Video description: Animation of Fig. 7 in the manuscript

H. Moldenhauer
Integration of Direction Fields with Standard Options in Finite Element Programs

Producing Figure 7 needs execution of the following steps:

1) Static run	Input: "Figure7-Static.inp" (reads "Figure7-BulkData.inp") Run the Abaqus Input "Figure7-Static.inp" Result: odb-File, which can be viewed with Abaqus/View A viewer option is: Report -> Field Output. With this option write the stress components Sxx, Syy, Sxy. Edit this file: add commas to separate the stress components. The result should look like the file "Figure7-Static.rpt"
2) Figure7.for	Fortran program, which reads "Figure7-Static.rpt" Input: Figure7-Static.rpt Output: Figure7-Static.out. This File contains local inclination angles with respect to the global x-axis, one line per element. These local directions are used for the orthotropic thermal analysis.
3) Thermal run	Input: "Figure7d.inp" (reads "Figure7-BulkData.inp" and “Figure7-Static.out”) Run the Abaqus Input "Figure7d.inp" (k1/k2 = 4096) Result: odb-File, which can be viewed with Abaqus/View and looks like Figure7d in the paper.
4) Animation	Input: "Figure7.inp" (reads "Figure7-BulkData.inp" and “Figure7-Static.out”) Run the Abaqus Input "Figure7.inp" (k1/k2 = 1,2,4, ... 4096) Result: odb-File, which can be viewed and animated with Abaqus/View.
Remark	Step 1 and 2 is optional. You can immediately start with step 3, as the file for the local conductivity systems is in the zip-archiv ("Figure7-Static.out").