cfMesh v1.0.1

User Guide

Document version: 1.0.1

Principal developer and document author:

Dr. Franjo Juretić, M. Eng., Assist. Prof.

Managing Director and Founding Partner

Creative Fields, Ltd.

Zagreb, October 2014

Contents

1. Introduction

cfMesh is a cross-platform library for automatic mesh generation that is built on top of **OpenFOAM®**[1]. It is licensed under GPL, and compatible with all recent versions of **OpenFOAM®** and foam-extend.

cfMesh supports various 3D and 2D workflows, built by using components from the main library, which are extensible and can be combined into various meshing workflows. The core library utilises the concept of mesh modifiers, which allows for efficient parallelisation using both shared memory parallelisation (SMP) and distributed memory parallelisation using MPI. In addition, special care has been taken on memory usage, which is kept low by implementing data containers (lists, graphs, etc.) that do not require many dynamic memory allocation operations during the meshing process.

2. Available meshing workflows

Meshing workflows implemented in **cfMesh** require input geometry in a form of surface triangulation, shown in *Figure 1*, and the user-specified settings, which will be explained in detail in subsequent sections. All workflows are parallelised for shared memory machines and use all available computer cores while running. The number of utilised cores can be controlled by the OMP_NUM_THREADS environment variable, which can be set to the desired number of cores.
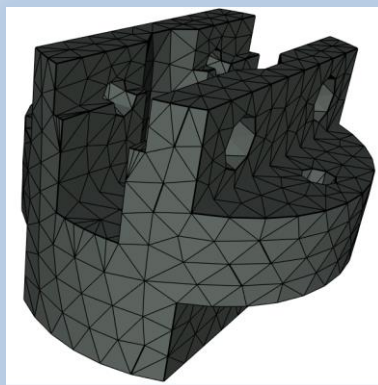

Figure 1 – Surface mesh

Currently available meshing workflows start the meshing process by creating a so-called mesh template from the input geometry and the user-specified settings. The template is later on adjusted to match the input geometry. The process of fitting the template to the input geometry is designed to be tolerant to poor quality input data, which does not need to be watertight. The available workflows differ by the type of cells generated in the template.

---

[1] None of the OpenFOAM® related offering by Creative Fields, Ltd., including cfMesh, is approved or endorsed by OpenCFD, Ltd. (ESI Group), producer of the OpenFOAM® software. OpenFOAM® and OpenCFD® are registered trade marks of ESI Group.

2.1. Cartesian

Cartesian workflow generates 3D meshes consisting of predominantly hexahedral cells with polyhedra in the transition regions between the cells of different size. It is started by typing *cartesianMesh* in a shell window. By default, it generates one boundary layer, which can be further refined on user request. In addition, this workflow can be run using MPI parallelisation, which is intended for generation of large meshes, which do not fit into the memory of a single available computer. An example of the mesh generated by *cartesianMesh* is shown in *Figure 2* and *Figure 3*.
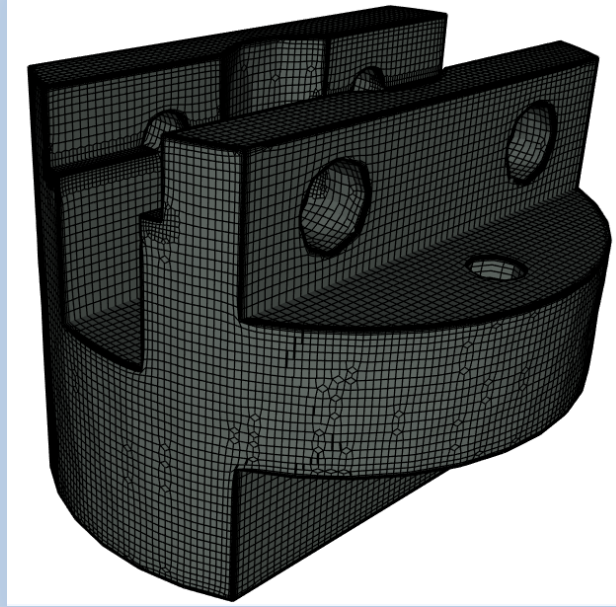
Figure 2 – Volume mesh generated by cartesianMesh
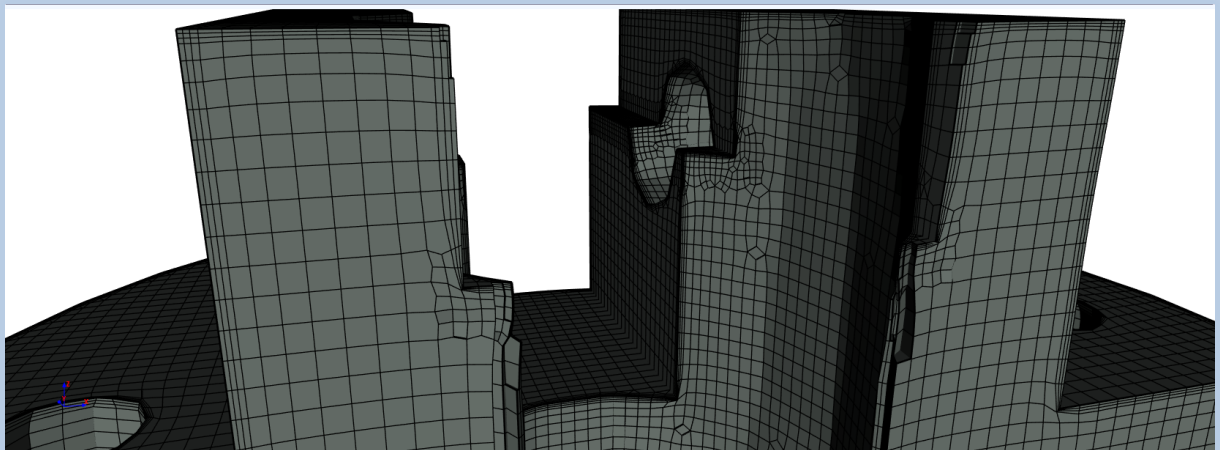
Figure 3 – Detail of the mesh generated by cartesianMesh

2.2. 2D Cartesian

It generates 2D meshes and is started by typing **cartesian2DMesh** in the console. By default it generates one boundary layer which can be further refined. The geometry is given in a form of a ribbon, shown in *Figure 4*. An example of final mesh is given in *Figure 5*.
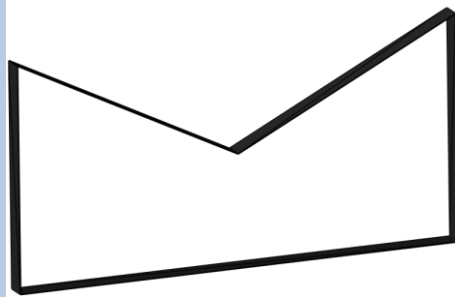
Figure 4 – Input geometry for the cartesian2DMesh
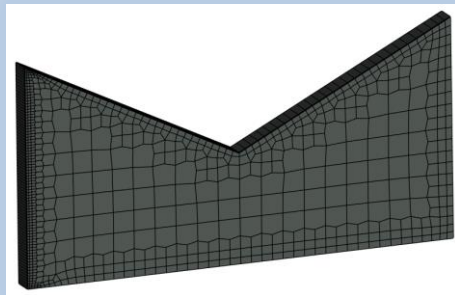
Figure 5 – Mesh generated by cartesian2DMesh

2.3. Tetrahedral

Tetrahedral workflow generates meshes consisting of tetrahedral cells and is started by typing *tetMesh* in a console. By default, it does not generate any boundary layers, and they can be added and refined on user request. An example of a tetrahedral mesh generated by this workflow is given in *Figure 6* and *Figure 7*.
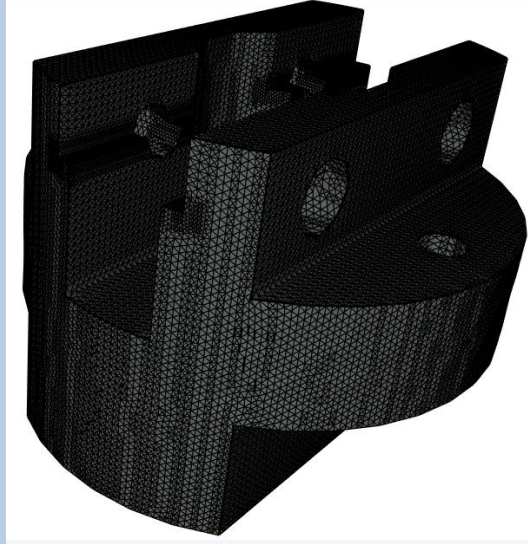
Figure 6 – Mesh generated by tetMesh
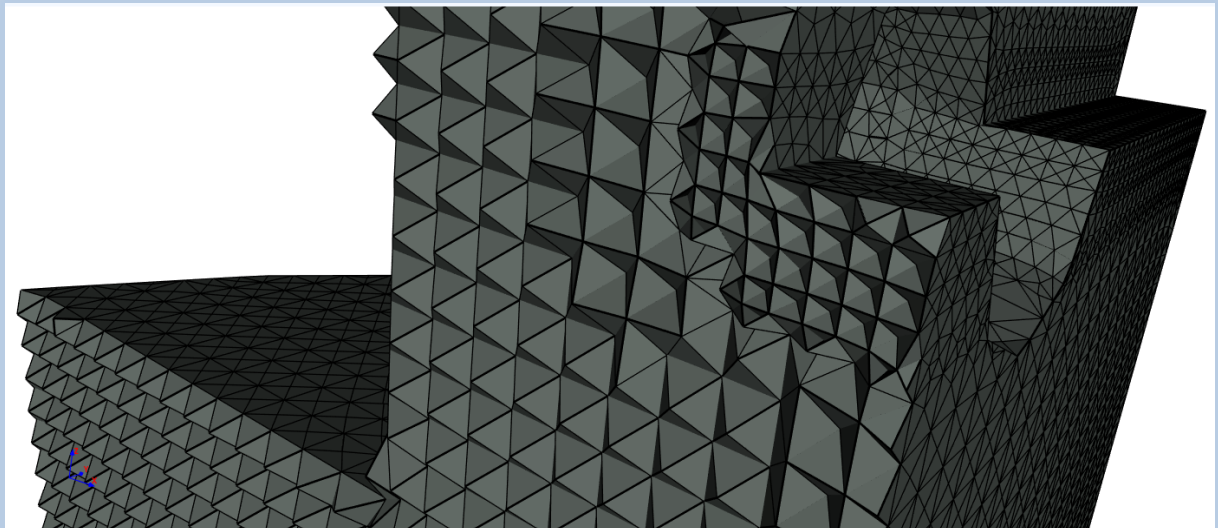
Figure 7 – Detail of the mesh generated by tetMesh

3. Input geometry

cfMesh requires geometries in the form of a surface triangulation. For 2D cases, the geometry is given in a form of a ribbon of triangles with boundary edges in the x-y plane (other orientations are not supported).

The geometry consists of the following entities:

> **List of points** – contains all points in the surface triangulation.

> **List of triangles** – contains all triangles in the surface mesh.

> **Patches** are entities that are transferred onto the volume mesh in the meshing process. Every triangle in the surface is assigned to a single patch, and cannot be assigned to more than one patch. Each patch is identified by its name and type. By default, all patch names and types are transferred on the volume mesh, and are readily available for definition of boundary conditions for the simulation.

> **Facet subsets** are entities, which are not transferred onto the volume mesh in the meshing process. They are used for definition of meshing settings. Each face subset contains indices of triangles in the surface mesh. Please note that a triangle in the surface mesh can be contained in more than one subset.

> **Feature edges** are treated as constraints in the meshing process. Surface points where three or more feature edges meet are treated as corners. Feature edges can be generated by the *surfaceFeatureEdges* utility.

The user must define all sharp features transferred by the cfMesh prior to the meshing process. The edges at the border between the two patches and the feature edges are handled as sharp features in the meshing process. Other edges in the triangulation are not constrained. Figure 8 shows a surface mesh with a patch highlighted in green, a facet subset coloured in blue and user-selected feature edges coloured in red.
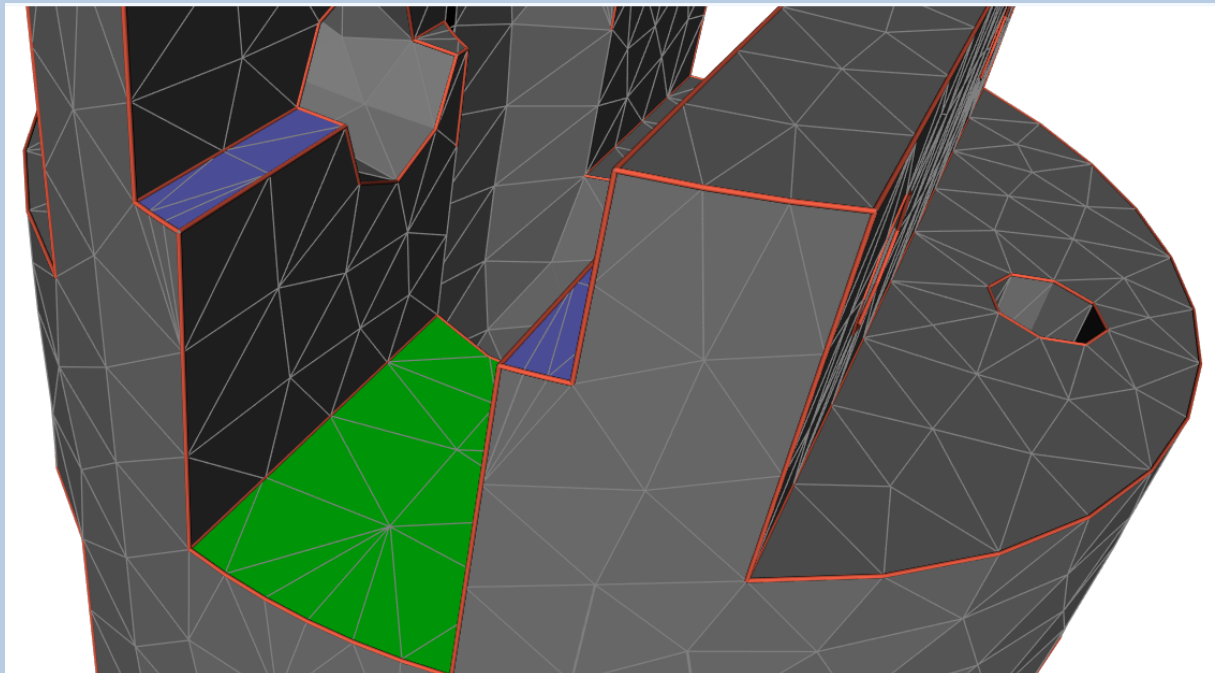

Figure 8 – Geometry with entities

The file formats suggested for meshing are: *fms*, *ftr*, and *stl*. In addition, the geometry can be imported in all formats supported by the ***surfaceConvert*** utility, which comes with **OpenFOAM®**. In addition, **cfMesh** also provides additional utilities for conversion of surface mesh into fms and from fms to other formats. However, the three suggested formats support definition of patches, which are transferred onto the volume mesh by default. Other formats can also be used for meshing but they do not support definition of patches in the input geometry and all faces at the boundary of the resulting volume mesh end up in a single patch.

The preferred format for **cfMesh** is *fms*, designed to hold all relevant information for setting up a meshing job. It stores patches, subsets, and feature edges in a single file. In addition, it is the only format, which can store all geometric entities into a single file, and the users are strongly encouraged to use it. An example of the surface mesh written in *fms* is depicted in *Figure 9*.

```
// patch names and types
6
(
bottom
patch

top
wall

fwd
patch

back
patch

left
patch

right
wall
)

// coordinates of surface points
8((0 0 0) (1 0 0) (1 1 0) (0 1 0) (0 0 1) (1 0 1) (1 1 1) (0 1 1))

// list of triangles
12
(
((0 1 2 ) 0) ((0 3 2 ) 0) ((5 4 7 ) 1) ((5 6 7 ) 1) ((0 1 5 ) 2) ((0 4 5 ) 2)
((2 6 7 ) 3) ((2 3 7 ) 3) ((0 3 7 ) 4) ((0 4 7 ) 4) ((2 1 5 ) 5) ((2 6 5 ) 5)
)

// list of feature edges
2((1 2) (3 4))

// list of point subsets
1
(
pointSubset0 4 4(1 3 4 6)
)

// list of facet subsets
1
(
facetSubset1 2 3(0 4 10)
)

// subsets of feature edges
0()
```

Figure 9 – Structure of the fms geometry file format

4. Required dictionaries and available settings

4.1. Dictionaries

The meshing process is steered by the settings provided in a **meshDict** dictionary located in the *system* directory of the case. For parallel meshing using MPI, a **decomposeParDict** located in the *system* directory of a case is required, and the number of nodes used for the parallel run must match the *numberOfSubdomains* entry in **decomposeParDict**. Other entries in **decomposeParDict** are not required.

The resulting volume mesh is written in the *polyMesh* directory inside the *constant* directory. The settings available in **meshDict** will be handled in the remainder of this section.

4.2. Mandatory settings in meshDict

cfMesh requires only two mandatory settings (see *Figure 10*) to start a meshing process:

- *surfaceFile* points to a geometry file. The path to the geometry file is relative to the path of the case directory.
- *maxCellSize* represent the default cell size used for the meshing job. It is the maximum cell size generated in the domain.

```
// path to the surface mesh
// relative from case or absolute
surfaceFile "surfaceMeshes/surf.fms";

// maximum cell size in the mesh (mandatory)
maxCellSize 3; // [m]
```

Figure 10 – Mandatory settings

4.3. Refinement settings in meshDict

Quite often a uniform cell size is not satisfactory, and there are many options for local refinement sources in **cfMesh**. *boundaryCellSize* option is used for refinement of cells at the boundary. It is a global option and the requested cell size is applied everywhere at the boundary. An example is given in *Figure 11*. *boundaryCellSizeRefinementThickness* specifies the distance from the boundary at which the *boundaryCellSize* is still applied. *minCellSize* is a global option which activates automatic refinement of the mesh template. This option performs refinement in regions where the cells are larger than the estimated feature size. The scalar value provided with this setting, specifies the smallest cell size allowed by this procedure, see *Figure 11*. This option is useful for quick simulation because it can generate meshes in complex geometry with low user effort. However, if high mesh quality is required, it provides hints where some mesh refinement is needed.

```
// size of the cells at the boundary (optional)
boundaryCellSize    1.5; // [m]

// distance from the boundary at which
// boundary cell size shall be used (optional)
boundaryCellSizeRefinementThickness 4.5; // [m]

// minimum cell size allowed in the automatic refinement procedure (optional)
minCellSize 0.375; // [m]
```

Figure 11 – boundaryCellSize and minCellSize

Creative Fields

localRefinement allows for local refinement regions at the boundary. It is a dictionary of dictionaries and each dictionary inside the main *localRefinement* dictionary is named by a patch or facet subset in the geometry that is used for refinement, see *Figure 12*. The requested cell size for an entity is controlled by the *cellSize* keyword and a scalar value, or by specifying *additionalRefinementLevels* keyword and the desired number of refinements relative to the maximum cell size. It is possible to specify patches via regular expressions. The thickness of the refinement zone can be specified by the *refinementThickness* option.

```
// refinement zones at the surface
// of the mesh (optional)
localRefinement
{
    // patch name
    "patch15.*" // accepts regex
    {
        // additional refinement levels
        // to the maxCellSize
        additionalRefinementLevels  1;

        // thickness of the refinement region
        // away from the patch
        refinementThickness 4.5;
    }

    // subset name
    subset1
    {
        cellSize    1.5; // [m]
    }
}
```

Figure 12 – localRefinement

objectRefinement is used for specifying refinement zones inside the volume. The supported objects that can be used for refinement are: lines, spheres, boxes, and truncated cones. It is specified as a dictionary of dictionaries, where each dictionary inside the *objectRefinement* dictionary represents the name of the object used for refinement. *refinementThickness* option can be used to specify the thickness of the refinement zone away from the object. The options required for each type of object is given in *Figure 13*.

```
// refinement zones inside the mesh
// based on primitive geometric objects (optional)
objectRefinements
{
    ear // name of the object
    {
        type                cone; // determined by two points and the radius at each point
        cellSize            7.51; // [m] cell size
        p0                  (-100 1873 -320);
        p1                  (-560 1400 0);
        radius0             200;
        radius1             200;
    }
    tail
    {
        type                box; // determined by the centre and th size in each coordinate
        cellSize            7.51;
        centre              (500 500 150);
        lengthX             100;
        lengthY             150;
        lengthZ             200;
    }
    insideTheBody
    {
        type                sphere; // determined by the centre and the radius
        cellSize            7.51;
        centre              (0 700 0);
        radius              50; // [m] radius
        refinementThickness 50; // [m] distance from the sphere
    }
    muzzlePiercing
    {
        type                line; // line is determined by two endpoints
        cellSize            7.51;
        p0                  (-750 1000 450);
        p1                  ( -750 1500 450);
        refinementThickness 40; // [m] distance from the line
    }
}
```

Figure 13 – objectRefinements subdictionary

surfaceMeshRefinement allows for using surface meshes as refinement zones in the mesh. It is specified as a dictionary of dictionary where each refinement zone is a sub-dictionary inside the *surfaceMeshRefinement* dictionary. Surface mesh used for the refinement zone is provided with the *surfaceFile* keyword, and the requested cell size for an entity is controlled by the *cellSize* keyword and a scalar value, or by specifying *additionalRefinementLevels* keyword and the desired number of refinements relative to the maximum cell size. Currently, the setting is used to refine the mesh in the region intersected by the surface mesh. It is possible to control the thickness of the refinement zone via *refinementThickness* keyword. An example of the available settings is given in *Figure 14*.

```
// refine regions intersected by surface meshes (optional)
surfaceMeshRefinement
{
    // name of the refinement region
    hull
    {
        // path to the surface file
        surfaceFile "refSurface.stl";

        // additional refinement levels
        // to the maxCellSize
        additionalRefinementLevels  3;

        // thickness of the refinement region
        // away from the surface
        refinementThickness 50;
    }
    deck
    {
        surfaceFile "refDeck.ftr";

        cellSize 6.125; // [m] cell size
    }
}
```

Figure 14 – surfaceMeshRefinement

4.4.   Keeping/removing of cells in user-defined regions

Meshing workflows implemented in **cfMesh** are based on the inside-out meshing, and the meshing process starts by generating the so-called mesh template based on the user-specified cell size. However, if the cell size is locally larger than the geometry feature size it may result with gaps in the geometry being filled by the mesh. On the contrary, the mesh in thin parts of the geometry can be lost if the specified cell size is larger than the local feature size.

keepCellsIntersectingBoundary is a global option which ensures that all cells in the template which are intersected by the boundary remain part of the template. By default, all meshing workflows keep only cells in the template which are completely inside the geometry. *keepCellsIntersectingBoundary* keyword must be followed by either 1 (active) or 0 (inactive), see *Figure 15*. Activation of this option can cause locally connected mesh over a gap, and the problem can be remedied by the *checkForGluedMesh* option which also must be followed by either 1 (active) or 0 (inactive).

```
// keep template cells intersecting boundary (optional setting)
keepCellsIntersectingBoundary   1; // 1 keep or 0 only internal cells are used

// remove cells where distinct parts of the mesh are joined together (optional)
removeGluedMesh 0; // 1 active or 0 inactive
```

Figure 15 – keepCellsIntersectingBoundary and checkForGluedMesh

keepCellsIntersectingPatches is an option which preserves cells in the template in the regions specified by the user. It is a dictionary of dictionaries, and each dictionary inside the main *keepCellsIntersectingPatches* dictionary is named by patch or facet subset, see *Figure 16*. This option is not active when the *keepCellsIntersectingBoundary* option is switched on. Patches can be specified using regular expressions.

```
// keep cells in the mesh template
// which intersect selected patches/subsets (optional)
// it is active when keepCellsIntersectingBoundary
// is switched off
keepCellsIntersectingPatches
{
    // patch name
    "patch1.*" // accepts regex
    {
        keepCells 1; // 1 active or 0 inactive
    }

    // subset name
    subset1
    {
        keepCells 1; // 1 active or 0 inactive
    }
}
```

Figure 16 – keepCellsIntersectingPatches subdictionary

removeCellsIntersectingPatches is an option which removes cells from the template in the regions specified by the user. It is a dictionary of dictionaries, and each dictionary inside the main *removeCellsIntersectingPatches* dictionary is named by a patch or a facet subset as shown in *Figure 17*. The option is active when the *keepCellsIntersectingBoundary* option is switched on. Patches can be specified using regular expressions.

```
// remove cells the cells intersected
// by the selected patches/subsets
// from the mesh template (optional)
// it is active when keepCellsIntersectingBoundary
// is switched on
removeCellsIntersectingPatches
{
    // patch name
    "patch1.*" // accepts regex
    {
        keepCells 0; // 0 remove or 1 keep
    }

    // subset name
    subset1
    {
        keepCells 0; // 0 remove or 1 keep
    }
}
```

Figure 17 – removeCellsIntersectingPatches subdictionary

4.5. Boundary layers

Boundary layers in **cfMesh** are extruded from the boundary faces of the volume mesh towards the interior, and cannot be extruded prior to the meshing process. In addition, their thickness is controlled by the cell size specified at the boundary and the mesher tends to produce layers of similar thickness to the cell size. Layers in **cfMesh** can span over multiple patches if they share concave edges or corners with valence greater than three. All boundary layer settings are provided inside a _boundaryLayers_ dictionary, shown in _Figure 18_. The options are:

- _nLayers_ specifies the number of layers which will be generated in the mesh. It is not mandatory. In case it is not specified the meshing workflow generates the default number of layers, which is either one or zero.
- _thicknessRatio_ is a ratio between the thickness of the two successive layer. It is not mandatory. The ratio must be larger than 1. The default value is 1.
- _maxFirstLayerThickness_ ensures that the thickness of the first boundary layer never exceeds the specified value. It is not mandatory.

patchBoundaryLayers setting is a dictionary which is used for specifying local properties of boundary layers for individual patches. It is possible to specify _nLayers_, _thicknessRatio_ and _maxFirstLayerThickness_ for each patch individually within a dictionary with the name equal to the patch name. By default, the number of layers generated at a patch is governed by the global number of layers, or the maximum number of layers specified at any of the patches which form a continuous layer together with the existing patch. _allowDiscontinuity_ option ensures that the number of layers required for a patch shall not spread to other patches in the same layer. Patches can be specified using regular expressions.

```
// settings for boundary layers
boundaryLayers
{
    // global number of layers (optional)
    nLayers 3;

    // thickness ratio (optional)
    thicknessRatio  1.2;

    // max thickness of te first layer (optional)
    maxFirstLayerThickness  0.5; // [m]

    // local settings for individual patches
    patchBoundaryLayers
    {
        // patch name
        "patch7.*" // accepts regex
        {
            // number of layers (optional)
            nLayers 4;

            // thickness ratio (optional)
            thicknessRatio  1.1;

            // max thickness of the first layer
            // (optional)
            maxFirstLayerThickness  0.5; // [m]

            // active 1 or inactive 0
            allowDiscontinuity  0;
        }
    }
}
```

Figure 18 – boundaryLayers subdictionary

4.6. Renaming patches

The settings presented in this section are used for changing of patch names and patch types during the meshing process. The settings are provided inside a *renameBoundary* dictionary with the following options:

- *newPatchNames* is a dictionary inside the *renameBoundary* dictionary. It contains dictionaries with names of patches which shall be renamed. For each patch it is possible to specify the new name or the new patch type with the settings:
  - o *newName* keyword is followed by the new name for the given patch. The setting is not mandatory.
  - o *type* keyword is followed by the new type for the given patch. The setting is not mandatory.
- *defaultName* is a new name for all patches except the ones specified in *newPatchNames* dictionary. The setting is not mandatory.
- *defaultType* sets the new type for all patches except the ones specified in *newPatchNames* directory. The setting is not mandatory.

Patch names can be specified using regular expressions. An example is given in *Figure 19*.

```
// setting used for assigning new names
// of patches in the surface mesh when
// transferring them onto the volume mesh (optional)
renameBoundary
{
    // new name of all patches except
    // the ones specified below (optional)
    defaultName walls;

    // new type of the default patch (optional)
    defaultType wall;

    newPatchNames
    {
        // patch name in the surface mesh
        "patch0.*" // accepts regex
        {
            // patch name in the volume mesh (optional)
            newName outlet;

            // patch type in the volume mesh (optional)
            type    patch;
        }
        patch1
        {
            newName inlet;
            type    patch;
        }
    }
}
```

Figure 19 – renameBoundary subdictionary

4.7. Enforcing geometry constraints

Sometimes it is not possible to capture all geometric features during the meshing process and the acceptance criteria are not met. *enforceGeometryConstraints* option stops the meshing process when it is not possible to capture all features of the input geometry. When active, it stops the meshing process and writes a subset of points that had to be moved away from the geometry in order to yield a valid mesh. An example is given in *Figure 20*.

```
// stops the meshing process
// when it is not possible to capture all
// geometric features (optional)
enforceGeometryConstraints 1; // 1 active or 0 inactive
```

Figure 20 – enforceGeometryConstraints

5. Utilities

copySurfaceParts

It copies surface facets in a specified facet subset into a new surface mesh.

extrudeEdgesInto2DSurface

This utility extrudes edges written as feature edges in the geometry into a ribbon of triangles required for 2D mesh generation. Generated triangles are stored in a single patch.

FLMAToSurface

It converts geometry from AVL's *flma* format into the format readable by **cfMesh**. Cell selections defined in the input file are transferred as facet subsets.

FMSToSurface

The utility converts the data in a fms file into several files which can be imported into ParaView.

FMSToVTK

Converts fms file into ParaView's vtm file, containing all data available in an fms file.

FPMAToMesh

Is a utility for importing volume meshes from AVL's *fpma* format. Selections defined on the input mesh are transferred as subsets.

importSurfaceAsSubset

The ustility matches the triangles of the surface mesh used for meshing with the other other surface, and create a facet subset in the original surface.

improveSymmetryPlanes

OpenFOAM is very sensitive to the position of points in the symmetry plane. This utility calculates the regression plane for all vertices belonging to a symmetry plane, and ensures that all vertices lie in a plane.

mergeSurfacePatches

The utility allow the user to specify the patches in the surface mesh which shall be merge together.

meshToFPMAposition

This utility converts the mesh into the AVL's *fpma* format.

patchesToSubsets

It converts patches in the geometry into facet subsets.

preparePar

It creates processor directories required for MPI parallelisation. The number of processor directories is dependent on the *numberOfSubdmains* specified in **decomposeParDict**.

removeSurfaceFacets

It is a utility for removing facets from the surface mesh. The facets that shall be removed are given by a patch name or a facet subset.

subsetToPatch

It creates a patch in the surface mesh consisting of facets in the given facet subset.

surfaceFeatureEdges

It is used for generating feature edges in the geometry. In case the output is a *fms* file, generated edges are stored as feature edges. Otherwise it generates patches bounded by the selected feature edges.

surfaceGenerateBoundingBox

It generates a box around the geometry. It does not resolve self-intersections in case the box intersects with the rest of the geometry.

surfaceToFMS

A converter from common surface triangulation formats into *fms*.

Python utilities

extractFeatureEdges.py

A script to extract feature edges from geometries created in Salome.

salomeTriSurf.py

A script for extracting a *fms* file from the geometry created in Salome.

Appendix A: Installation instructions

The simplest way to install **cfMesh** is to download the installer for **OpenFOAM®** and **cfMesh** from the company website www.c-fields.com.

If you want to build it from source, please download the source of **cfMesh** from www.c-fields.com. Open the archive by typing *tar xzf cfMesh-v1.0.1.tgz* in a Linux shell, and check that you have the **OpenFOAM®** environment set up. The build is started by the *Allwmake* script within the **cfMesh** directory.