Abaqus user subroutines for numerical modelling of evolving arcades and rock pillars during natural erosion

Alexander Safonov

*Center for Design, Manufacturing and Materials, Skolkovo Institute of Science and Technology, Moscow, Russia*

We performed an advanced numerical modeling to model various shapes of arcades and rock pillars during the erosion of rock masses dissected by discontinuities. We demonstrate that the erosion model, in which erosion takes place when the maximum principal stress is below a certain critical value, can adequately describe the formation of arcades. In the modeling, we set higher critical values for stresses at discontinuities than in a homogeneous material (representing a rock mass) to represent the higher tendency for disintegration of the discontinuity material, which was weakened by the discontinuity formation processes.

For numerical modeling of the erosion process, we used the ABAQUS finite-element software suite (version 6.14). Abaqus allows a researcher to get realistic solutions for nonlinear mechanical problems quickly. Besides, its user subroutines mechanism makes it possible to locally change mechanical properties and loads distribution during computation. In this study we used following user subroutines: USDFLD, URDFIL, DLOAD, and UEXTERNALDB. We used USDFLD to consider changes in mechanical properties in finite elements during the erosion process and URDFIL to access the results of the solution of maximum stress distribution. URDFIL was also used to get the coordinates of the integration points of finite elements and form the array containing the list of adjacent elements for each given finite element. We used DLOAD to assign the surface and bulk forces at each iteration step, depending on the advance of the erosion front or on the number of the iteration step (arcade evolution modeling). UEXTERNALDB was used to assign algorithm parameters before modeling. We used allocatable arrays to transfer data arrays between user subroutines during computations.

A more detailed description of the problem is given in the article

*Alexander Safonov, Michal Filippi, David Mašín, Jiří Bruthans. (2020). Modeling of cracking during pultrusion of large-size profiles. Geomorphology.*

The preprint of this article is in the folder.

For information about Abaqus user subroutines, see [Abaqus User Subroutines Reference Guide](http://abaqus-pc:2080/v6.13/books/sub/default.htm) and [“User subroutines: overview,” Section of the Abaqus Analysis User's Guide](http://abaqus-pc:2080/v6.13/books/usb/usb-link.htm#usb-anl-asubroutineover).

Software: Abaqus/CAE 6.14-4