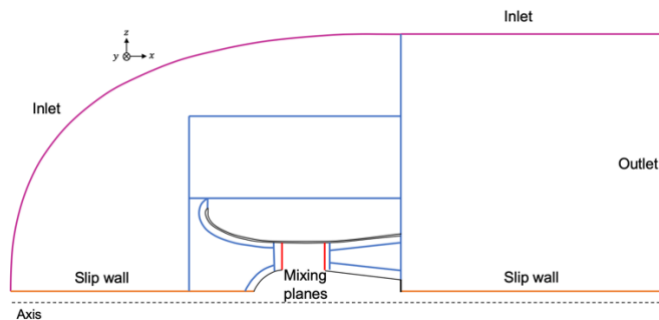


PointWise Mesh Generation

Step 1: Make 2D mesh of geometry

- Outline the geometry you want to mesh in 2D, including farfield blocks which boundary conditions will later be applied to (the exact boundary conditions, inlet/outlet/slip wall etc., can be applied in Pointwise but often easier to do this in MATLAB after the mesh is completed)



- TURBOSTREAM works in polar coordinates so the slip wall must be a slight distance above the x-axis, for example point (0,0,0) should be placed at (0,0,1)
- CFD mesh can be structured or unstructured, for eVTOL rig CFD the mesh is structured
- Nodes can be clustered near regions of interest using the distribute tool, for example clustering near duct walls
- Once the mesh is generated, the quality of different properties of the mesh should be examined using:
 - Examine > area ratio >
 - Examine > skewness > max of around 0.5
- Use PointWise Y+ calculator to calculate number of cells in boundary layers
 - Reference length = length of diameter
- Improve cell distribution
 - Click on line > grid > distribute > click on arrow to click which side you want to change cell distribution > change distribution by typing in spacing number and clicking apply
- Improve cell distribution further
 - Highlight domains > grid > solve > run
 - Under edge attributes tab in solve, change boundary condition to floating which allows connectors to move to improve cell distribution further, again click solve tab and run
- Settings shell
 - Can change wall thickness
 - 100% infill is basically solid
- Mask tool lets you only click on things you want to click on

Step 2: Make 3D mesh

For the eVTOL inlet and outlet duct mesh, the flow is assumed axisymmetric and so the 2D mesh is rotated to form a sector.

- Create > Extrude > rotate
- Choose angle of sector you want to model (10 degrees say with 10 steps run)
- Choose the axis and point to rotate around (XYZ rotate around 0 0 0, not 0 0 1)

Step 3: Export mesh

- Ensure solver is set to gridgen generic
 - CAE > select solver
- Ensure all blocks are aligned in left handed coord system as this is what TURBOSTREAM is in
 - Highlight all blocks > Edit > orient > align so all the same (can click left hand alignment button)
- Add boundary conditions
 - CAE> Set boundary conditions > highlight the part you want to set a boundary condition for and label what it is, i.e. inlet/outlet/free stream etc.
 - This part can be skipped if setting BC in MATLAB
- The CAE (computer aided engineering) tab lets you export your complete grid
- Export as gridgen generic into three files

Step 4: Convert Pointwise files to TURBOSTREAM input files

- The files exported from Pointwise are in '.inp', '.rot' and '.dat' format
- Copy Pointwise output files to Darwin (**scp**)
- Log into Darwin terminal from local and go into eVTOL folder, where the g and bcs files should be (**ssh**)
- Load TurboStream environment
source /rds/project/hpc/rds-hpc-pullan/ts3/bashrc_module_ts362
- The script '**CONVERT_MESH.py**' will convert Pointwise files into correct TURBOSTREAM input file format ('.xdmf', '.hdf5'), must edit the script to update the names of files it is converting before running
- Copy these files from Darwin to home computer