Tutorial: From TexGen to Finite Element Analysis

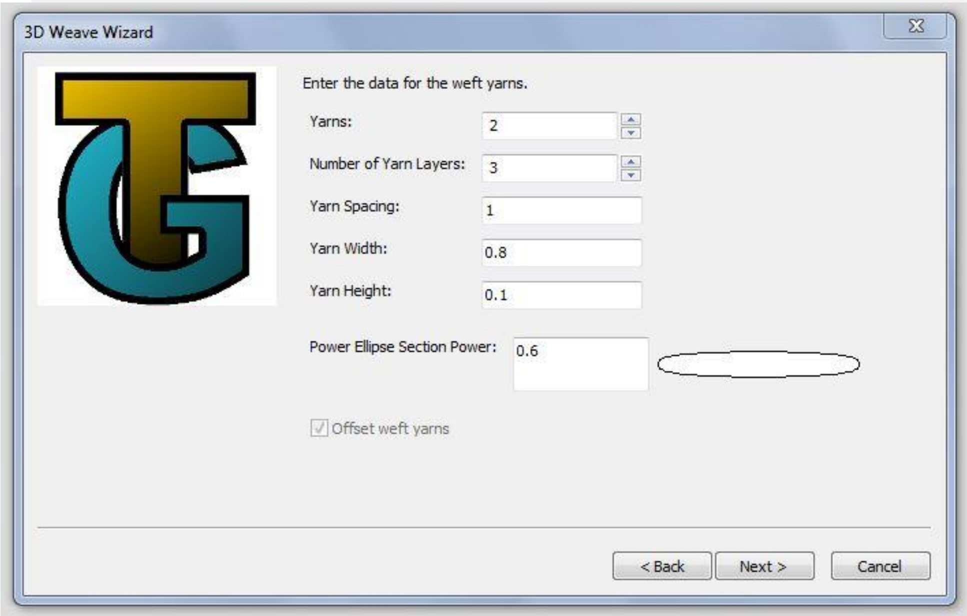
Xuesen Zeng

Division of Materials, Mechanics & Structures
Faculty of Engineering
University of Nottingham
University Park
Nottingham
UK

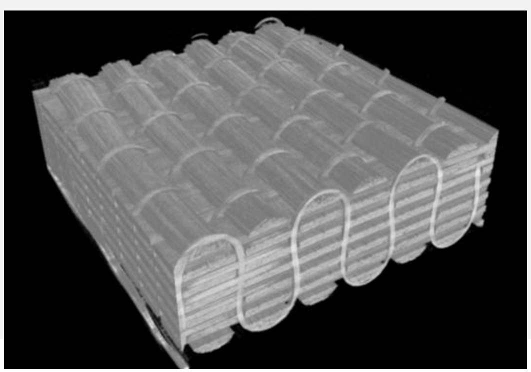
TexGen FE Tutorial

- Geometric modelling of textile composites
- Automation of model generation
- Parameterized geometry definition
- Realistic representation of fibre architecture
- Pre-process for ABAQUS
- Mesh
- Boundary conditions
- Load cases
- Extract elastic constants from simulation
- Pre-process for ANSYS CFX
- Mesh
- Boundary conditions
- Permeability calculation

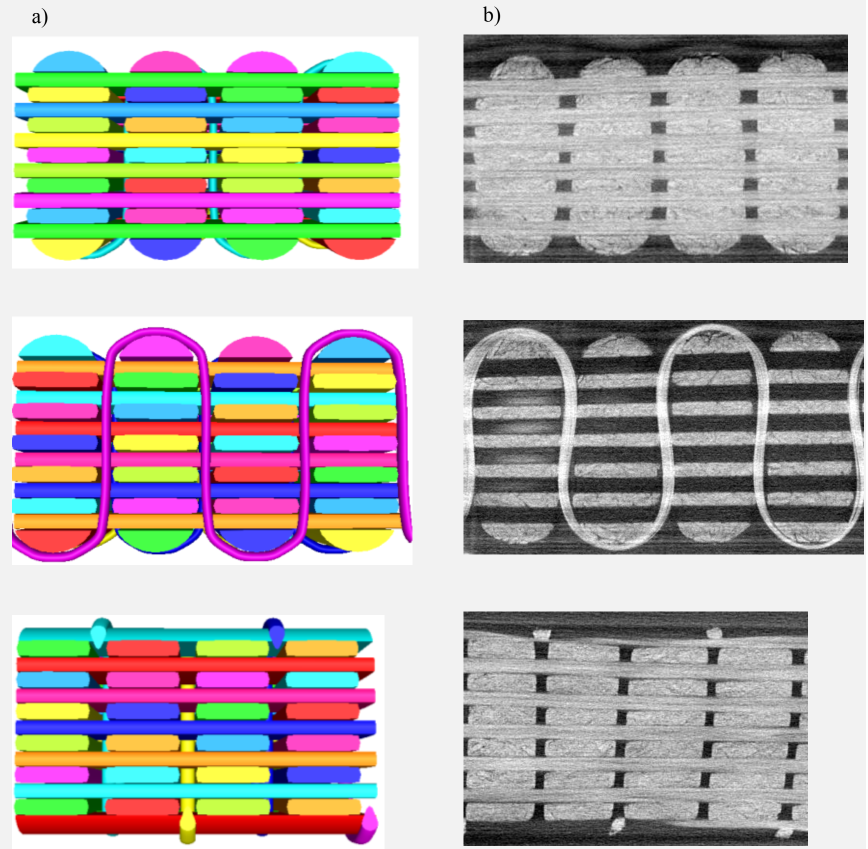
Textile geometry modelling



TexGen 3D Weave Wizard

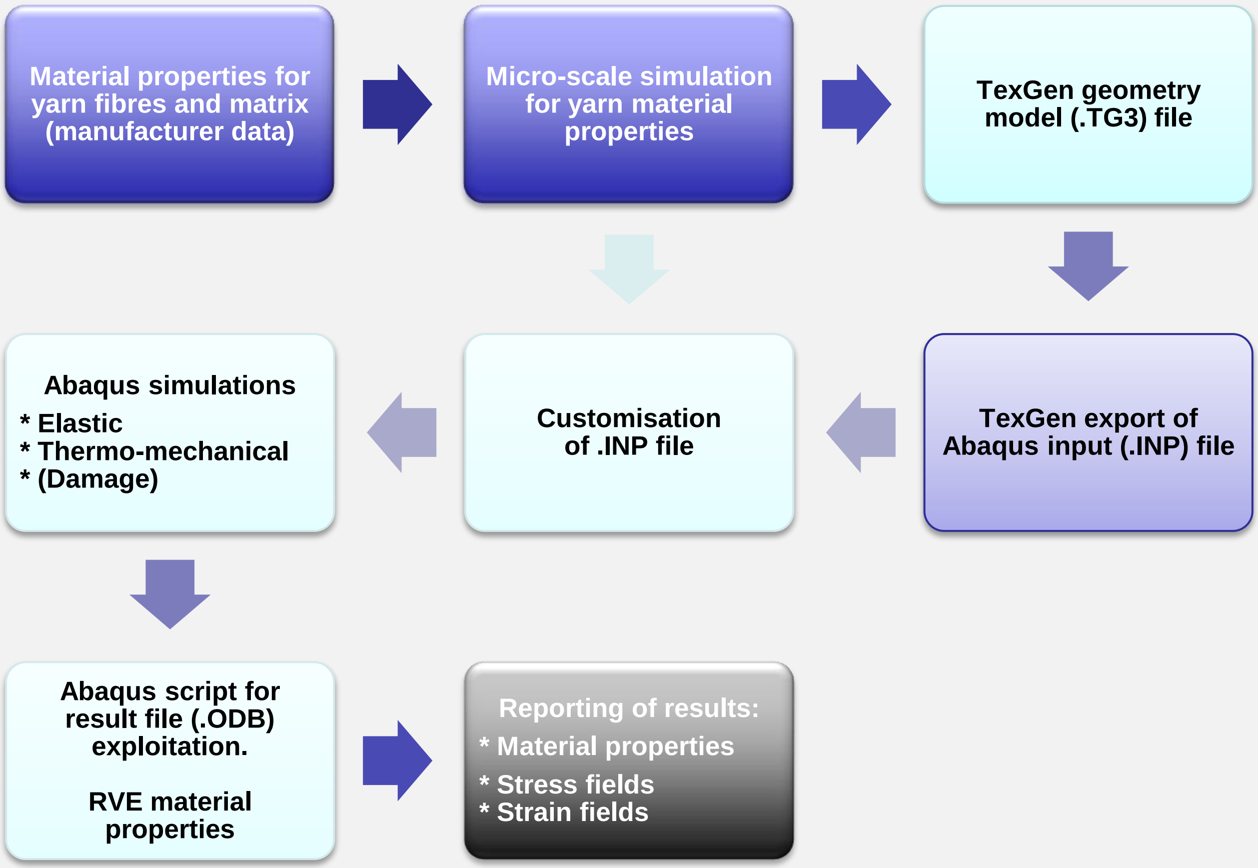


3Tex orthogonal weave scanned by x-ray computed tomography (µCT)



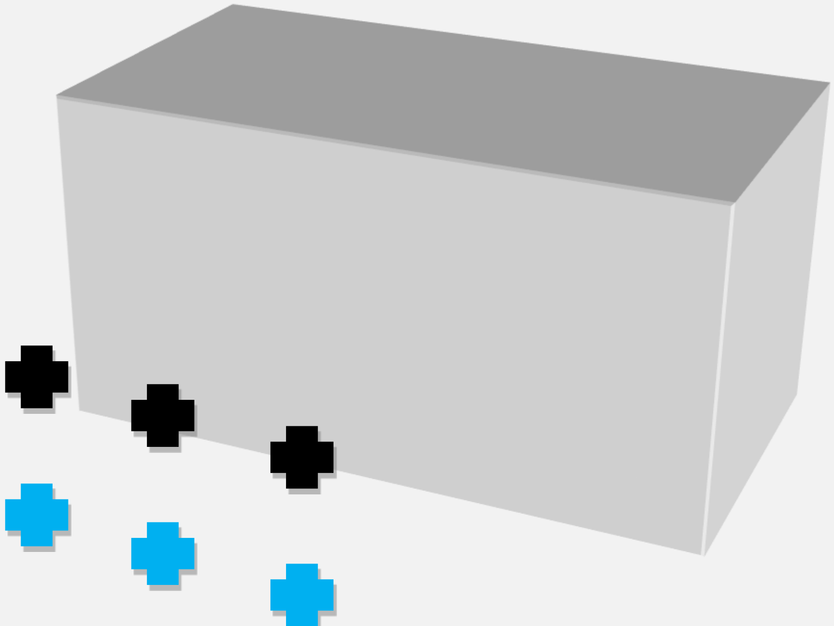
Orthogonal 3D carbon fabric: a) weft and warp views of the TexGen model; b) µCT images of corresponding weft and warp cross-sections.

Workflow for finite element elastic analysis



Material continuum assumption

- Displacement field under overall macroscopic strains:
  - 3 direct strains
  - 3 shear strains

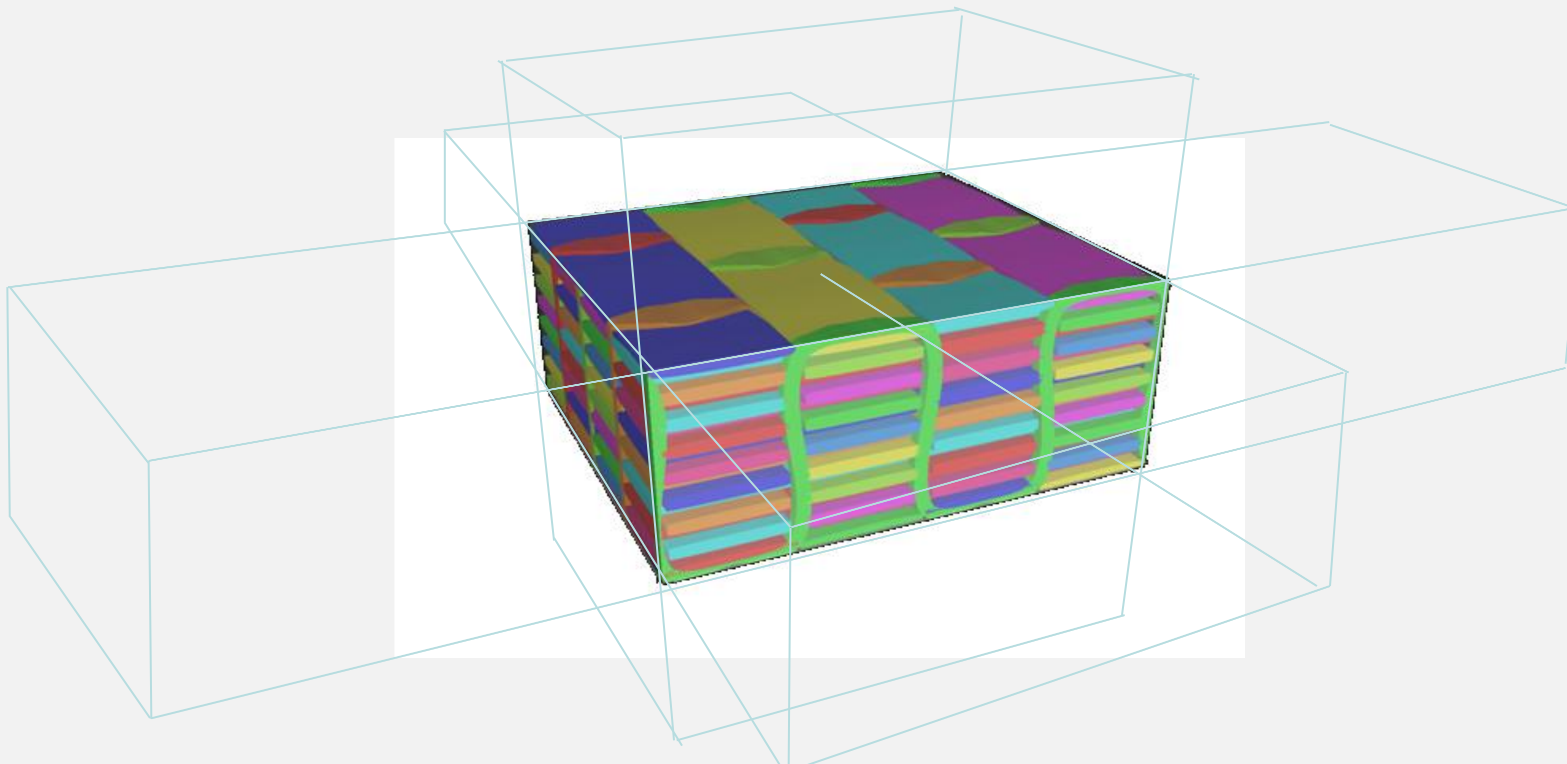


- “Correct” BC approach: equivalent continuum, tie to 6 “driver points”
- Recover relationship between stresses and strains (elastic props)

Periodic BCs: 3D and “untied”

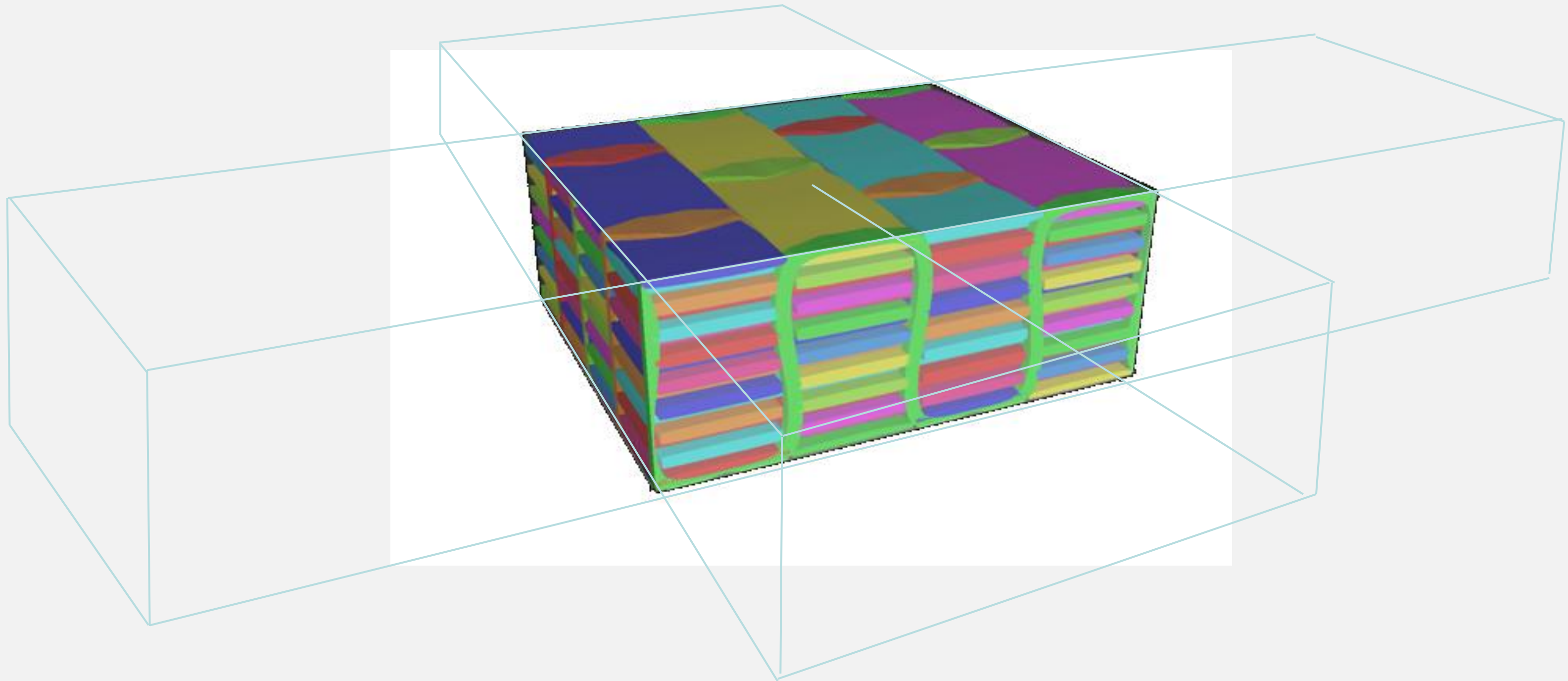
The University of Nottingham
UNITED KINGDOM · CHINA · MALAYSIA

- “Correct” BC approach of Li and Wongsto
- Full 3D boundary conditions: infinitely repeating 3D continuum
- Not true for tests!



Periodic BCs: 3D and “untied”

- Also tried “untying” the through-thickness constraints
- Represents repeating units only in-plane, free surfaces top and bottom
- Differences in results actually insignificant



Evaluation of yarn component properties

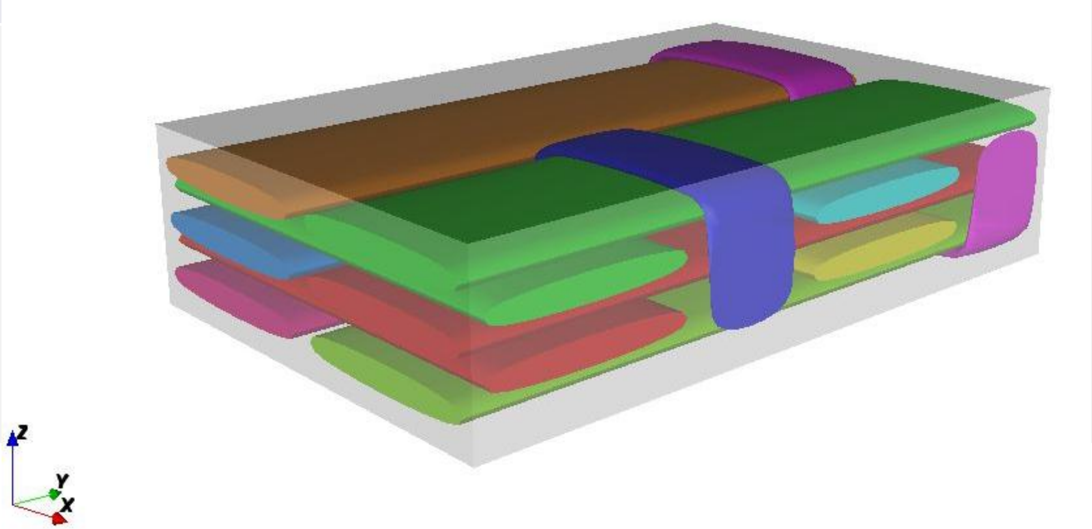
Fibre properties (HTS40 F13)

	E_x	v
Fibre	238.6	0.20
Resin	3.1	0.35

Input yarn material properties:

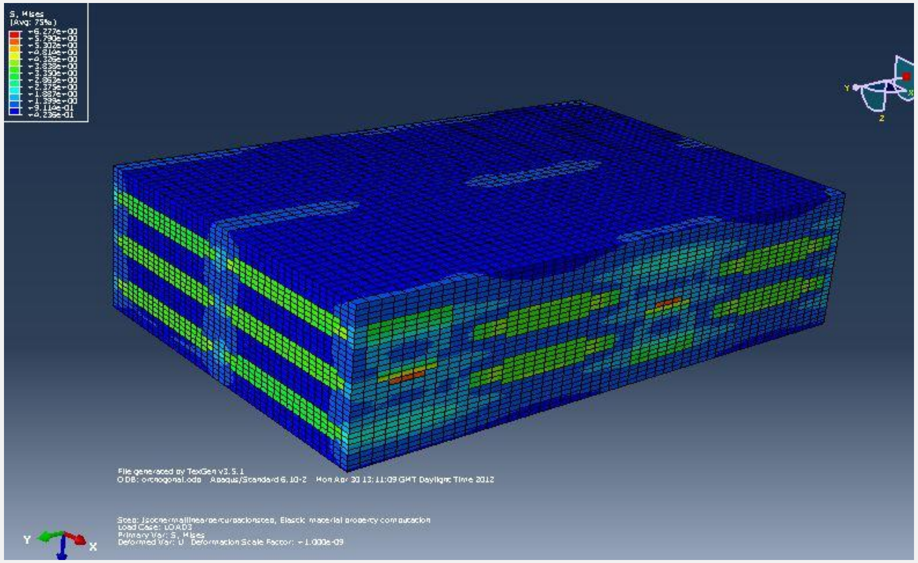
7micron fibre at 76% vol.
UD hexagonal packing

E_1	E_2	E_3	v_{13}	v_{12}	v_{32}	G_{12}	G_{23}	G_{31}
183.1	9.67	9.67	0.23	0.23	0.43	5.66	3.37	5.66



FE prediction

The University of Nottingham
UNITED KINGDOM · CHINA · MALAYSIA



Mesh size	E_1	E_2	v_{12}	v_{21}	G_{12}
40-50-30	34	64	0.03	0.05	3.06

$$E = \frac{\sigma}{\varepsilon} = \frac{F}{V\varepsilon} = \frac{1}{\varepsilon}$$

$$v_{xy} = \frac{\varepsilon_y}{\varepsilon_x}$$

Step 1: export ABAQUS .inp file

Step 2: lauch ABAQUS Solver with command line:
ABAQUS job=… ask_delete=off interactive

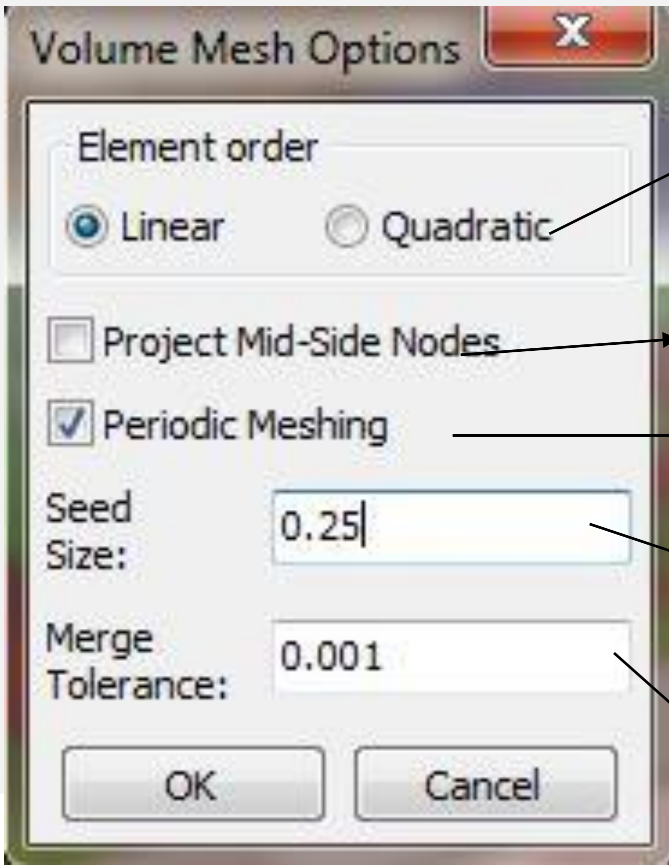
Step 3: parce the results from .odb file: Creat XY data -> ODB field output -> select Unique Nodal and U: spatial displacement -> go to Tab: Elements/nodes, select …CONSTRAINTDRIVER0, 1, 3 -> SAVE ->Go to main tab Report, XY… to export .rpt file

Step 4: calculate the elastic constants: Refer to Result.xls.

Tutorial - Flow simulation using TexGen and CFX

1. Exporting volume mesh from TexGen

Open a TexGen model ready in TexGen window. Click “File -> Export -> Volume Mesh…”, a dialog window appears as below. Once the parameters are set and click ok, a volume mesh file .vtk is created.



1.a This controls the number of element nodes. Linear results in 4 noded tetrahedrons while Quadratic results in 10 noded tetrahedrons.

1.b Option for Quadratic meshing only.

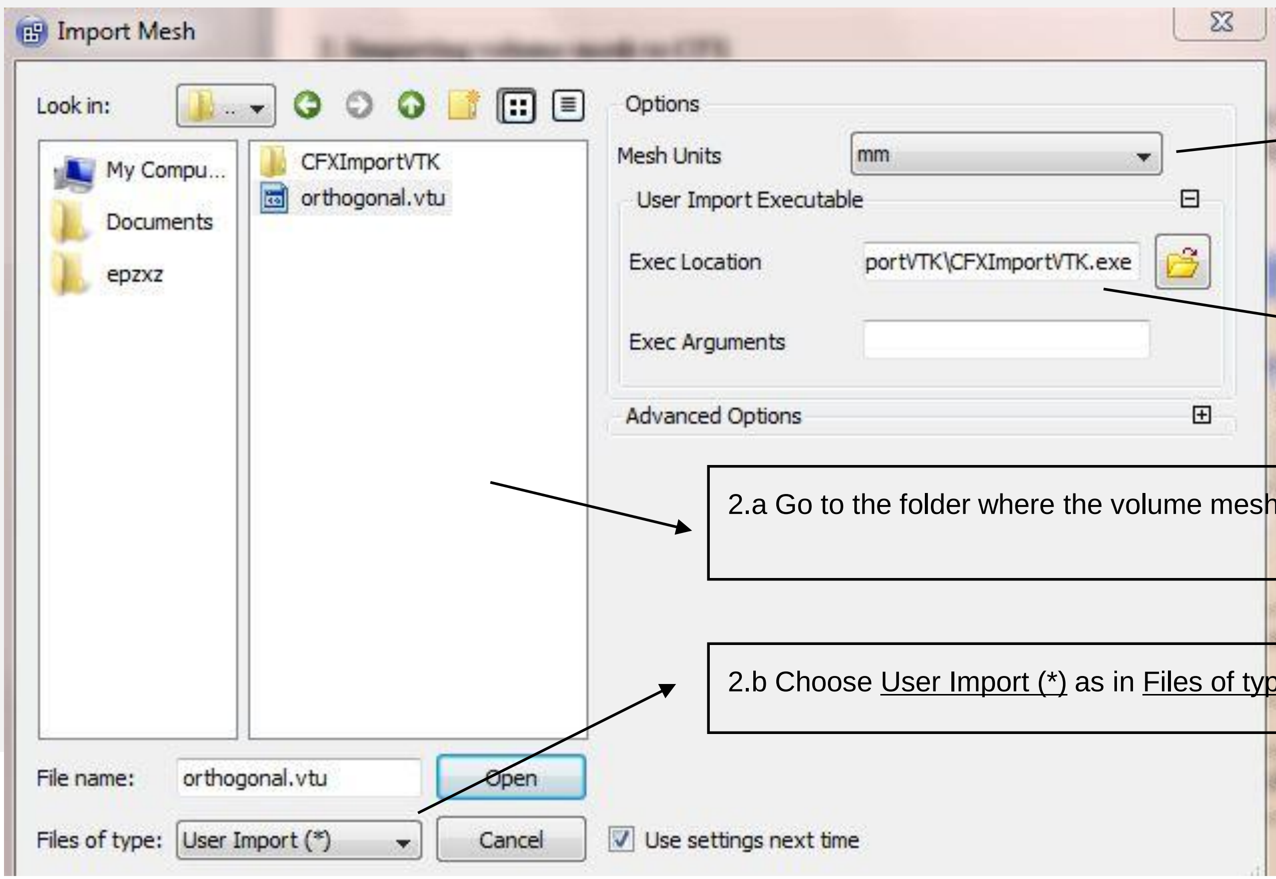
1.c Create the matching meshes on opposite faces to ensure periodic interface.

1.d Controls the mesh density and it approximates the edge length of elements.

1.e Any gap between yarns smaller than this tolerance will be merged to avoid bad quality elements.

2. Importing volume mesh to CFX

Launch ANSYS CFX -> CFX Pre 13.0. Go to File and then New Case, choose General as Simulation Type and click OK. Go to File -> Import -> Mesh, an Import Mesh window comes up. After the steps 2.a-d, click Open. The mesh is now imported.



2.a Go to the folder where the volume mesh .vtu file is allocated

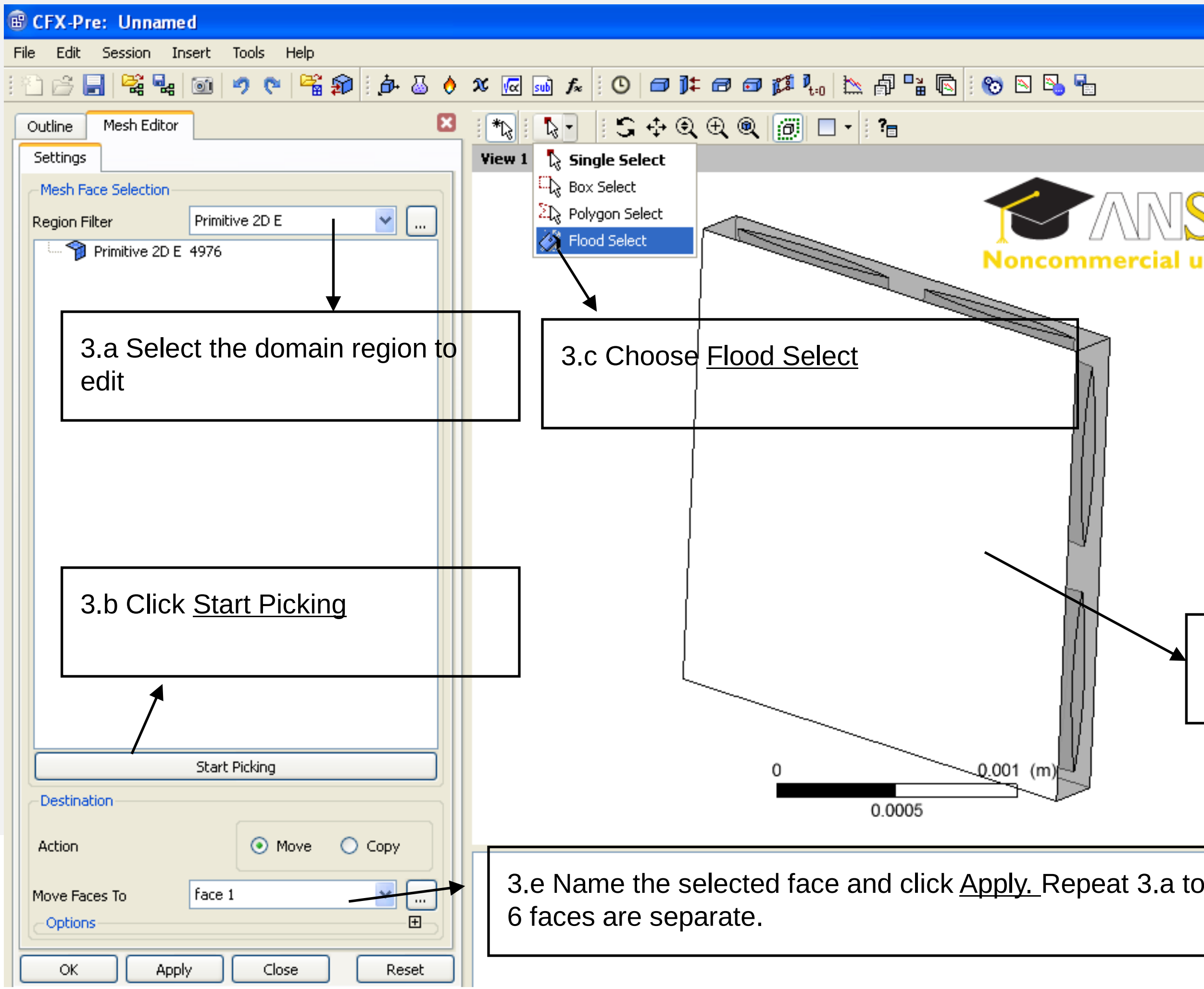
2.b Choose User Import (*) as in Files of type

2.c Choose mm as mesh unit

2.d Choose CFXImportVTK.exe which is provide by this Workshop.

3. Editing mesh in CFX

To organise the domain surfaces mesh into 6 separate regions which will be used for defining boundary conditions.



3.a Select the domain region to edit

3.b Click Start Picking

3.c Choose Flood Select

3.d Click on the face to be moved

3.e Name the selected face and click Apply. Repeat 3.a to 3.e till all 6 faces are separate.

4. Defining domains in CFX

To define the **fluid domain** for the gaps between yarns only.

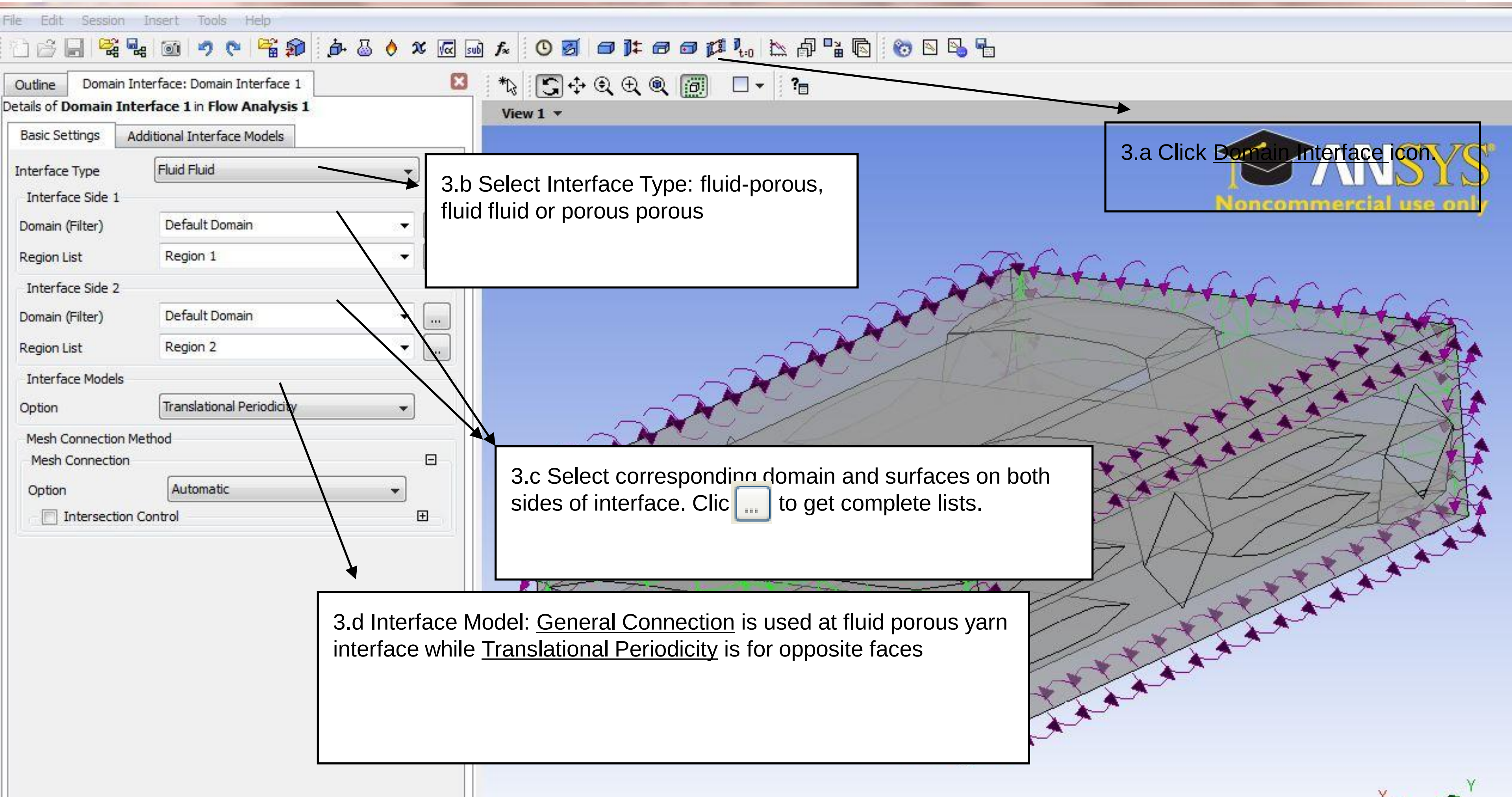
New domains are created by selecting **Insert** > **Domain** or clicking the *Domains* icon. Note that creation of domains from the menu bar or toolbar may subsequently require selection of the appropriate analysis type. Domains can also be created by right-clicking the appropriate analysis type in the **Outline** view.

For the fluid domain, the default setting is recommend in the **Basic Settings Tab**, and in **Fluid Models Tab** a **None (Laminar)** option is chosen for Turbulence Model.

Click OK/Apply

5. Domain interface

2 sets of interfaces is to be defined. 2 sets of **fluid fluid** interface with translational periodicity at opposite faces of the domain;



6. Boundary conditions

To define pressure drop on two faces along Z direction. Click Boundary icon 

In **Basic Settings Tab**, Boundary Type is chosen as Opening. Since the boundary surface is very close to fabric, Opening option allows the fluid to cross the boundary surface in either direction. Choose the appropriate region for Location.

In **Boundary Details Tab,** set the relative pressure in Pa.

7. Initialisation

Click the global initialisation icon 

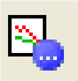
to assign all the unspecified solution field values (an Initial Guess).

8. Solver Control

Right click Solver Control in **Outline,** select Edit.

In **Basic Settings Tab, Convergence control,** set the maximum iteration to 1000. **Convergence Criteria:** Set **Residual Type** as **RMS, Residual Target** to 1E-5.

9. Run Solver and Monitor

To start the analysis by clicking 

The simulation will terminate once the convergence criteria is satisfied or it reaches the maximum iterations.

10. CFD results

In CFX-Solver Monitor Window, when the simulation terminates normally, check Post-Process Results and click OK or launch CFD-Post by click Icon 

How to calculate fabric permeability?
Go to **Calculator Tab,** choose **Function Calculator.** To obtain the values of flow cross-section area (*A*), areaAve pressure at the inlet (*P1*) and at the outlet (*P2*), mass flow rate at either inlet or outlet (m'). Given the air at 25c, its density (ρ) is 1.185kg/m^3, and its dynamic viscosity (μ) is 1.831E-5 Pa s. L is the unit cell thickness. Fabric permeability (*k*) follows Darcy's law

$$\frac{m'}{\rho} = \frac{kA}{\mu}\frac{P_1 - P_2}{L}$$

End