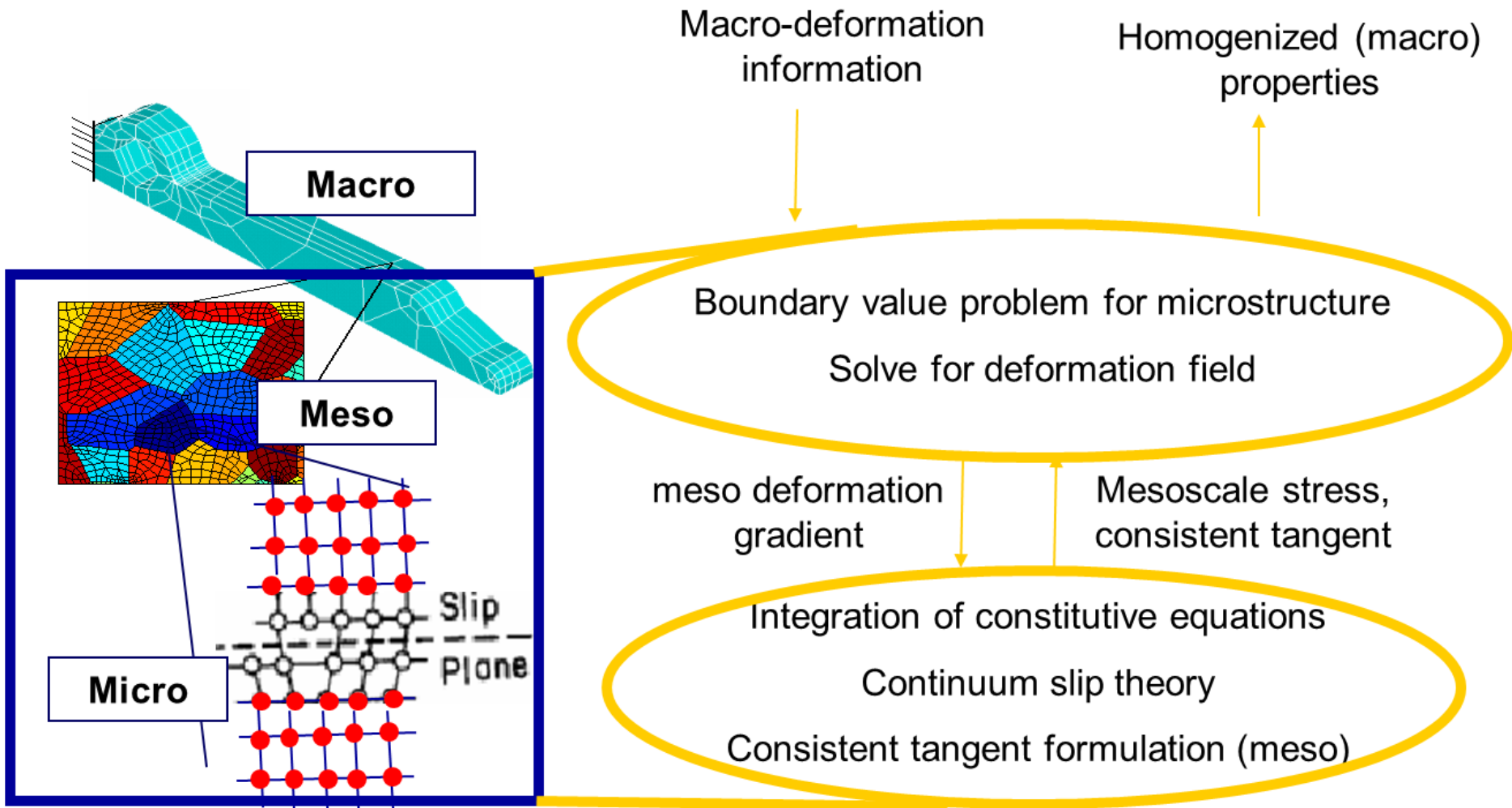# PRISMS-Plasticity: An Open Source Crystal plasticity FE Code

Mohammadreza Yaghoobi

Department of Materials Science and Engineering, University of Michigan, Ann Arbor

PRISMS Center Training Session
August 15, 2022

U.S. DEPARTMENT OF ENERGY | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRISMS

Crystal plasticity FEM
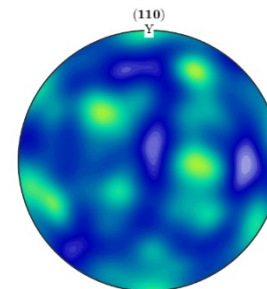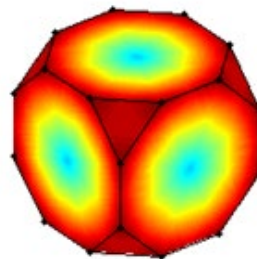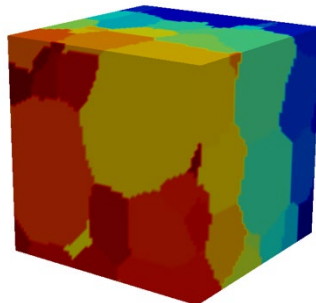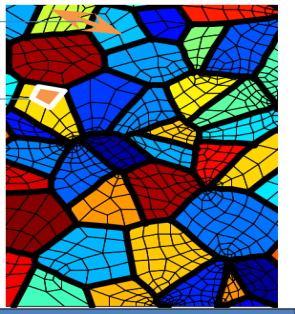
PRISMS-Plasticity

An Open Source Crystal plasticity Finite Elements Code
(github.com/prisms-center/plasticity)

Advanced Capabilities:

- Rate-independent and Rate-dependent Crystal plasticity Constitutive models.
- PTR and TDT Twinning model.
- Isotropic and Kinematic hardenings.
- Easily connected to the common pre-processors (DREAM3D, Neper).
- Integration with Materials Commons.
- High-Performance: Ideal scaling for >1,000 processors.
- Multiphase problems.
- Effect of grain size.

User-Friendly:

- Simple input file.
- Detailed online user guide for the code.
- post-processors (MTEX and Paraview).
- Detailed online user guide for linking pre-processing and post-processing.
- Applications to get you started.
- Highly modular and easy to add new features to the code.
- Framework to implement different crystal plasticity constitutive models.
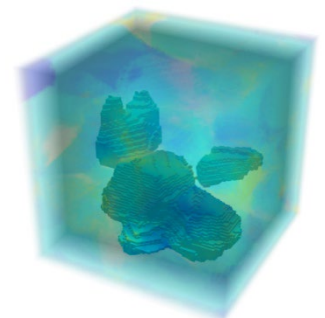
PRISMS Plasticity Scaling Study Versus Abaqus

Cyclic response of polycrystalline Al alloy 7075-T6

- Microstructure with Random texture and ~7,500 grains; 90^3 FE discretization
- Periodic BCs; Uniaxial cyclic loading; Kinematic Hardening;
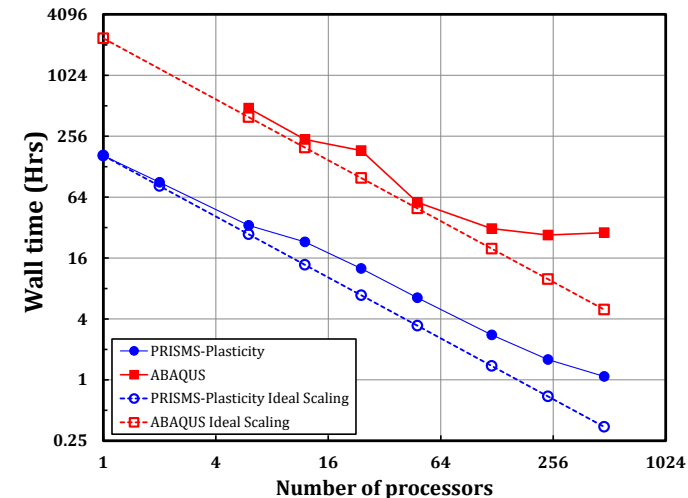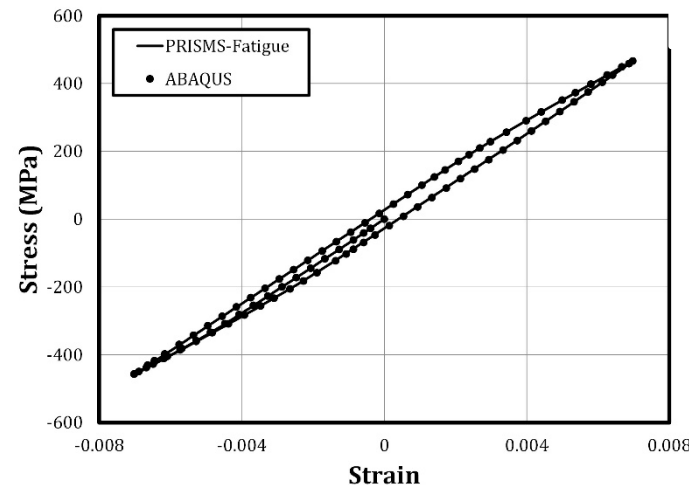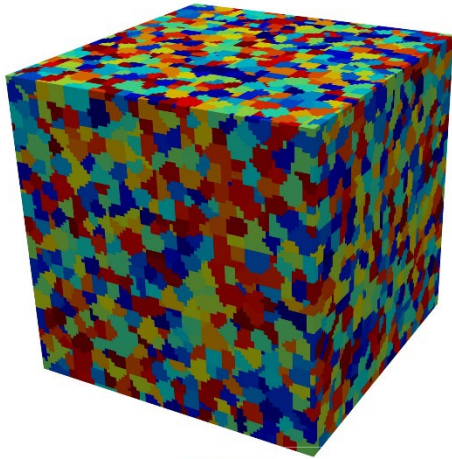
The results shows that PRISMS-Plasticity can simulate the sample in 1.08 hours using 480 processors, while ABAQUS requires 28.5 hours using the same number of processors.

PRISMS-Plasticity website

- Google prisms-center/plasticity github

- Going over Training_Materials, applications, docs, src, include folders

- Going over an application folder:
  plasticity/applications/crystalPlasticity/fcc/compression/

- PRISMS-Plasticity required input files:

1) Main input file (prm.prm)
2) Grain ID file
3) Orientation file
4) BCs file
5) Latent Hardening Ratio file
6) Slip Direction/Normal (Twin Direction/Normal)

General Linux Terminal Commands

ls   Used to show the contents of the current directory

pwd   Prints the current working directory (where you are in the file structure)

cd   Change directory (cd .. to go up one directory, cd [insert directory here] to go into a directory)

scp   Copy a file or directory (cp [file to be copied] [where to copy it to], cp –r [directory to be copied] [where to copy it to])

mv   Move a file or directory (mv [file to be moved] [where to move it to], mv –r [directory to be moved] [where to move it to])

rm   Delete a file or directory (rm [file to be deleted], rm –r [directory to be moved])

mkdir   Create a new directory (mkdir [directory name])

U.S. DEPARTMENT OF **ENERGY** | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Microstructure generation

orientations.txt:

The first row is dummy (does not matter).
After that, each line has four columns:
Grain ID and three Rodrigues vector components:
grainID, Rod1,rod2,rod3

Multiphase simulations:

If you have a multiphase simulations, an additional column should be added to the orientations.txt as a phaseID (Check user manual for more detail).

Additional Voxel info:

In addition to orientations (and possibly phase ID), if you have any other grain level information you want to pass, such as grain size, residual strain, etc., you can use a feature of "Additional Voxel info", and add that as a new column (Check user manual for more detail)

```
C:\Users\yaghoobi\Downloads\plasticity-master-2021\Training_Materials\Pre-Processing\HCP\orientations.txt - Notepad++
File  Edit  Search  View  Encoding  Language  Settings  Tools  Macro  Run  Plugins  Window  ?

orientations.txt

 1  0 0 0 0
 2  1 0.706527 0.45221 0.198347
 3  2 14.9515 9.41504 -10.786
 4  3 -1.10348 1.2271 1.17724
 5  4 2.82433 -0.543489 -3.4207
 6  5 1.45413 2.19316 -0.529163
 7  6 -0.108922 -2.01401 0.405957
 8  7 -0.180539 -0.545749 0.177164
 9  8 3.37263 -4.8475 -1.7216
10  9 -6.04981 -10.4885 -11.4154
11  10 0.551727 0.222017 -0.0430181
12  11 -9.2471 -5.3786 6.10939
13  12 -0.955772 0.056508 -0.789907
14  13 0.488871 -0.507244 0.74779
15  14 -22.0284 -39.3318 -0.739059
16  15 -12.2119 5.63069 -6.85142
17  16 -0.303606 0.372028 -0.42964
18  17 3.66351 -0.0208949 -1.37557
19  18 0.9062 0.490992 -0.19592
20  19 0.435739 1.06345 4.58736
21  20 5.67255 -0.288694 0.955549
22  21 0.421963 1.1515 -0.125092
23  22 -2.40555 -3.54481 -0.539058
24  23 -0.609045 0.61898 0.0524783
25  24 -0.0637942 -3.05449 -2.13411
26  25 1.25453 0.240392 -1.02011
27  26 -3.79384 0.0290665 -1.37979
28  27 0.254935 -0.314292 0.257108
```
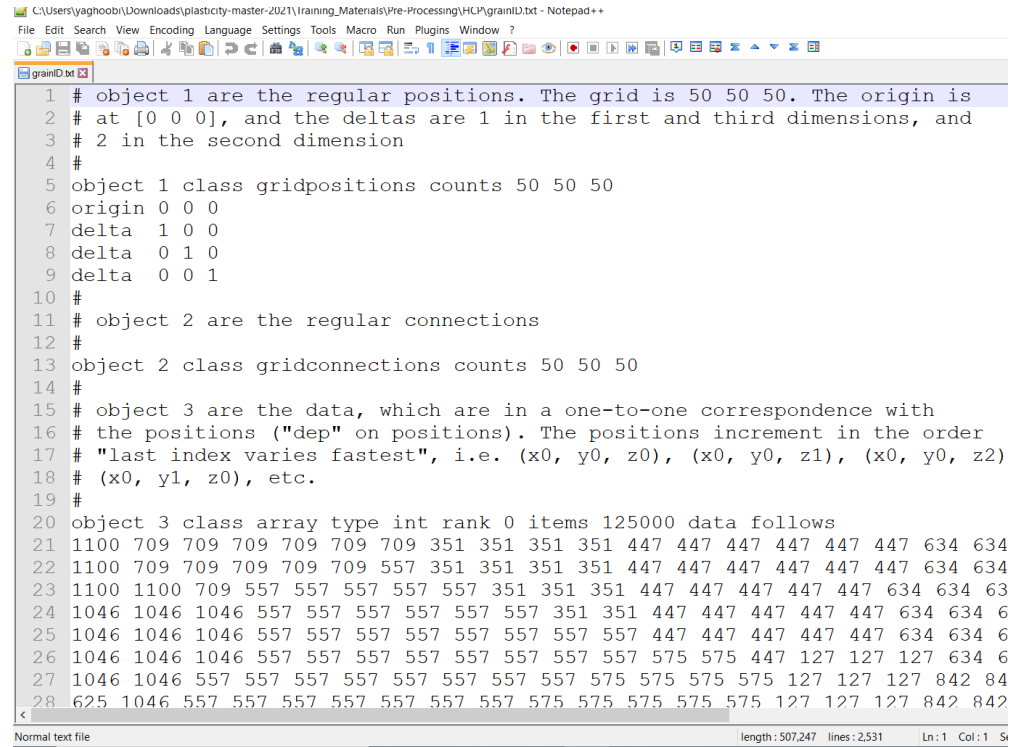
Microstructure generation

grainID.txt:

This file assigns the grainID for each voxel. Important information from this file is:
- Number of header lines
- Number of Voxel in each direction

In the presented file, we have 20 lines as header. After that, the numbers are grainID for each voxel. A 3d microstructure of $N1 \times N2 \times N3$ voxels is mapped into a 2D array. The corner voxel closest to the origin may be indexed as (1,1,1) because it is in the first voxel in x, y and z directions, respectively. Using this notation, any voxel can be indexed as (i, j, k) with $i \in \{1, 2, \dots, N1\}, j \in \{1, 2, \dots, N2\}, k \in \{1, 2, \dots, N3\}$. Then the grain ID of such a voxel will be present in the $(j + (i - 1) \times N2)^{th}$ row and in the kth column of grainID.txt (after header lines).


```
C:\Users\yaghoobi\Downloads\plasticity-master-2021\Training_Materials\Pre-Processing\HCP\grainID.txt - Notepad++
File  Edit  Search  View  Encoding  Language  Settings  Tools  Macro  Run  Plugins  Window  ?

grainID.txt
  1  # object 1 are the regular positions. The grid is 50 50 50. The origin is
  2  # at [0 0 0], and the deltas are 1 in the first and third dimensions, and
  3  # 2 in the second dimension
  4  #
  5  object 1 class gridpositions counts 50 50 50
  6  origin 0 0 0
  7  delta   1 0 0
  8  delta   0 1 0
  9  delta   0 0 1
 10  #
 11  # object 2 are the regular connections
 12  #
 13  object 2 class gridconnections counts 50 50 50
 14  #
 15  # object 3 are the data, which are in a one-to-one correspondence with
 16  # the positions ("dep" on positions). The positions increment in the order
 17  # "last index varies fastest", i.e. (x0, y0, z0), (x0, y0, z1), (x0, y0, z2)
 18  # (x0, y1, z0), etc.
 19  #
 20  object 3 class array type int rank 0 items 125000 data follows
 21  1100 709 709 709 709 709 709 351 351 351 351 447 447 447 447 447 447 634 634
 22  1100 709 709 709 709 709 557 351 351 351 351 447 447 447 447 447 447 634 634
 23  1100 1100 709 557 557 557 557 557 351 351 351 447 447 447 447 447 634 634 63
 24  1046 1046 1046 557 557 557 557 557 557 351 351 447 447 447 447 447 634 634 6
 25  1046 1046 1046 557 557 557 557 557 557 557 557 447 447 447 447 447 634 634 6
 26  1046 1046 1046 557 557 557 557 557 557 557 575 575 447 127 127 127 634 6
 27  1046 1046 557 557 557 557 557 557 557 557 575 575 575 575 127 127 127 842 84
 28  625 1046 557 557 557 557 557 557 557 575 575 575 575 575 127 127 127 842 842

Normal text file                                    length : 507,247  lines : 2,531    Ln : 1  Col : 1  S
```

Main Input file (prm.prm)

FE parameters

set Order of finite elements = 1

Polynomial order of interpolation function (1 => linear basis functions)

set Order of quadrature = 2

Quadrature point order (2 => 2^n) quadrature points where n is the physical dimension)

Domain parameters

* Assuming that the simulation domain is a cuboid of arbitrary dimensions.

set Number of dimensions = 3

Number of physical dimensions for the simulation

set Domain size X = 1.0

set Domain size Y = 1.0

set Domain size Z = 1.0

The size of the domain in the x, y, and z directions.

U.S. DEPARTMENT OF **ENERGY** | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Main Input file

Mesh parameters

Meshing can be performed through deal.II or by reading an external mesh through a file generated using Gmsh.

Deal.II mesh generator

The mesh generation starts with a single unit cell, and slices it in x, y and z directions as many times as the subdivisions or refinement factor indicates.

> *set Subdivisions X = 2*

> *set Subdivisions Y = 2*

> *set Subdivisions Z = 2*

The number of mesh subdivisions in the x, y, and z direction (2by2by2 mesh so far).

> *set Refine factor = 2*

The number of initial refinements of the coarse mesh (2 => each coarse element is subdivided into $2^2 = 4$ smaller elements) -> 8by8by8 is the mesh.

It is always more efficient to use the set Refine factor to refine the mesh rather than using the subdivisions in the x,y, and z directions.

There are some exceptions in which you cannot use set refinement: (1) Applying periodic BCs (2) Applying grainID deletion. In these two cases, one can only use set Subdivisions for mesh refinement.

Main Input file

External Mesh Generation

The external mesh with the format of Gmsh can be read.

>   *set Use external mesh = true*

Flag to indicate whether to use external mesh.

>   *set Name of file containing external mesh = n200-id4_hex.msh*

Name of external mesh file.

>   *set External mesh parameter = 0.05*

The external mesh parameter: The ratio of defined region size to the Domain size.
When an external mesh is read, x, y, and z positions are not precisely defined due to roundoff errors. This matters in the case of BCs assignment. Accordingly, here, a margin is defined in which if a point is located in this margin, it will be considered as BCs. The size of this region in each direction is equal *to (External mesh parameter)×(Domain size in that direction)*.

Main Input file

Output parameters

In this section, all the features related to output generation are discussed..

> *set Write Output = true*

Flag to write output vtu and pvtu files.

> *set Output Directory = results*

Output Directory name

> *set Skip Output Steps = 1*

Number of Output Steps to skip. The outputs will be generated every N steps, which N is defined in this line.

> *set Output Equivalent strain = true*
> *set Output Equivalent stress = true*
> *set Output Grain ID = true*
> *set Output Twin fractions = true*

By setting each of these flags to true, that will be outputted.

Advanced feature of adding variable to vtu and pvtu outputs!!! I will discuss this on the session 2 (Tuesday) of training.

U.S. DEPARTMENT OF **ENERGY** | Office of Science

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Output parameters (Cntd.)

set Write Quadrature Output = true

Flag to write Quadrature Output, which is the information for each FE integration point.

set Skip Quadrature Output Steps = 1

Number of Quadrature Output Steps to skip.

What variables are outputted using Quadrature_Output feature?

https://github.com/prisms-center/plasticity/blob/master/src/materialModels/crystalPlasticity/updateAfterIncrement.cc
Starting line 248 which is: "*temp.push_back(cellOrientationMap[cellID]);*" up to the line 489 before
"*} addToQuadratureOutput(temp);*"

Whatever variable is inside temp.push_back(variable); will be added to the QuadratureOutput.
If you do not want to print out a specific variable, you can comment it out by adding // to the start of that line.
These three lines in QuadratureOutput section of updateAfterIncrement.cc (lines 258-260) defines the spatial coordinate of the integration point (x, y, and z of the integration point):

temp.push_back(fe_values.get_quadrature_points()[q][0]);
temp.push_back(fe_values.get_quadrature_points()[q][1]);
temp.push_back(fe_values.get_quadrature_points()[q][2]);

If you apply any changes to updateAfterIncrement.cc, you need to recompile so the changes can be applied.

13

U.S. DEPARTMENT OF ENERGY | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Output parameters (Cntd.)

> *set Tabular Output = true*

One can use this feature to output the results (Both visualization and Quadrature outputs) at specific time table they define here. This is the flag for tabular time output.

> *set Tabular Time Output Table = 0,0.25,0.48*

Simulation times at which outputs are generated.

Even if all output generation fields are disabled, a stressstrain.txt file is generated for each step which includes the average Green-Lagrange strain tensor, Cauchy stress tensor, twin activity volume (TwinRealVF), twin volume fraction (TwinMade), and slip activity (SlipTotal). If one wants to compare the average slip and twin activities, i.e., TwinRealVF vs SlipTotal, they should consider that average twin activity volume is the twin pseudo-slip systems shear activity divided by the characteristic twin shear constant.

The default Quadrature Output: GrainID, Phase ID (If Advanced Twinning model is not used), det(J), twin (1 if it is reoriented due to twinning and 0 if not), x,y,z (coordinates of quadrature point),rotnew(1), rotnew(2), rotnew(3) (updated Rodrigues vector of orientation), 9 components of Fe (elastic deformation gradient tensor), 9 components of Fp, 9 components of Cauchy stress tensor, slip activity for 84 slip systems (if there are less systems, it will be appeared as 0), and 6 twin activity volume. Again, the twin activity volume is the twin pseudo-slip systems shear activity divided by Characteristic twin shear constant.

14

Center for PRedictive Integrated Structural Materials Science

Main Input file

Boundary conditions information

Boundary conditions (BCs) can be applied by defining the applied displacements or velocity gradient tensor. Different boundary conditions can be applied to the sample including:

Simple BCs (Session 1)
Velocity gradient tensor (Session 1)
Cyclic BCs
Tabular BCs (Session 1)
SEM-DIC (Digital Image Correlation) BCs
Periodic BCs (Session 2)

Nodal BCs
Tabular Torsion BCs
Tabular Numann BCs

Simple BCs:

set Use Simple BCs = true

This flag defines if one use Simple BCs. It will be considered true if it is not defined. In the case of other BCs types, it should be defined as false.

set Boundary condition filename = BCinfo.txt

File name containing BC information.

set BC file number of header lines = 2

Number of header lines in BC file

set Number of boundary conditions = 4

Number of boundary conditions applied on the sample

Main Input file

Boundary conditions information (Cntd.)

A sample boundary condition specification could look like:

```
 BCinfo.txt ✕
1  # Header lines = 2
2  # FaceID DoF FinalDisplacement
3  1 1 0
4  1 2 0
5  1 3 0
6  2 1 -0.05
7
```

Column 1: FaceID: refers to the face identifier and follows the numbering as indicated here:

Column 2: DoF: refers to the direction of application of displacement, where 1, 2 and 3 refer to the x, y and z directions respectively:

Column 3: Final displacement: The final displacement applied at the end of total time. The displacement will be linearly varied to reach this *Final displacement*.

U.S. DEPARTMENT OF **ENERGY** | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS
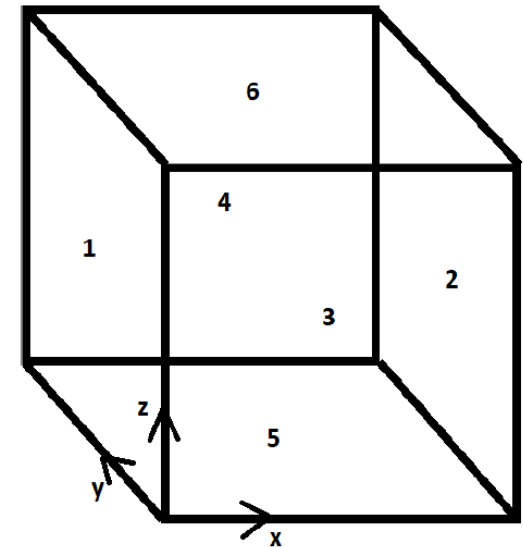
Main Input file

Boundary conditions information (Cntd.)

Velocity gradient tensor:
set Use Simple BCs = false
This flag defines if one use Simple BCs. In this case, it should be defined as false.

set Use velocity gradient BC = true
Flag to indicate whether to use velocity gradient tensor to apply BCs.

set Velocity gradient row 1 = -0.005, 0, 0
set Velocity gradient row 2 = 0, -0.005, 0
set Velocity gradient row 3 = 0, 0, 0.01
Velocity gradient tensor including the multiplication factor.

Tabular BCs:
set Use Simple BCs = false
This flag defines if one use Simple BCs. In this case, it should be defined as false.

set Use Tabular BCs = true
Flag to indicate whether to use Tabular BCs.

set Tabular Boundary condition filename = BCinfoTable.txt
Flag to indicate whether to use Tabular BCs.

set Number of time data for Tabular BCs = 5
Number of time data for Tabular BCs (it includes the initial BCs).

Center for PRedictive Integrated Structural Materials Science

Boundary conditions information (Cntd.)

Tabular BCs (Cntd.):

> *set Tabular Time Table = 0,5,15,25,30*

Table for Time intervals of Tabular BCs

The format for the BCinfoTable.txt (input file for Tabular BCs) is as follows:

1 1 0 0 0 0 0
3 2 0 0 0 0 0
5 3 0 0 0 0 0
6 3 0 -0.05 0.05 -0.05 0
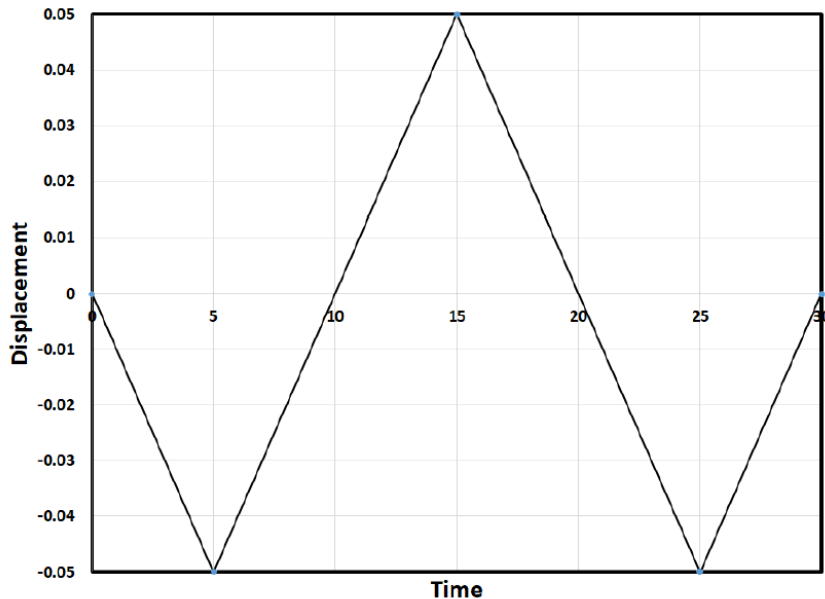
The format is very similar to the Simple BCs input file. However, there is no header line for Tabular BCs input file. The first column is the FaceID. The second column is DoF. The following columns are the displacement at the times defined in the Tabular Time Table. For example, the Tabular BCs defined above is fixing faces 1,3 and 5 throughout the simulations. Face 6 is loaded in the 3 direction, i.e., z direction in Fig. 2.

18

U.S. DEPARTMENT OF ENERGY | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Main Input file

Solver parameters

      set Time increments = 0.1
ΔT for every increment

      set Total time = 10
Maximum iterations for linear solver

Time increments depend on the total applied strain and total time. For a set of material system and loading/BCs, you need to first obtain the optimized ΔT for your simulation. As a general suggestion, your starting guess should be applying **strain increment** of 0.0001 in each step and then double it or divide it by 2.

      set Maximum linear solver iterations = 50000
Maximum iterations for linear solver

      set Relative linear solver tolerance = 1.0e-10
Relative linear solver tolerance

These two parameters controls the iterative solving the equilibrium equations using the BiCG method. The first one controls the maximum number of iteration, and the second one controls the criteria for the required residual to be satisfied. The convergence criteria threshold is the l2_norm of force vector multiplied by the relative tolerance we define here. The smaller the threshold, the more accurate the solution is but it takes longer.

Be careful, very large threshold may lead to divergence. For a set of material system and loading/BCs, you need to first obtain the optimized maximum iterations number and relative linear solver tolerance.

U.S. DEPARTMENT OF **ENERGY** | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Input microstructure

set Voxels in X direction = 20

set Voxels in Y direction = 20

set Voxels in Z direction = 20

Number of voxels in X, Y, and Z directions. This should match the number of voxels in grain ID file.

set Grain ID file name = grainID.txt

Grain IDs file name.

set Header Lines GrainID File = 20

Number of header Lines in grain ID file (these are to be skipped).

set Orientations file name = orientations.txt

Grain orientations file name

Center for PRedictive Integrated Structural Materials Science

Main Input file

Constitutive Model (Rate-Independent Model)

Multiplicative decomposition

$$\mathbf{F} = \mathbf{F}^e \mathbf{F}^p$$

Key idea of crystal plasticity

$$\mathbf{L}^p = \sum_\alpha \dot{\gamma}^\alpha \mathbf{S}^\alpha \, \mathrm{sign}(\tau^\alpha)$$

Yield surface

$$f^\alpha = |\tau^\alpha| - s^\alpha$$

Evolution of slip resistance

$$\dot{s}^\alpha = \sum_\beta h^{\alpha\beta} \dot{\gamma}^\beta$$

$$h^{\alpha\beta} = \begin{cases} h_0^\beta \left[1 - \dfrac{s^\beta}{s_s^\beta} \right]^{a^\beta} & \text{if } \alpha = \beta \; (\text{coplanar systems}) \\[2em] h_0^\beta q \left[1 - \dfrac{s^\beta}{s_s^\beta} \right]^{a^\beta} & \text{if } \alpha \neq \beta \end{cases}$$

21
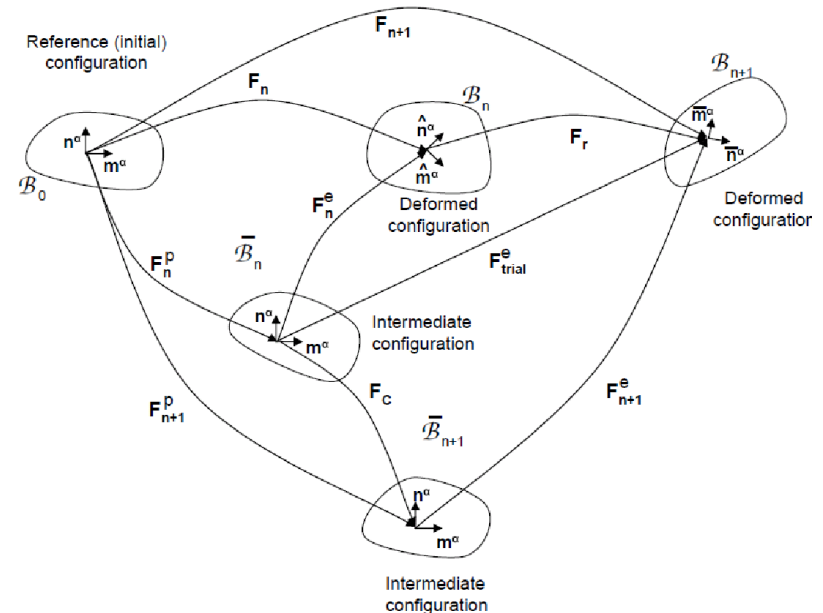
Elasticity parameters

set Elastic Stiffness row 1 = 170.0e3, 124.0e3, 124.0e3, 0, 0,0
set Elastic Stiffness row 2 = 124.0e3, 170.0e3, 124.0e3, 0, 0,0
set Elastic Stiffness row 3 = 124.0e3, 124.0e3, 170.0e3, 0, 0,0
set Elastic Stiffness row 4 = 0, 0, 0, 75.0e3, 0, 0
set Elastic Stiffness row 5 = 0, 0, 0, 0, 75.0e3, 0
set Elastic Stiffness row 6 = 0, 0, 0, 0, 0, 75.0e3

Input elastic stiffness matrix in Voigt notation as separate rows. The units are MPa.

Slip parameters

set Number of Slip Systems = 18

Number of Slip Systems

set Latent Hardening Ratio filename   = LatentHardeningRatio.txt

A text file including the latent hardening ratio matrix. This file does not have any headers. The size of the matrix is the (total systems×total systems)

set Initial Slip Resistance =0.25,0.25,0.25,10,10,10,10,10,10,10,10,10,15,15,15,15,15,15

This is the initial slip resistances: s_0^α

Main Input file

Slip parameters (Cntd.)

> *set Initial Hardening Modulus = 5.0, 5.0, 5.0, 100, 100.0, 100.0, 100.0, 100.0, 100.0, 100.0, 100.0, 100.0, 200.0, 200.0, 200.0, 200.0, 200.0, 200.0*

Initial hardening moduli of slip systems h_0^α

> *set Power Law Exponent = 0, 0, 0, 1, 1, 1, 1, 1, 1, 1, 1, 1,1, 1, 1, 1, 1, 1*

Power law coefficient in slip rate equation a^β

> *set Saturation Stress = 185.0, 185.0, 185.0, 160.0, 160.0,160.0, 160.0, 160.0, 160.0, 160.0, 160.0, 160.0, 200.0,200.0, 200.0, 200.0, 200.0, 200.0*

Saturation Stress s_s^α

> *set Slip Directions File = slipDirections.txt*

Slip Directions File

> *set Slip Normals File = slipNormals.txt*

Slip Normals File

U.S. DEPARTMENT OF ENERGY | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Uniaxial Compression in Cu with random Orientation

Let's start the simulation!!!

Moving to the fcc compression uniaxial folder with random texture:

cd plasticity/applications/crystalPlasticity/fcc/FCC_RandomOrientationBlock/

Open the main input file with your preferred text editor (I'm using vim):

Vim prm.prm

This is a cubic of unit length, with 5*8*10 grains, each grain is modeled with an FE mesh.

Let's change the visualization setup, go to line 52, change false to true as:

set Write Output   = true

Let's change the visualization skip output steps to generate the visualization results every 25 time steps, go to line 59, change 10000 to 25 as:

set Skip Output Steps   = 25

ΔT=0.005; Total time=5

Save the changes and exit. In Vim, you can click on esc, and then type **:wq** and then hit enter.

Center for PRedictive Integrated Structural Materials Science

Uniaxial Compression in Cu with random Orientation

Now, open the BCs file to explore as:

> *vim boundaryconditions.txt*

(you can type a few character of the file and then hit Tab for auto completion)

Symmetry boundary conditions are imposed on adjacent x, y, and z faces and the uniaxial compression is applied in the z direction on the opposite z face.

The final displacement in z direction is -0.05. Considering the unit length of cube, the final strain is -5%.

$\Delta T = 0.005$; Total time=5, $\Delta\varepsilon = \frac{0.005\times\%5}{5} = 0.00005$

Now, start the simulation as:

> *mpirun -n 2 ../../main prm.prm*

Number of processors depends on the capabilities of your system.

Kill the simulation after 50 increments using ctrl+c.

```
# Header lines = 2
# FaceID DoF FinalDisplacement
1 1 0
3 2 0
5 3 0
6 3 -0.05
~
~
~
```

FaceIDs
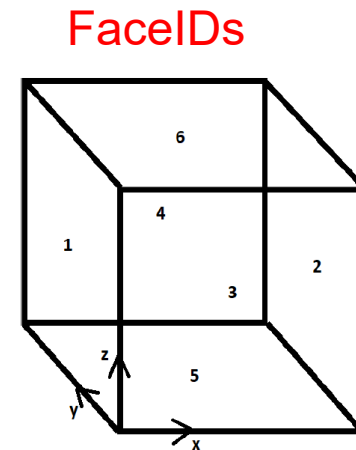
Uniaxial Compression in Cu with random Orientation

Go to the results folder:

cd results

Check the list of files:

ls

stressstrain.txt: file is generated for each step which includes the average Green-Lagrange strain tensor, Cauchy stress tensor, twin activity volume (TwinRealVF), twin volume fraction (TwinMade), and slip activity (SlipTotal).

solution-N.pvtu: The visualization file of **nodal displacements.** You can open the file with paraview. **N** is increment number that output is generated. The .vtu files are small block building the .pvtu file which is generated for each processor for specific time step.

projectedFields-N.pvtu: The visualization file you need to open with paraview for **variables defined in the outputs section of prm.prm**. **N** is increment number that output is generated. The .vtu files are small block building the .pvtu file which is generated for each processor for specific time step. Unlike the solution (nodel displacements) which are directly obtained for each node, these variables were calculated for each integration point. The nodal values need then to be calculated by solving a linear equation.
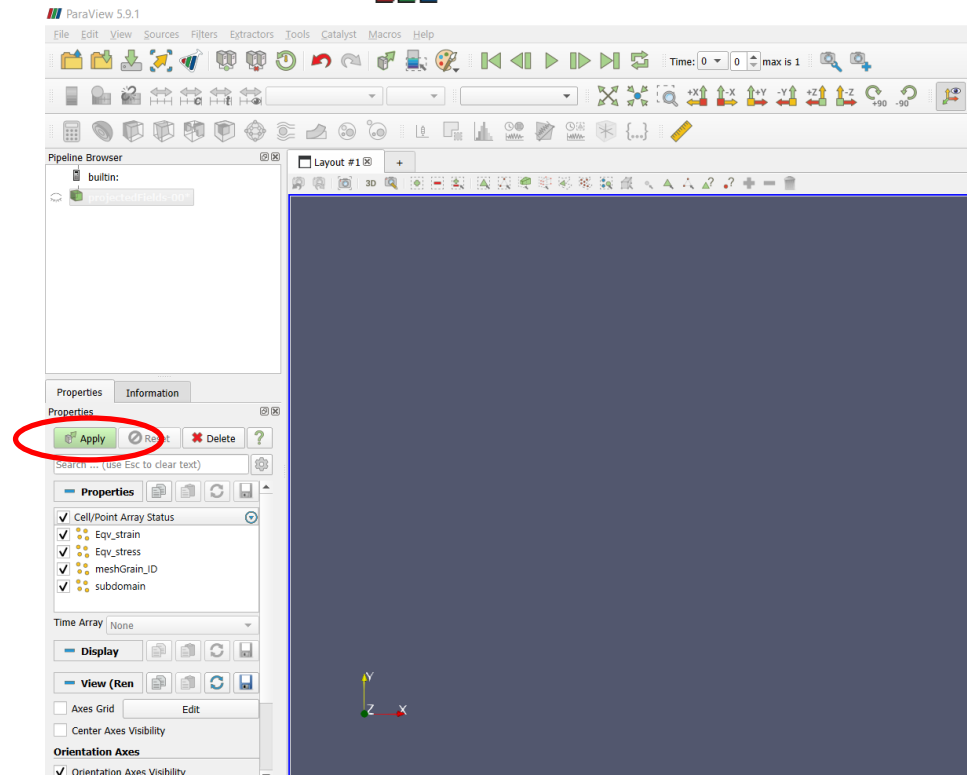
Uniaxial Compression in Cu with random Orientation

To visualize the outputs from CPFE we can go through the following steps using **Paraview**. (https://www.paraview.org/download/
or *sudo apt install paraview*)

• Browse to **ParaView-5.91.0-Windows-64bit** -> **bin** in the main directory containing the Paraview installation, and double-click on the Paraview icon

• Then click on **File-> Open**, and choose the file with **projectedFields-...pvtu** extension that you wish to visualize. This will load the file into Paraview.
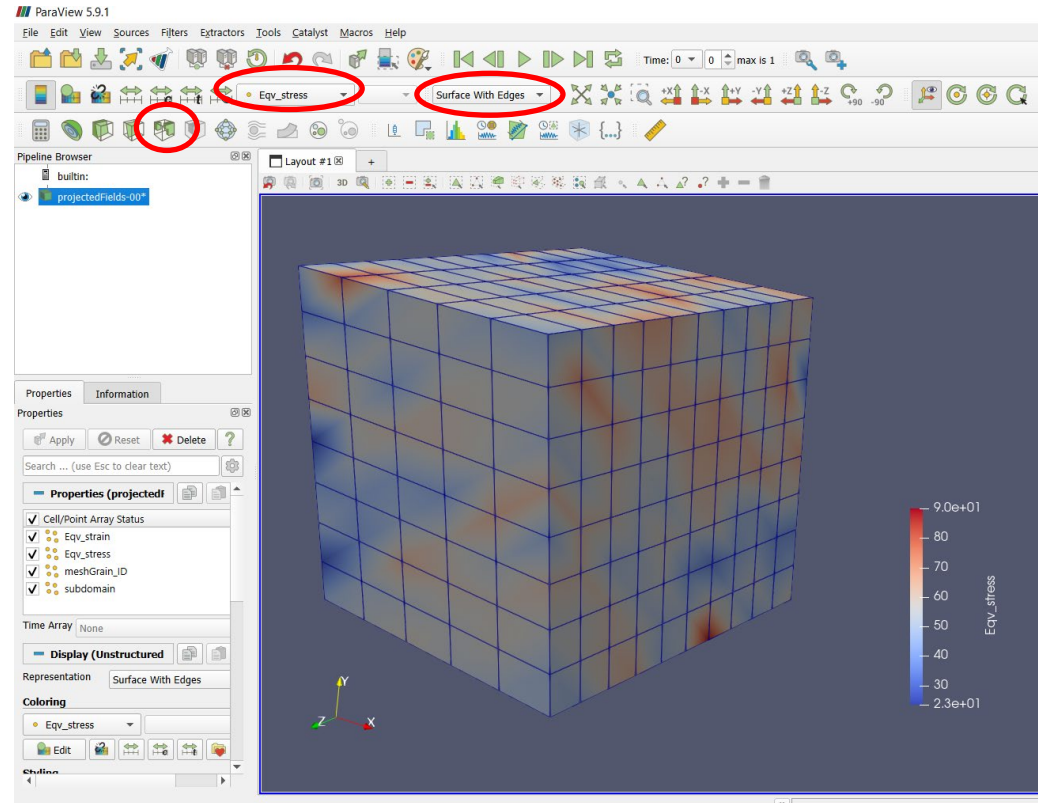
• The filename is visible in a small section to the left hand side of the window. Click on **Apply** in the properties and make sure that the file is visible.

Uniaxial Compression in Cu with random Orientation

In the toolbar, there is a drop down menu that reads **Solid Color**. Click on that menu to choose a particular variable of choice. Adjacent to it is another drop down men that reads **Outline**. Click on it and choose **Surface With Edges**. The variation of that variable in space is now visualized with a colorbar for reference. The viewer should look something like below.

•The same can be repeated for other variables as well.

•Now you can delete this data by right clicking on the name of projectedFields-00* and select delete.

Visualizing a deforming mesh

•The deformed mesh can be visualized using the solution pvtu files and the Warp By Vector filter in Paraview by following the steps below.

•Load Paraview and load a **solution** file with **.pvtu** extension, and then click on apply to open it.

•Choose the field variable as **u**(for displacement), the **Z** component and select **Surface** for representation.

•Click on the **Filters** dropdown in the menu bar, click on **Alphabetical** and click on the **Warp By Vector filter**.

29
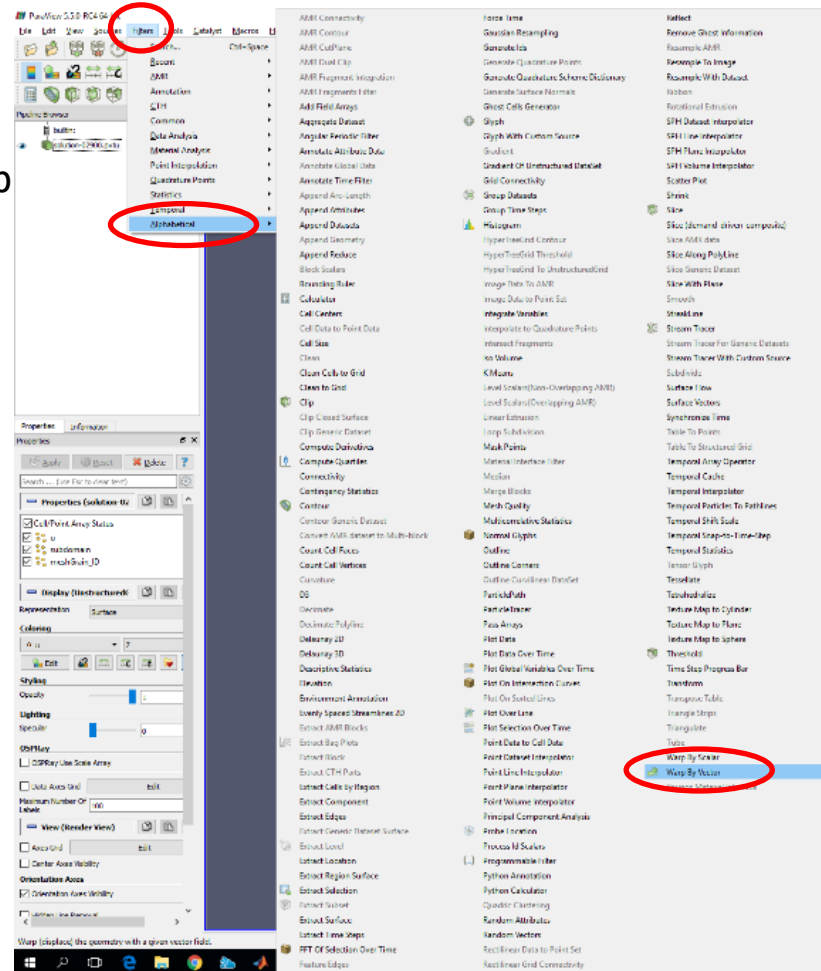
Uniaxial Compression in Cu with random Orientation

Visualizing a deforming mesh (Cntd.)

•This appears as a filter in the pipeline below the solution file. Then click on **Apply** button, and the deformed mesh will be visible in the viewer.

•Change the **Scale Factor** to 50.

•You can march in time by clicking on the **green arrow** (if you have more than one steps output were generated).

Center for PRedictive Integrated Structural Materials Science
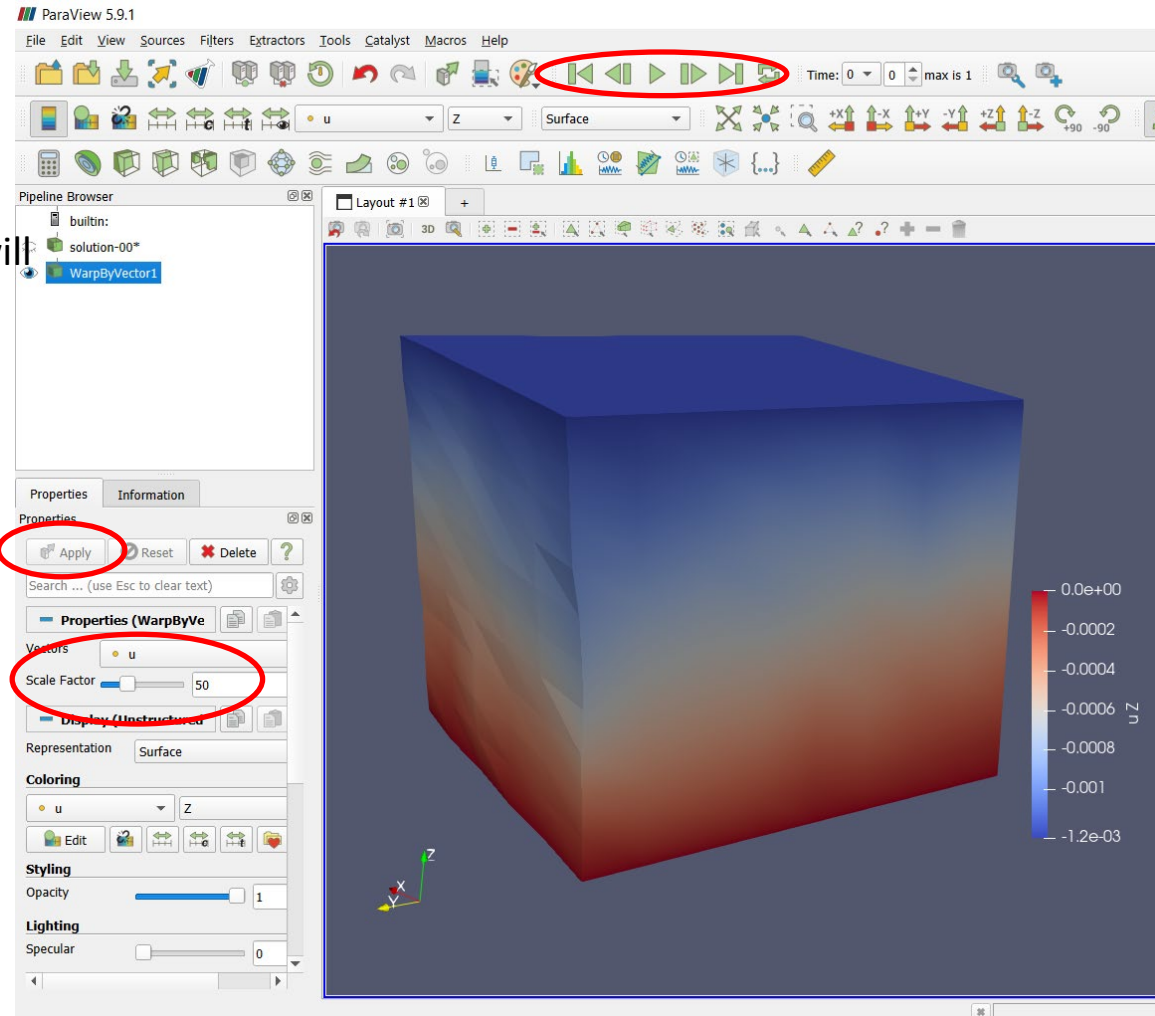
Uniaxial Compression in Cu with random Orientation

Stress-strain curve

- Open MS Excel (ubuntu libreoffice calc).
- Click on File -> Open, and then choose the file **stressstrain.txt**. You might need to change the default from **All Excel Files** to **All Files** in the search box.

.

- Text Import wizard will be shown next. Click on Finish.

- Select the Ezz column (clicking on the column C) hold ctrl button and select the Tzz column by clicking on the I column. Both columns should be selected now.
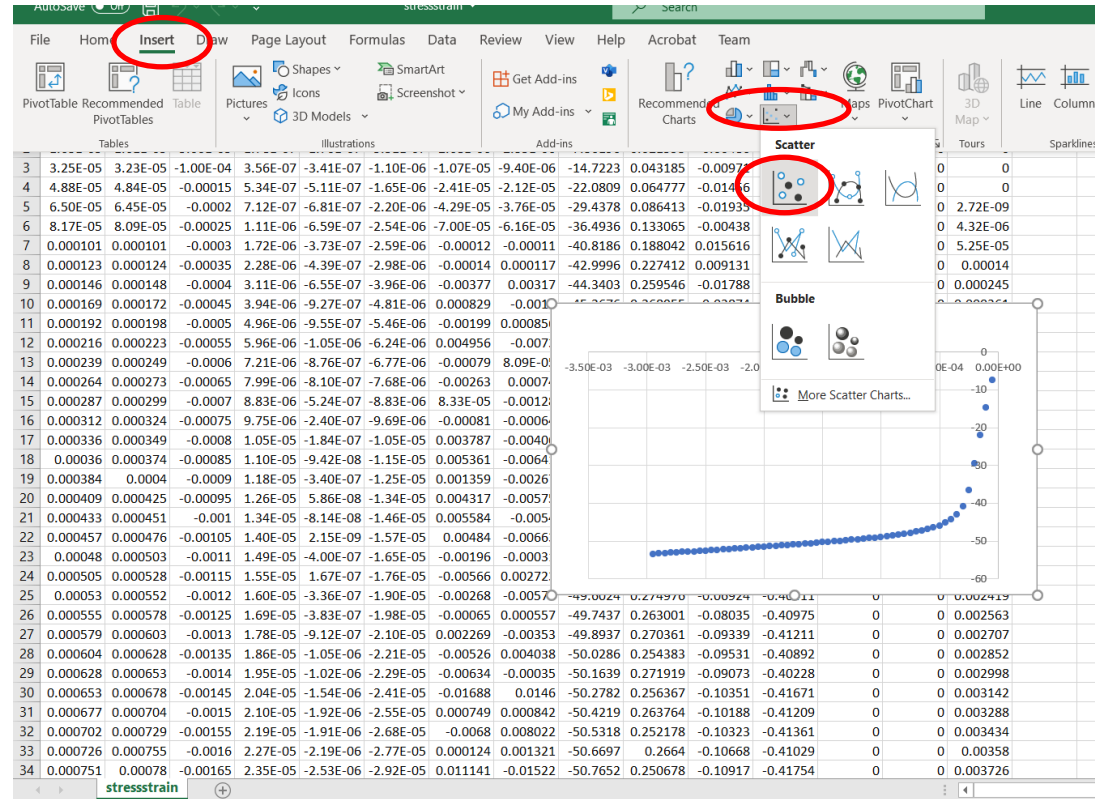
- Click on the **Insert** tab, choose the **Scatter** option and **Scatter** suboption.

- You can right click on the plot and **move** it into a **new Sheet**.

- In the case of ubuntu libreoffice calc, you can select insert->Chart->XY (Scatter).

Center for PRedictive Integrated Structural Materials Science

Uniaxial Compression in Cu with random Orientation

Playing with the input files:

1) Change the loading to uniaxial tension.

2) Use a more refined mesh:

> *set Refine factor = 2*
> *set Skip Output Steps = 1*

3) Output QuadratureOutput and investigate the output file with just grainID, position and orientation:

> *set Refine factor = 0*
> *set Skip Output Steps = 25*
> *set Write Quadrature Output = true*
> *set Skip Quadrature Output Steps = 25*

If you are in the FCC_RandomOrientationBlock folder as your current directory:

> *vim ../../../../src/materialModels/crystalPlasticity/updateAfterIncrement.cc*

Commenting all lines in the QuadratureOutput region except those you want:

Recompile the code using "*make release*" in the plasticity/applications/crystalPlasticity/fcc folder.
> *cd fcc/FCC_RandomOrientationBlock*
> *mpirun -n 2 ../../main prm.prm*

U.S. DEPARTMENT OF **ENERGY** | Office of Science

UNIVERSITY OF MICHIGAN

Center for PRedictive Integrated Structural Materials Science

PRiSMS

Playing with the input files:

4) Applying a cyclic loading using the Tabular BCs (session 2).

.