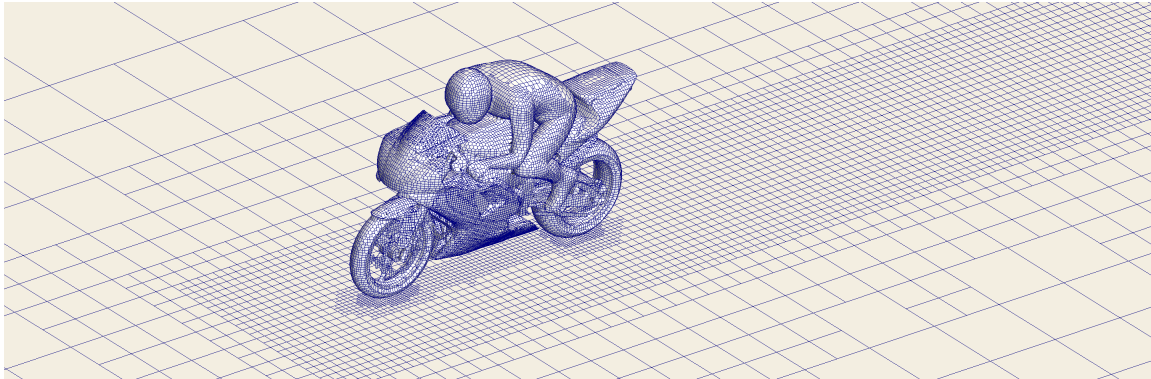


Laboratory Exercise: Grid Generation with OpenFOAM



You will use OpenFOAM's built-in grid generation utilities to generate a series of grids for different problem types.

The utility `blockMesh` can be used to create relatively simple meshes with uniform or stretched cells and straight or curved edges based on the dictionary in `constant/polyMesh/blockMeshDict`. You can find a detailed description here:

<https://cfd.direct/openfoam/user-guide/v6-blockmesh/>

For complex geometries (like aircraft, ships, ground vehicles, etc), `snappyHexMesh` is often used. This utility uses a background grid (often generated with `blockMesh`) and a surface geometry in either STL or OBJ format to create the body-fitted grid through three main steps: castellation, snapping, layer addition. See the description here:

<https://cfd.direct/openfoam/user-guide/v6-snappyhexmesh/>

Complete the following tasks to explore grid generation with OpenFOAM:

- (1) Modify the sample backward step grid from HW7. Increase or decrease the length of the inlet section. Change the grid spacing to refine or coarsen the grid. Try stretching the cells using the `simpleGrading` entry.
- (2) Use `snappyHexMesh` to generate a grid around a motor bike geometry. The sample grid generation case setup is uploaded to our class git repository:

<https://github.com/gpfilip/NA423-Fall2018/tree/master/lab-grid-gen>

You can also download the case from Canvas. Experiment with the background grid parameters specified in `blockMeshDict` to coarsen or refine the grid. Try generating a mesh with more prism layers to help in the near-wall gradient calculations during the flow solve.