Gmsh

C. Geuzaine and J.-F. Remacle

Université de Liège and Université catholique de Louvain

October 20, 2020
Some background

- I am a professor at the University of Liège in Belgium, where I lead a team of about 15 people in the Montefiore Institute (EECS Dept.), at the intersection of applied math, scientific computing and engineering physics.
- Our research interests include modeling, analysis, algorithm development, and simulation for problems arising in various areas of engineering and science.
- Current applications: low- and high-frequency electromagnetics, geophysics, biomedical problems.
Some background

- I am a professor at the Université catholique de Louvain in Belgium, where I lead a team of a dozen researchers in the Institute of Mechanics, Materials and Civil Engineering
- We work mainly on mesh generation and high-order methods for PDEs
- Our current main research topic is hex meshing, both unstructured and structured
- I have been co-operating with Christophe for more than 20 years, a fruitful collaboration that has led to the creation of Gmsh
What is Gmsh?

- Gmsh (http://gmsh.info) is an open source 3D finite element mesh generator with a built-in CAD engine and post-processor
- Includes a graphical user interface (GUI) and can drive any simulation code through ONELAB
- Today, Gmsh represents about 500k lines of C++ code
  - still same 2 core developers; about 100 with ≥ 1 commit
  - about 1,000 people on mailing lists
  - about 10,000 downloads per month (70% Windows)
  - about 500 citations per year – the Gmsh paper is cited about 4,500 times
  - Gmsh has probably become one of the most popular (open source) finite element mesh generators?
∼ 20 years of Gmsh development in 1 minute

A warm thank you to all the contributors!
A little bit of history

- Gmsh was started in 1996, as a side project
- 1998: First public release
- 2003: Open Sourced under GNU GPL
- 2006: OpenCASCADE integration (Gmsh 2)
- 2009: IJNME paper and switch to CMake
- 2012: Curvilinear meshing and quad meshing
- 2013: Homology and ONELAB solver interface
- 2015: Multi-Threaded 1D and 2D meshing (coarse-grained)
- 2017: Boolean operations and switch to Git (Gmsh 3)
- 2018: C++, C, Python and Julia API (Gmsh 4)
- 2019: Multi-Threaded 3D meshing (fine-grained), robust STL remeshing
Strategic choices

- Design goals: fast, light and user-friendly
  - Written in simple C++
  - GUIs: FLTK (desktop), UIKit (iOS), Android
  - OpenGL graphics
  - Highly portable (OSes & compilers)
  - Easy to distribute & install: zero dependencies on installation

- Handling of numerous third party libraries
  - Build system based on CMake – everything is optional
  - Some libs integrated and redistributed directly in gmsh/contrib (BAMG, Metis, Concorde, ...)

- Funding
  - Hobby until 2006, then industry, Wallonia, Belgium & EU
Strategic choices

• Community infrastructure
  • Our own (using GitLab) to enable public/private parts (https://gitlab.onelab.info/gmsh/gmsh)
  • Continuous integration and delivery (CI/CD) of Gmsh app and Gmsh SDK on Windows, Linux and macOS
  • Two mailing lists and a web site (http://gmsh.info) with documentation, tutorials, etc.
  • Scientific aspects of algorithms detailed in journal papers

• Licensing
  • Gmsh is distributed under the GNU General Public License v2 or later, with exceptions to allow for easier linking with external libraries
  • We double-license to enable embedding in commercial codes
Basic concepts

- Gmsh is based around four modules: Geometry, Mesh, Solver and Post-processing
- Gmsh can be used at 3 levels
  - Through the GUI
  - Through the dedicated .geo language
  - Through the C++, C, Python and Julia API
- Main characteristics
  - All algorithms are written in terms of abstract model entities, using a Boundary REPresentation (BREP) approach
  - Gmsh never translates from one CAD format to another; it directly accesses each CAD kernel API (OpenCASCADE, Built-in, ...)


Basic concepts

The goal is to deal with very different underlying data representations in a transparent manner.
Geometry module

Under the hood, 4 types of model entities are defined:

1. Model points $G^0_i$ that are topological entities of dimension 0
2. Model curves $G^1_i$ that are topological entities of dimension 1
3. Model surfaces $G^2_i$ that are topological entities of dimension 2
4. Model volumes $G^3_i$ that are topological entities of dimension 3
Geometry module

- Model entities are topological entities, i.e., they only deal with adjacencies in the model; a bi-directional data structure represents the graph of adjacencies

\[ G_i^0 \Leftrightarrow G_i^1 \Leftrightarrow G_i^2 \Leftrightarrow G_i^3 \]

- Any model is able to build its list of adjacencies of any dimension using local operations

- The BRep is extended with non-manifold features: adjacent entities, and embedded (internal) entities

- Model entities can be either CAD entities (e.g. from the built-in or OpenCASCADE kernel) or discrete entities (defined by a mesh, e.g. STL)
Geometry module

The geometry of a CAD model entity depends on the solid modeler kernel for its underlying representation. Solid modelers usually provide a parametrization of the shapes, i.e., a mapping:

\[ p \in \mathbb{R}^d \mapsto x \in \mathbb{R}^3 \]

1. The geometry of a model point \( G^0_i \) is simply its 3-D location \( x_i = (x_i, y_i, z_i) \)
2. The geometry of a model curve \( G^1_i \) is its underlying curve \( C_i \) with its parametrization \( p(t) \in C_i, \ t \in [t_1, t_2] \)
3. The geometry of a model surface \( G^2_i \) is its underlying surface \( S_i \) with its parametrization \( p(u, v) \in S_i \)
4. The geometry associated to a model volume is \( \mathbb{R}^3 \)
Point \( p \) located on a curve \( C \) that is itself embedded in a surface \( S \)
Geometry module

Operations on CAD model entities are performed directly within their respective CAD kernels:

- There is no common internal geometrical representation
- Rather, Gmsh directly performs the operations (translation, rotation, intersection, union, fragments, ...) on the native geometrical representation using each CAD kernel’s own API
Geometry module

Discrete model entities are defined by a mesh (e.g. STL):

- They can be equipped with a geometry through a reparametrization procedure
- The parametrization is then used for meshing, in exactly the same way as for CAD entities
Mesh module

- Gmsh implements several meshing algorithms with specific characteristics
  - 1D, 2D and 3D
  - Structured, unstructured and hybrid
  - Isotropic and anisotropic
  - Straight-sided and curved
  - From standard CAD data or from STL through reparametrization
- Built-in interfaces to external mesh generators (BAMG, MMG3D, Netgen)
Mesh module

Typical CAD kernel idiosyncrasies: seam edges and degenerated edges
Mesh module

- Mesh data is made of *elements* (points, lines, triangles, quadrangles, tetrahedra, hexahedra, ...) defined by an ordered list of their *nodes*

- Elements and nodes are stored (*classified*) in the model entity they discretize:
  - A model point will thus contain a mesh element of type point, as well as a mesh node
  - A model curve will contain line elements as well as its interior nodes, while its boundary nodes will be stored in the bounding model points
  - A model surface will contain triangular and/or quadrangular elements and all the nodes not classified on its boundary or on its embedded entities (curves and points)
  - A model volume will contain tetrahedra, hexahedra, etc. and all the nodes not classified on its boundary or on its embedded entities (surfaces, curves and points)
This mesh data structure allows to easily and efficiently handle the creation, modification and destruction of conformal finite element meshes.
Solver module

- Gmsh implements a ONELAB (http://onelab.info) server to pilot external solvers, called “clients”
- Example client: GetDP finite element solver (http://getdp.info)

- The ONELAB interface allows to call such clients and have them share parameters and modeling information
- Parameters are directly controllable from the GUI
Solver module

- The implementation is based on a client-server model, with a server-side database and local or remote clients communicating in-memory or through TCP/IP sockets
  - Contrary to most solver interfaces, the ONELAB server has no a priori knowledge about any specifics (input file format, syntax, ...) of the clients
  - This is made possible by having any simulation preceded by an analysis phase, during which the clients are asked to upload their parameter set to the server
  - The issues of completeness and consistency of the parameter sets are completely dealt with on the client side: the role of ONELAB is limited to data centralization, modification and re-dispatching
Post-processing module

- Post-processing data is made of *views*
- A view stores both display *options* and *data* (unless the view is an *alias* of another view)
- View data can contain several *steps* (e.g. to store time series) and can be either linked to one or more models (*mesh-based* data, as stored in `.msh` or `.med` files) or independent from any model (*list-based* data, as stored in parsed `.pos` files)
- Data is interpolated through arbitrary polynomial interpolation schemes; automatic mesh refinement is used for adaptive visualization of high-order views
- Various *plugins* exist to create and modify views
Post-processing module

- Cuts, iso-curves and vectors
- Elevation maps
- Streamlines
- Adaptive high-order visualization
Recent developments

- Constructive Solid Geometry
- Application Programming Interface (API)
- Multi-Threaded meshing
- Robust STL remeshing based on parametrizations
Constructive Solid Geometry

http://en.wikipedia.org/wiki/Constructive_solid_geometry
Constructive Solid Geometry

SetFactory("OpenCASCADE"); // use OpenCASCADE kernel

R = DefineNumber[ 1.4 , Min 0.1 , Max 2 , Step 0.01,
        Name "Parameters/Box dimension" ];
Rs = DefineNumber[ R*.7 , Min 0.1 , Max 2 , Step 0.01,
        Name "Parameters/Cylinder radius" ];
Rt = DefineNumber[ R*1.25 , Min 0.1 , Max 2 , Step 0.01,
        Name "Parameters/Sphere radius" ];


Sphere(2) = {0,0,0, Rt};

BooleanIntersection(3) = { Volume{1}; Delete; }
        { Volume{2}; Delete; };
        // delete object and tool

Cylinder(4) = {-2*R,0,0, 4*R,0,0, Rs};
Cylinder(5) = {0,-2*R,0, 0,4*R,0, Rs};
Cylinder(6) = {0,0,-2*R, 0,0,4*R, Rs};

BooleanUnion(7) = { Volume{4}; Delete; }
        { Volume{5,6}; Delete; };

BooleanDifference(8) = { Volume{3}; Delete; }
        { Volume{7}; Delete; };
Constructive Solid Geometry

```
gmsh/demos/boolean/boolean.geo
```
Constructive Solid Geometry

SetFactory("OpenCASCADE");

DefineConstant [
    z = {16, Name "Parameters/z position of box"}
    sph = {0, Choices{0,1}, Name "Parameters/Add sphere?"}
];

a() = ShapeFromFile("component8.step"); // import STEP shape
b() = 2;
Box(b(0)) = {0,156,z, 10,170,z+10};

If(sph)
    b() += 3;
    Sphere(b(1)) = {0,150,0, 20};
EndIf

// fragmentation intersects everything
r() = BooleanFragments{ Volume{a()}; Delete; }{ Volume{b()}; Delete; };
Save "merged.brep"; // save into native OpenCASCADE format

Physical Volume("Combined volume", 1) = {r()};
Physical Surface("Combined boundary", 2) = CombinedBoundary{ Volume{r()}; }
Constructive Solid Geometry

gmsh/demos/boolean/import.geo
Constructive Solid Geometry

- All existing .geo commands are conserved
- New or modified .geo commands:
  - Shapes (with explicit numbering): Circle, Ellipse, Wire, Surface, Sphere, Box, Torus, Rectangle, Disk, Cylinder, Cone, Wedge, ThickSolid, ThruSections, Ruled ThruSections
  - Operations (implicit numbering): ThruSections, Ruled ThruSections, Fillet, Extrude
  - Boolean operations (explicit or implicit numbering): BooleanUnion, BooleanIntersection, BooleanDifference, BooleanFragments
  - Other: ShapeFromFile, Recursive Delete
Gmsh 4 introduces a new stable Application Programming Interface (API) for C++, C, Python and Julia, with the following design goals:

- Allow to do everything that can be done in .geo files
  - ... and then much more!
- Be robust, in particular to wrong input data (i.e. “never crash”)
- Be efficient; but still allow to do simple things, simply
- Be maintainable over the long run
To achieve these goals the Gmsh API

- is purely functional
- only uses basic types from the target language (C++, C, Python or Julia)
- is automatically generated from a master API description file
- is fully documented
import gmsh

gmsh.initialize()
gmsh.model.add("boolean")

R = 1.4; Rs = R*.7; Rt = R*1.25

gmsh.model.occ.addSphere(0,0,0,Rt, 2)
gmsh.model.occ.intersect([(3, 1)], [(3, 2)], 3)
gmsh.model.occ.addCylinder(-2*R,0,0, 4*R,0,0, Rs, 4)
gmsh.model.occ.addCylinder(0,-2*R,0, 0,4*R,0, Rs, 5)
gmsh.model.occ.addCylinder(0,0,-2*R, 0,0,4*R, Rs, 6)
gmsh.model.occ.fuse([(3, 4), (3, 5)], [(3, 6)], 7)
gmsh.model.occ.cut([(3, 3)], [(3, 7)], 8)

gmsh.model.occ.synchronize()
gmsh.model.mesh.generate(3)
gmsh.fltk.run()
gmsh.finalize()
#include <gmsh.h>

int main(int argc, char **argv)
{
    gmsh::initialize(argc, argv);
    gmsh::model::add("boolean");

    double R = 1.4, Rs = R*.7, Rt = R*1.25;

    std::vector<std::pair<int, int>> ov;
    std::vector<std::vector<std::pair<int, int>>> ovv;
    gmsh::model::occ::addBox(-R,-R,-R, 2*R,2*R,2*R, 1);
    gmsh::model::occ::addSphere(0,0,0,Rt, 2);
    gmsh::model::occ::intersect({{3, 1}}, {{3, 2}}, ov, ovv, 3);
    gmsh::model::occ::addCylinder(-2*R,0,0, 4*R,0,0, Rs, 4);
    gmsh::model::occ::addCylinder(0,-2*R,0, 0,4*R,0, Rs, 5);
    gmsh::model::occ::addCylinder(0,0,-2*R, 0,0,4*R, Rs, 6);
    gmsh::model::occ::fuse({{3, 4}, {3, 5}}, {{3, 6}}, ov, ovv, 7);
    gmsh::model::occ::cut({{3, 3}}, {{3, 7}}, ov, ovv, 8);

    gmsh::model::occ::synchronize();

    gmsh::model::mesh::generate(3);
    gmsh::fltk::run();
    gmsh::finalize();
    return 0;
}

gmsh/demos/api/boolean.cpp
Application Programming Interface

In addition to CAD creation and meshing, the API can be used to

- Access mesh data (getNodes, getElements)
- Generate interpolation (getBasisFunctions) and integration (getJacobians) data to build Finite Element and related solvers (see e.g. `gmsh/demos/api/poisson.py`)
- Create post-processing views
- Run the graphical user-interface
- Build custom graphical user-interfaces, e.g. for domain-specific codes (see `gmsh/demos/api/custom_gui.py`) or co-post-processing via ONELAB
In order to make this API easy to use, we publish a binary Software Development Toolkit (SDK):

- Continuously delivered (for each commit in master), like the Gmsh app
- Contains the dynamic Gmsh library together with the corresponding C++/C header files, and Python and Julia modules
Multi-Threaded meshing

Most meshing algorithms are now multi-threaded using OpenMP:

- 1D and 2D algorithms are multithreaded using coarse-grained approach, i.e. several curves/surfaces are meshed concurrently.
- The new 3D Delaunay-based algorithm is multi-threaded using a fine-grained approach. It currently lacks several features (embedded entities, hybrid meshes, ...), which will eventually be supported.

You need to recompile Gmsh with `-DENABLE_OPENMP=1` to enable this; then e.g. `gmsh file.geo -3 -nt 8 -algo hxt`
Multi-Threaded meshing

(a) 4-core Intel® Core™ i7-6700HQ CPU.

(b) 64-core Intel® Xeon Phi™ 7210 CPU.

<table>
<thead>
<tr>
<th># vertices</th>
<th>$10^4$</th>
<th>$10^5$</th>
<th>$10^6$</th>
<th>$10^7$</th>
<th>$10^8$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ours</td>
<td>0.032</td>
<td>0.13</td>
<td>0.85</td>
<td>7.40</td>
<td></td>
</tr>
<tr>
<td>Geogram</td>
<td>0.041</td>
<td>0.19</td>
<td>1.73</td>
<td>17.11</td>
<td></td>
</tr>
<tr>
<td>CGAL</td>
<td>0.037</td>
<td>0.24</td>
<td>2.20</td>
<td>23.37</td>
<td></td>
</tr>
</tbody>
</table>
# Multi-Threaded meshing

## Truck tire

| # threads | # tetrahedra | Timings (s) | |
|-----------|-------------|-------------|
|           |             | BR | Refine | Total |
| 1         | 123 640 429 | 75.9 | 259.7 | 364.7 |
| 2         | 123 593 913 | 74.5 | 166.8 | 267.1 |
| 4         | 123 625 696 | 74.2 | 107.4 | 203.6 |
| 8         | 123 452 318 | 74.2 | 95.5  | 190.0 |

## Aircraft

| # threads | # tetrahedra | Timings (s) | |
|-----------|-------------|-------------|
|           |             | BR | Refine | Total |
| 1         | 672 209 630 | 45.2 | 1348.5 | 1418.3 |
| 2         | 671 432 038 | 42.1 | 1148.9 | 1211.5 |
| 8         | 665 826 109 | 39.6 | 714.8  | 774.8  |
| 64        | 664 587 093 | 38.7 | 322.3  | 380.9  |
| 127       | 663 921 974 | 38.1 | 255.0  | 313.3  |

AMD EPYC 64 core
Multi-Threaded meshing

<table>
<thead>
<tr>
<th>100 thin fibers</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td># threads</td>
<td># tetrahedra</td>
</tr>
<tr>
<td>1</td>
<td>325 611 841</td>
</tr>
<tr>
<td>2</td>
<td>325 786 170</td>
</tr>
<tr>
<td>4</td>
<td>325 691 796</td>
</tr>
<tr>
<td>8</td>
<td>325 211 989</td>
</tr>
<tr>
<td>16</td>
<td>324 897 471</td>
</tr>
<tr>
<td>32</td>
<td>325 221 244</td>
</tr>
<tr>
<td>64</td>
<td>324 701 883</td>
</tr>
<tr>
<td>127</td>
<td>324 190 447</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>500 thin fibers</th>
</tr>
</thead>
<tbody>
<tr>
<td># threads</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>4</td>
</tr>
<tr>
<td>8</td>
</tr>
<tr>
<td>16</td>
</tr>
<tr>
<td>32</td>
</tr>
<tr>
<td>64</td>
</tr>
<tr>
<td>127</td>
</tr>
</tbody>
</table>

AMD EPYC 64 core
Robust STL remeshing

New pipeline to remesh discrete surfaces (represented by triangulations):

- Automatic construction of a set of parametrizations that form an atlas of the model
- Each parametrization is guaranteed to be one-to-one, amenable to meshing using existing algorithms
- New nodes are guaranteed to be on the input triangulation (“no modelling”)
- Optional pre-processing (i.e. edge detection) to color sub-patches if sharp features need to be preserved
Robust STL remeshing

Batman STL mesh
Robust STL remeshing

Automatic atlas creation: each patch is provably parametrizable by solving a linear PDE, using mean value coordinates
Robust STL remeshing
Robust STL remeshing

Automatic atlas creation, this time with feature edge detection
Robust STL remeshing

Remeshing with feature edge detection
CT scan of an artery: 101 patches created, most because of the large aspect ratio
Robust STL remeshing

Remeshing of a skull: 715 patches created for reparametrization; mesh adapted to curvature
Remeshing of an X-ray tomography image of a silicon carbide foam by P. Duru, F. Muller and L. Selle (IMFT, ERC Advanced Grant SCIROCCO): 1,802 patches created for reparametrization
Conclusions and perspectives

- Overview of Gmsh and recent developments:
  - Constructive Solid Geometry
  - Application Programming Interface
  - New multi-threaded algorithms
  - Robust STL remeshing based on parametrizations

- Many exciting developments in the pipeline:
  - Improved high-order remeshing
  - Hex-dominant meshes
  - (Semi-)automatic bloc-structured decompositions
  - Boundary layers
Post-Scriptum

• To download Gmsh: http://gmsh.info
• For references, see http://gmsh.info/#References

• For fun, go to the
  • Google Play Store (if you are on Android)
  • Apple AppStore (if you are on iOS)
and download the ONELAB app: it contains a full-featured version of Gmsh + the finite element solver GetDP
... so you can impress your friends by solving finite element models on your smartphone!