Mesh conversion

- OpenFOAM® gives users a lot of flexibility when it comes to meshing.
- You are not constrained to use OpenFOAM® meshing tools.
- To convert a mesh generated with a third party software to OpenFOAM® **polyMesh** format, you can use the OpenFOAM® mesh conversion utilities.
- If your format is not supported, you can write your own conversion tool.
- By the way, many of the commercially available meshers can save the mesh in OpenFOAM® **polyMesh** format or in a compatible format.

Mesh conversion

- In the directory **`$FOAM_UTILITIES`** (use the alias `util` to go there) you will find the following sub-directories containing the source code for the utilities available in the OpenFOAM® installation (version 3.0.x):

  - **mesh**
  - **miscellaneous**
  - **parallelProcessing**
  - **postProcessing**
  - **preProcessing**
  - **surface**
  - **thermophysical**

- In the sub-directory **`mesh`** you will find the source code for the mesh utilities included in the OpenFOAM® installation.

Mesh conversion

- Let us visit the **`mesh`** directory. In the terminal type:
  - `$> util`
  - `$> cd mesh`
  - `$> ls -al`

- In this directory you will find the directories containing the source code for the following mesh utilities
  - **advanced**
  - **conversion**
  - **generation**
  - **Manipulation**

- In the directory **`conversion`** you will find the source code for the mesh conversion utilities. Let us visit this directory, in the terminal type:
  - `$> cd conversion`
  - `$> ls -al`

Mesh conversion

- In the directory **`$FOAM_UTILITIES/mesh/conversion`** you will find the following mesh conversion utilities:

  - **ansysToFoam**
  - **cfx4ToFoam**
  - **datToFoam**
  - **fluent3DMeshToFoam**
  - **fluentMeshToFoam**
  - **foamMeshToFluent**
  - **foamToStarMesh**
  - **foamToSurface**
  - **gambitToFoam**
  - **gmshToFoam**
  - **ideasUnvToFoam**
  - **kivaToFoam**
  - **mshToFoam**
  - **netgenNeutralToFoam**
  - **Optional/ccm26ToFoam**
  - **plot3dToFoam**
  - **sammToFoam**
  - **star3ToFoam**
  - **star4ToFoam**
  - **tetgenToFoam**
  - **vtkUnstructuredToFoam**
  - **writeMeshObj**