# 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*  $\rightarrow$  *New Entity*  $\rightarrow$  *Group*  $\rightarrow$  *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.

1

🛪 Create Group 🍥	? <b>.</b> ×
Shape Type	1
○ + ○ / ○ ■ ○	<i>a</i>
Group Name	1
Name New Boundary	
Main Shape And Sub-Shapes	 [
Main Shape 🕜 Box_1	
Second Shape	
Main Shape Selection restriction	Select All
<ul> <li>No restriction</li> <li>Geometrical parts of the Second Shape</li> </ul>	Add
○ Only Sub-Shapes of the Second Shape	Remove
Apply and Close Apply Close	Help

• 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.

Object Browser		1 CT V
Name	🐨 Create Groups from Geometry 🍥	
Geometry Box_1 New Boundary * Box_1 Translations_1 * Box_1 *	Mesh Mesh_1 Elements Geometry R New Boundary	

• 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.

🛃 📔 Salome_5.1.3	📔 system
Name 👻	
🥌 controlDict	
🥏 controlDict~	
🥌 decomposeParDict	
🛒 fvSchemes	
🛒 fvSolution	
🔊 fvSolution~	
setFieldsDict	
🧑 setFieldsDict~	

#### 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.

🛃 📄 Salome_5.1.3 📄 SalomeMesh	and the second s
Name 🗸	Size
襘 system	4.0 KB
Mesh_1.unv	33.7 MB

# 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:

👌 Create New		•	
<b>↑</b> <u>U</u> р	Alt+Up	1	
🗲 <u>B</u> ack	Alt+Left		
Eorward	Alt+Right		
붭 <u>P</u> aste Clipboard Contents	Ctrl+V		
📱 Open with Cervisia		100	
Open With			
Preview In		•	
Actions		16	<u>c</u> reate Data DVD with K3b…
Сору То		•	Create Data CD with K3b
Move To		•	Browse with Gwenview
<u>P</u> roperties		(	👂 <u>O</u> pen with Kaffeine
			😼 Copy <u>t</u> o Public Folder
		4	💡 <u>A</u> rchive & Encrypt Folder
		Ę	📕 Open Terminal <u>H</u> ere
			.2

### Actions $\rightarrow$ Open Terminal here

• In the terminal type *ideasUnvToFoam Mesh\_1.unv*, where Mesh\_1.unv is the full file name of your mesh.

/SalomeMe	esh - Shell - Konsole 🎯			×
Session Edit View Bookmarks Settir	ngs Help			
:~/OpenFOAM/ 1.unv /*	′SalomeMesh>	ideasUnvToFoam	Mesh_ *\	
=======     \\ / F ield     \\ / O peration     \\ / A nd     \\/ M anipulation	OpenFOAM: The Open Source CFD Version: 1.6 Web: www.OpenFOAM.org	Toolbox		
Build : 1.6-53b7f692aa41 Exec : ideasUnvToFoam Mesh_ Date : Sep 17 2010	1.unv		*/	

• 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:

/SalomeMesh/constant/polyMesh		
be	Modified Permissi	
urce File	17/09/2010 01:38 -rw-rr	
urce File	17/09/2010 01:38 -rw-rr	
urce File	17/09/2010 01:38 -rw-rr	
urce File	17/09/2010 01:38 -rw-rr	
urce File	17/09/2010 01:38 -rw-rr	
urce File urce File		

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.



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.