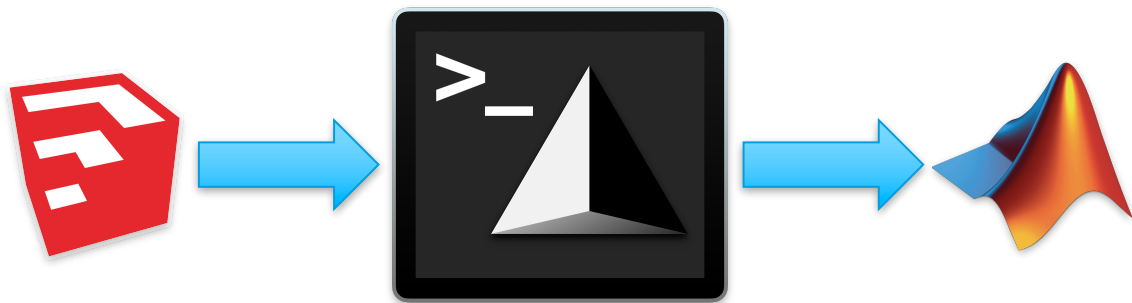

Code tutorial of the finite volume method on diffusion equation



Author:
Quentin GOESTCHEL

Supervisors:
Marteen HORNIKX, Raúl PAGÁN MUÑOZ, Sai CHARAN TRIKOOTAM

June 29, 2018

Contents

1 Sketchup: creating the model	0
2 Gmsh: meshing the volume	2
3 Matlab: processing the code	3

Introduction

This tutorial is made to familiarise the user with the process of modelling the diffuse sound field in rooms, starting from Sketchup 3D models. As a first step, make sure to use the folder "**DiffFVMTool**" as a work folder. It should contain:

- 1 folders: "**GeoModels**" with .geo samples
- 6 Matlab codes: "**Area_vec.m**", "**Bound_Assmbl.m**", "**Diffusion_unstructured_FVM.m**", "**Input_mesh.m**", "**Newid.m**" and "**Orth_idx.m**"
- The Gmsh executor "**Gmsh.exe**" also available at <http://gmsh.info/#Download>

If these files are all inside the work folder "**DiffFVMTool**", you just need to add the the gptoolbox folders developed by Alex Jacobson and others, available at <https://github.com/alecjacobson/gptoolbox/> and everything is ready to start.

1 Sketchup: creating the model

To begin the model, you need to have Sketchup installed with at least a student licence, and you need to install the extension "**Meshkit**" from the extension warehouse. Then, you should have access to a new set of buttons in Sketchup, their utilities are displayed on Figure 1.

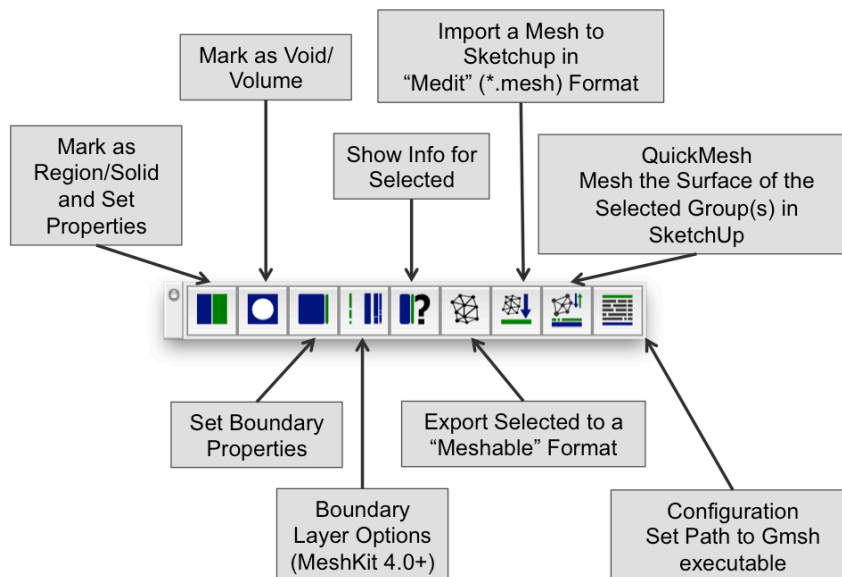


Figure 1: Overview of the Meshkit buttons and their utilities

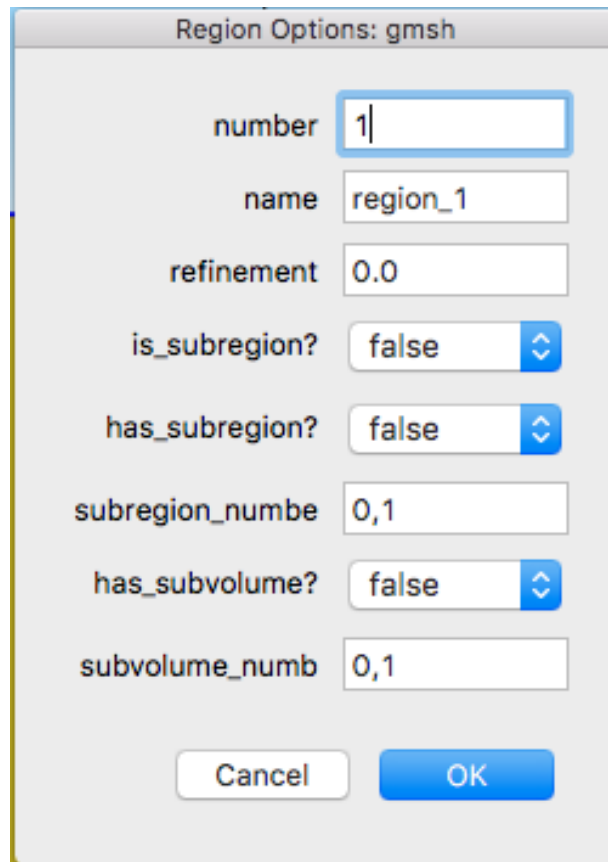
More explanations and tutorial about Meshkit are available there:

<https://extensions.sketchup.com/en/content/sketchy-tetgen-tools>

Once you have built your volume in Sketchup as you would usually do it, the exportation to a "meshable" object can be done following these steps:

- Make sure your volume is totally closed. (Some extensions like solid inspector can do it for you).

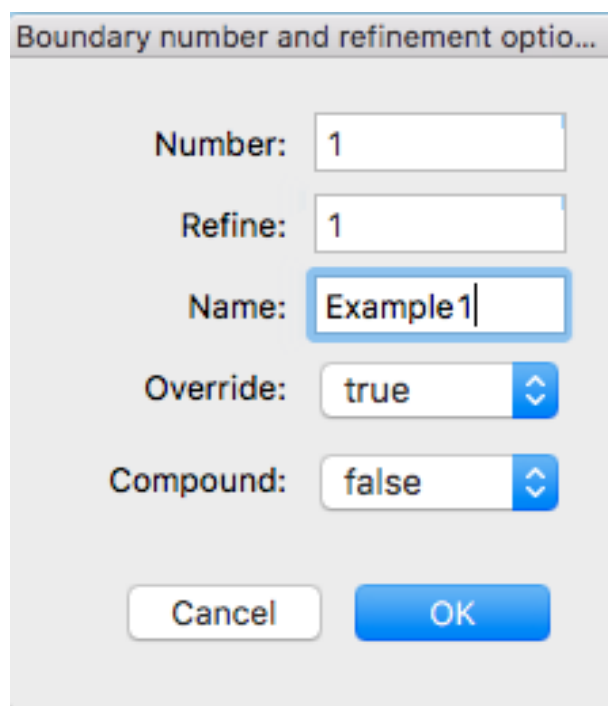
- Select all the components of your volume (Faces, edges, points), **right click** → **Make Group**
- Select the group you created, and use the **Mark as a Region/Solid and Set Properties** button. This window should appear:



The image shows a dialog box titled "Region Options: gmsh". It contains several input fields and dropdown menus for configuring a region. The fields are: "number" (text input with "1"), "name" (text input with "region_1"), "refinement" (text input with "0.0"), "is_subregion?" (dropdown menu with "false"), "has_subregion?" (dropdown menu with "false"), "subregion_numbe" (text input with "0,1"), "has_subvolume?" (dropdown menu with "false"), and "subvolume_numb" (text input with "0,1"). At the bottom, there are "Cancel" and "OK" buttons.

Leave the parameters as they are displayed. You can change the region name if you want to. Click **OK**.

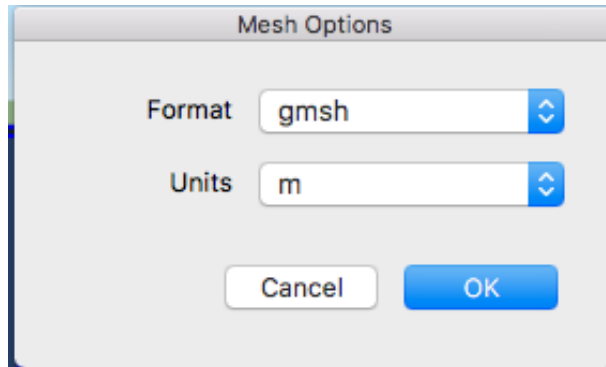
- Enter inside the group you have created by **double left click**
- For each surface, you need to enter a name related to an absorption coefficient. To do so, **double left click** on the surface you want to parameter and use the **Set Boundaries Properties** button. This window should appear:



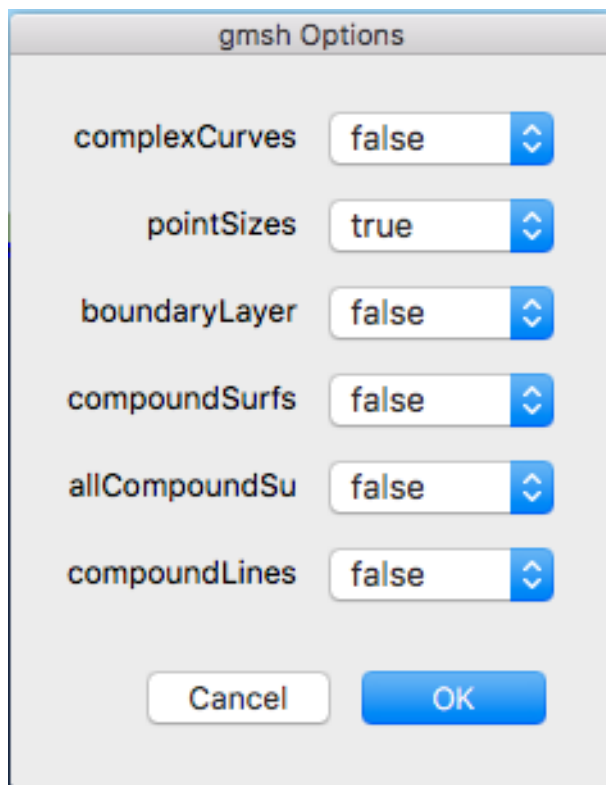
The image shows a dialog box titled "Boundary number and refinement options". It contains several input fields and dropdown menus for configuring boundary properties. The fields are: "Number:" (text input with "1"), "Refine:" (text input with "1"), "Name:" (text input with "Example1"), "Override:" (dropdown menu with "true"), and "Compound:" (dropdown menu with "false"). At the bottom, there are "Cancel" and "OK" buttons.

In this one you need to set the surface refinement to **1**, and the most important: choose a name for the surface like "**concrete**" for instance. This name will be used later.

- Proceed like mentioned in the precedent point for each surface of the volume. You can set multiple surface properties by selecting several surfaces and pushing the button **Set Boundaries Properties** afterwards.
- Leave the group you have created by clicking randomly out of the volume you are designing.
- Select the group & use the button **Export Selected to a "Meshable" format** This window should appear:



Set the format to **gms** and the unit to **m**, click **OK**. This window should appear:



The only parameter which should be set to **True** is **pointSizes**, click **OK**.

- Save the .geo file with the name of your choice in the folder "**GeoModels**".

The geometry you have designed is now ready to be used.

2 Gmsh: meshing the volume

This part is only for your information. Gmsh is an open-source meshing software developed by researchers. Every information about it can be found on:

<http://gmsh.info/>

In this procedure, Gmsh is controlled by Matlab so the user does not have to deal with it.

3 Matlab: processing the code

To get the results, you need to open the codes "**Bound_Assmbl.m**" and "**Diffusion_unstructured_FVM.m**" with Matlab. These codes have been developed with Matlab version 2018a for 64 bit OS X. The compatibility with windows OS has been checked but beware of any compatibility problem if you are using an older version of Matlab. The procedure to get the results is:

- Open "**Bound_Assmbl.m**".
- For each surface name you have defined in your Sketchup model, you need to assign an absorption coefficient. For this, add one condition "**elseif**" for each surface as it follows:
elseif mesh.ENTBelements(i).elset,21,1 == "<MaterialNameInSketchup>"
alpha = <AbsorptionCoeffValue(ex: 0.1)>;
Belements(i).Abn = Acoef(alpha);
Between the < > you need to implement your own parameters. This piece of code is also at your disposal in the Matlab code as comments.
- Save "**Bound_Assmbl.m**".
- Go in "**Diffusion_unstructured_FVM.m**".
- Run the code. You have to enter your filename and choose the simulation parameters. Be aware that every 3D plot costs substantial computational power. Just below you can find an inventory of the different geometric files ("**.geo**") already stored in the "**GeoModels**" folder:

Cinema	Model of a cinema without absorption information
CinemaAbs	Model of a cinema with absorption information
Cube	Reference model of a 10m×10m×10m cube
Meshtest	Experimental cross street model
NewTransfSphere	Experimental orthogonality control
RevRoom	Model of the reverberation chamber from echo building
Sphere	Reference model of a 5m radius sphere

- The results should be displayed to you. Before the end you get the choice between a time depending plot of the sound energy (take a while) or a plot of the error depending on the time.

For a better understanding of what is happening inside the Matlab code, it is recommended reading the scientific report related to it. Regarding the simulation parameters, explanation can be found for most of them in different parts of this scientific report:

Characteristic length of the mesh [in meter]	Report section 3.2.1
Boundary conditions types	Report section 1.2.2
Mesh orthogonality	Report section 1.6
Recording time & Source signal duration	Enter times long enough to actually model a diffuse field

If any problems persist, my contact is: quentin.goestchel@ens-cachan.fr