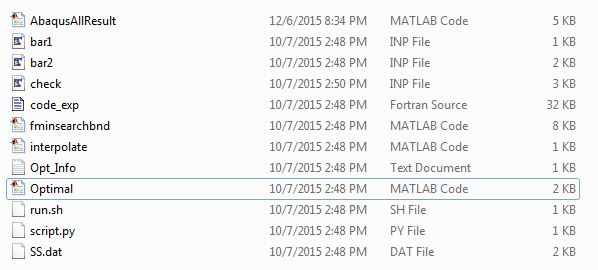
**Calibration of Model Parameters**

* 1. **Software requirements**: Abaqus, Python and Matlab
  2. **Files Required**



File-1: AbaqusAllResult.m: this file creates an Abaqus input file by combining bar1.inp and bar2.inp and adding material properties. Runs the resulting Abaqus input file. After the analysis is finished, it calls “script.py” to obtain the engineering stress strain curve. Then it imports the target stress strain data from SS.dat and evaluates the RMS error in the stress prediction and its derivative with respect to strain. The outputs of AbaqusAllResult.m are: 1) f1: RMS of estimated stress data; 2) f2: RMS of estimated stress derivative with respect to strain divided by stress and multiplied by an importance factor (100-default value); and 3) mean square root error.

File-2: bar1.inp and bar2.inp: incomplete Abaqus input files of a single element with periodic boundary conditions.

File-3: check.inp: Abaqus input file generated by AbaqusAllResult.m.

File-4: code\_exp.f: isotropic rate-dependent plasticity UMAT

File-5: fminsearchbnd.m: Matlab script for finding optimum model parameters.

File-6: interpolate.m: Matlab script for linear interpolation of model parameters. AbaqusAllResult.m calls interpolate.m in order to linearly interpolate stress values at intermediate strain levels.

File-7: Opt\_info.dat: data file containing the values of model parameters and RMS errors.

File-8: Optimal.m: an interface between the end-user and these set of codes: the end-user can specify starting, upper and lower bounds of the model parameters.

File-9: run.sh: batch script that runs “Optimal.m”.

File-10: script.py: python script to obtain engineering stress strain curve from single element analysis.

File-11: SS.dat: ASCII file of stress strain data for calibrating the model parameters.

* 1. **Usage**

Step-1: copy the target stress strain data in to “SS.dat”.

Step-2: edit “Optimal.m” to specify the upper and lower bounds of the model parameters. Also, specify the starting values of the model parameters in this matlab script.

Note: this code is fairly generic and hence the number of model parameters can be changed. However if the number of model parameters are changed in “Optimal.m”, corresponding changes should be made in “AbaqusAllResult.m”.

Step-3: submit the “run.sh” batch script. This batch script runs “Optimal.m” matlab script which in turn invokes “fminsearchbnd.m” that searches for the optimum set of model parameters. The matlab script “fminsearchbnd.m” needs the mean squared error value (m=f1+f2; f1-RMS error in stress and f2-RMS error in stress derivative with respect to strain multiplied by an importance factor) corresponding to the input model parameters. This is evaluated by invoking “AbaqusAllResult.m”.

Step-4: if everything is used appropriately, the program will iteratively find the best fit model parameters. The model parameters in each cycle along with RMS error values will be printed to “Opt\_info.dat” file.