

Meshing in OpenFOAM®

- OpenFOAM® comes with the following meshing applications:
 - `blockMesh`
 - `snappyHexMesh`
 - `foamyHexMesh`
 - `foamyQuadMesh`
- We are going to work with `blockMesh` and `snappyHexMesh`.
- **blockMesh** is a multi-block mesh generator.
- **snappyHexMesh** is an automatic split hex mesher, refines and snaps to surface.
- If you are not comfortable using OpenFOAM® meshing applications, you can use an external mesher.
- OpenFOAM® comes with many mesh conversion utilities. Many popular meshing formats are supported. To name a few: `gambit`, `cfx`, `fluent`, `gmsh`, `ideas`, `netgen`, `plot3d`, `starccm`, `VTK`.
- In this module, we are going to address how to mesh using OpenFOAM® technology, how to convert meshes to OpenFOAM® format, and how to assess mesh quality in OpenFOAM®.

Meshing in OpenFOAM®

By the end of this module, you will realize that

You will use **snappyHexMesh** to mesh the sphinx

You will use **blockMesh** to mesh the pyramids

