Abaqus user subroutines for numerical simulation of cracking during pultrusion of 80 mm diameter GFRP rod

Alexander Safonov1, Alexander Konstantinov2

1Center for Design, Manufacturing and Materials, Skolkovo Institute of Science and Technology, Moscow, Russia

2[Research Institute of Mechanics](https://www.researchgate.net/institution/Nizhny_Novgorod_State_University/department/Research_Institute_of_Mechanics), [Nizhny Novgorod State University](https://www.researchgate.net/institution/Nizhny_Novgorod_State_University), Nizhny Novgorod, Russia

We simulate pultrusion process mechanics, including the temperature distribution, degree of polymerization, and residual stresses. The pultrusion is a high-production process allowing the fabrication of high-performance composite profiles having a constant cross section.

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

*Safonov, A., Gusev, M., Saratov, A., Konstantinov, A., Sergeichev, I., Konev, S., … Akhatov, I. (2019). Modeling of cracking during pultrusion of large-size profiles. Composite Structures, 111801.* [*https://doi.org/10.1016/j.compstruct.2019.111801*](https://doi.org/10.1016/j.compstruct.2019.111801)

The preprint of this article is in the folder.

Additional description of mathematical relations can be found in the file Description.docx. When using the developed Abaqus user subroutines, please cite to the original article.

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