Here is an example showing how to run the CPFE-based creep simulation of 151-grain polycrystal structure with cohesive element grain boundaries

1. Abaqus input file:

“CZM151G\_p28d6.inp” – A 151-grain microstructure applied by a pressure 28.6 MPa at the front surface in X-direction.

Linked files:

* “CZElement\_151G.inp” – the information of cohesive elements at grain boundaries
* “CZM.para” – Set of parameters used in cohesive zone model

1. Fortran file: “umat\_uel.f”

* This Fortran file includes both crystal plasticity model and cohesive zone model
* The Fortran file is linked to several .f files and the folder “minpack”

1. Python file: “CZM151G\_p28d6.py” – a post-process to plot creep curve
2. Shell script “CZM151G\_p28d6.sh” – to run with with multiple CPUs

“qsub –l h=node001 CZM151G\_p28d6.sh”

1. All above input files, Fortran .f files and folder “minpack” must be put together in the working folder “tmp”