contact information in the message file. This could help you to find out the ABAQUS contact convergence problems. Also use the command `“*PRINT, PLASTICITY=YES”` to get the integration point numbers and element output for material issues. You can use the `“*PRINT”` command to add more information in message files (.msg, .sta). Be patient, and we will explain to you how to use this command.

First, you need to know the input file (.inp). The Input file is one of the ABAQUS files that contains model data such as load, step, etc. It is like the “.cae” file but has less size, and you can open it in a text file and change whatever you want. When you have created your model completely and then created a Job for it, before you run it, you can create an input file for the model by clicking on the “Write Input” button in the “Job Manager” window (see figure

6). You can open the input file in a text file and change whatever you want; then, to use it in the ABAQUS, open the file according to Figure 7.

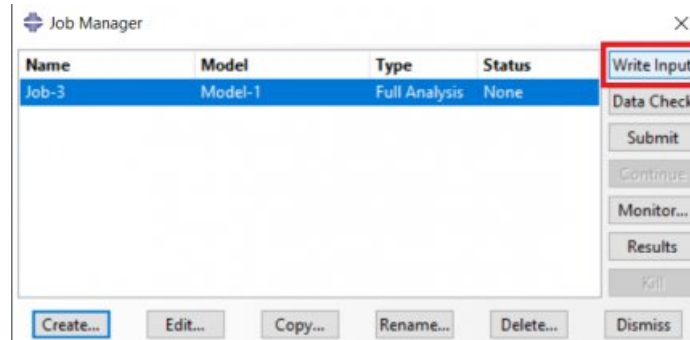


Figure 6 Create an input file

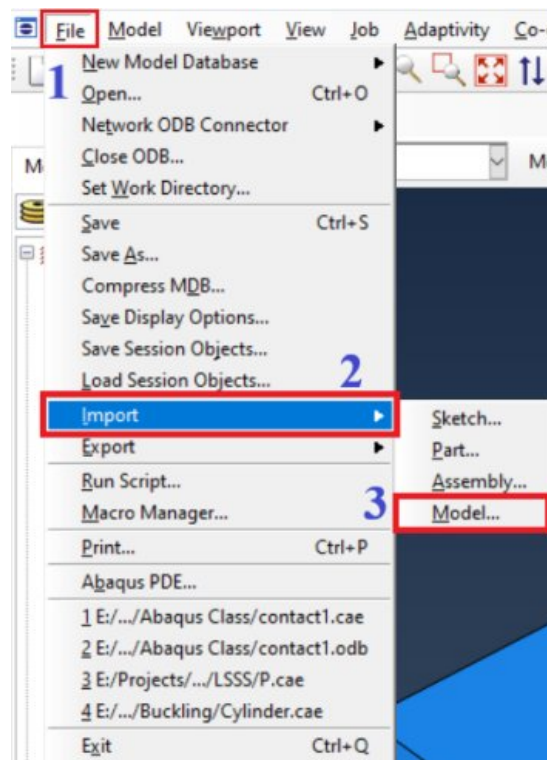
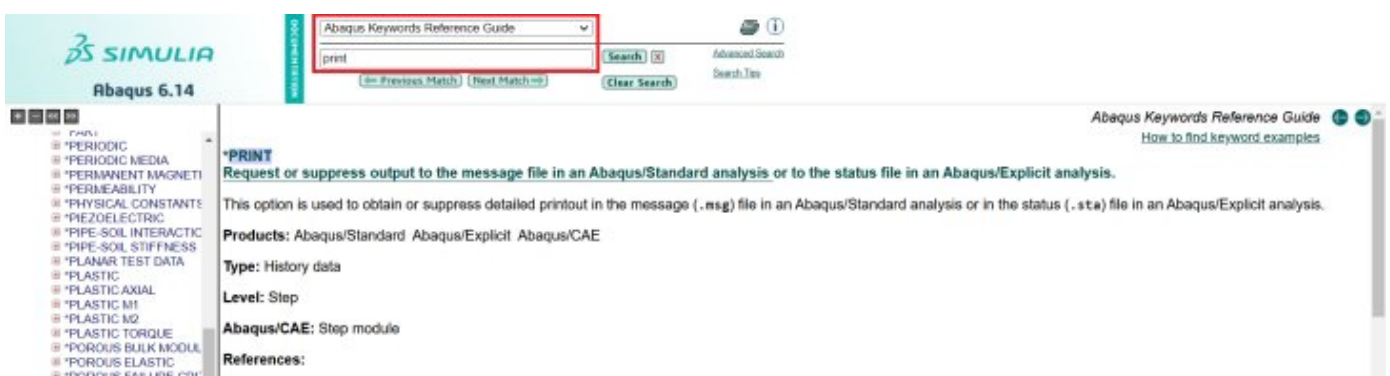


Figure 7: Open the Input file in the software

Now, as you know what an input file is, let's use the “\*PRINT” command. You can find the instructions about the “\*PRINT” command or any other keywords in the Keywords in the ABAQUS documentation (Figure 8).





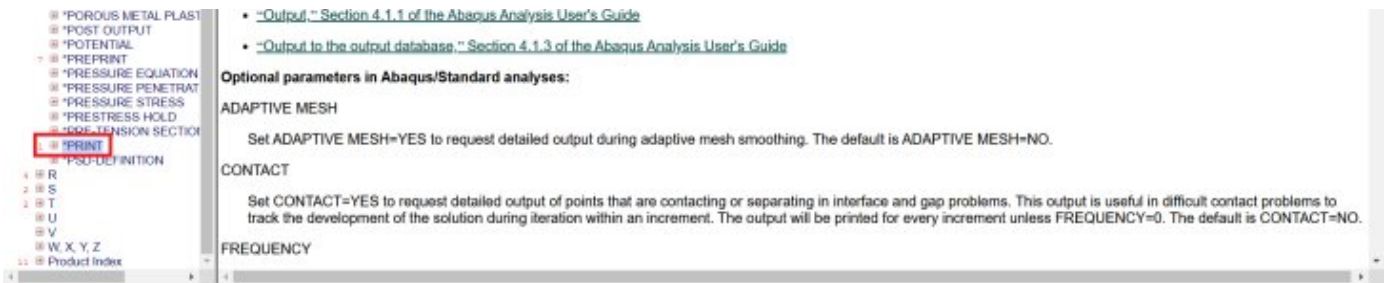


Figure 8 Finding PRINT keyword in ABAQUS documentation

As you can see in Figure 9, open your input file through the Edit Keywords window; then, find the lines that define the loading conditions; after these lines and before “\*END STEP,” you can add your “\*PRINT” commands then click OK button. After running the Job, you would see the results in the .msg (Standard solver) and .sta (Explicit solver) files. Now, you know how to write an input file, use it, and modify it.

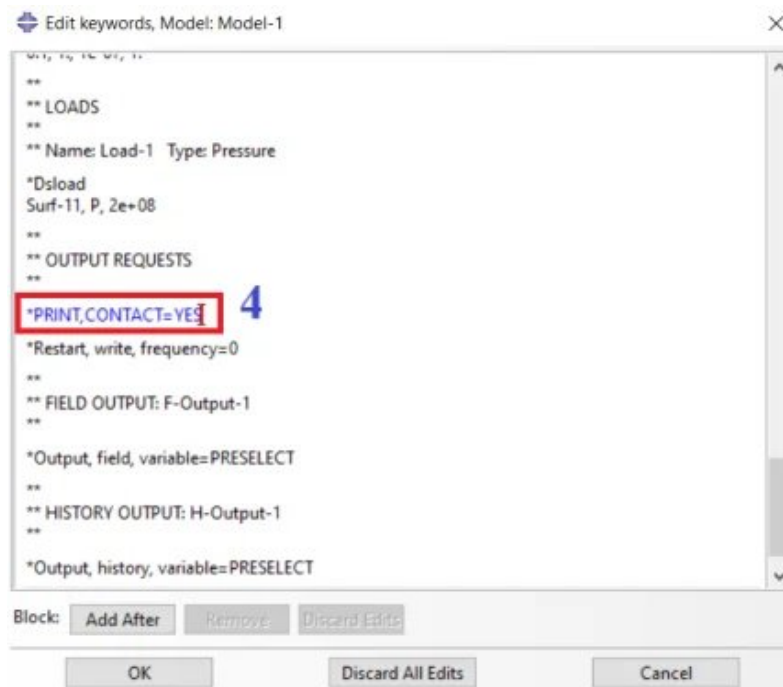


Figure 9 Enter the PRINT command

In this post, we have attempted to address ABAQUS convergence problems. We have explored how the finite element method relies on a primary equation, and when this equation remains unsolved, leading to an FEA convergence problem. Additionally, we have discussed the symptoms of ABAQUS convergence issues, why we encounter them, methods for identifying ABAQUS convergence problems, and the reasons behind FEA convergence problems.

We, the **CAE Assistant** team, hope you have got enough information about the Abaqus convergence problem and Abaqus contact convergence issues in this post. The upcoming articles will focus on tools and methods for resolving Abaqus errors, particularly convergence issues. You can click on [“Debugging of ABAQUS errors”](#) for more information.

It would be useful to see **Abaqus Documentation** to understand how it would be hard to start an Abaqus simulation without any **Abaqus tutorial**. Additionally, you can obtain a PDF version of this post to review the FEA convergence problem by clicking on “Abaqus convergence issues PDF.”

☆☆☆ **Free Abaqus Course** | 10 hours Video 👤+1000 Students 🌐 Lifetime Access

✔ Module by Module Training Analyses Tutorial

✔ Subroutines (UMAT) Training Examples

✔ Standard/Explicit

✔ Python Scripting Lesson & Examples

[Start Free Abaqus Course](#)

Reference: Troubleshooting Finite-Element Modeling with Abaqus

- ✦ What are the reasons behind the Abaqus convergence issues?
- ✦ What does convergence in Abaqus mean?
- ✦ What should we do when faced with a convergence issue?
- ✦ What is the error “Too many attempts made for this increment”?
- ✦ How can I extract the reasons for convergence issues in the .sta and .msg files?

• ABAQUS

• Convergence



Older

# 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)



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

