**main\_IBIIProcessing\_V3\_0ir.m**

Three main codes were used to process the results in the modulus paper. The first code is called ‘main\_IBIIProcessing\_V3\_0ir.m’. For the reduced orthotropic (UD90°) case, this code calculates the kinematic fields from which average stress and strain values are derived along vertical slices on the sample. For the off-axis case, the kinematic fields are first rotated into material coordinates, and then acceleration and strain fields are interpolated to angled slices on the sample. The transverse and shear stress gauge equations are then used and combined with the strains to plot average stress-strain curves for each slice and time step. Each stress-strain curve (on each slice) is fitted linearly to obtain modulus values (on each slice). These modulus values are averaged over a defined region on the sample, which is the middle 50% of the sample length in the UD90° case and over the full range of x0 values for the UD45° case, where x0 is the distance from the top of the slice to the sample edge (see paper for full definition of x0 for each case). The chart below shows the workflow for this code. Here the initialisation file is called ‘processingParameters\_v3\_0ir.mat’, which is generated by the code ‘main\_initProcessingParams\_XXXX\_v3\_0ir.m’, where ‘XXXX’ is the sample designation *e.g.* ‘UD90\_S1’, ‘UD45\_S1’ or ‘MD45\_S1’.

main\_IBIIProcessing\_V3\_0ir.m

AllProcessedData.mat

main\_initProcessingParams\_XXXX\_S1\_v3\_0ir.m

processingParameters\_v3\_0ir.mat

**main\_IBIIProcessing\_v3\_0ir\_ImportFEFields.m**

If the ε11 correction is performed, the code ‘main\_offAxisPostProcessor\_v2.m’ (the third code) uses the file ‘avg11\_FE.mat’, which contains the average ε11 strain fields from an FE simulation. However, the fields need to be interpolated to each sample where the correction is performed, as the sizing of each sample is different. If this is not done, a different number of slices for the experimental and FE fields results and the correction will not be performed. Here you would get a ‘matrix dimensions must agree’ error in Matlab. So first we need to get the FE avg11 fields, interpolated to each sample being analysed. This is done as follows:

First run the Abaqus simulation by executing the files ‘CFRPPulseXXXX.py’ and ‘CFRPPulseXXXX\_DataExtraction.py’ in Abaqus, with XXXX = UD90 or UD45 (for this example we will use the UD90 case). This will create data files for the simulation. These files are located in ..\ModulusPaper\FE\_UD90. Then navigate to ..\FE\_UD90\AbaqusToMatlab and open the FE init file ‘main\_initFESimProcessingParams\_2D\_CFRPPulse\_UD90\_v1.m’ in Matlab. This will read the specimen, time and pos structs from the simulation configuration file. It creates the initialisation file containing this information called ‘FESimProcessingParameters\_v1.mat’ and the folder where that and the upcoming ‘.mat’ file (generated with ‘Import2DFEModelFields.m’) will be saved. Currently, this is set to the same folder that the init file is in ..\FE\_UD90\AbaqusToMatlab.

Next, open the script ‘Import2DFEModelFields.m’ (located in the same folder) in matlab and execute. This converts the FE displacement fields to the mat file ‘FESimKinematicFields\_2D\_CFRPPulse\_UD90.mat.’

Then execute the code ‘main\_IBIIProcessing\_v3\_0ir\_ImportFEFields.m’. This code imports the FE init file ‘FESimProcessingParameters\_v1.mat’ and displacements ‘FESimKinematicFields\_2D\_CFRPPulse\_UD90.mat.’ Then it loads the experimental init file 'processingParameters\_v3\_0ir.mat' (generated in code 1) for a particular sample that you wish to interpolate the FE fields to, saving the file ‘avg11\_FE.mat’.

You can also first process the FE displacements without interpolating to the sample, checking the kinematic fields, general diagnostics (load pulse etc), stress-strain curves and modulus vs. position plots. This is done in the code ‘main\_IBIIProcessing\_v3\_0ir\_ImportFEFields.m’ with switches.FEValidMode = 1.

Python script for Abaqus simulation

avg11\_FE.mat

main\_IBIIProcessing\_v3\_0ir\_ImportFEFields.m

FESimKinematicFields\_2D\_CFRPPulse\_XXXX.mat

Import2DFEModelFields.m

FE fields

main\_initFESimProcessingParams\_2D\_CFRPPulse\_XXXX\_v1.m

FESimProcessingParameters\_v1.mat

**main\_offAxisPostProcessor\_v2.m**

The third code is the post processor called ‘main\_offAxisPostProcessor\_v2’. This code calculates the UD45° shear modulus values with the linear elastic limit determined by progressive chord fits to the shear stress-strain curves.

It also corrects the UD45° or UD90° E22 values for the fibre strains (or Q12 stiffness) using FE ε11 values in the file the file ‘avg11\_FE.mat’, which is calculated in main\_IBIIProcessing\_v3\_0ir\_ImportFEFields.m as explained on the previous page.

Finally, it can be used to obtain strain-normalised strain rate values. The chart below shows the work flow for the processing code.

Note that the code requires ‘main\_IBIIProcessing\_V3\_0ir.m’ to be run first, as it uses the save data file ‘AllProcessedData.mat’ for its calculations. When that is done, run the init file and execute the code, setting the switches for what analysis you wish to perform.

main\_offAxisPostProcessor\_v2.m

Normalised strain rate

Chord fitting

e11 correction

AllPostProcessedData.mat

avg11\_FE.mat

init\_postProcessingVariables\_XXXX\_SX

post-ProcessingParameters\_v1.mat