Modeling constant strain rate control in the microMegas loading mode (8)
mode (8) = using inhomogeneous stress field defined in a regular grid

March 17, 2016

Strain rate control can be applied in the mM loading mode (8). To activate this solution, the stress factor coefficient in the file 'FE_Sig' must be set to 0! As usual, the strain rate control is inspired by Hook's law. The stress increment is calculated as

$$\Delta\sigma = C(\dot{\varepsilon}_{imp} - \dot{\varepsilon}_p)\Delta t \tag{1}$$

C is an apparent young modulus.
ε_{imp} is the strain rate we want to impose.
ε_p is the effective simulation plastic strain rate.

The simulation plastic strain rate is calculated over the whole simulated domain in a reference direction which is defined in the control file (loading axis for the uniaxial simulations).

In the loading mode (8), $\Delta\sigma$ has no meaning and the control is made by applying variation of the stress factor SF used to multiply the reference stress field defined in the file 'FE_Sig'.

Stress factor calculation: We calculate this stress factor (SF) as

$$\mathrm{SF} = (\dot{\varepsilon}_{\mathrm{imp}} - \dot{\varepsilon}_{\mathrm{p}})\Delta \mathrm{t} \tag{2}$$

Once the SF is known, the stress components from finite element (FE) simulation are factorized by SF as

$$\sigma_{\mathrm{app}} = \sigma_{\mathrm{FE}} \times \mathrm{SF}/\Delta\varepsilon_0 \tag{3}$$

$\Delta\varepsilon_0$ is a reference strain needed for reason of dimensionality. This quantity is the strain amplitude associated to the stress field solution defined in the 'FE_Sig' file in the direction used for the strain rate control.

This new parameter is defined in the 'FE_Sig' file. In order to facilitate simulations, a lower limit ($\mathrm{SF_l}$) and an upper limit ($\mathrm{SF_u}$) are also defined in the 'FE_Sig' file. $\mathrm{SF_l}$ is the initial value taken for SF in the simulations.

Technicalities: For compression or indentation $\Delta\varepsilon_0 = -|\Delta\varepsilon_0|$.
If SF in 'FE_Sig' is initialized to '0' the simulation will run for strain rate control mode, with the constant rate provided in the control file. If SF in 'FE_Sig' is set to a non-zero value the system will follow the provided FE stress distributions multiplied by SF.