Salome to OpenFOAM mesh conversion tutorial

1. Part one – Mesh preparation for conversion

1.1 Intro and Mesh merging

This tutorial deals with conversion from Salome to OpenFOAM. For this reason it will be assumed that the mesh has already been prepared in Salome and is ready to be converted to OpenFOAM format.

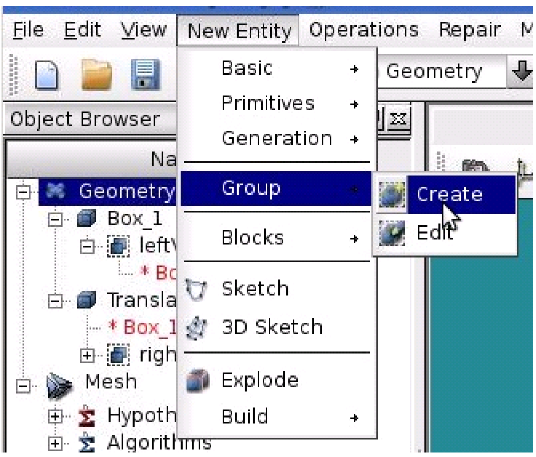
Note that the mesh object by itself should be one. If the model is not covered by one single mesh, then the meshes should be converted into sub meshes of one “overall” mesh. The reason for this lies in the exporting process.

1.2 Patches

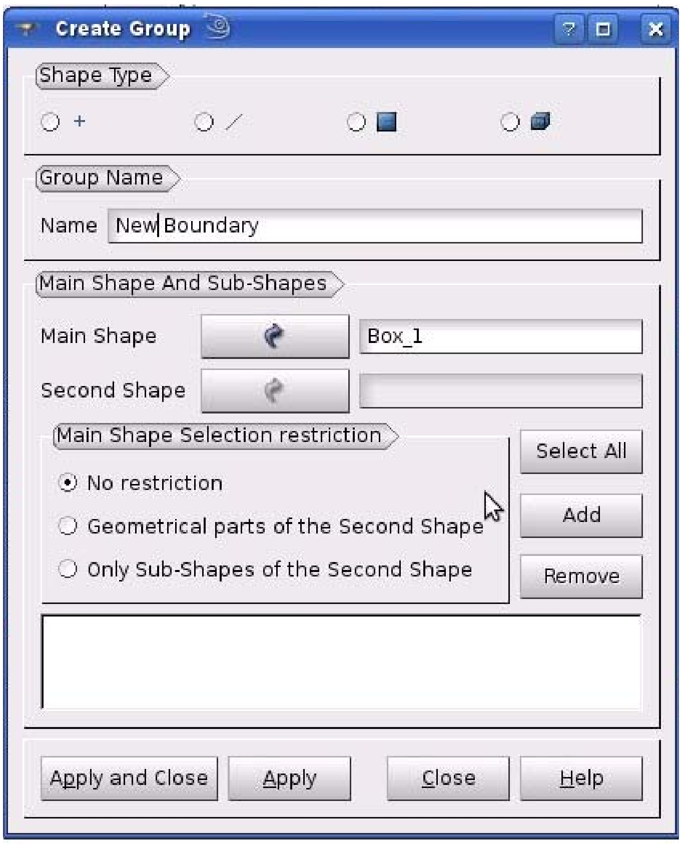
Once the mesh has been merged together to form a single mesh there is needs to define boundary patches for OpenFOAM. This is done as follows:

- In the Geometry tab do the following

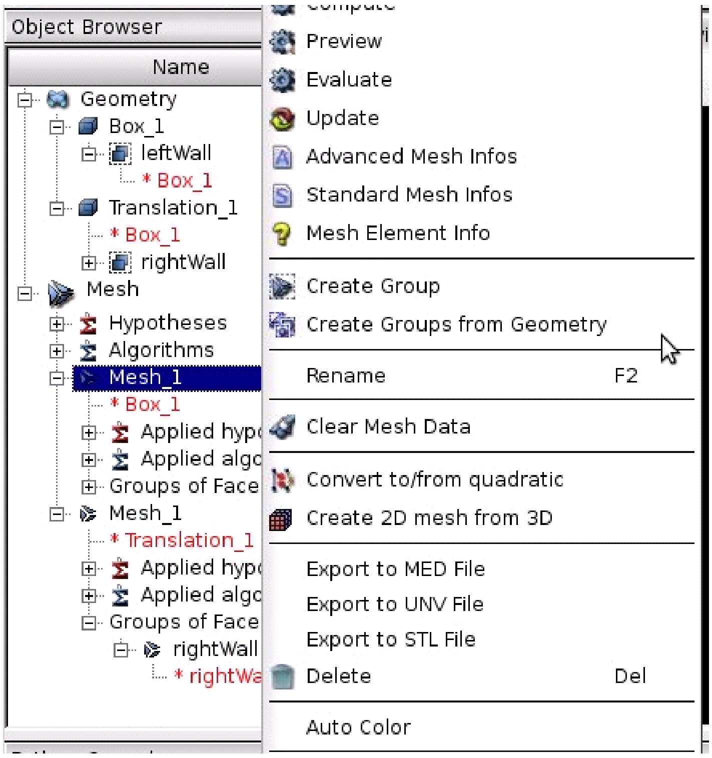
Top Menu → New Entity → Group → Create



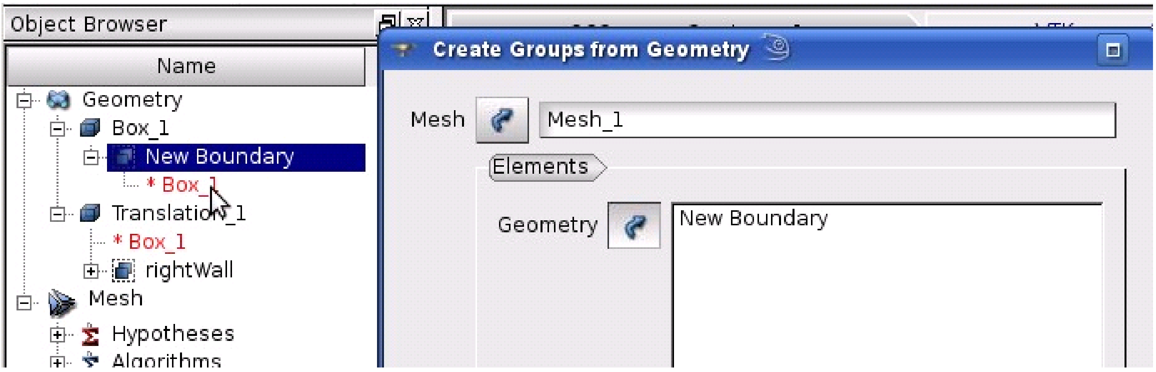
- In the popup box, choose ***Group on faces***, and select the faces that constitute the boundary. This group gives the name to the boundary, so it should be given the intended boundary name. Click ***Apply*** when finished.



- Now, select the ***Mesh*** tab found on the right side menu and right click the ***global mesh***.



- In the context menu, choose ***Create Groups from Geometry***.
- Then select the global mesh and the necessary boundary as geometry.

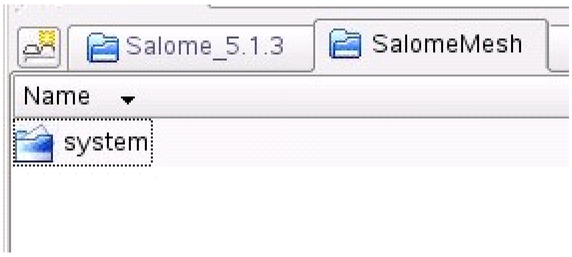


- This will create the boundary.

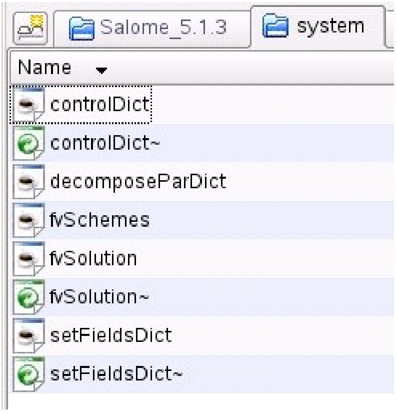
2. Part two – Mesh export and directory structure

2.1 Directory preparation

Before exporting the mesh into a separate directory, a certain directory tree must be prepared. The exact structure can be created as follows:

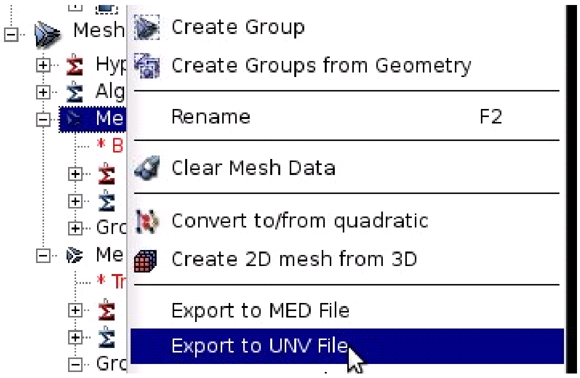


- Case folder has to contain folder *system*.
- Folder *system* has to contain the file *controlDict* with simulation parameters. Note that the conversion is not affected by this file. However, OpenFOAM checks for it during the conversion process.

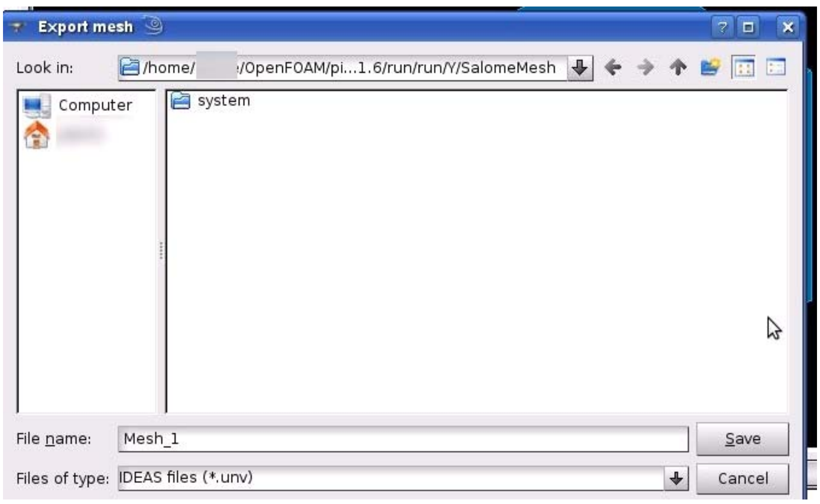


2.2 Mesh Export

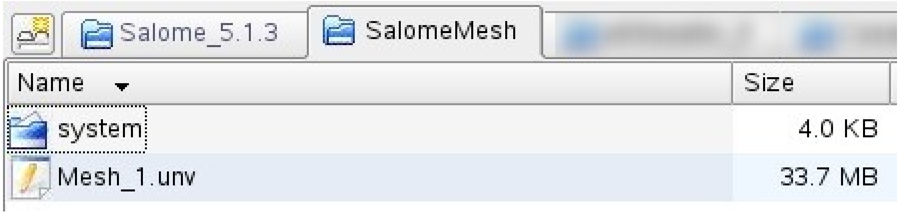
Mesh is exported from OpenFOAM in the following way: Right click on the global mesh, and choose ***export to UNV file***,



then select prepared case directory and save.



The mesh has now been exported.



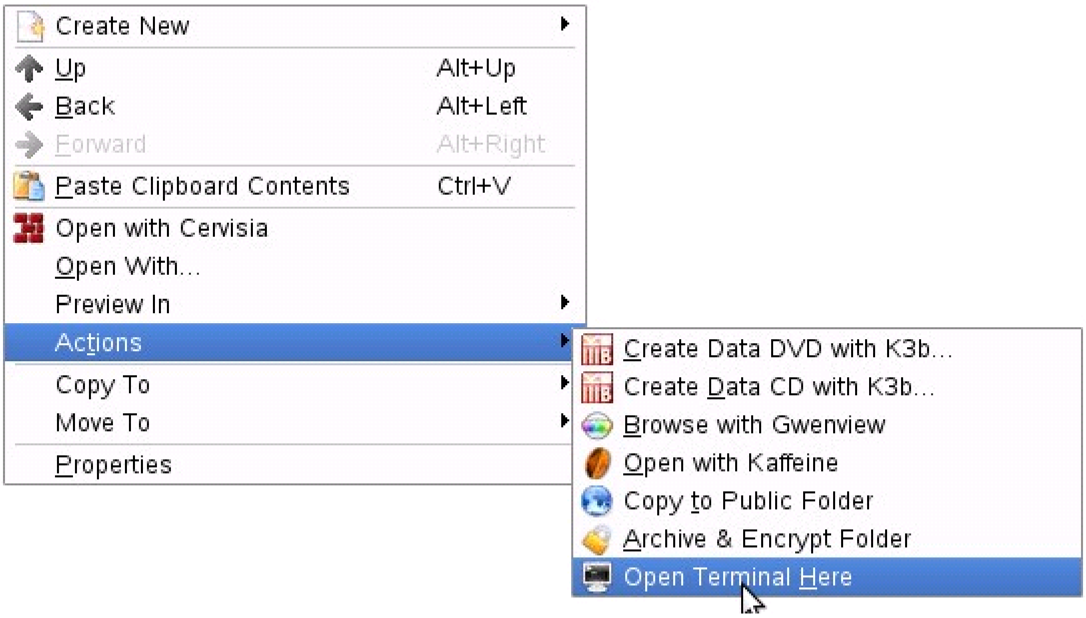
3. Part three – Mesh conversion and final result

3.1 Conversion

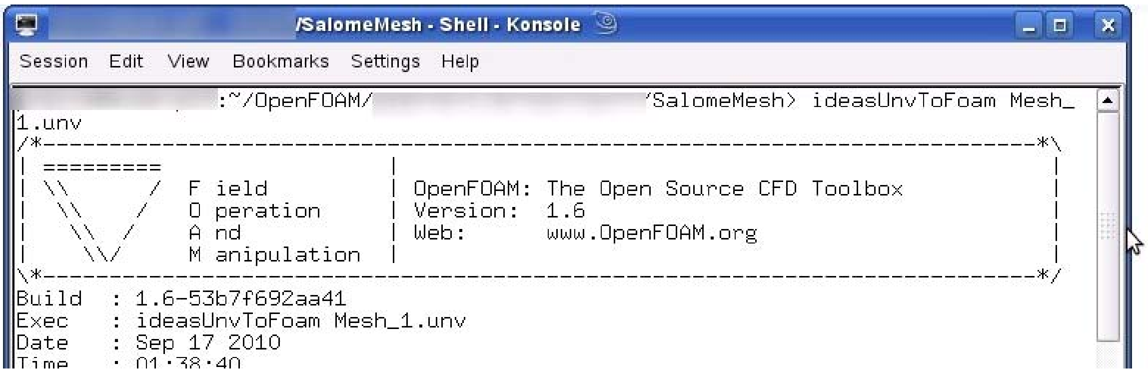
The exported UNV file has to be converted to OpenFOAM format. The steps are as follows

- Open terminal in case folder where the mesh is found. A short way of doing this would be by right clicking and choosing:

Actions → Open Terminal here



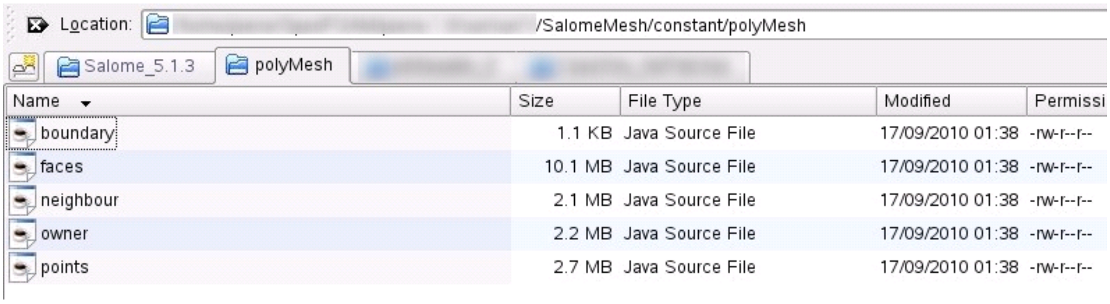
- In the terminal type ***ideasUnvToFoam Mesh_1.unv***, where Mesh_1.unv is the full file name of your mesh.



- The mesh will now be converted.

3.2 Basic validation

After a successful conversion a folder *constant* should appear inside case root folder. The folder will contain subfolder *polyMesh*, which in turn contains OpenFOAM mesh files:



You can check the status of the converted mesh by running ***checkMesh***. If no errors are reported you can check the patches in the *boundary* file: All boundary mesh faces grouped as discussed in Section 1.2 will be assigned into their respective patch – in this case we should find a face defined as *leftWall*, whilst all other external faces are assigned a *default* patch.

Salome_5.1.3 | boundary

```
/*--------------------------------*- C++ -*----------------------------------
| =========                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbo:
|  \\    /   O peration     | Version:  1.6
|   \\  /    A nd           | Web:      www.OpenFOAM.org
|    \\/     M anipulation  |
\*---------------------------------------------------------------------------
FoamFile
{
    version     2.0;
    format      ascii;
    class       polyBoundaryMesh;
    location    "constant/polyMesh";
    object      boundary;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *

2
(
    leftWall
    {
        type            patch;
        nFaces          12500;
        startFace       367500;
    }
    defaultFaces
    {
        type            patch;
        nFaces          2500;
        startFace       380000;
    }
)
```

This work was originally written by Mr Yury Kulakov as part of his summer work with the Physics Department in 2010 which was financed by the University of Malta.