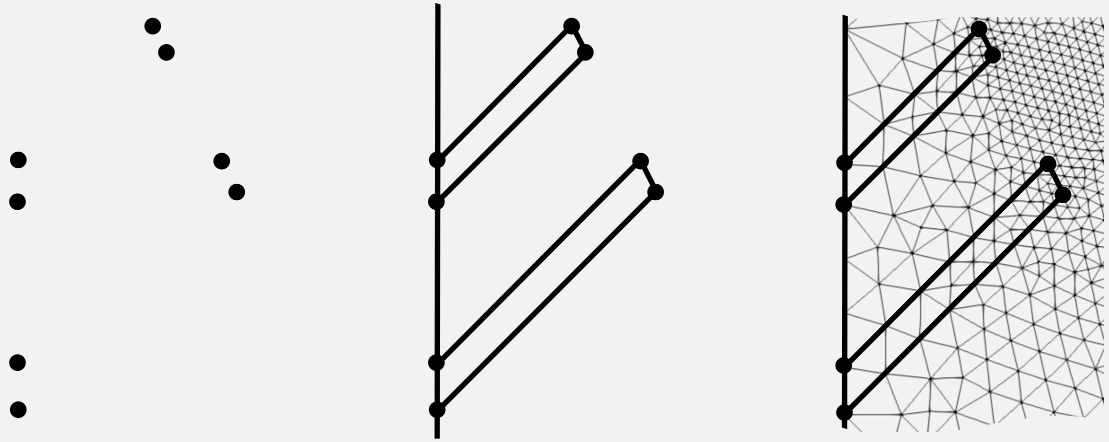
Input

Domain Division

vertices coordinates → vertices indices in each domain → Mesh generation (p.3)



(Plastic) Right Cauchy-Green Tensor Field

domain_id	$[\mathrm{C}^{\mathrm{p}}_{\mathrm{ij}}]$
⋮	⋮
6	$\begin{bmatrix} 1 & 0.8 \\ 0.8 & 1.8 \end{bmatrix}$
⋮	⋮

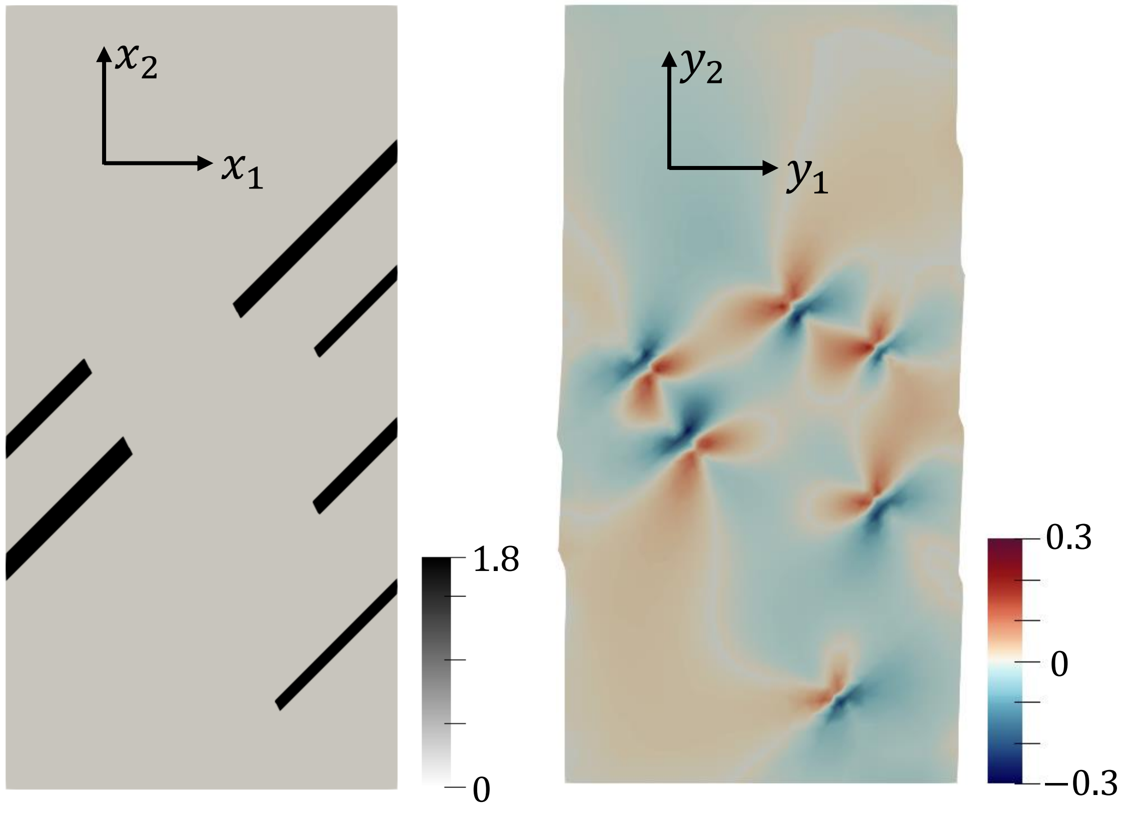
Output

A vtk file containing various fields

Output.vtk

$$\min_{\mathbf{y}(\mathbf{x})} \mathcal{F}[\mathbf{y}(\mathbf{x}), \mathbf{C}^{\mathrm{p}}(\mathbf{x})]$$

$\mathcal{F}$: elastic energy

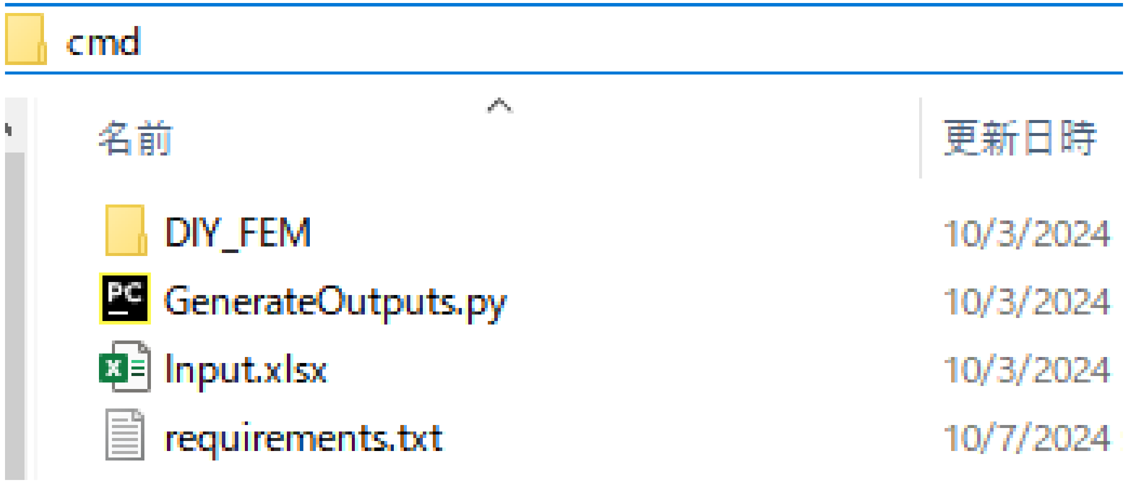


Ref. config. colored with $\mathrm{C}^{\mathrm{p}}_{12}(\mathbf{x})$

Deformed config. colored with shear stress σ_{12}

Install libraries | Input of parameters | Start

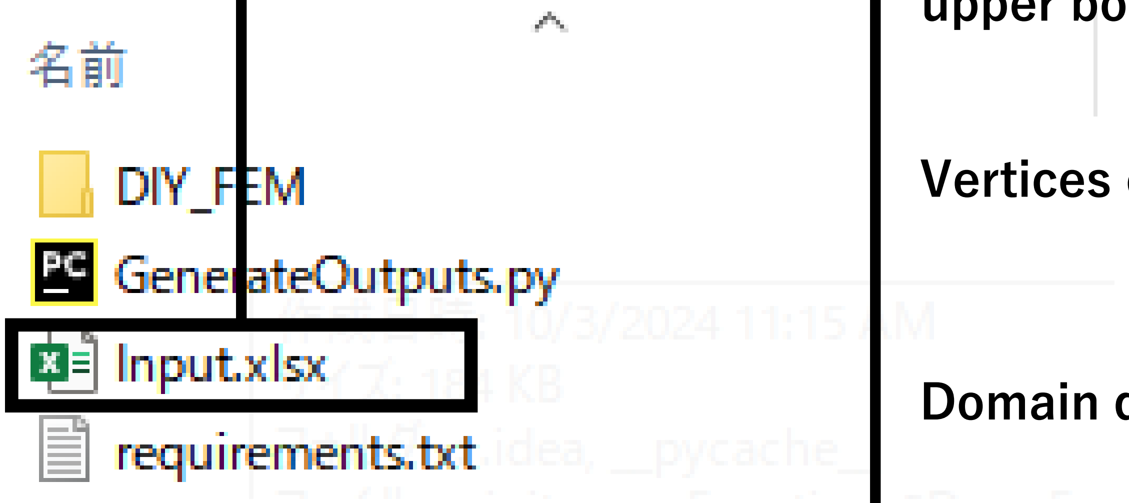
- Enter the ‘Sample’ folder, type ‘cmd’ in the navigation bar and press ‘Enter.’



- After pressing ‘Enter,’ type ‘pip install –r requirements.txt’ on the command line that pops up to complete the installation of the required libraries.

Install libraries → **Input of parameters** → Start

- Open ‘Input.xlsx’ and enter various parameters required to solve stress equilibrium equations.



Elasticity

Shear modulus [GPa]	Poisson’s ratio
21	0.26

Elasticity | Bound_Condition

Vertical displacement at the upper boundary

Vertical Loading Displacement [um]	Output File Name
−0.2	Output_2
0	Output_3

Elasticity | Bound_Condition | Vert

Vertices coordinates

x_1 [um]	x_2 [um]	MeshSize [um]
0	5	1
3	2	0.2
3.14	2.26	0.2

Vertices_Coords | Domain_

Domain division

	vertex index of P0 at domain bound.	vertex index of P1 at domain bound.	vertex ir
domain0	0	1	
domain1	4	5	
domain2	8	9	
domain3	12	13	

Vertices_Coords | Domain_VertID | Domain_Cp

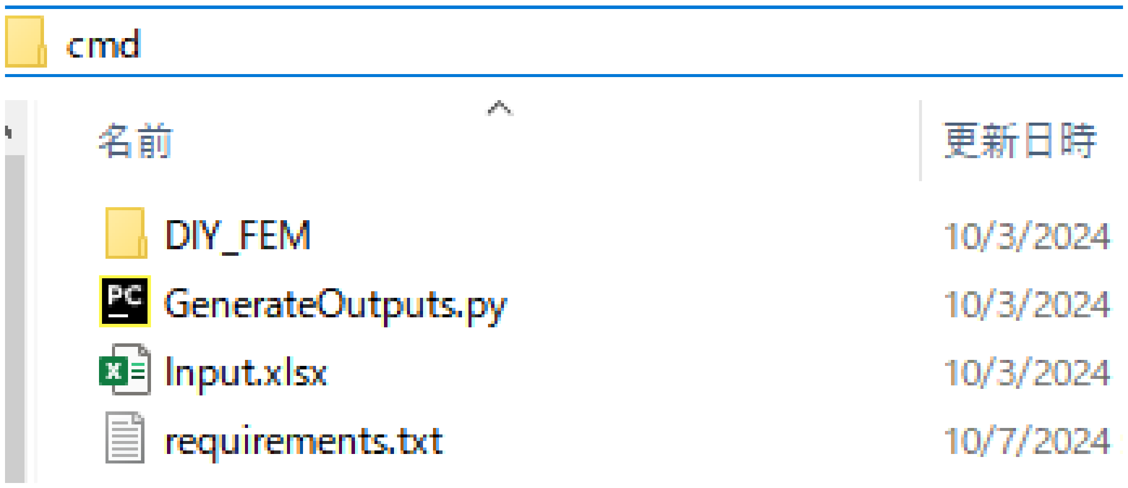
Plastic right Cachy-Green tensor C^p in each domain

	Cp_11	Cp_12	Cp_22
domain0	1.78	0.18	0.58
domain1	1.78	0.18	0.58
domain2	1.78	0.18	0.58

Vertices_Coords | Domain_Cp | Domain_VertID

Install libraries | Input of parameters | **Start**

- Again, enter the ‘Sample’ folder, type ‘cmd’ in the navigation bar and press ‘Enter.’



- After pressing ‘Enter,’ type ‘python GenerateOutputs.py’ on the command line that pops up to start to solve the stress equilibrium equations.

- Outputs (With names defined in ‘Bound_condition’ in ‘input.xlsx.’ Each file corresponds to one bound. condition.):