





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.



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.



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."

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

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.

				,	ob: Job	-2 Status: Ab	orted						
					Step	Increment	Att	Severe Discon	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
					2	46	10	0	4	4	1.98482	0.984819	3.26994e-08
					2	46	2	0	1	1	1.98482	0.984819	1e-08
					2	47	10	0	4	4	1.98482	0.984819	1.5e-08
					2	47	20	0	4	4	1.98482	0.984819	1e-08
					2	47	3	0	0	0	1.98482	0.984819	1e-35
					Log !	Errors ! Warr	nings (Output Dat	ta File Me	ssage File	Status File		
	🗢 lob Manager 🛛 🕹												
Job Manager				×	The nig cause of displace	eom flag was n onvergence pro ement response	ot active oblems if historie	in the previo the displace s should also	us general (ments at the be expected	non-perturi end of the	bation) step. The previous genera	e activation of nige al step are not smal	om in the current step r I. Discontinuities in
Job Manager	Model	Туре	Status	× Write Input	The nig cause o displace	eom flag was n onvergence pro ement response ve distortion at	ot active oblems if histories a total of	in the previo the displace should also	us general (ments at the be expected	non-perturi end of the f	bation) step. The previous genera ntinuum) elemer	e activation of nige al step are not smal	om in the current step r I. Discontinuities in
Job Manager lame ob-1	Model Model-1	Type Full Analysis	Status Completed	× Write Input Data Check	The nlg cause o displace Excession	eom flag was n onvergence pro ement response ve distortion at	ot active oblems if histories a total of	in the previo the displace s should also 3673 integra	us general (ments at the be expected tion points i	non-perturt r end of the 1. in solid (cor	bation) step. The previous genera itinuum) elemer	e activation of nIge al step are not smal nts	om in the current step r I. Discontinuities in
≱ Job Manager lame ob-1 ob-2	Model Model-1 Model-1	Type Full Analysis Full Analysis	Status Completed Aborted	X Write Input Data Check	The nlg cause o displace Excessiv The stra	eom flag was n onvergence pro ement response ve distortion at in increment h	ot active oblems if historie a total of as exceed	in the previo the displace s should also f 3673 integra ded fifty time	us general (ments at the be expected tion points i s the strain (non-perturt end of the d. in solid (cor to cause firs	bation) step. The previous genera ntinuum) elemer it yield at 10949 (e activation of nIge al step are not smal nts points	om in the current step r I. Discontinuities in
≱ Job Manager ame /b-1 &-2	Model Model-1 Model-1	Type Full Analysis Full Analysis	Status Completed Aborted	Write Input Data Check Submit	The nig cause of displace Excessive The strate The strate	eom flag was n onvergence pro ement resoonse ve distortion at in increment h in increment h	ot active sblerns if histories a total of as exceed as exceed	in the previo the displace s should also 13673 integra ded fifty time ded fifty time	us general (ments at the <u>be expectes</u> tion points i s the strain (s the strain)	non-perturt end of the f. in solid (cor to cause firs to cause firs	bation) step. The previous genera ntinuum) elemer it yield at 10949 ; it yield at 438 po	e activation of nIge al step are not smal nts points sints	om in the current step r I. Discontinuities in 2
i Job Manager ame b-1 5-2	Model Model-1 Model-1	Type Full Analysis Full Analysis	Status Completed Aborted	X Write Input Data Check Submit	The nig cause o displace Excession The strate The strate	eom flag was n onvergence pro ement resoonse ve distortion at in increment h in increment h	ot active oblems if historie a total of as excees as excees	in the previo the displace <u>s should also</u> (3673 integra ded fifty time ded fifty time	us general (ments at the <u>be expected</u> tion points i is the strain t is the strain t	non-perturit end of the in solid (cor to cause firs to cause firs	bation) step. The previous genera ntinuum) elemer it yield at 10949 j it yield at 438 po	e activation of nIge al step are not small nts points nints ulation at 4 exists.	om in the current step r 8. Discontinuities in 2
≽ Job Manager ame ab-1 5-2	Model Model-1 Model-1	Type Full Analysis Full Analysis	Status Completed Aborted	X Write Input Data Check Submit Comment	The nlg cause o displace Excession The strate The strate The strate	eom flag was n onvergence pro ement response ve distortion at in increment h in increment h	ot active oblems if histories a total of as excees as excees so large	in the previo the displace s should also f 3673 integra ded fifty time ded fifty time that the prog	us general (ments at the be expected tion points i s the strain t s the strain t gram will no	non-perturt end of the in solid (cor to cause firs to cause firs t attempt th	bation) step. The previous genera ntinuum) elemer at yield at 10949 it yield at 438 po ne plasticity calc	e activation of nIge al step are not small nts points ints ulation at 4 points	om in the current step r I. Discontinuities in 2
≱ Job Manager Jame Jab-1 Jab-2	Model Model-1 Model-1	Type Full Analysis Full Analysis	Status Completed Aborted	Write Input Data Check Submit Monitor Results	The nlg cause c displace Excession The strate The strate Search	eom flag was n onvergence pre ement response ve distortion at in increment h in increment h in increment is Text	ot active oblems if historie a total of as excees as excees so large	in the previo the displace s should also f 3673 integra ded fifty time ded fifty time that the prog	us general (ments at the <u>be expected</u> tion points i is the strain t s the strain t gram will no	non-perturt end of the f. in solid (cor to cause firs to cause firs t attempt th	bation) step. The previous genera ntinuum) elemen it yield at 10949 (it yield at 438 po ve plasticity calc	e activation of nIge al step are not small nts points vints ulation at 4 points	om in the current step r I. Discontinuities in 2
Job Manager tame ob-1 ob-2	Model Model-1 Model-1	Type Full Analysis Full Analysis	Status Completed Aborted	X Write Input Data Check Submit Monitor Results	The nig cause c displace Excession The stra The stra The stra Search Text to f	eom flag was n onvergence pre ement response ve distortion at in increment h in increment h in increment is Text Text	ot active oblems if a total of as excees as excees so large	in the previo the displace <u>s should also</u> (3673 integra ded fifty time ded fifty time that the prog	us general (ments at the <u>be expected</u> tion points i is the strain t gram will no gram will no	non-perturt end of the in solid (cor to cause firs to cause firs t attempt th tch case 8	bation) step. The previous genera ntinuum) elemen at yield at 10949 it yield at 438 po we plasticity calc i. Next ① Prev	e activation of nIge al step are not small nts points ints ulation at 4 points vious	om in the current step 1 6. Discontinuities in 2

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



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.

Type: Static, General		0.4		
Sasic Increm	entation	Other		
ype: () Autor	matic 🔿 Fi	ixed		
Aaximum num	ber of incre	ements: 100		
	Initial	Minimum	Maximum	
crement size:	0.1	15 007		
		1E-007	1	
		1E-007	1	
		12-007	1	
		12-007		

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.

Name	Mode	el	Туре	Status	Write Input
Job-3	Mode	el-1	Full Analysis	None	Data Check
					Submit
					Continue
					Monitor
					Results
					Kall

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).

BS SIMULIA Abaqus 6.14	Abeque Keywords Reference Guide v print (im Previoes Match) (Rest Match-III)	(Search) () Advanced Search (Clear Search) Search.lite	
CAN C	*PRINT Request or suppress output to the message file in an A	Abaqus/Standard analysis or to the status file in	Abaqus Keywords Reference Guide O How to find keyword examples n an Abaqus/Explicit analysis.
*PERMEABILITY **PHYSICAL CONSTANTS **PIE20ELECTRIC **PIPE-SOIL INTERACTIC **PIPE-SOIL STIFFNERS	This option is used to obtain or suppress detailed printout i Products: Abequa/Standard Abequa/Explicit Abequa/CAB	n the message (.esg) file in an Abaqus/Standard a	analysis or in the status (.sta) file in an Abaqus/Explicit analysis.
PLANAR TEST DATA PLASTIC PLASTIC AXIAL PLASTIC M1 PLASTIC M1	Type: History data Level: Step		
"PLASTIC N2 "PLASTIC TORQUE "POROUS BULK MODUL "POROUS BLASTIC "POROUS FAILURE CRI'	Abaqus/CAE: Step module References:		



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 <table-cell> +1000

Students @ Lifetime Access

Module by Module Training
Analyses Tutorial

* Subroutines (UMAT) Training
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?



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 O All Rights Reserved

