



fc oogmsh Matlab toolbox, User's Guide*

version 0.2.4

François Cuvelier[†]

January 5, 2023

Abstract

This Matlab toolbox make it possible to generate mesh files from *.geo* files by using `gmsh`. It's also possible with the `ooGMSH2` and `ooGMSH4` classes to read the mesh file (respectively for MSH file format version 2.2 and version 4.x). This toolbox must be regarded as a very simple interface between `gmsh` files and Matlab. So you are free to create any data structures or objects you want from an `ooGMSH2` object or an `ooGMSH4` object.

0 Contents

1	Introduction	2
2	Installation	2
2.1	Installation automatic, all in one (recommanded)	2
3	gmsh interface	4
3.1	function <code>fc_oogmsh.gmsh.buildmesh2d</code>	4
3.2	function <code>fc_oogmsh.gmsh.buildmesh3d</code>	6
3.3	function <code>fc_oogmsh.gmsh.buildmesh3ds</code>	7
3.4	function <code>fc_oogmsh.gmsh.buildpartmesh2d</code>	7
3.5	function <code>fc_oogmsh.gmsh.buildpartmesh3d</code>	8
3.6	function <code>fc_oogmsh.gmsh.buildpartmesh3ds</code>	8
3.7	function <code>fc_oogmsh.gmsh.buildPartRectangle</code>	8
4	ooGmsh4 class (version 4.x)	9
4.1	Methods	10
4.1.1	<code>ooGms4</code> constructor	10
4.1.2	<code>info</code> method	11
4.1.3	<code>get_ElementaryTags</code> method	13

* $\text{\LaTeX}$ manual, revision 0.2.4.a, compiled with Matlab 2022a, and toolboxes `fc-oogmsh`[0.2.4], `fc-tools`[0.0.33], `fc-bench`[0.1.3], `fc-amat`[0.1.3], `fc-meshtools`[0.1.4], `fc-graphics4mesh`[0.1.5], and using `gmsh` 4.11.0

[†]LAGA, UMR 7539, CNRS, Université Paris 13 - Sorbonne Paris Cité, Université Paris 8, 99 Avenue J-B Clément, F-93430 Villetaneuse, France, cuvelier@math.univ-paris13.fr.

This work was supported by the ANR project DEDALES under grant ANR-14-CE23-0005.

4.1.4	<code>get_PhysicalTags</code> method	13
4.1.5	<code>get_me_ElementaryTag</code> method	14
4.1.6	<code>get_me_PhysicalTag</code> method	14
4.1.7	<code>get_localmesh_ElementaryTag</code> method	15
4.1.8	<code>get_localmesh_PhysicalTag</code> method	16
4.2	Description of properties	17
4.3	Sample 1	19
4.4	Sample 2	19
4.5	Sample 3	20
5	ooGmsh2 class (version 2.2)	20
5.1	Methods	21
5.1.1	<code>ooGmsh2</code> constructor	21
5.1.2	<code>info</code> method	22
5.1.3	<code>get_ElementaryTags</code> method	24
5.1.4	<code>get_PhysicalTags</code> method	24
5.1.5	<code>get_me_ElementaryTag</code> method	25
5.1.6	<code>get_me_PhysicalTag</code> method	25
5.1.7	<code>get_localmesh_ElementaryTag</code> method	26
5.1.8	<code>get_localmesh_PhysicalTag</code> method	27
5.2	Sample 1	28
5.3	Sample 2	28
5.4	Sample 3	29
A	Element type	29
B	Other functions	31
B.1	function <code>fc_oogmsh.gmsh.elm_type_desc</code>	31
B.2	method <code>plotElementaryElements</code>	31
B.3	method <code>plotPhysicalElements</code>	32
B.4	method <code>plotPartitionElements</code>	33
B.5	method <code>plotInterfaceElements</code>	34

1 Introduction

The `ooGmsh` Matlab toolbox is closely related to `gmsh`, see [2] or [3], which is a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. `gmsh` can also build two-dimensional meshes and three-dimensional surface meshes. This toolbox was initially created to make it possible from Matlab to rapidly

- generate mesh file from `.geo` file by using `gmsh`
- efficiently read this mesh file and store its contents in `ooGmsh` Matlab object easy to manipulate.

The `ooGmsh` Matlab object can be used to create, from a `.msh` file, any data structures or objects needed by your project. For example, the `fc-simesh` Matlab toolbox uses this toolbox to create the `siMesh` object containing all the simplices elements of the mesh.

This toolbox was only tested on Ubuntu 22.04.1 with Matlab R2022a.

Firstly, we explain how to configure the `ooGmsh` toolbox for using `gmsh`. Thereafter, we describe the `ooGmsh` 's functions which use `gmsh` to create mesh files.

2 Installation

2.1 Installation automatic, all in one (recommended)

For automatic installation, one has to get/download the install file

`mfc_oogmsh_install.m`

or get it on the dedicated web page. Thereafter, one run it under Matlab. This command download, extract and configure the *fc-oogmsh* toolbox and all the required toolboxes in the current directory.

By default, the gmsh binary is supposed to be located in

- <USERDIR>/bin/gmsh under linux and MacOS,
- <USERDIR>/Softwares/GMSH/gmsh.exe under Windows.

To specify an other location one can do

```
>> mfc_oogmsh_install('gmsh_bin', GMSH)
```

where *GMSH* is the gmsh binary with path as a string. It's also possible, after installation, to change the gmsh binary by using the Matlab command

```
>> fc_oogmsh.configure('gmsh_bin','~/gmsh-4.11.0/bin/gmsh')
```

For example, to install this toolbox in ~/Matlab directory, one have to copy the file *mfc_oogmsh_install.m* in the ~/Matlab directory. Then in a Matlab terminal run the following commands

```
>> cd ~/Matlab
>> mfc_oogmsh_install
```

There is the output of the *mfc_oogmsh_install* command on a Linux computer:

```
Parts of the <fc-oogmsh> Matlab toolbox.
Copyright (C) 2017-2022 F. Cuvelier

1- Downloading and extracting the toolboxes
2- Setting the <fc-oogmsh> toolbox
Write in ~/Matlab/fc-oogmsh-full/fc_oogmsh-0.2.4/configure_loc.m ...
3- Using toolboxes :
->          fc-tools : 0.0.34
->          fc-bench : 0.1.3
->          fc-amat : 0.1.3
->          fc-meshtools : 0.1.4
->          fc-graphics4mesh : 0.1.5
with          fc-oogmsh : 0.2.4
*** Using instructions
To use the <fc-oogmsh> toolbox:
addpath('~/Matlab/fc-oogmsh-full/fc_oogmsh-0.2.4')
fc_oogmsh.init()

See ~/Matlab/mfc_oogmsh_set.m
```

The complete toolbox (i.e. with all the other needed toolboxes) is stored in the directory
~/Matlab/fc-oogmsh-full
and, for each Matlab session, one has to set the toolbox by:

```
>> addpath('~/Matlab/fc-oogmsh-full/fc_oogmsh-0.2.4')
>> fc_oogmsh.init()
```

If it's the first time the *fc_oogmsh.init()* function is used, then its output is

```
Try to use default parameters!
Use fc_tools.configure to configure.
Write in ~/Matlab/fc-oogmsh-full/fc_tools-0.0.34/configure_loc.m ...
Try to use default parameters!
Use fc_bench.configure to configure.
Write in ~/Matlab/fc-oogmsh-full/fc_bench-0.1.3/configure_loc.m ...
Try to use default parameters!
Use fc_amat.configure to configure.
Write in ~/Matlab/fc-oogmsh-full/fc_amat-0.1.3/configure_loc.m ...
Try to use default parameters!
Use fc_meshtools.configure to configure.
Write in ~/Matlab/fc-oogmsh-full/fc_meshtools-0.1.4/configure_loc.m ...
Try to use default parameters!
Use fc_graphics4mesh.configure to configure.
Write in ~/Matlab/fc-oogmsh-full/fc_graphics4mesh-0.1.5/configure_loc.m ...
Using fc_oogmsh[0.2.4] with fc_tools[0.0.34], fc_bench[0.1.3], fc_amat[0.1.3], ...
fc_meshtools[0.1.4], fc_graphics4mesh[0.1.5].
Configured to use gmsh 4.11.0 with default MSH file format version 4.1
```

Otherwise, the output of the *fc_oogmsh.init()* function is

```
Using fc_oogmsh[0.2.4] with fc_tools[0.0.34], fc_bench[0.1.3], fc_amat[0.1.3], ...
fc_meshtools[0.1.4], fc_graphics4mesh[0.1.5].
Configured to use gmsh 4.11.0 with default MSH file format version 4.1
```

For **uninstalling**, one just has to delete directory:

~/Matlab/fc-oogmsh-full

3 gmsh interface

All functions provided in this section use **gmsh** to create a mesh file from a **gmsh** geometry script file (extension *.geo*).

3.1 function `fc_oogmsh.gmsh.buildmesh2d`

This function uses **gmsh** and a *.geo* file (describing a 2D-geometry) to generate a 2D-mesh.

Syntaxe

```
meshfile=fc_oogmsh.gmsh.buildmesh2d(geofile,N)
meshfile=fc_oogmsh.gmsh.buildmesh2d(geofile,N,Name,Value)
```

Description

`meshfile=fc_oogmsh.gmsh.buildmesh2d(geofile,N)` create a 2D-mesh using **gmsh** and the *geo* file *geofile* (without path). The integer **N** has two functions : numbering the name of the generated mesh as `<geofile without extension and path> + <-N.msh>` and passing this number to **gmsh** via the option `"-setnumber N <N>".` Usually we used this parameter in **gmsh** to set the prescribed mesh element size at the points. (see given *geo* files)

As output return a file name (with full path) corresponding to the mesh generated by **gmsh**.

`meshfile=fc_oogmsh.gmsh.buildmesh2d(geofile,N,Name,Value, ...)` specifies function options using one or more **Name,Value** pair arguments. The **Name** options can be

- **'geodir'** : to specify the directory of the *geo* file *geofile*,
- **'meshdir'** : to specify the directory where the mesh file will be written,
- **'meshfile'** : to specify the name of the mesh file (with path and *.msh* extension),
- **'check'** : to perform various consistency checks on mesh with **gmsh**, if **Value** is **true**. (default : **false**)
- **'force'** : to force meshing even if the mesh file already exists if **Value** is **true** (default : **false**)
- **'verbose'** : to specify the degree of verbosity (0, silence; 2, default; ...)
- **'strings'** : cells array of strings corresponding to **gmsh** options given with `-string "..."` (default empty) (see **gmsh** documentation). For example, **Value** could be:

```
{'Mesh.Algorithm=1;', 'Mesh.ScalingFactor=2;'}
```

- **'MshFileVersion'** : to specify the MSH file format version. **Value** could be
 - **'2.2'** if **gmsh** version $\geq$ 2.16.0,
 - **'4.0'** if **gmsh** version $\geq$ 4.0.0,
 - **'4.1'** if **gmsh** version $\geq$ 4.1.0.

Examples All the following examples use the *.geo* file **condenser11.geo** which is in the directory **geodir** of the toolbox.

Matlab code with output

```
disp('****_fc_oogmsh.gmsh.buildmesh2d_:1st_call')
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser11',25,'force',true);
disp('****_fc_oogmsh.gmsh.buildmesh2d_:2nd_call')
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser11',25);
```

```
**** fc_oogmsh.gmsh.buildmesh2d : 1st call
[fc-oogmsh] Input file : <fc-oogmsh>/geodir/2d/condenser11.geo
[fc-oogmsh] Starting building mesh <fc-oogmsh>/meshes/condenser11-25.msh with gmsh 4.11.0
[fc-oogmsh] Using command : gmsh -2 -setnumber N 25 -string "Mesh.MeshFileVersion=4.1;" <fc-oogmsh>/geodir/2d/condenser11.geo -o ...
<fc-oogmsh>/meshes/condenser11-25.msh
Be patient...
[fc-oogmsh] Using gmsh 4.11.0 to write MSH file format version 4.1 in <fc-oogmsh>/meshes/condenser11-25.msh
**** fc_oogmsh.gmsh.buildmesh2d : 2nd call
[fc-oogmsh] Input file : <fc-oogmsh>/geodir/2d/condenser11.geo
[fc-oogmsh] Mesh file <fc-oogmsh>/meshes/condenser11-25.msh [version 4.1] already exists.
-> Use "force" flag to rebuild if needed.
```

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser11',25,'force',true,...  
    'verbose',4, 'strings',{'Mesh.Algorithm=1;', 'Mesh.ScalingFactor=2;'});
```

```
[fc-oogmsh] Input file : <fc-oogmsh>/geodir/2d/condenser11.geo  
[fc-oogmsh] Overwriting mesh file <fc-oogmsh>/meshes/condenser11-25.msh  
[fc-oogmsh] Starting building mesh <fc-oogmsh>/meshes/condenser11-25.msh with gmsh 4.11.0  
[fc-oogmsh] Using command : '/home/cuvellier/bin/gmsh' -2 -setnumber N 25 -string "Mesh.Algorithm=1;Mesh.ScalingFactor=2;Mesh.MshFileVersion=4.1;" ...  
    '/home/cuvellier/Travail/Recherche/Matlab/fc-config/build/tmpdir/packages/fc_oogmsh-0.2.4/geodir/2d/condenser11.geo' -o ...  
    '/home/cuvellier/Travail/Recherche/Matlab/fc-config/build/tmpdir/packages/fc_oogmsh-0.2.4/meshes/condenser11-25.msh'  
Be patient...  
[fc-oogmsh] Starting building mesh <fc-oogmsh>/meshes/condenser11-25.msh with gmsh 4.11.0  
[fc-oogmsh] Using command : gmsh -2 -setnumber N 25 -string "Mesh.Algorithm=1;Mesh.ScalingFactor=2;Mesh.MshFileVersion=4.1;" ...  
    <fc-oogmsh>/geodir/2d/condenser11.geo -o <fc-oogmsh>/meshes/condenser11-25.msh  
Be patient...  
[fc-oogmsh] gmsh output :  
Info : Running '/home/cuvellier/bin/gmsh -2 -setnumber N 25 -string Mesh.Algorithm=1;Mesh.ScalingFactor=2;Mesh.MshFileVersion=4.1;" ...  
    <fc-oogmsh>/geodir/2d/condenser11.geo -o <fc-oogmsh>/meshes/condenser11-25.msh' [Gmsh 4.11.0, 1 node, max. 1 thread]  
Info : Started on Wed Dec 21 12:43:14 2022  
Info : Reading '<fc-oogmsh>/geodir/2d/condenser11.geo'...  
Info : Reading '<fc-oogmsh>/geodir/2d/options01.data.geo'...  
Info : Done reading '<fc-oogmsh>/geodir/2d/options01.data.geo'  
Info : Reading '<fc-oogmsh>/geodir/2d/shape_functions.geo'...  
Info : Done reading '<fc-oogmsh>/geodir/2d/shape_functions.geo'  
Warning : Skipping unknown surface 115 in physical surface 1  
Warning : Skipping unknown surface 125 in physical surface 3  
Warning : Skipping unknown surface 135 in physical surface 5  
Warning : Skipping unknown surface 145 in physical surface 7  
Warning : Skipping unknown surface 115 in physical surface 1  
Warning : Skipping unknown surface 125 in physical surface 3  
Warning : Skipping unknown surface 135 in physical surface 5  
Warning : Skipping unknown surface 145 in physical surface 7  
Info : Removing duplicate mesh nodes...  
Info : Found 0 duplicate nodes  
Info : No duplicate nodes found  
Warning : Skipping unknown surface 115 in physical surface 1  
Warning : Skipping unknown surface 125 in physical surface 3  
Warning : Skipping unknown surface 135 in physical surface 5  
Warning : Skipping unknown surface 145 in physical surface 7  
Info : Done reading '<fc-oogmsh>/geodir/2d/condenser11.geo'  
Info : Meshing 1D...  
Info : [ 0%] Meshing curve 101 (Line)  
Info : [ 10%] Meshing curve 102 (Line)  
Info : [ 10%] Meshing curve 103 (Line)  
Info : [ 10%] Meshing curve 104 (Line)  
Info : [ 10%] Meshing curve 106 (Circle)  
Info : [ 20%] Meshing curve 107 (Circle)  
Info : [ 20%] Meshing curve 108 (Circle)  
Info : [ 20%] Meshing curve 109 (Circle)  
Info : [ 20%] Meshing curve 111 (Circle)  
Info : [ 30%] Meshing curve 112 (Circle)  
Info : [ 30%] Meshing curve 113 (Circle)  
Info : [ 30%] Meshing curve 114 (Circle)  
Info : [ 30%] Meshing curve 116 (Circle)  
Info : [ 40%] Meshing curve 117 (Circle)  
Info : [ 40%] Meshing curve 118 (Circle)  
Info : [ 40%] Meshing curve 119 (Circle)  
Info : [ 40%] Meshing curve 121 (Circle)  
Info : [ 50%] Meshing curve 122 (Circle)  
Info : [ 50%] Meshing curve 123 (Circle)  
Info : [ 50%] Meshing curve 124 (Circle)  
Info : [ 50%] Meshing curve 126 (Circle)  
Info : [ 60%] Meshing curve 127 (Circle)  
Info : [ 60%] Meshing curve 128 (Circle)  
Info : [ 60%] Meshing curve 129 (Circle)  
Info : [ 60%] Meshing curve 131 (Circle)  
Info : [ 70%] Meshing curve 132 (Circle)  
Info : [ 70%] Meshing curve 133 (Circle)  
Info : [ 70%] Meshing curve 134 (Circle)  
Info : [ 70%] Meshing curve 136 (Circle)  
Info : [ 80%] Meshing curve 137 (Circle)  
Info : [ 80%] Meshing curve 138 (Circle)  
Info : [ 80%] Meshing curve 139 (Circle)  
Info : [ 80%] Meshing curve 141 (Circle)  
Info : [ 90%] Meshing curve 142 (Circle)  
Info : [ 90%] Meshing curve 143 (Circle)  
Info : [ 90%] Meshing curve 144 (Circle)  
Info : [ 90%] Meshing curve 146 (Circle)  
Info : [100%] Meshing curve 147 (Circle)  
Info : [100%] Meshing curve 148 (Circle)  
Info : [100%] Meshing curve 149 (Circle)  
Info : Done meshing 1D (Wall 0.0106941s, CPU 0.033143s)  
Info : Meshing 2D...  
Info : [ 0%] Meshing surface 105 (Plane, MeshAdapt)  
Info : [ 20%] Meshing surface 110 (Plane, MeshAdapt)  
Info : [ 40%] Meshing surface 120 (Plane, MeshAdapt)  
Info : [ 50%] Meshing surface 130 (Plane, MeshAdapt)  
Info : [ 70%] Meshing surface 140 (Plane, MeshAdapt)  
Info : [ 90%] Meshing surface 150 (Plane, MeshAdapt)  
Info : Done meshing 2D (Wall 0.222733s, CPU 0.358376s)  
Info : 3236 nodes 6605 elements  
Info : Writing '<fc-oogmsh>/meshes/condenser11-25.msh'...  
Info : Done writing '<fc-oogmsh>/meshes/condenser11-25.msh'  
Info : Stopped on Wed Dec 21 12:43:15 2022 (From start: Wall 0.261739s, CPU 0.429871s)  
  
[fc-oogmsh] Using gmsh 4.11.0 to write MSH file format version 4.1 in <fc-oogmsh>/meshes/condenser11-25.msh
```

3.2 function `fc_oogmsh.gmsh.buildmesh3d`

This function uses `gmsh` and a `.geo` file (describing a 3D-geometry) to generate a 3D-mesh. See function `gmsh.buildmesh2d` for usage and options (section 3.1).

3.3 function `fc_oogmsh.gmsh.buildmesh3ds`

This function uses `gmsh` and a `.geo` file (describing a 3D surface geometry or a 3D-geometry) to generate a 3D surface mesh. See function `gmsh.buildmesh2d` for usage and options (section 3.1).

3.4 function `fc_oogmsh.gmsh.buildpartmesh2d`

This function uses `gmsh` and a `.msh` file (containing a 2D-mesh) to generate a 2D partitioned mesh.

Syntaxe

```
partmeshfile=fc_oogmsh.gmsh.buildpartmesh2d(meshfile,np)
partmeshfile=fc_oogmsh.gmsh.buildpartmesh2d(meshfile,np,Name,Value)
```

Description

`partmeshfile=fc_oogmsh.gmsh.buildpartmesh2d(meshfile,np)` create a 2D partitioned mesh using `gmsh` and the `msh` file `meshfile` (with path). The integer `np` is the number of partitions. As output return a file name (with full path) corresponding to the partitioned mesh generated by `gmsh`. The output file name is construct as following : `<meshfile without extension>-part<np>.msh`

`partmeshfile=fc_oogmsh.gmsh.buildpartmesh2d(meshfile,np,Name,Value, ...)` specifies function options using one or more `Name,Value` pair arguments. The `Name` options can be

- `'savedir'` : to specify the directory where the partitioned mesh file will be written,
- `'check'` : to perform various consistency checks on mesh with `gmsh`, if `Value` is `true`. (default : `false`)
- `'force'` : to force meshing even if the mesh file already exists if `Value` is `true` (default : `false`)
- `'verbose'` : to specify the degree of verbosity (0, silence; 2, default; ...)
- `'strings'` : cells array of strings corresponding to `gmsh` options given with `-string "..."` (default empty) (see `gmsh` documentation)
- `'MshFileVersion'` : to specify the MSH file format version. `Value` could be
 - `'2.2'` if `gmsh` version $\geq 2.16.0$,
 - `'4.0'` if `gmsh` version $\geq 4.0.0$,
 - `'4.1'` if `gmsh` version $\geq 4.1.0$.

Examples

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser11',25,'verbose',0);
pmfile=fc_oogmsh.gmsh.buildpartmesh2d(meshfile,5,'force',true);
```

```
[fc-oogmsh] Input file : <fc-oogmsh>/meshes/condenser11-25.msh
[fc-oogmsh] Starting building mesh <fc-oogmsh>/meshes/condenser11-25-part5.msh with gmsh 4.11.0
[fc-oogmsh] Using command : gmsh -2 -part 5 -string "Mesh.MshFileVersion=4.1;" -saveall <fc-oogmsh>/meshes/condenser11-25.msh -o ...
<fc-oogmsh>/meshes/condenser11-25-part5.msh
Be patient...
[fc-oogmsh] Using gmsh 4.11.0 to write MSH file format version 4.1 in <fc-oogmsh>/meshes/condenser11-25-part5.msh
```

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser11',25,'verbose',0);
pmfile=fc_oogmsh.gmsh.buildpartmesh2d(meshfile,5,'force',true,'verbose',4,...
    'strings',{'Mesh.MetisAlgorithm=3;'});
```

```
[fc-oogmsh] Input file : <fc-oogmsh>/meshes/condenser11-25.msh
[fc-oogmsh] Overwriting mesh file <fc-oogmsh>/meshes/condenser11-25-part5.msh
[fc-oogmsh] Starting building mesh <fc-oogmsh>/meshes/condenser11-25-part5.msh with gmsh 4.11.0
[fc-oogmsh] Using command : '/home/cuvellier/bin/gmsh' -2 -part 5 -string "Mesh.MetisAlgorithm=3;Mesh.MshFileVersion=4.1;" -saveall ...
'/home/cuvellier/Travail/Recherch/Matlab/fc-config/build/tmpdir/packages/fc_oogmsh-0.2.4/meshes/condenser11-25.msh' -o ...
'/home/cuvellier/Travail/Recherch/Matlab/fc-config/build/tmpdir/packages/fc_oogmsh-0.2.4/meshes/condenser11-25-part5.msh'
Be patient...
[fc-oogmsh] Starting building mesh <fc-oogmsh>/meshes/condenser11-25-part5.msh with gmsh 4.11.0
[fc-oogmsh] Using command : gmsh -2 -part 5 -string "Mesh.MetisAlgorithm=3;Mesh.MshFileVersion=4.1;" -saveall <fc-oogmsh>/meshes/condenser11-25.msh -o ...
<fc-oogmsh>/meshes/condenser11-25-part5.msh
Be patient...
[fc-oogmsh] gmsh output :
Info : Running '/home/cuvellier/bin/gmsh -2 -part 5 -string Mesh.MetisAlgorithm=3;Mesh.MshFileVersion=4.1; -saveall <fc-oogmsh>/meshes/condenser11-25.msh -o ...
<fc-oogmsh>/meshes/condenser11-25-part5.msh' [Gmsh 4.11.0, 1 node, max. 1 thread]
Info : Started on Wed Dec 21 12:43:37 2022
Info : Reading '<fc-oogmsh>/meshes/condenser11-25.msh'...
Info : 95 entities
Info : 3227 nodes
Info : 6556 elements
Info : Done reading '<fc-oogmsh>/meshes/condenser11-25.msh'
Info : Meshing 1D...
Info : Done meshing 1D (Wall 6.7828e-05s, CPU 0.000129s)
Info : Meshing 2D...
Info : Done meshing 2D (Wall 4.1839e-05s, CPU 7.7e-05s)
Info : 3236 nodes 6605 elements
Info : Partitioning mesh...
Info : Running METIS with npart:5, sizeof(idxt):64, ptype:default, ufactor:default, ctype:schem, rtype:greedy, objtype:cut, minconn:default
Info : 5 partitions, 156 total edge-cuts
Info : Done partitioning mesh (Wall 0.0174086s, CPU 0.030171s)
Info : - Repartition of 49 points: 5(min) 22(max) 9.8(avg)
Info : - Repartition of 360 lines: 62(min) 92(max) 72(avg)
Info : - Repartition of 6196 triangles: 1239(min) 1240(max) 1239.2(avg)
Info : Creating partition topology...
Info : - Creating partition curves
Info : - Creating partition points
Info : Done creating partition topology (Wall 0.00285845s, CPU 0.017419s)
Info : Writing '<fc-oogmsh>/meshes/condenser11-25-part5.msh'...
Info : Done writing '<fc-oogmsh>/meshes/condenser11-25-part5.msh'
Info : Stopped on Wed Dec 21 12:43:37 2022 (From start: Wall 0.0753202s, CPU 0.141083s)

[fc-oogmsh] Using gmsh 4.11.0 to write MSH file format version 4.1 in <fc-oogmsh>/meshes/condenser11-25-part5.msh
```

3.5 function `fc_oogmsh.gmsh.buildpartmesh3d`

This function uses `gmsh` and a `.msh` file (containing of a 3D-mesh) to generate a 3D partioned mesh. See function `gmsh.buildpartmesh2d` for usage and options (section 3.4).

3.6 function `fc_oogmsh.gmsh.buildpartmesh3ds`

This function uses `gmsh` and a `.msh` file (containing of a 3D surface mesh) to generate a 3D partioned surface mesh. See function `gmsh.buildpartmesh2d` for usage and options (section 3.4).

3.7 function `fc_oogmsh.gmsh.buildPartRectangle`

This function uses `gmsh` and the `geodir/rectanglepart.geo` file to generate a 2D regular partioned mesh of the rectangle $[0, L_x] \times [0, L_y]$ with $N_x \times N_y$ partitions.

Syntaxe

```
meshfile=fc_oogmsh.gmsh.buildpartrectangle(Lx,Ly,Nx,Ny,N)

meshfile=fc_oogmsh.gmsh.buildpartrectangle(Lx,Ly,Nx,Ny,N, Name, Value)
```

Description

`meshfile=fc_oogmsh.gmsh.buildpartrectangle(Lx,Ly,Nx,Ny,N)` create a 2D regular partitioned mesh using `gmsh` of the rectangle $[0, L_x] \times [0, L_y]$ with $N_x \times N_y$ partitions. The `N` parameter is passed to `gmsh` to set the prescribed mesh element size at the points
As output return a file name (with full path) corresponding to the partitioned mesh generated by `gmsh`. The default output file name is construct as following :

```
sprintf('rectanglepart-Lx%.3f-Ly%.3f-Nx%d-Ny%d-N%d.msh',Lx,Ly,Nx,Ny,N)
```

`meshfile=fc_oogmsh.gmsh.buildpartrectangle(Lx,Ly,Nx,Ny,N,Name,Value, ...)` specifies function options using one or more `Name,Value` pair arguments (see the `fc_oogmsh.gmsh.buildmesh2d`, section 3.1).

Examples All the following examples ...

Matlab code with output

```
pmfile=fc_oogmsh.gmsh.buildpartrectangle(1,1,3,2,100,'force',true);
```

```
[fc-oogmsh] Input file : <fc-oogmsh>/geodir/2d/rectanglepart.geo
[fc-oogmsh] Starting building mesh <fc-oogmsh>/meshes/rectanglepart-Lx1.000-Ly1.000-Nx3-Ny2-N100.msh with gmsh 4.11.0
[fc-oogmsh] Using command : gmsh -2 -string "Mesh.MshFileVersion=4.1;" -setnumber N 100 -setnumber NX 3 -setnumber NY 2 -setnumber LX 1 -setnumber LY 1 ...
<fc-oogmsh>/geodir/2d/rectanglepart.geo -o <fc-oogmsh>/meshes/rectanglepart-Lx1.000-Ly1.000-Nx3-Ny2-N100.msh
Be patient...
[fc-oogmsh] Using gmsh 4.11.0 to write MSH file format version 4.1 in <fc-oogmsh>/meshes/rectanglepart-Lx1.000-Ly1.000-Nx3-Ny2-N100.msh
```

Matlab code with output

```
pmfile=fc_oogmsh.gmsh.buildpartrectangle(1,1,3,2,100,'verbose',4, ...
'force',true,'meshfile','./toto.msh');
```

```
[fc-oogmsh] Input file : <fc-oogmsh>/geodir/2d/rectanglepart.geo
[fc-oogmsh] Starting building mesh ./toto.msh with gmsh 4.11.0
[fc-oogmsh] Using command : /home/cuvelier/bin/gmsh -2 -string "Mesh.MshFileVersion=4.1;" -setnumber N 100 -setnumber NX 3 -setnumber NY 2 -setnumber LX 1 ...
-setnumber LY 1 /home/cuvelier/Travail/Recherche/Matlab/fc-config/build/tmpdir/packages/fc-oogmsh-0.2.4/geodir/2d/rectanglepart.geo' -o './toto.msh'
Be patient...
[fc-oogmsh] Starting building mesh ./toto.msh with gmsh 4.11.0
[fc-oogmsh] Using command : gmsh -2 -string "Mesh.MshFileVersion=4.1;" -setnumber N 100 -setnumber NX 3 -setnumber NY 2 -setnumber LX 1 -setnumber LY 1 ...
<fc-oogmsh>/geodir/2d/rectanglepart.geo -o ./toto.msh
Be patient...
[fc-oogmsh] gmsh output :
Info : Running /home/cuvelier/bin/gmsh -2 -string Mesh.MshFileVersion=4.1; -setnumber N 100 -setnumber NX 3 -setnumber NY 2 -setnumber LX 1 -setnumber LY 1 ...
<fc-oogmsh>/geodir/2d/rectanglepart.geo -o ./toto.msh' [Gmsh 4.11.0, 1 node, max. 1 thread]
Info : Started on Wed Dec 21 12:44:00 2022
Info : Reading '<fc-oogmsh>/geodir/2d/rectanglepart.geo'...
Info : Reading '<fc-oogmsh>/geodir/2d/partitions01.data.geo'...
Info : Done reading '<fc-oogmsh>/geodir/2d/partitions01.data.geo'
Info : Reading '<fc-oogmsh>/geodir/2d/partitions_shape.geo'...
Info : Done reading '<fc-oogmsh>/geodir/2d/partitions_shape.geo'
Info : Done reading '<fc-oogmsh>/geodir/2d/rectanglepart.geo'
Info : Meshing 1D...
Info : [ 0%] Meshing curve 1 (Line)
Info : [ 10%] Meshing curve 2 (Line)
Info : [ 20%] Meshing curve 3 (Line)
Info : [ 20%] Meshing curve 4 (Line)
Info : [ 30%] Meshing curve 5 (Line)
Info : [ 30%] Meshing curve 6 (Line)
Info : [ 40%] Meshing curve 7 (Line)
Info : [ 50%] Meshing curve 8 (Line)
Info : [ 50%] Meshing curve 9 (Line)
Info : [ 60%] Meshing curve 10 (Line)
Info : [ 60%] Meshing curve 11 (Line)
Info : [ 70%] Meshing curve 12 (Line)
Info : [ 80%] Meshing curve 13 (Line)
Info : [ 80%] Meshing curve 14 (Line)
Info : [ 90%] Meshing curve 15 (Line)
Info : [ 90%] Meshing curve 16 (Line)
Info : [100%] Meshing curve 17 (Line)
Info : Done meshing 1D (Wall 0.00298852s, CPU 0.008101s)
Info : Meshing 2D...
Info : [ 0%] Meshing surface 19 (Plane, Frontal-Delaunay)
Info : [ 20%] Meshing surface 21 (Plane, Frontal-Delaunay)
Info : [ 40%] Meshing surface 23 (Plane, Frontal-Delaunay)
Info : [ 50%] Meshing surface 25 (Plane, Frontal-Delaunay)
Info : [ 70%] Meshing surface 27 (Plane, Frontal-Delaunay)
Info : [ 90%] Meshing surface 29 (Plane, Frontal-Delaunay)
Info : Done meshing 2D (Wall 0.525289s, CPU 0.663312s)
Info : 12087 nodes 24486 elements
Info : Writing './toto.msh'...
Info : Done writing './toto.msh'
Info : Stopped on Wed Dec 21 12:44:00 2022 (From start: Wall 0.593461s, CPU 0.748516s)

[fc-oogmsh] Using gmsh 4.11.0 to write MSH file format version 4.1 in ./toto.msh
```

4 ooGmsh4 class (version 4.x)

The `ooGMSH4` class can be used to read `gmsh` mesh files with the MSH ASCII file format version 4.1 since `gmsh` 4.1.0 ([4], section 9.1) or version 4.0 since `gmsh` 4.0.0.

The `gmsh`'s native "MSH" file format (version 4.x) is used to store meshes and associated post-processing datasets either save as an ASCII file or a binary file with extension `.msh`. The focus of the `ooGMSH4` class is to read only meshes contained in an ASCII file. Currently, it is not planned to read post-processing datasets.

As described in [4], section 9.1: *the MSH file format version 4 (current revision: version 4.1) contains one mandatory section giving information about the file (\$MeshFormat), followed by several optional sections defining the physical group names (\$PhysicalName), the elementary geometrical enti-*

ties (`$Entities`), the partitioned entities (`$PartitionedEntities`), the nodes (`$Nodes`), the elements (`$Elements`), the periodicity relations (`$Periodic`), the ghost elements (`$GhostElements`) and the post-processing datasets (`$NodeData`, `$ElementData`, `$ElementNodeData`).

For each section, the `ooGmsh4` class has a property with corresponding name. The properties of this class are:

Properties of <code>ooGmsh4</code> class	
<code>dim</code>	: space dimension (2 or 3)
<code>nq</code>	: number of nodes/vertices.
<code>q</code>	: nodes/vertices array with dimension <code>dim-by-nq</code> .
<code>toGlobal</code>	: ...
<code>MeshFormat</code>	: structure
<code>PhysicalNames</code>	: (optional), array of <code>PhysicalName</code> structure
<code>Entities</code>	: structure
<code>PartitionedEntities</code>	: (optional) structure
<code>Nodes</code>	: structure
<code>Elements</code>	: structure
<code>PeriodicLinks</code>	: (optional), array of <code>PeriodicLink</code> structure

The structures `MeshFormat`, `PhysicalNames`, `Entities`, `PartitionedEntities`, `Nodes`, `Elements` and `PeriodicLinks` are described in section 4.2. In the following subsections, `Gh` is an `ooGmsh4` object.

4.1 Methods

4.1.1 `ooGmsh4` constructor

The `ooGmsh4` class have only one constructor :

```
Gh=fc_ooGmsh.ooGmsh4( meshfile )
Gh=fc_ooGmsh.ooGmsh4( meshfile , 'verbosity' , Value )
```

where `meshfile` is the name of ... a mesh file. The '`verbosity`' Key/Value option can be used to print some informations, when reading the file `meshfile`, if `Value` is `true`. Default is `false`

Matlab code with output

```
fprintf('1) Building the mesh\n')
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser',10, ...
    'verbose',0,'force',true);
fprintf('2) Reading the mesh\n');
Gh = fc_oogmsh.ooGmsh4(meshfile,'verbose',true);
fprintf('=> Gh is an ooGmsh4 object containing a MSH file version ...
    %s\n',Gh.MeshFormat.version)
fprintf('3) Displaying Gh\n');
Gh
```

```
1) Building the mesh
2) Reading the mesh
Optional string "$PhysicalNames" not found
Reading $Entities section seem OK
Optional string "$PartitionedEntities" not found
Reading $Nodes section seem OK
Reading $Elements section seem OK
Optional string "$Periodic" not found
-> Gh is an ooGmsh4 object containing a MSH file version 4.1
3) Displaying Gh

Gh =

fc_oogmsh.ooGmsh4 with properties:
    q: (2x8162 double)
    nq: 8162 double
    dim: 2 double
    d: 2 double
    toGlobal: (1x8162 double)
    meshfile: (1x110 char)
    partitionedfile: 0 logical
    MeshFormat: (1x1 struct)
    PhysicalNames: []
    Entities: (1x1 struct)
    PartitionedEntities: []
    Nodes: (1x1 struct)
    Elements: (1x1 struct)
    PeriodicLinks: []
    Info: (1x1 struct)
    orders: 1 double
```

4.1.2 info method

```
info(Gh)
Gh.info()
Gh.info(Key, Value, ...)
```

Description

Gh.info()

print informations on class fields with 3 levels of recursivity (i.e. field of field of field).

Gh.info(Key, Value, ...)

specifies function options using one or more **Key,Value** pair arguments. The **Key** options can be

- **'maxlevel'** : level of recursivity, default is 3.
- **'tab'** : number of space characters between two levels of recursivity, default is 4.

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser',6,'verbose',0,'force',true);
Gh = fc_oogmsh.oogmsh4(meshfile);
Gh.info('maxlevel',2);
```

```
fc_oogmsh.oogmsh4 with properties:
[1] q : [2 3036] double
[1] nq : [1 1] double
[1] dim : [1 1] double
[1] d : [1 1] double
[1] toGlobal : [1 3036] double
[1] meshfile : [1 109] char
[1] partitionedfile : [1 1] logical
[1] MeshFormat : [1 1] struct
[2] version : [1 3] char
[2] file_type : [1 1] double
[2] data_size : [1 1] double
[1] PhysicalNames : [0 0] double
[1] Entities : [1 1] struct
[2] numPoints : [1 1] double
[2] Points : [1 11] struct
[2] numCurves : [1 1] double
[2] Curves : [1 10] struct
[2] numSurfaces : [1 1] double
[2] Surfaces : [1 1] struct
[2] numVolumes : [1 1] double
[2] Volumes : [1 0] struct
[1] PartitionedEntities : [0 0] double
[1] Nodes : [1 1] struct
[2] numEntityBlocks : [1 1] double
[2] numNodes : [1 1] double
[2] minNodeTag : [1 1] double
[2] maxNodeTag : [1 1] double
[2] EntityBlocks : [1 21] struct
[1] Elements : [1 1] struct
[2] numEntityBlocks : [1 1] double
[2] numElements : [1 1] double
[2] minElementTag : [1 1] double
[2] maxElementTag : [1 1] double
[2] EntityBlocks : [1 19] struct
[2] ElementTypes : [1 19] double
[1] PeriodicLinks : [0 0] double
[1] Info : [1 1] struct
[2] meshfile : [1 109] char
[1] orders : [1 1] double
```

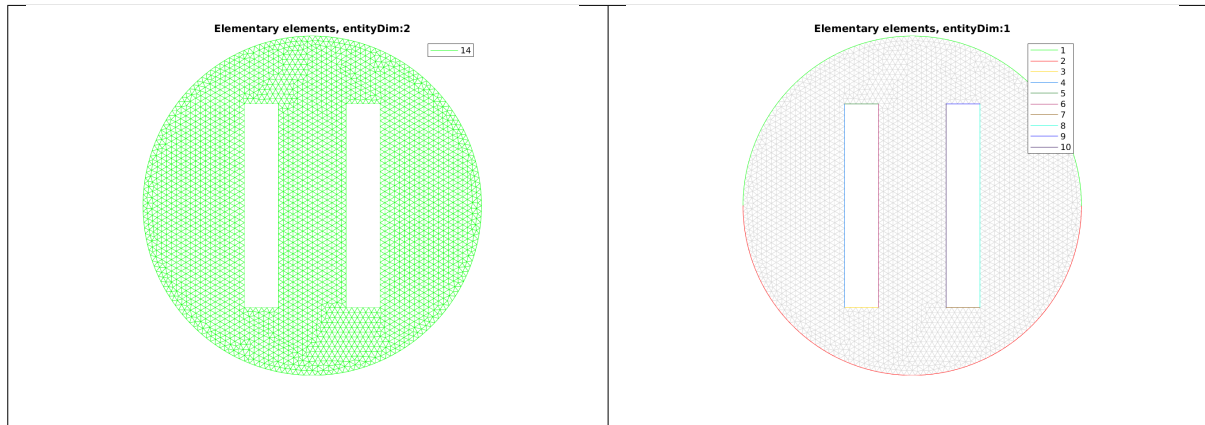


Figure 1: *Elementary Tag* elements of the *geofile* condenser.geo

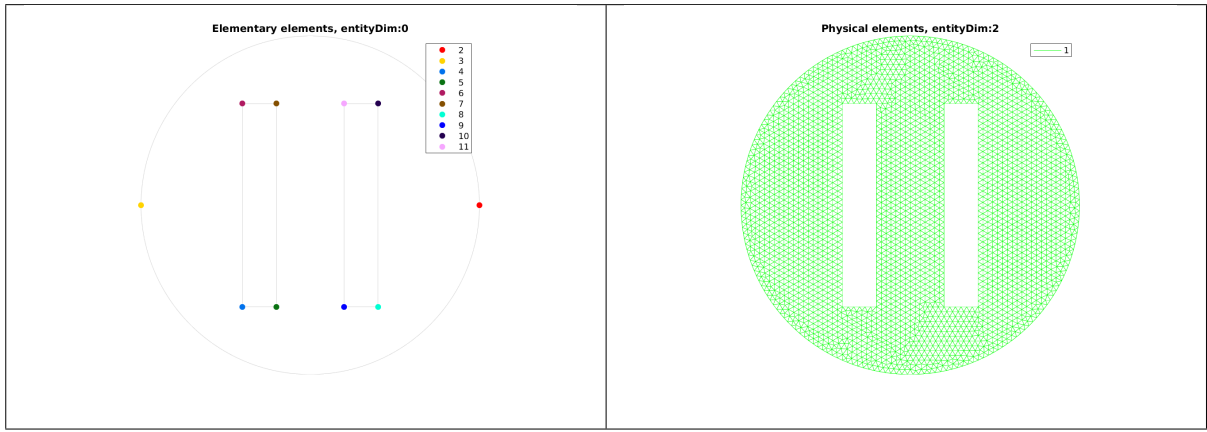


Figure 2: *Physical Tag* elements of the *geofile condenser.geo*

In the *geofile condenser.geo* the *Physical Tags* are created from the *Elementary Tags* as follow

```
...
Physical Line(1) = {1, 2};
Physical Line(98) = {5, 6, 3, 4};
Physical Line(99) = {9, 8, 7, 10};
Physical Surface(1) = {14};
```

4.1.3 `get_ElementaryTags` method

```
eltags=get_ElementaryTags(Gh, EltType)
eltags=Gh.get_ElementaryTags(EltType)
```

Description

```
eltags=Gh.get_ElementaryTags(EltType)
```

returns all the elementary tags associated with elements of type **EltType** as an array with unique elements. **EltType** is described in section ?? . For example, **EltType** is 1 for 2-nodes **line** (i.e 1-simplex of order 1), **EltType** is 2 for 3-nodes **triangle** (i.e 2-simplex of order 1) and **EltType** is 4 for 4-nodes **tetrahedron** (i.e 3-simplex of order 1).

Matlab code with output

```
eltags1=Gh.get_ElementaryTags(1)
eltags2=Gh.get_ElementaryTags(2)
```

```
eltags1 =
    1    2    3    4    5    6    7    8    9   10

eltags2 =
    14
```

4.1.4 `get_PhysicalTags` method

```
phtags=get_PhysicalTags(Gh, EltType)
phtags=Gh.get_PhysicalTags(EltType)
```

Description

```
phtags=Gh.get_PhysicalTags(EltType)
```

returns all the elementary tags associated with elements of type **EltType** as an array with unique elements.

Matlab code with output

```
phtags1=Gh.get_PhysicalTags(1)
phtags2=Gh.get_PhysicalTags(2)
```

```
phtags1 =
    1    98    99
```

```
phtags2 =
    1
```

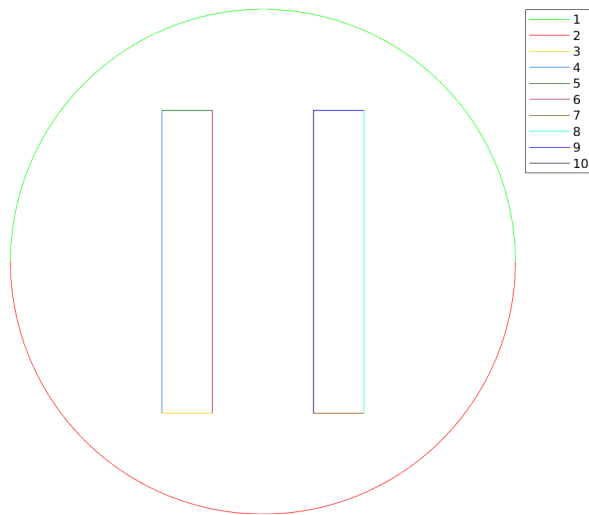
4.1.5 get_me_ElementaryTag method

```
me=get_me_ElementaryTag(Gh,EltType,EltTag)
me=Gh.get_me_ElementaryTag(EltType,EltTag)
```

Description

```
me=Gh.get_me_ElementaryTag(EltType,EltTag)
```

returns **me** the connectivity array of mesh elements of type and *elementary tag* given respectively by **EltType** and **EltTag**. This array is associated with the **Gh.q** nodes/vertices array.



```
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser',6,'verbose',0);
Gh = fc_oogmsh.ooGmsh4(meshfile);
eltags1=Gh.get_ElementaryTags(1);
n1=length(eltags1);
colors = fc_tools.graphics.selectColors(length(eltags1));
figure(1)
hold on
for i=1:n1
    me=Gh.get_me_ElementaryTag(1,eltags1(i));
    h(i)=fc_graphics4mesh.plotmesh(Gh.q,me,'color',colors(i,:));
    clegend{i}=num2str(eltags1(i));
end
legend(h,clegend,'Location','NorthEastOutside','AutoUpdate','off');
axis image;axis off
fc_tools.graphics.SaveAllFigsAsFiles('ooGmsh4_get_me_ElementaryTags', SaveOptions{:})
```

Listing 1: Plot curves mesh elements by using get_me_ElementaryTags function

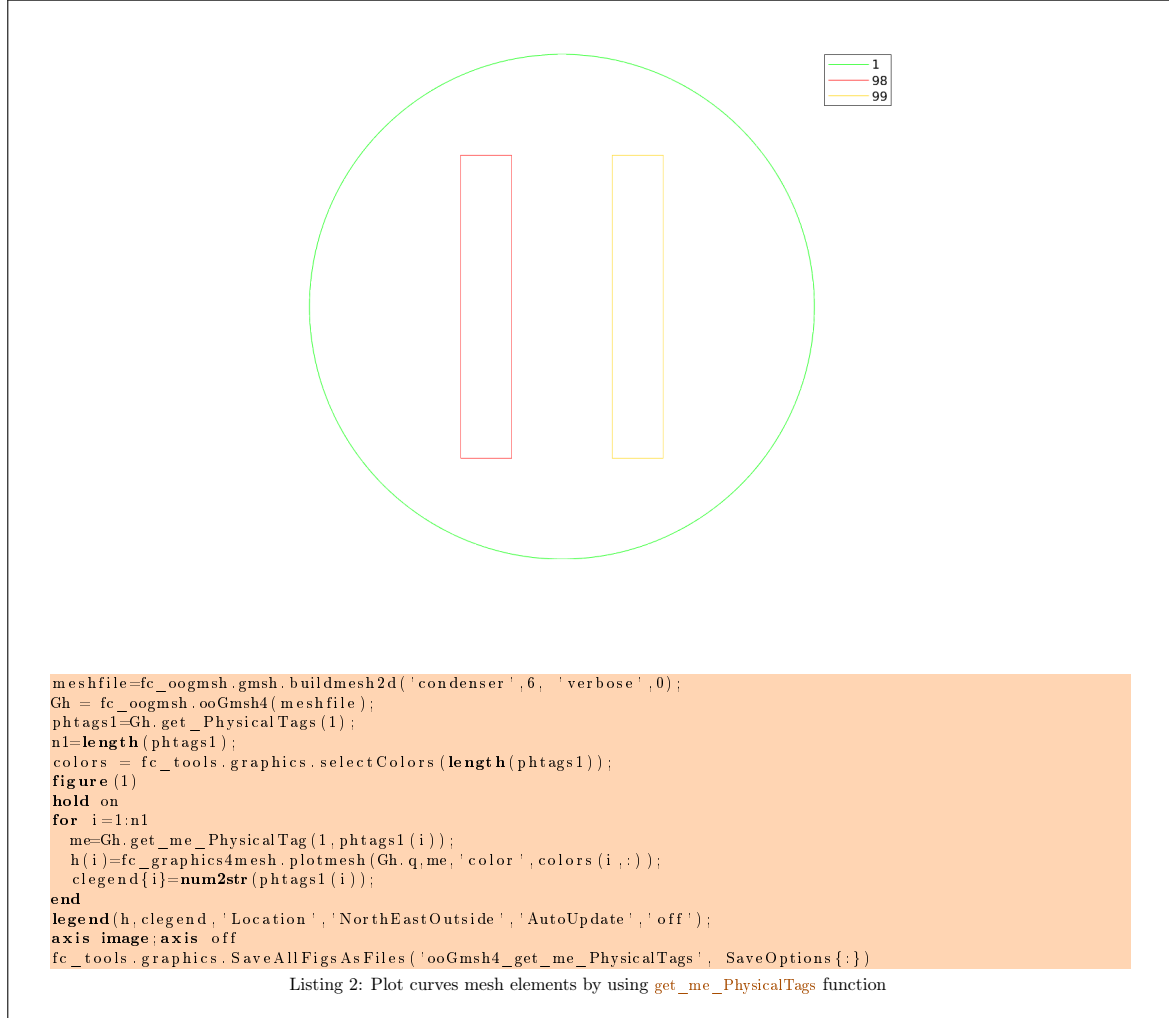
4.1.6 get_me_PhysicalTag method

```
me=get_me_PhysicalTag(Gh,EltType,PhyTag)
me=Gh.get_me_PhysicalTag(EltType,PhyTag)
```

Description

```
get_me_PhysicalTag(Gh,EltType,PhyTag)
```

returns **me** the connectivity array of mesh elements of type and *physical tag* given respectively by **EltType** and **PhyTag**. This array is associated with the **Gh.q** nodes/vertices array.



4.1.7 get_localmesh_ElementaryTag method

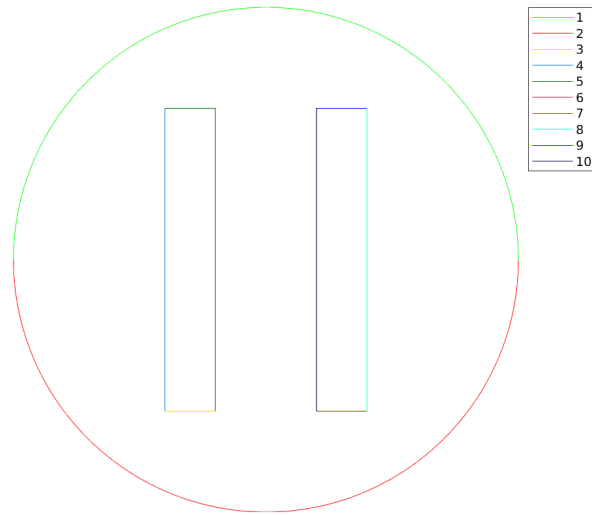
```
[q,me]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)
[q,me,toGlobal]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)
```

```
[q,me]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)
```

returns the *local* nodes/vertices array **q** and the *local* connectivity array **me** of the element of type **EltType** and with *elementary tag* given by **EltTag**.

```
[q,me,toGlobal]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)
```

Also returns the *global* tags array **toGlobal** such that **Gh.q(:,toGlobal)** is equal to **q**.



```

meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser',6, 'verbose',0);
Gh = fc_oogmsh.ooGmsh4(meshfile);
eltags1=Gh.get_ElementaryTags(1); % 1: 2-nodes line
n1=length(eltags1);
colors = fc_tools.graphics.selectColors(length(eltags1));
figure(1)
hold on
for i=1:n1
    [q,me]=Gh.get_localmesh_ElementaryTag(1,eltags1(i));
    h(i)=fc_graphics4mesh.plotmesh(q,me,'color',colors(i,:));
    clegend{i}=num2str(eltags1(i));
end
legend(h,clegend,'Location','NorthEastOutside','AutoUpdate','off');
axis image;axis off
fc_tools.graphics.SaveAllFigsAsFiles('ooGmsh4_get_localmesh_ElementaryTag', SaveOptions{:})

```

Listing 3: Plot 2-nodes line mesh elements by using `get_localmesh_ElementaryTag` function

4.1.8 `get_localmesh_PhysicalTag` method

```

[q,me]=Gh.get_localmesh_PhysicalTag(EltType,PhyTag)
[q,me,toGlobal]=Gh.get_localmesh_PhysicalTag(EltType,PhysicalTag)

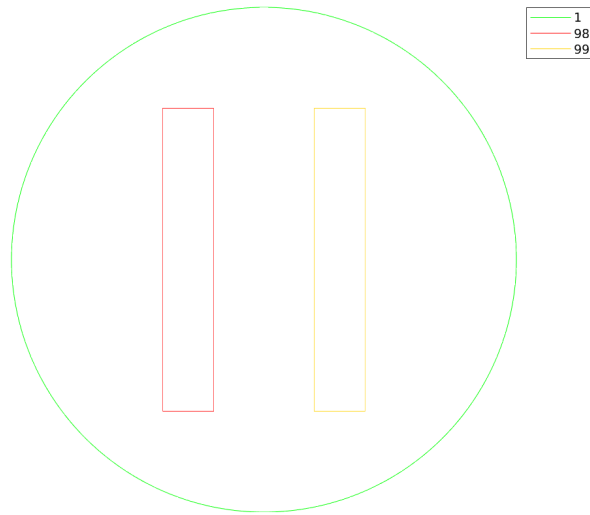
```

```
[q,me]=Gh.get_localmesh_PhysicalTag(EltType,PhyTag)
```

returns the *local* nodes/vertices array `q` and the *local* connectivity array `me` of the elements of type `EltType` and with *PhyTag* given by `PhysicalTag`.

```
[q,me,toGlobal]=Gh.get_localmesh_PhysicalTag(EltType,PhyTag)
```

Also returns the *global* tags array `toGlobal` such that `Gh.q(:,toGlobal)` is equal to `q`.



```
meshfile=fc_oogmsh.gmsh.buildmesh2d('condenser',6,'verbose',0);
Gh = fc_oogmsh.ooGmsh4(meshfile);
phtags1=Gh.get_PhysicalTags(1); % 1: 2-nodes line
n1=length(phtags1);
colors = fc_tools.graphics.selectColors(length(phtags1));
figure(1)
hold on
for i=1:n1
    [q,me]=Gh.get_localmesh_PhysicalTag(1,phtags1(i));
    h(i)=fc_graphics4mesh.plotmesh(q,me,'color',colors(i,:));
    clegend{i}=num2str(phtags1(i));
end
legend(h,clegend,'Location','NorthEastOutside','AutoUpdate','off');
axis image;axis off
fc_tools.graphics.SaveAllFigsAsFiles('ooGmsh4_get_localmesh_PhysicalTag', SaveOptions{:})
```

Listing 4: Plot 2-nodes line mesh elements by using `get_localmesh_PhysicalTag` function

4.2 Description of properties

Fields of **MeshFormat** structure

version	: string, version of the mesh file format.
file_type	: integer, 0 for ASCII mode, 1 for binary mode.
data_size	: integer, <code>sizeof(size_t)</code>

Fields of the (optional) **PhysicalName** structure

dimension	: integer.
physicalTag	: integer.
name	: string

Fields of the **Entities** structure

numPoints	:	integer.
Points	:	array of Point structure.
numCurves	:	integer.
Curves	:	array of Curve structure.
numSurfaces	:	integer.
Surfaces	:	array of Surface structure.
numVolumes	:	integer.
Volumes	:	array of Volume structure.

Fields of (optional) **PartitionedEntities** structure

numPartitions	:	integer.
numGhostEntities	:	integer.
GhostEntities	:	array of structure.
numPoints	:	integer
Points	:	array of structure.
numCurves	:	integer
Curves	:	array of structure.
numSurfaces	:	integer
Surfaces	:	array of structure.
numVolumes	:	integer.
Volumes	:	array of structure.

Fields of **Nodes** structure

numEntityBlocks	:	integer.
numNodes	:	integer.
minNodeTag	:	integer.
maxNodeTag	:	integer
EntityBlocks	:	array of EntityBlock structure.

Fields of **EntityBlocks** structure of **Nodes**

entityDim	:	integer.
entityTag	:	integer.
parametric	:	integer.
numNodes	:	integer.
nodeTags	:	1-by- numNodes array of integer.
Nodes	:	3-by- numNodes array of double.

Fields of **Elements** structure

numEntityBlocks	:	integer.
numElements	:	integer.
minElementTag	:	integer.
maxElementTag	:	integer
EntityBlocks	:	array of EntityBlock structure.
ElementTypes	:	array of .

Fields of **EntityBlocks** structure of **Elements**

entityDim	:	integer.
entityTag	:	integer.
elementType	:	integer.
elementDesc	:	structure returned by function <code>gmsh.elm_type_desc(elementType)</code> .
numElementsBlock	:	integer.
nodeTags	:	n -by- numElementsBlock array. n depends of elementType : $n = \text{elementDesc.nb_nodes}$
elementTags	:	1-by- numElementsBlock array

Fields of **PeriodicLink**

entityDim	:	integer.
entityTag	:	integer.
entityTagMaster	:	integer.
numAffine	:	.
values	:	.
numCorrespondingNodes	:	.
nodeTags	:	.
nodeTagMasters	:	.

4.3 Sample 1

The 2d .geo file *condenser.geo* is used to create a .msh file : *condenser-25.msh*. This .msh file contains only 1 (2-node line) and 2 (3-node triangle) *elm-type*.

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh('condenser',25,'verbose',0,'force',true);
Gh = fc_oogmsh.ooGmsh4(meshfile)
```

```
Gh =
fc_oogmsh.ooGmsh4 with properties:
  q: (2x49338 double)
  nq: 49338 double
  dim: 2 double
  d: 2 double
  toGlobal: (1x49338 double)
  meshfile: (1x110 char)
  partitionedfile: 0 logical
  MeshFormat: (1x1 struct)
  PhysicalNames: []
  Entities: (1x1 struct)
  PartitionedEntities: []
  Nodes: (1x1 struct)
  Elements: (1x1 struct)
  PeriodicLinks: []
  Info: (1x1 struct)
  orders: 1 double
```

4.4 Sample 2

The 3d .geo file *cylinderkey.geo* is used to create a .msh file : *cylinderkey-10.msh*. This .msh file contains 1 (2-node line), 2 (3-node triangle) and 4 (4-node tetrahedron) *elm-type*.

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh3d('cylinderkey',10, ...  
    'verbose',0,'force',true);  
Gh = fc_oogmsh.ooGmsh4(meshfile)
```

```
Gh =  
  
fc_oogmsh.ooGmsh4 with properties:  
    q: (3x5855 double)  
    nq: 5855 double  
    dim: 3 double  
    d: 3 double  
    toGlobal: (1x5855 double)  
    meshfile: (1x112 char)  
    partitionedfile: 0 logical  
    MeshFormat: (1x1 struct)  
    PhysicalNames: []  
    Entities: (1x1 struct)  
    PartitionedEntities: []  
    Nodes: (1x1 struct)  
    Elements: (1x1 struct)  
    PeriodicLinks: []  
    Info: (1x1 struct)  
    orders: 1 double
```

4.5 Sample 3

The 3d .geo file *ball8.geo* is used to create a 3d surface .msh file : *ball8-50.msh*. This .msh file contains 1 (2-node line), 2 (3-node triangle) and 15 (1-node point) *elm-type*.

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh3ds('ball8',25, 'verbose',0,'force',true);  
Gh = fc_oogmsh.ooGmsh4(meshfile)
```

```
Gh =  
  
fc_oogmsh.ooGmsh4 with properties:  
    q: (3x9740 double)  
    nq: 9740 double  
    dim: 3 double  
    d: 3 double  
    toGlobal: (1x9740 double)  
    meshfile: (1x106 char)  
    partitionedfile: 0 logical  
    MeshFormat: (1x1 struct)  
    PhysicalNames: []  
    Entities: (1x1 struct)  
    PartitionedEntities: []  
    Nodes: (1x1 struct)  
    Elements: (1x1 struct)  
    PeriodicLinks: []  
    Info: (1x1 struct)  
    orders: 1 double
```

5 ooGmsh2 class (version 2.2)

The `ooGMSH2` class can be used to read `gmsh` mesh files with the MSH ASCII file format (version 2.2) described for example in [3], section 9.1. A MSH file can contain various mesh elements which are identified by an *elm-type* integer given in Appendix A. One can also refer to the `fc_oogmsh.gmsh.elm_type_desc` function, described in Appendix B.1, to obtain information on a given *elm-type*.

When reading a MSH file (format 2.2) generated by `gmsh`, we split the mesh elements by *elm-type* and generate an array of `Elmt` structure. The dimension of this array is the number of different *elm-type* founds on the .msh file.

The `Elmt` structure is given by

Fields of **Elmt** structure

type	: integer, refers to the type of the element : 1 for 2-node line, 2 for 3-node triangle, ... See the <i>elm-type</i> description of [3], section 9.1. Informations on a given type can be obtained by using <code>elt=fc_oogmsh.gmsh.elm_type_desc(type)</code> .
geo	: string, contains the kind of geometry: 'line', 'triangle', 'tetrahedron', ...
d	: integer, space dimension or d -simplex.
order	: integer, order of the element.
nme	: integer, number of mesh elements.
me	: array of nb_nodes -by- nme integers, connectivity array. nb_nodes is equal to <code>elt.nb_nodes</code> where <code>elt=fc_oogmsh.gmsh.elm_type_desc(type)</code> .
phys_lab	: array of nme -by-... integers, physical labels of the elements.
geo_lab	: array of nme -by-... integers, geometrical labels of the elements.
nb_parts	: array of nme -by-1 integers, number of mesh partitions to which the element belongs.
part_lab	: array of nme -by- max(nb_parts) integers, <code>part_lab(i, 1 : nb_parts(i))</code> contains all the partitions index to which the <i>i</i> -th element belongs.

The **ooGMSH2** class was created to store a maximum of(all the) information(s) contained in the .msh file. The properties of this class are:

Properties of **ooGMSH** class

dim	: integer space dimension
nq	: integer number of vertices/nodes
q	: dim-by-n _q array of reals array of vertex coordinates
types	: array of integers List of the element types found in the mesh file.
orders	: array of integers List of the orders of the element types found in the mesh file.
sElts	: array of Elmt structure One Elmt structure by element type, such that <code>sElts(i)</code> contains all the elements of type <code>types(i)</code> and order <code>orders(i)</code> .

5.1 Methods

5.1.1 **ooGmsh2** constructor

The **ooGmsh2** class have only one constructor :

```
Gh=fc_oogmsh.ooGmsh2( meshfile )
```

where meshfile is the name of ... a mesh file

Matlab code with output

```
fprintf('1) Building the mesh\n')
meshfile=fc_oogmsh.gmsh.buildmesh2d('disk3holes',15, ...
    'verbose',0,'force',true,'MshFileVersion','2.2');
fprintf('2) Reading the mesh\n');
Gh = fc_oogmsh.ooGmsh2(meshfile);
fprintf('=> Gh is an ooGmsh2 object containing a MSH file version ...
    %s\n',Gh.MeshFormat.version)
fprintf('3) Displaying Gh\n');
Gh
```

```
1) Building the mesh
2) Reading the mesh
-> Gh is an ooGmsh2 object containing a MSH file version 2.2
3) Displaying Gh
```

Gh =

```
fc_oogmsh.ooGmsh2 with properties:
    q: (2x910 double)
    dim: 2 double
    nq: 910 double
    d: 2 double
    sElts: (2x1 struct)
    toGlobal: (1x910 double)
    partitionedfile: 0 logical
    orders: 1 double
    types: [ 1 2 ] (1x2 int32)
    MeshFormat: (1x1 struct)
    meshfile: (1x111 char)
    Info: (1x1 struct)
    debug: (1x1 struct)
```

5.1.2 info method

```
info(Gh)
Gh.info()
Gh.info(Key, Value, ...)
```

Description

Gh.info()

print informations on class fields with 3 levels of recursivity (i.e. field of field of field).

Gh.info(Key, Value, ...)

specifies function options using one or more **Key,Value** pair arguments. The **Key** options can be

- **'maxlevel'** : level of recursivity, default is 3.
- **'tab'** : number of space characters between two levels of recursivity, default is 4.

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh2d('disk3holes',15, ...
    'verbose',0,'force',true,'MshFileVersion','2.2');
Gh = fc_oogmsh.oogmsh2(meshfile);
Gh.info('maxlevel',2);
```

```
fc_oogmsh.oogmsh2 with properties:
[1] q : [2 910] double
[1] dim : [1 1] double
[1] nq : [1 1] double
[1] d : [1 1] double
[1] sElts : [2 1] struct
[2] type : [1 1] int32
[2] geo : [1 4] char
[2] d : [1 1] double
[2] order : [1 1] double
[2] me : [2 146] double
[2] nme : [1 1] double
[2] phys_lab : [146 1] double
[2] geo_lab : [146 1] double
[2] part_lab : [0 0] double
[2] nb_parts : [146 1] double
[1] toGlobal : [1 910] double
[1] partitionedfile : [1 1] logical
[1] orders : [1 1] double
[1] types : [1 2] int32
[1] MeshFormat : [1 1] struct
[2] version : [1 3] char
[2] file_type : [1 1] double
[2] data_size : [1 1] double
[1] meshfile : [1 111] char
[1] Info : [1 1] struct
[2] meshfile : [1 111] char
[1] debug : [1 1] struct
[2] Mt : [1824 55] int32
```

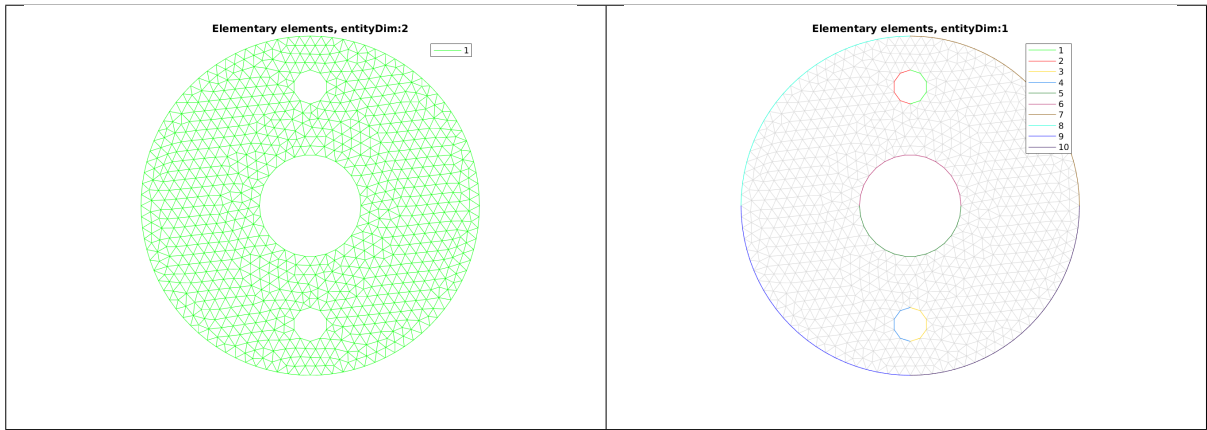


Figure 3: *Elementary Tag* elements of the geofile disk3holes.geo

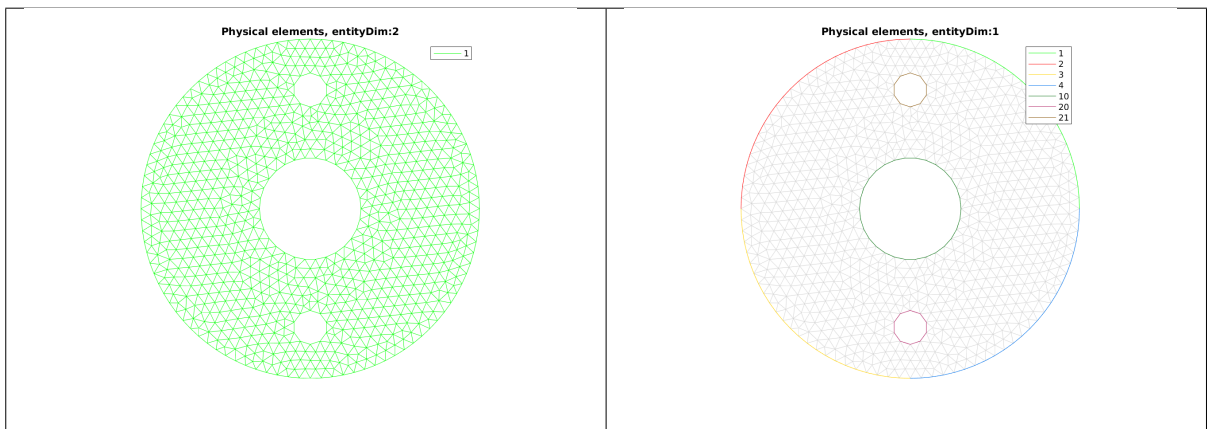


Figure 4: *Physical Tag* elements of the geofile disk3holes.geo

In the geofile disk3holes.geo the *Physical Tags* are created from the *Elementary Tags* as follow

```

...
Physical Line(10) = {6, 5};
Physical Line(21) = {2, 1};
Physical Line(20) = {4, 3};
Physical Line(1) = {7};
Physical Line(2) = {8};
Physical Line(3) = {9};
Physical Line(4) = {10};
Physical Surface(1) = {1};

```

5.1.3 `get_ElementaryTags` method

```

eltags=get_ElementaryTags(Gh,EltType)
eltags=Gh.get_ElementaryTags(EltType)

```

Description

```
eltags=Gh.get_ElementaryTags(EltType)
```

returns all the elementary tags associated with elements of type **EltType** as an array with unique elements. **EltType** is described in Section A. For example, **EltType** is 1 for 2-nodes **line** (i.e 1-simplex of order 1), **EltType** is 2 for 3-nodes **triangle** (i.e 2-simplex of order 1) and **EltType** is 4 for 4-nodes **tetrahedron** (i.e 3-simplex of order 1).

Matlab code with output

```

eltags1=Gh.get_ElementaryTags(1)
eltags2=Gh.get_ElementaryTags(2)

```

```

eltags1 =
     1     2     3     4     5     6     7     8     9    10

eltags2 =
     1

```

5.1.4 `get_PhysicalTags` method

```

phtags=get_PhysicalTags(Gh,EltType)
phtags=Gh.get_PhysicalTags(EltType)

```

Description

```
phtags=Gh.get_PhysicalTags(EltType)
```

returns all the physical tags associated with elements of type **EltType** as an array with unique elements. **EltType** is described in Section A.

Matlab code with output

```

phtags1=Gh.get_PhysicalTags(1)
phtags2=Gh.get_PhysicalTags(2)

```

```

phtags1 =
     1     2     3     4    10    20    21

phtags2 =
     1

```

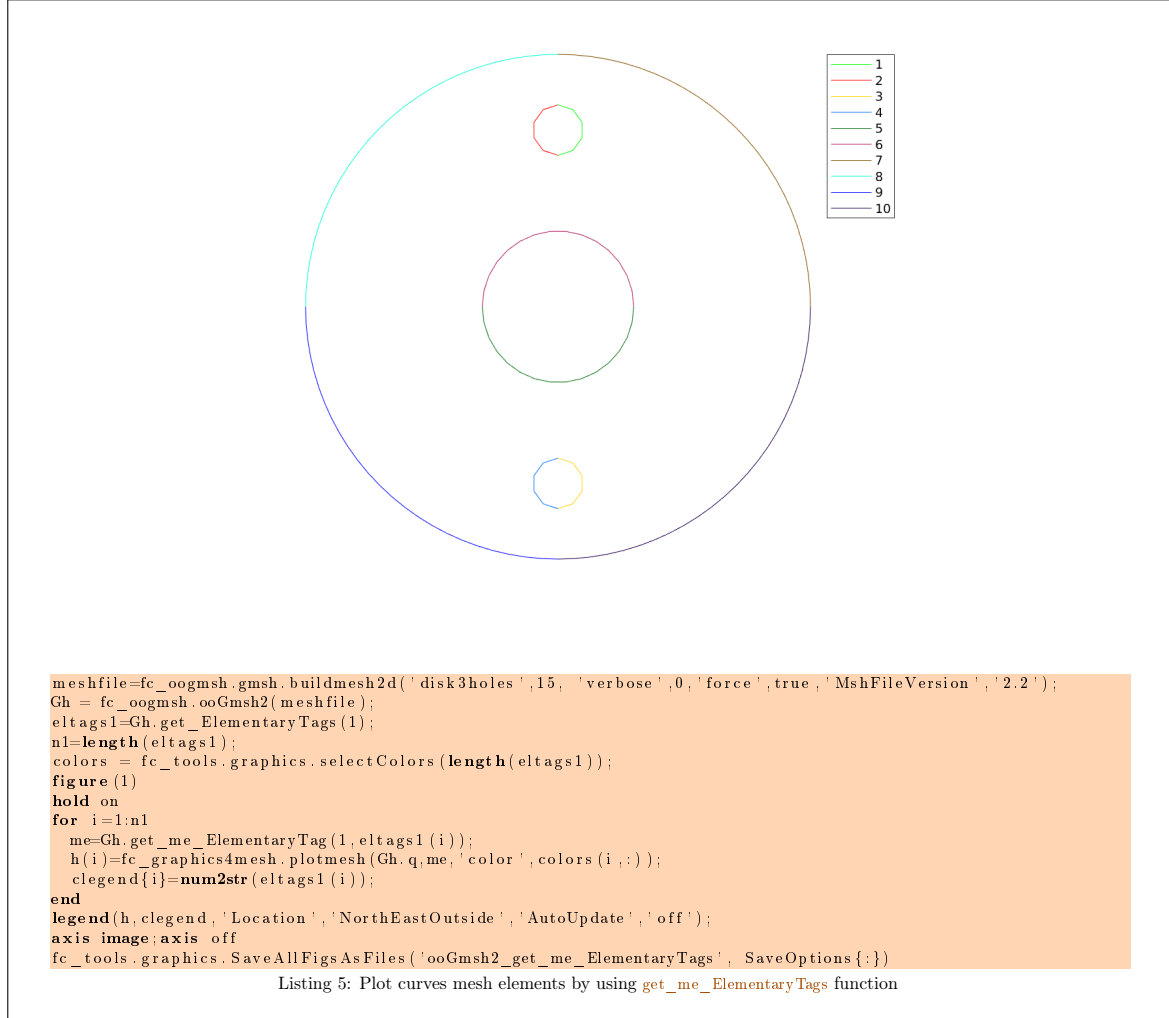
5.1.5 `get_me_ElementaryTag` method

```
me=get_me_ElementaryTag(Gh,EltType,EltTag)
me=Gh.get_me_ElementaryTag(EltType,EltTag)
```

Description

```
me=Gh.get_me_ElementaryTag(EltType,EltTag)
```

returns **me** the connectivity array of mesh elements of type and *elementary tag* given respectively by **EltType** and **EltTag**. This array is associated with the **Gh.q** nodes/vertices array.



5.1.6 `get_me_PhysicalTag` method

```
me=get_me_PhysicalTag(Gh,EltType,PhysicalTag)
me=Gh.get_me_PhysicalTag(EltType,PhysicalTag)
```

Description

```
get_me_PhysicalTag(Gh,EltType,PhysicalTag)
```

returns **me** the connectivity array of mesh elements of type and *physical tag* given respectively by **EltType** and **PhysicalTag**.



5.1.7 `get_localmesh_ElementaryTag` method

```

[q,me]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)
[q,me,toGlobal]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)

```

```

[q,me]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)

```

returns the *local* nodes/vertices array `q` and the *local* connectivity array `me` of the element of type `EltType` and with *elementary tag* given by `EltTag`.

```

[q,me,toGlobal]=Gh.get_localmesh_ElementaryTag(EltType,EltTag)

```

Also returns the *global* tags array `toGlobal` such that `Gh.q(:,toGlobal)` is equal to `q`.



5.1.8 `get_localmesh_PhysicalTag` method

```

[q,me]=Gh.get_localmesh_PhysicalTag(EltType,PhysicalTag)
[q,me,toGlobal]=Gh.get_localmesh_PhysicalTag(EltType,PhysicalTag)

```

```

[q,me]=Gh.get_localmesh_PhysicalTag(EltType,PhysicalTag)

```

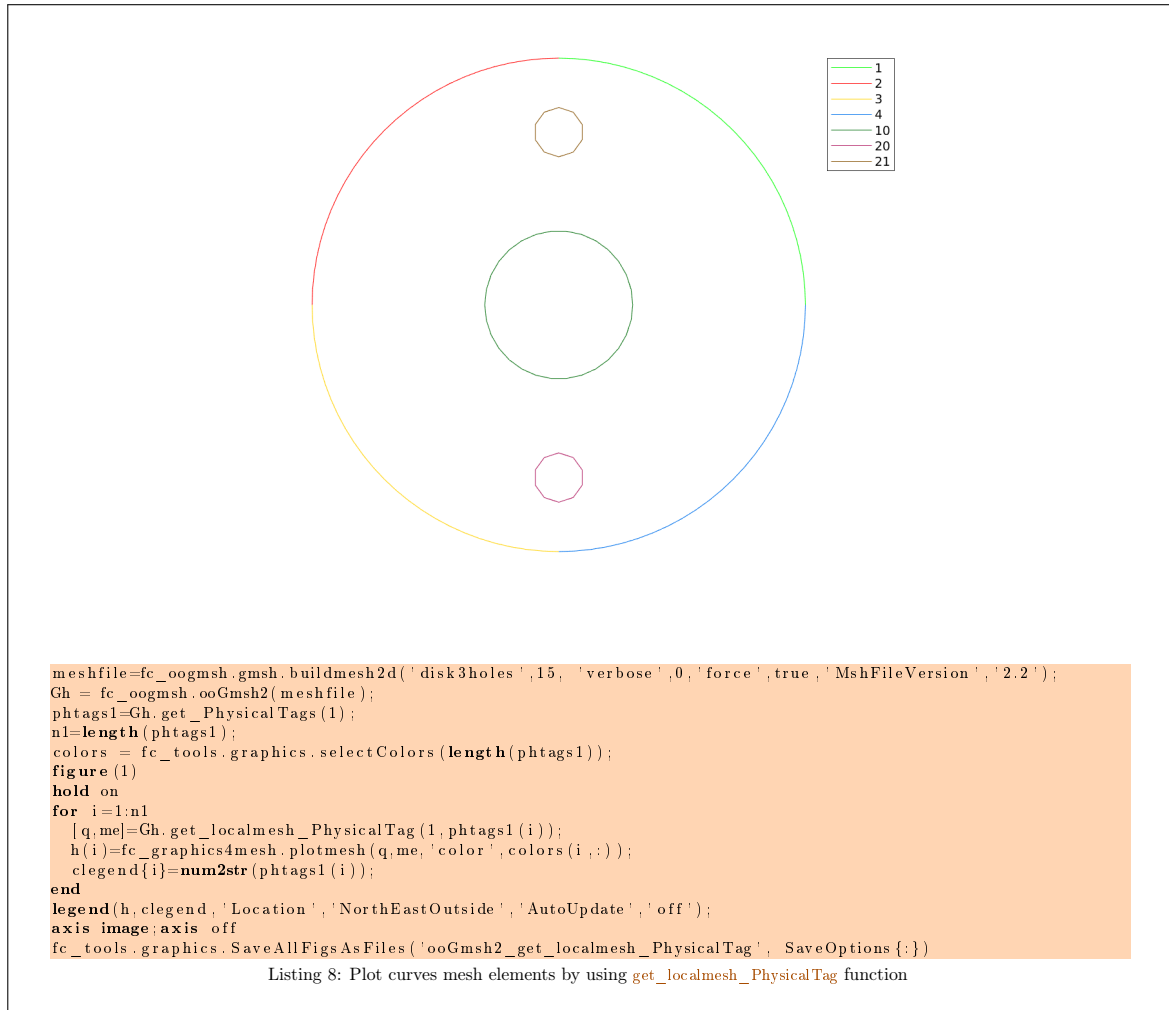
returns the *local* nodes/vertices array `q` and the *local* connectivity array `me` of the elements of type `EltType` and with *PhysicalTag* given by `PhysicalTag`.

```

[q,me,toGlobal]=Gh.get_localmesh_PhysicalTag(EltType,PhysicalTag)

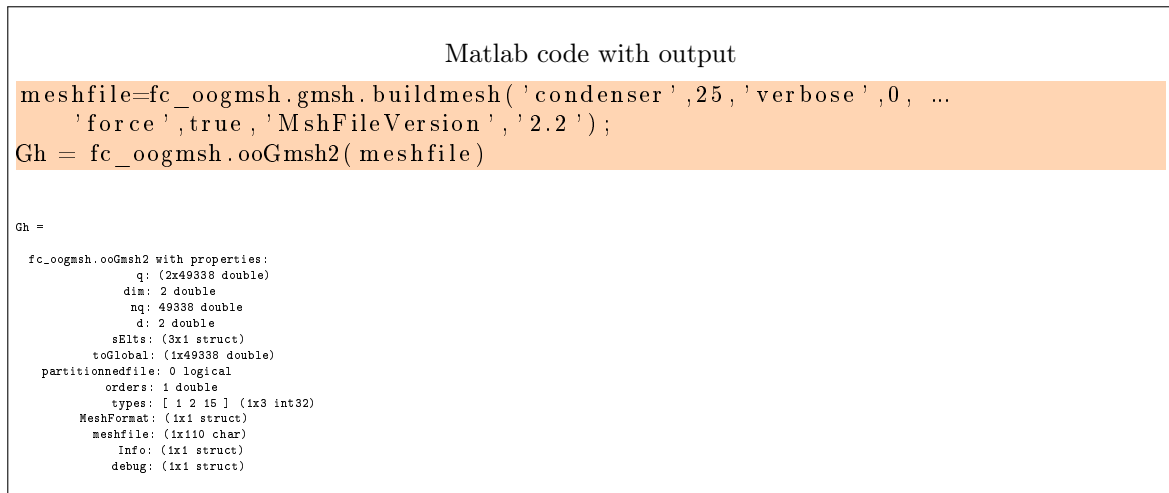
```

Also returns the *global* tags array `toGlobal` such that `Gh.q(:,toGlobal)` is equal to `q`.



5.2 Sample 1

The 2d .geo file *condenser.geo* is used to create a .msh file : *condenser-25.msh*. This .msh file contains only 1 (2-node line) and 2 (3-node triangle) *elm-type*.



5.3 Sample 2

The 3d .geo file *cylinderkey.geo* is used to create a .msh file : *cylinderkey-10.msh*. This .msh file contains 1 (2-node line), 2 (3-node triangle) and 4 (4-node tetrahedron) *elm-type*.

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh3d('cylinderkey',10,'verbose',0, ...
    'force',true,'MshFileVersion','2.2');
Gh = fc_oogmsh.ooGmsh2(meshfile)
```

```
Gh =
fc_oogmsh.ooGmsh2 with properties:
    q: (3x5855 double)
    dim: 3 double
    nq: 5855 double
    d: 3 double
    sElts: (3x1 struct)
    toGlobal: (1x5855 double)
    partitionnedfile: 0 logical
    orders: 1 double
    types: [ 1 2 4 ] (1x3 int32)
    MeshFormat: (1x1 struct)
    meshfile: (1x112 char)
    Info: (1x1 struct)
    debug: (1x1 struct)
```

5.4 Sample 3

The 3d .geo file *ball8.geo* is used to create a 3d surface .msh file : *ball8-50.msh*. This .msh file contains 1 (2-node line), 2 (3-node triangle) and 15 (1-node point) *elm-type*.

Matlab code with output

```
meshfile=fc_oogmsh.gmsh.buildmesh3ds('ball8',50,'verbose',0, ...
    'force',true,'MshFileVersion','2.2');
Gh = fc_oogmsh.ooGmsh2(meshfile)
```

```
Gh =
fc_oogmsh.ooGmsh2 with properties:
    q: (3x37799 double)
    dim: 3 double
    nq: 37799 double
    d: 2 double
    sElts: (3x1 struct)
    toGlobal: (1x37799 double)
    partitionnedfile: 0 logical
    orders: 1 double
    types: [ 1 2 15 ] (1x3 int32)
    MeshFormat: (1x1 struct)
    meshfile: (1x106 char)
    Info: (1x1 struct)
    debug: (1x1 struct)
```

A Element type

In a .msh file the kind of mesh elements are identified by their *elm-type* integer values :

Table 1:

<i>elm-type</i>	description
1	2-node line
2	3-node triangle
3	4-node quadrangle
4	4-node tetrahedron
5	8-node hexahedron
6	6-node prism
7	5-node pyramid

Continued on next page

Table 1: (Continued)

8	3-node second order line (2 nodes associated with the vertices and 1 with the edge)
9	6-node second order triangle (3 nodes associated with the vertices and 3 with the edges)
10	9-node second order quadrangle (4 nodes associated with the vertices, 4 with the edges and 1 with the face)
11	10-node second order tetrahedron (4 nodes associated with the vertices and 6 with the edges)
12	27-node second order hexahedron (8 nodes associated with the vertices, 12 with the edges, 6 with the faces and 1 with the volume)
13	18-node second order prism (6 nodes associated with the vertices, 9 with the edges and 3 with the quadrangular faces)
14	14-node second order pyramid (5 nodes associated with the vertices, 8 with the edges and 1 with the quadrangular face)
15	1-node point
16	8-node second order quadrangle (4 nodes associated with the vertices and 4 with the edges)
17	20-node second order hexahedron (8 nodes associated with the vertices and 12 with the edges)
18	15-node second order prism (6 nodes associated with the vertices and 9 with the edges)
19	13-node second order pyramid (5 nodes associated with the vertices and 8 with the edges)
20	9-node third order incomplete triangle (3 nodes associated with the vertices, 6 with the edges)
21	10-node third order triangle (3 nodes associated with the vertices, 6 with the edges, 1 with the face)
22	12-node fourth order incomplete triangle (3 nodes associated with the vertices, 9 with the edges)
23	15-node fourth order triangle (3 nodes associated with the vertices, 9 with the edges, 3 with the face)
24	15-node fifth order incomplete triangle (3 nodes associated with the vertices, 12 with the edges)
25	21-node fifth order complete triangle (3 nodes associated with the vertices, 12 with the edges, 6 with the face)
26	4-node third order edge (2 nodes associated with the vertices, 2 internal to the edge)
27	5-node fourth order edge (2 nodes associated with the vertices, 3 internal to the edge)
28	6-node fifth order edge (2 nodes associated with the vertices, 4 internal to the edge)
29	20-node third order tetrahedron (4 nodes associated with the vertices, 12 with the edges, 4 with the faces)
30	35-node fourth order tetrahedron (4 nodes associated with the vertices, 18 with the edges, 12 with the faces, 1 in the volume)
31	56-node fifth order tetrahedron (4 nodes associated with the vertices, 24 with the edges, 24 with the faces, 4 in the volume)

Continued on next page

Table 1: (Continued)

92	64-node third order hexahedron (8 nodes associated with the vertices, 24 with the edges, 24 with the faces, 8 in the volume)
93	125-node fourth order hexahedron (8 nodes associated with the vertices, 36 with the edges, 54 with the faces, 27 in the volume)

B Other functions

B.1 function `fc_oogmsh.gmsh.elm_type_desc`

This function returns a structure which contains some informations on a `gmsh elt-type` described in Appendix A.

Syntaxe

```
elt=fc_oogmsh.gmsh.elm_type_desc(type)
```

Matlab code with output

```
elt2=fc_oogmsh.gmsh.elm_type_desc(2)
elt4=fc_oogmsh.gmsh.elm_type_desc(4)
elt11=fc_oogmsh.gmsh.elm_type_desc(11)

elt2 =
  struct with fields:
    define: 'MSH_TRI_3'
    elm_type: 2
    desc: '3-node triangle'
    nb_nodes: 3
    order: 1
    incomplete: 0
    d: 2
    geo: 'triangle'

elt4 =
  struct with fields:
    define: 'MSH_TET_4'
    elm_type: 4
    desc: '4-node tetrahedron'
    nb_nodes: 4
    order: 1
    incomplete: 0
    d: 3
    geo: 'tetrahedron'

elt11 =
  struct with fields:
    define: 'MSH_TET_10'
    elm_type: 11
    desc: '10-node second order tetrahedron (4 nodes associated with the vertices and 6 with the edges)'
    nb_nodes: 10
    order: 2
    incomplete: 0
    d: 3
    geo: 'tetrahedron'
```

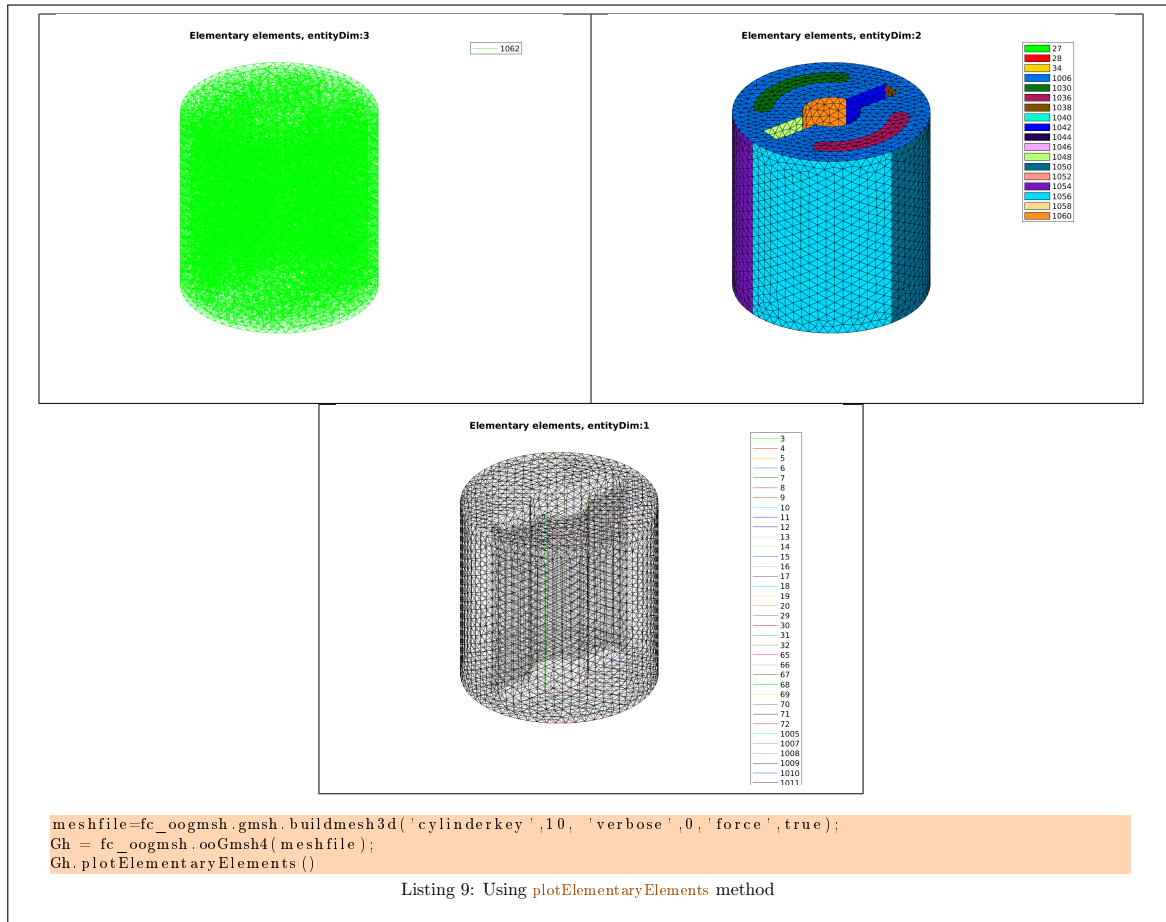
B.2 method `plotElementaryElements`

This function plot *Elementary Elements* of an `ooGmsh2` or `ooGmsh4` object of *Element Type*

- 1, *2-node line* elements,
- 2, *3-node triangle* elements,
- 4, *4-node tetrahedron* elements.

This function uses the `fc-graphics4mesh` toolbox [1] version 0.1.5.

```
Gh.plotElementaryElements()
plotElementaryElements(Gh)
```



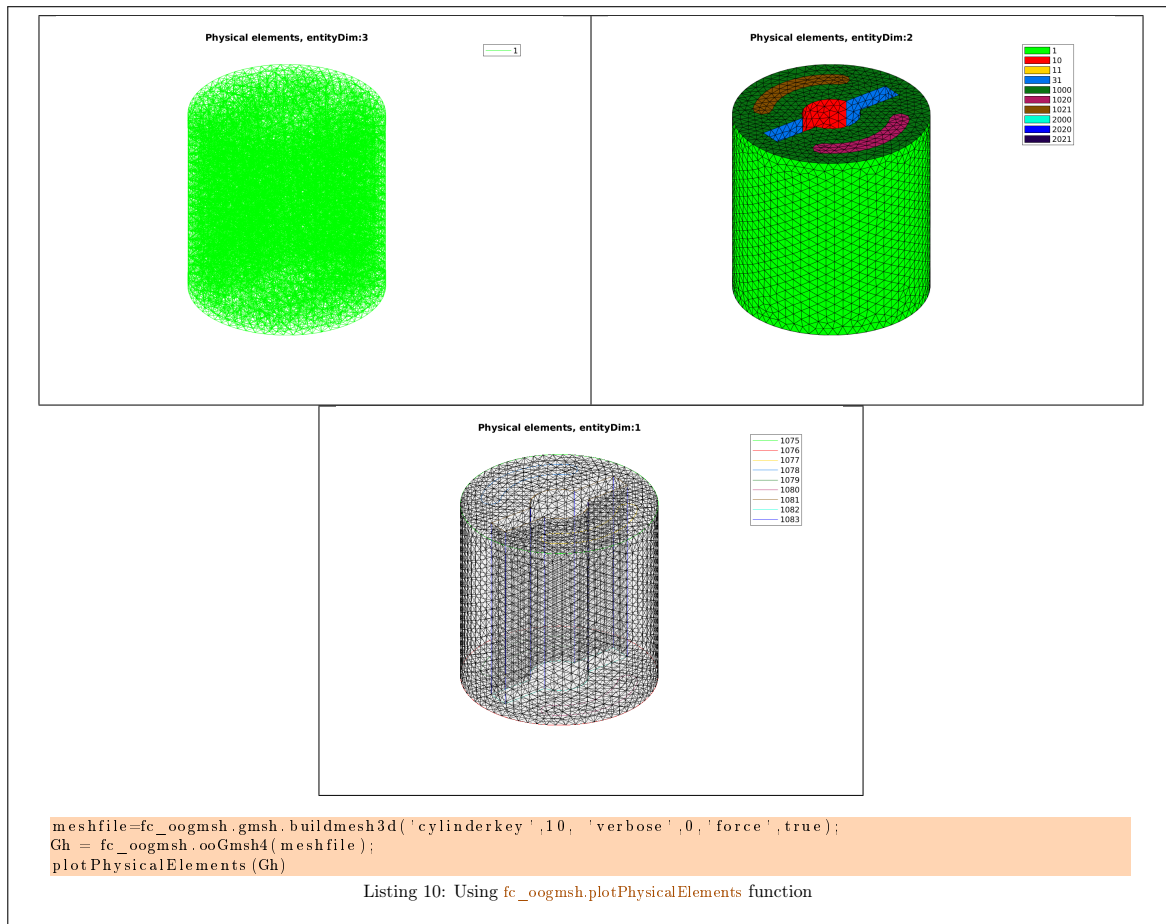
B.3 method `plotPhysicalElements`

This function plot *Physical Tags* of an `ooGmsh2` or `ooGmsh4` object of *Element Type*

- 1, 2-node line elements,
- 2, 3-node triangle elements,
- 4, 4-node tetrahedron elements.

This function uses the `fc-graphics4mesh` toolbox [1] version 0.1.5.

```
Gh.plotPhysicalElements()
plotPhysicalElements(Gh)
```



B.4 method `plotPartitionElements`

This function can be used with partitioned mesh file built with one of the following functions:

```

fc_oogmsh.gmsh.buildpartmesh3d,
fc_oogmsh.gmsh.buildpartmesh3ds,
fc_oogmsh.gmsh.buildpartmesh2d.

```

This function plot *Partition Tags* of an `ooGmsh2` or `ooGmsh4` object of *Element Type*

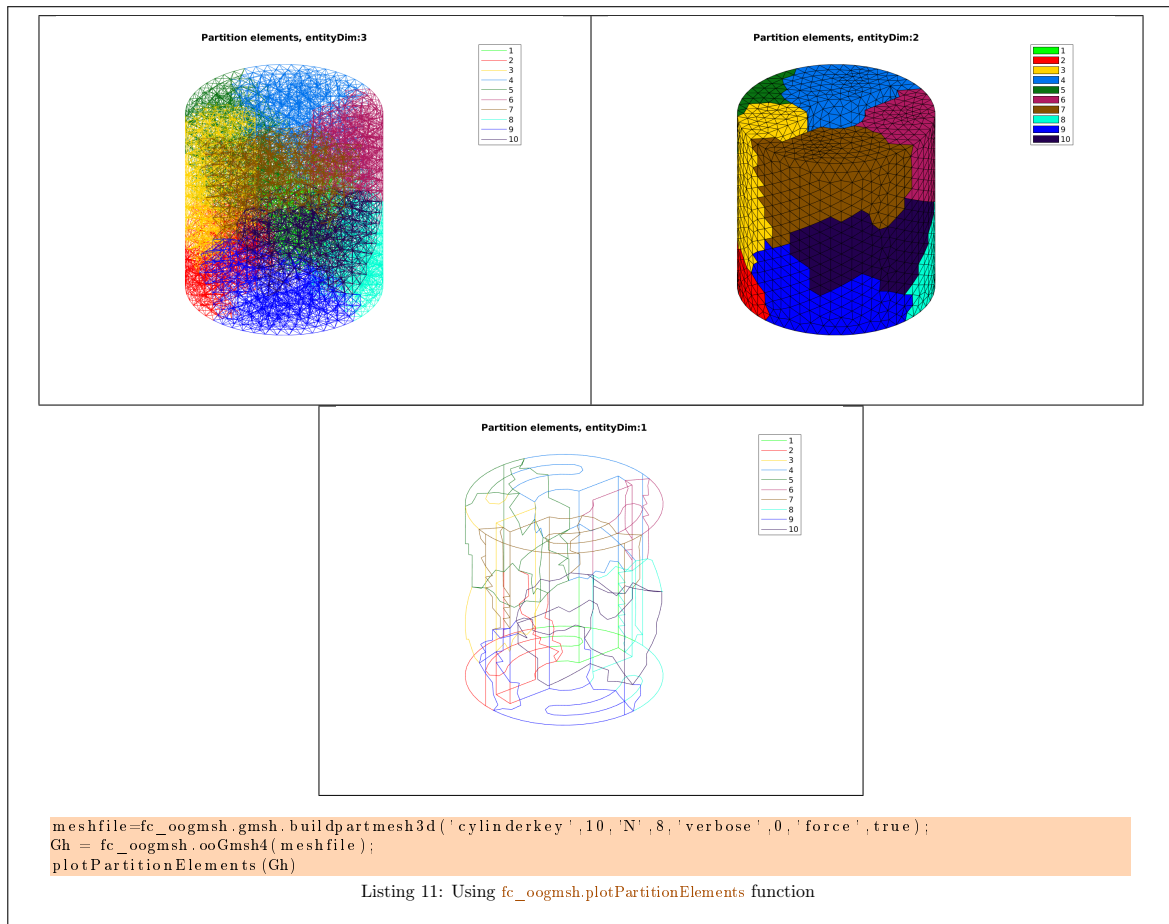
- 1, 2-node line elements,
- 2, 3-node triangle elements,
- 4, 4-node tetrahedron elements.

This function uses the `fc-graphics4mesh` toolbox [1] version 0.1.5.

```

Gh.plotPartitionElements()
plotPartitionElements(Gh)

```



B.5 method `plotInterfaceElements`

This function can be used with partitioned mesh file built with one of the following functions:

```

fc_oogmsh.gmsh.buildpartmesh3d,
fc_oogmsh.gmsh.buildpartmesh3ds,
fc_oogmsh.gmsh.buildpartmesh2d.

```

This function plot *Interface Tags* of the interfaces between partitions of an `ooGmsh2` or `ooGmsh4` object of *Element Type*

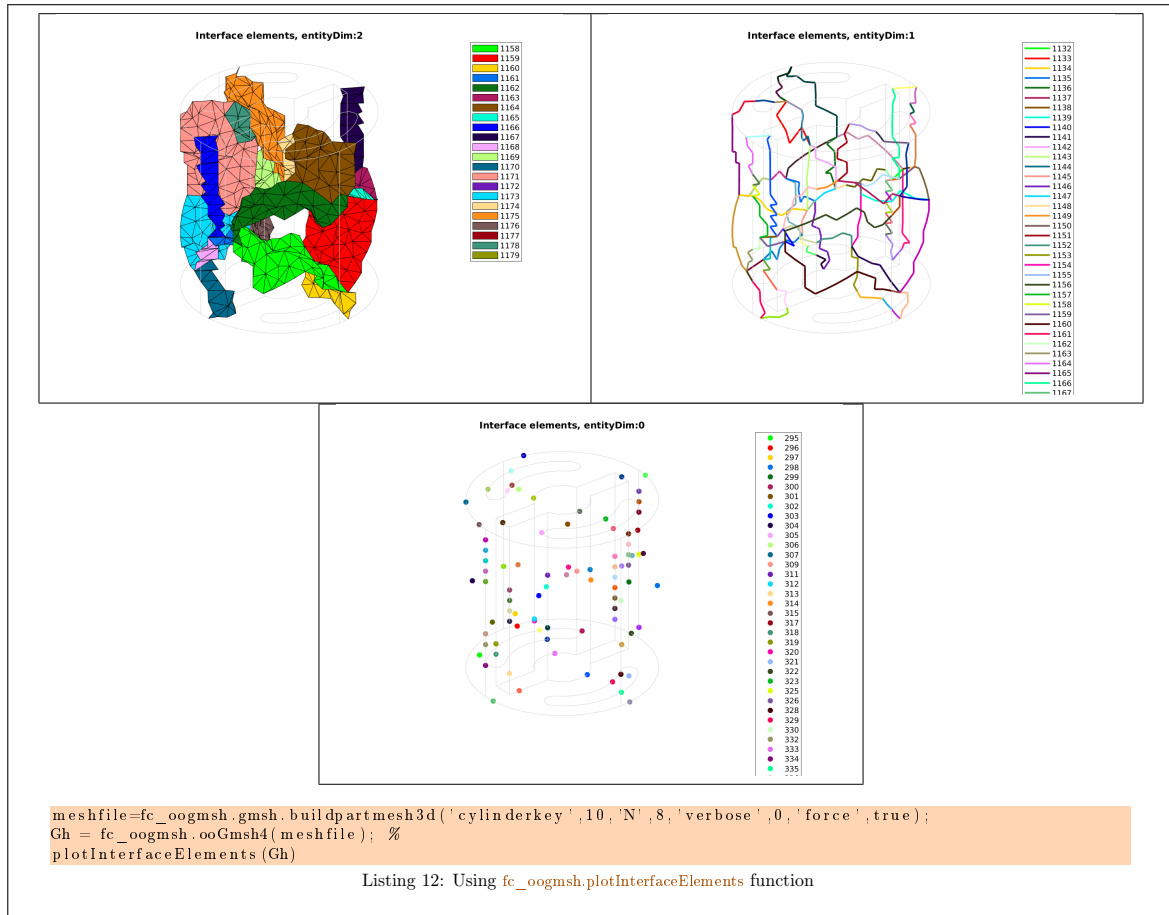
- 1, *2-node line* elements,
- 2, *3-node triangle* elements,
- 4, *4-node tetrahedron* elements.

This function uses the `fc-graphics4mesh` toolbox [1] version 0.1.5.

```

Gh.plotInterfaceElements()
plotInterfaceElements(Gh)

```



B References

- [1] F. Cuvelier. `fc_graphics4mesh`: a Matlab toolbox for displaying simplices meshes or datas on simplices meshes. <http://www.math.univ-paris13.fr/~cuvelier/software/>, 2017. User’s Guide.
- [2] C. Geuzaine and J.-F. Remacle. Gmsh: A 3-D finite element mesh generator with built-in pre- and post-processing facilities. *International Journal for Numerical Methods in Engineering*, 79(11):1309–1331, 2009.
- [3] C. Geuzaine and J.-F. Remacle. Gmsh 2.15.0. <http://gmsh.info>, 2016.
- [4] C. Geuzaine and J.-F. Remacle. Gmsh 4.2.1. <http://gmsh.info>, 2019.

Informations for git maintainers of the oogmsh Matlab toolbox

git informations on the toolboxes used to build this manual

```
-----
name : fc-oogmsh
tag : 0.2.4.a
commit : 9dff8e9ed08333f79eca8c3ba4595b66700d393f
date : 2022-12-21
time : 11-35-57
status : 0
-----
name : fc-tools
tag : 0.0.33
commit : 453fdb4bfec0bb0ce1932157d3a3aa4dcb4428d3
date : 2022-02-16
time : 15-32-56
status : 0
-----
name : fc-bench
tag : 0.1.3
commit : 6c8969bb49b90c7cae34e70c88dd2f968766376b
date : 2022-12-17
time : 13-58-43
status : 0
-----
name : fc-amat
tag : 0.1.3
commit : 90dbaa2839188cf8e01629817e5f8ba1b7188052
date : 2022-12-19
time : 07-41-33
status : 0
-----
name : fc-meshtools
tag : 0.1.4.a
commit : 57826462a8280dec5aa1eff0d424877097187b41
date : 2022-12-19
time : 08-27-27
status : 0
-----
name : fc-graphics4mesh
tag : 0.1.5.a
commit : b994b7b2a57399848541708cae3c33f79787cb1a
date : 2022-12-19
time : 08-28-40
status : 0
-----
```

git informations on the $\LaTeX$ package used to build this manual

```
-----
name : fctools
tag :
commit : c9a33ce7b4daf90f66e5e49856e69afaidac0a3
date : 2022-12-17
time : 07:57:20
status : 1
-----
```

Using the remote configuration repository:

```
url      ssh://lagagit/MCS/Cuvelier/Matlab/fc-config
commit   1ddb965ef261410ecee4ce178be4ceb85939de04
```