

DISPLACEMENT CONTROLS FOR GEOTECHNICAL PROBLEMS IN ABAQUS

Input specifications – an ABAQUS tutorial

Professor: William Mario Fuentes Lacouture
Student: Melany Gil Rueda
Department of Civil and Environmental Engineering
Universidad del Norte
Barranquilla, Colombia
2018

DISPLACEMENT CONTROLS FOR GEOTECHNICAL PROBLEMS IN ABAQUS

Using the displacement controls to achieve convergence on a geostatic step.

```
** STEP: Geostatic
**
*Step, name=Geostatic, unsymm=YES
*Geostatic, utol=tolerance for the maximum change of displacements
Initial time increment, Time period of the step, Minimum time increment allowed, Maximum time
increment allowed
**
** LOADS
**
** Name: gravity   Type: Gravity
*Dload, amplitude=instant
, GRAV, 10., 0., 0., -1.
**
*Controls, reset
*CONTROLS, PARAMETERS=FIELD, FIELD=DISPLACEMENT
 $R_n^\alpha$ ,  $C_n^\alpha$ 
, ,  $\epsilon_d^\alpha$ 
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
```

Using the displacement controls to achieve convergence on a geostatic step.

```
*Controls, reset  
*CONTROLS, PARAMETERS=FIELD, FIELD=DISPLACEMENT  
Rnα, Cnα  
, , Edα
```

Convergence criterion

$$R_n^\alpha = \frac{\text{Largest residual force}}{\text{Average force}} = \frac{r_{\max}^\alpha}{\overline{q}^\alpha}$$

Convergence criterion

$$C_n^\alpha = \frac{\text{Largest correction to displacement}}{\text{Largest increment of displacement}} = \frac{C_{\max}^\alpha}{\Delta U_{\max}^\alpha}$$

Criterion for zero displacement increment

$$\epsilon_d^\alpha = \frac{\text{Largest increment of displacement}}{\text{Average size of the element}} = \frac{\Delta U_{\max}^\alpha}{f_\beta^\alpha}$$

Default values

$$R_n^\alpha = 5 \times 10^{-3}$$

$$C_n^\alpha = 10^{-2}$$

$$\epsilon_d^\alpha = 10^{-8}$$

Using the displacement controls to achieve convergence on a geostatic step.

EXAMPLE

Given a characteristic element length in the model, $f_{\beta}^{\alpha} = 0.4 \text{ m}$

For average geotechnical practice,
 $\Delta U_{\max}^{\alpha} \approx 1\text{mm} = 0.001\text{m}$

$$\epsilon_d^{\alpha} = \frac{\text{Largest increment of displacement}}{\text{characteristic element length}} = \frac{\Delta U_{\max}^{\alpha}}{f_{\beta}^{\alpha}} = \frac{0.001\text{m}}{0.4 \text{ m}} = 0.0025$$

$$R_n^{\alpha} = \frac{\text{Largest residual force}}{\text{Average force}} = \frac{r_{\max}^{\alpha}}{\bar{q}^{\alpha}} = 0.01 \quad *$$

$$C_n^{\alpha} = \frac{\text{Largest correction to displacement}}{\text{Largest increment of displacement}} = \frac{C_{\max}^{\alpha}}{\Delta U_{\max}^{\alpha}} = 0.01 \quad *$$

```
*Controls, reset
*CONTROLS, PARAMETERS=FIELD, FIELD=DISPLACEMENT
0.01, 0.01
, , 0.0025
```



Note: The characteristic element length in the model (f_{β}^{α}) can be found in the message file (.msg)

* : Recommended values

Learn more at:

Abaqus 6.11
SIMULIA

DOCUMENTATION

My Support  

[Search All Books](#)
[Clear Search](#)
[Advanced Search](#)
[Search Tips](#)

Modeling and Visualization
[Abaqus/CAE User's Manual](#)

Analysis
[Abaqus Analysis User's Manual](#)

Examples
[Abaqus Example Problems Manual](#)
[Abaqus Benchmarks Manual](#)

Tutorials
[Getting Started with Abaqus: Interactive Edition](#)
[Getting Started with Abaqus: Keywords Edition](#)

Information
[Using Abaqus Online Documentation](#)

Installation and Licensing
[Abaqus Installation and Licensing Guide](#)

Reference
[Abaqus Keywords Reference Manual](#)
[Abaqus Theory Manual](#)
[Abaqus Verification Manual](#)
[Abaqus Keywords Reference Manual](#)
This manual contains a complete description of all the input options that are available in Abaqus/Standard and Abaqus/Explicit.
[Abaqus Scripting User's Manual](#)
[Abaqus Scripting Reference Manual](#)
[Abaqus GUI Toolkit User's Manual](#)
[Abaqus GUI Toolkit Reference Manual](#)

Interfaces
[Abaqus Interface for MSC.ADAMS User's Manual](#)
[Abaqus Interface for Moldflow User's Manual](#)

Abaqus 6.11 Update Information
[Abaqus Release Notes](#)

DISPLACEMENT CONTROLS FOR GEOTECHNICAL PROBLEMS IN ABAQUS

Learn more at:

The screenshot displays the Abaqus 6.11 Keywords Reference Manual interface. The top navigation bar includes the Abaqus 6.11 logo, a search bar with the text "controls", and navigation buttons for "Previous Match" and "Next Match". The left sidebar shows a tree view of keywords, with "*CONTROLS" selected. The main content area displays the following information for the *CONTROLS keyword:

***CONTROLS**
Reset solution controls.

Warning: This option is not needed in most nonlinear analyses, except for use with the parameter ANALYSIS=DISCONTINUOUS. However, if extreme nonlinearities occur, this option may be needed to obtain a solution. *“Commonly used control parameters,” Section 7.2.2 of the Abaqus Analysis User’s Manual, contains a discussion of the types of problems that may occur and the use of the *CONTROLS option to overcome these problems. This option can also be used in some cases to obtain a solution in a more efficient manner. Use of the option for this latter purpose is intended for experienced users only.*

Products: Abaqus/Standard Abaqus/CFD Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- [“Convergence and time integration criteria: overview,” Section 7.2.1 of the Abaqus Analysis User’s Manual](#)
- [“Commonly used control parameters,” Section 7.2.2 of the Abaqus Analysis User’s Manual](#)
- [“Convergence criteria for nonlinear problems,” Section 7.2.3 of the Abaqus Analysis User’s Manual](#)
- [“Time integration accuracy in transient problems,” Section 7.2.4 of the Abaqus Analysis](#)