To use any subroutine in Abaqus it is necessary to link Abaqus with Fortran compiler. There are few methods to this. Simplest method is the below method:

* Install Microsoft Visual Studio. Only „programming desktop applications in C++” is mandatory
* Install Intel OneAPI Base Toolkit. Only „DPC++/C++ compiler” is mandatory
* Install Intel HPC Toolkit. Only „fortran compiler” is mandatory
* Find the „launcher.bat” file in the SIMULIA folder and edit it in notepad. After the line:

@echo off

add the line:

call "C:\Program Files (x86)\Intel\oneAPI\compiler\latest\env\vars.bat" intel64 vs2019

or vs2022 depending on version of visual studio.

If any problems occur, try the following solutions:

* in SIMULIA folder find file "abaqus\_v6.env", edit in notepad and add at the end the line:

link\_sl='LINK /NODEFAULTLIB:LIBCMT.LIB /dll /def:%E /out:%U %F %A %L %B'

* in SIMULIA folder find file "custom\_v6.env", edit in notepad and add at the end the line:

compile\_fortran += ['/names:lowercase',]

* use visual studio 2019 instead of 2022.

In order to correctly use the crack propagation criterion, it is necessary to prepare the files for the Abaqus system. The above described criterion is located in a text file with the extension ".for" (fortran). First, it is necessary to prepare the ".inp" file, which is a text file with all the information about the model intended for calculations. This file is generated automatically in the Abaqus graphical environment (Abaqus/CAE) by right-clicking on one of the Jobs and selecting "Write input" (by default, this file appears in the "temp" directory of the Windows system), but a few changes must be made. Under the MATERIALS keyword for the selected material to be cracked, add the lines that are responsible for starting the UDMGINI procedure:

\*Damage Initiation, criterion=user, failure mechanisms=1, properties=1

ft

\*Damage Evolution, type=ENERGY, failure index=1

GIc

In the above lines, substitute own values ​​for ft (tensile strength), and GIc (critical strain energy release rate in mode I).

Below, the following two lines shoul be added:

\*User Output Variables

0

They are usually used to define User Output Variables, but they are not used in the above algorithms. In this case, these two lines are only used to call the UVARM procedure.

Then, after the keyword "Step" the information about the first step of the calculation, the following lines should be added:

\*node file

philsm,coord,u

\*el file

coord,s

First, they are used to call the URDFIL procedure, secondly, they inform the solver what results should be written to the "Results" file so that the procedure can read them. These are the data in the nodes, i.e. phi level set (PHILSM), coordinates of the nodes (COORD) and displacements in the nodes (U), and data in the integration points, i.e. coordinates and stresses (S). In the case where there is more than one part in the simulation, it is necessary to ensure that the stresses and displacements are read only from the part that is subjected to the fracture simulation. In order to avoid saving information about unnecessary parts of the model in the "Results" file, modify the line:

\*el file, elset=X

and

\*node file, nset=Y

where X and Y are the names of the set of elements and nodes that are to be cracked. X is the same set in which the enriched function for the X-FEM method occurs, the information about which can be found after the "Assembly" keyword:

\*Enrichment, name=Crack-1, type=PROPAGATION CRACK, elset=X

To properly prepare the models themselves in the Abaqus graphical environment, the only important condition is to select the mesh of four-node unreduced elements CPS4 in the case of a two-dimensional task and CAX4 in the case of an axisymmetric task.

All files subjected to calculations using the author's crack propagation prediction methods are prepared in this way.

To perform the simulation, create a new Job by selecting "Jobs" in Abaqus/CAE and then "create". Then as "source" select "Input file" and provide the path to the prepared ".inp" file. Then in the "Edit job" window in the "General" tab in the "User subroutine file" field provide the path to the procedure file with the ".for" extension.