3D Simulation of Gerotor with Deforming Mesh by using OpenFOAM

Castilla, R, Gamez-Montero, PJ, Raush, G, and Codina, E
LABSON – Dpt. of Fluid Mechanics
Universitat Politècnica de Catalunya

UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola Superior d'Enginyeries Industrial, Aeroespacial i Audiovisual de Terrassa

Objectives of the work

- The main objective is to **numerically simulate the fluid flow inside a gerotor pump with an affordable approach** and, from these simulations
- Contribute to the cost effective process for the design of new gerotor pumps with satisfactory performance and efficiency parameters

There are several commercial numerical codes that are able to do that, but they are expensive and, moreover, aimed to standard simulations, so that new numerical models cannot be tested

The present work is completely performed with OpenFOAM with the support of python scripting.

The main advantages are:

- It is free, open and scalable
- It allows coding and model testing (with the proper skills…)
- There is a big users community in the cyberspace willing to help

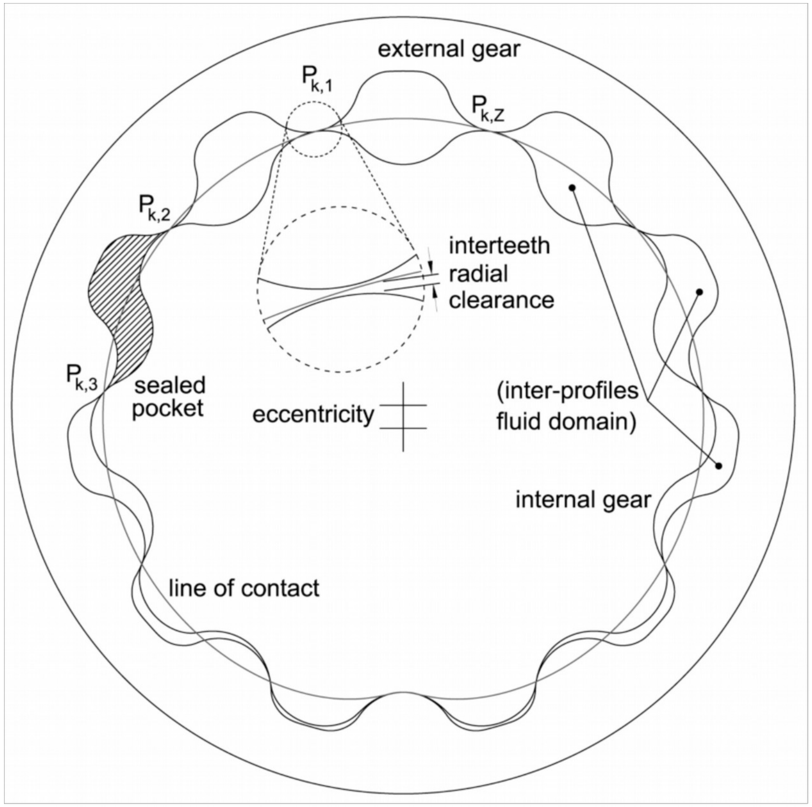
The main drawbacks are:

- The learning curve is really steep
- Although there are GUIs, the default commands environment is not friendly

What is a gerotor pump?

A pair of gears with trochoidal-teeth profile

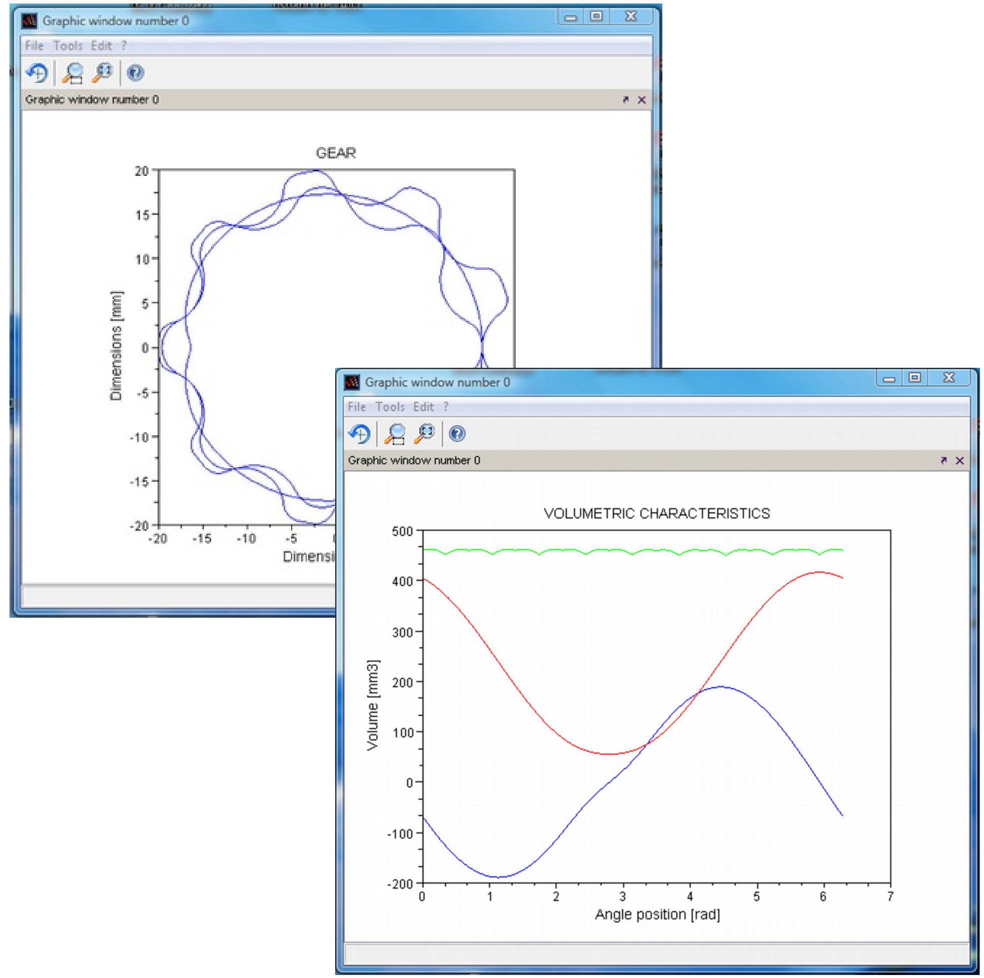
- Internal/Inner gear with external teeth
- External/Outer gear with internal teeth
- Gear are eccentric and rotate
  - In the same direction
  - Different speed (inner profile is faster than external)
- In theory, each internal tooth is in sliding contact → line of contact



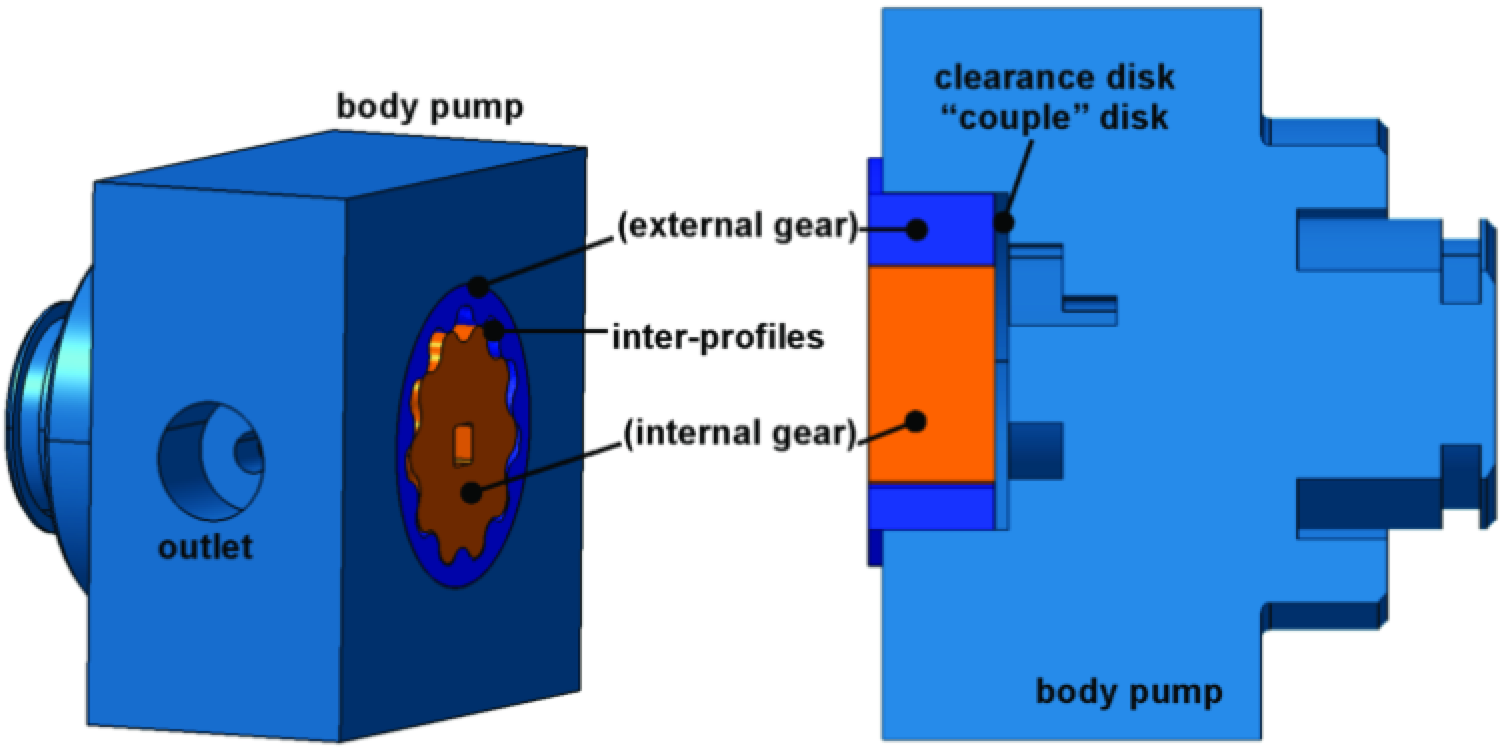
Our mini gerotor pump (mGp)

New-born design and construction.
Using GeroLAB (www.gerolab.es)

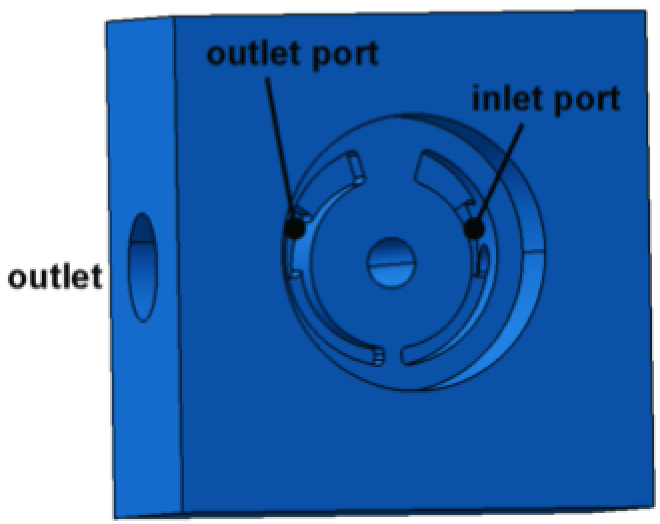
- Geometry
- Volumetric characteristics
- Contact stress

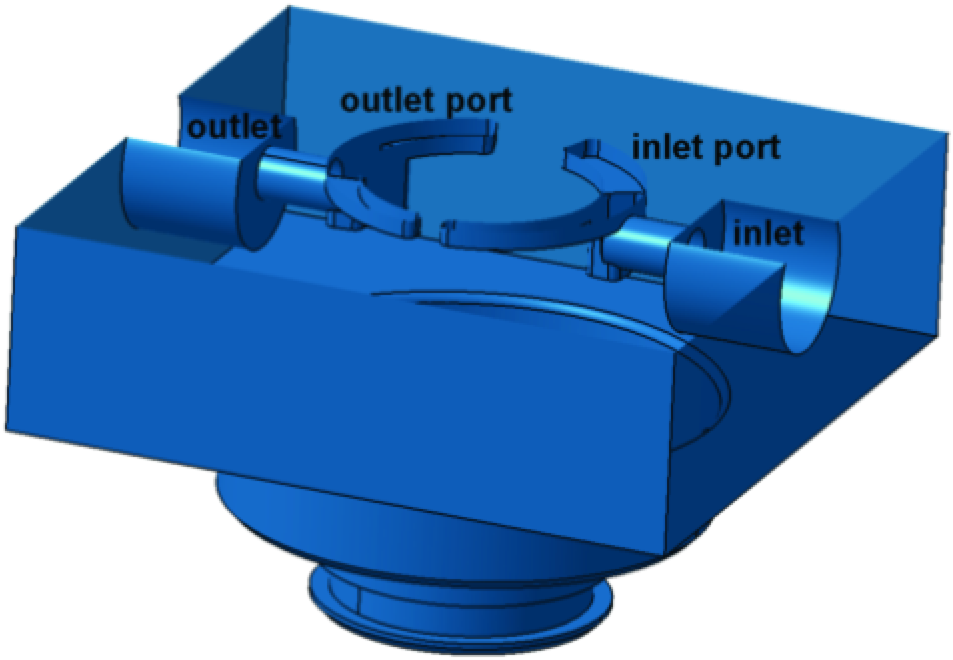


Our mini gerotor pump (mGp). Parts



Our mini gerotor pump (mGp). Parts





CFD. Fundamentals

Computational Fluid Dynamics uses numerical analysis and data structures to solve and analyze problems that involve fluid flows (wikipedia.org)

The basic aspects are:

- Proper geometry needed
- Discretization of both space domain (meshing) and time domain.
- Solver for solving transient Navier-Stokes equation
- Postprocessing

In the present case, there are some special features

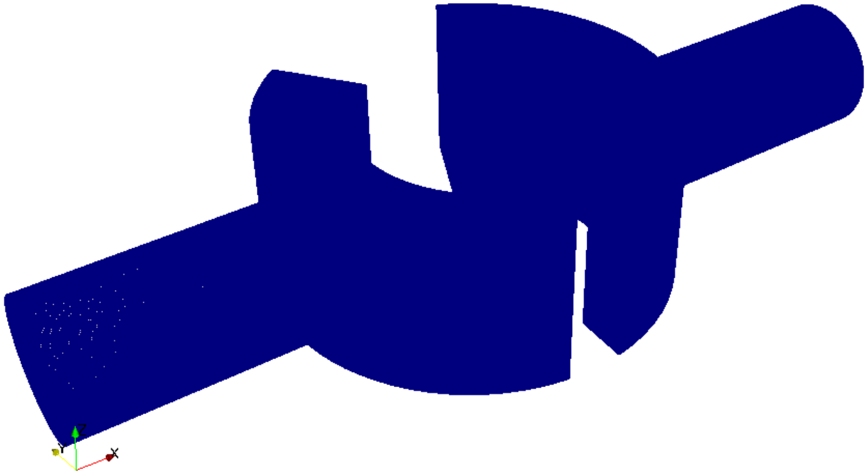
- Dynamic (deforming) mesh
- Solid-solid contact point
- Coupling of fluid flow between domains

CFD. Meshing the domain. The case

The case (pump body) has been meshed whit the tool snappyHexMesh (sHM), from the geometry in STL format.

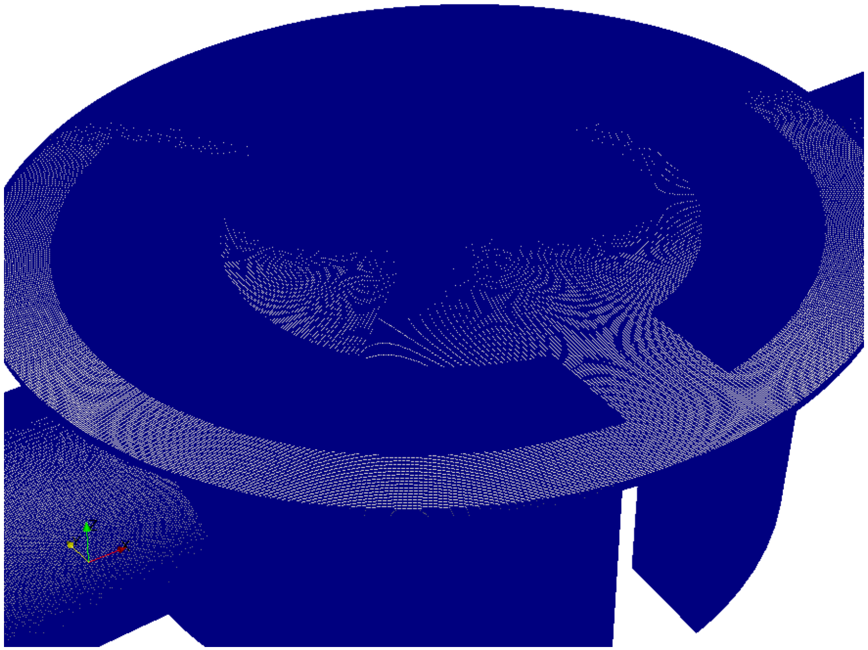
sHM creates a hexahedron dominated mesh with good quality (quality parameters are user-defined) for complex geometries

It starts from a background structured mesh (blockMesh) that is processed by refinement and elimination of cells and snap of the remaining



CFD. Meshing the domain. The clearance disk

- The clearance disk is discretized with a cylindrical structured mesh.
- In this way the number of layers in the clearance can be controlled.
- The coupling of disk and inlet/outlet ports are done with ACMI (Arbitrary Coupled Mesh Interface)



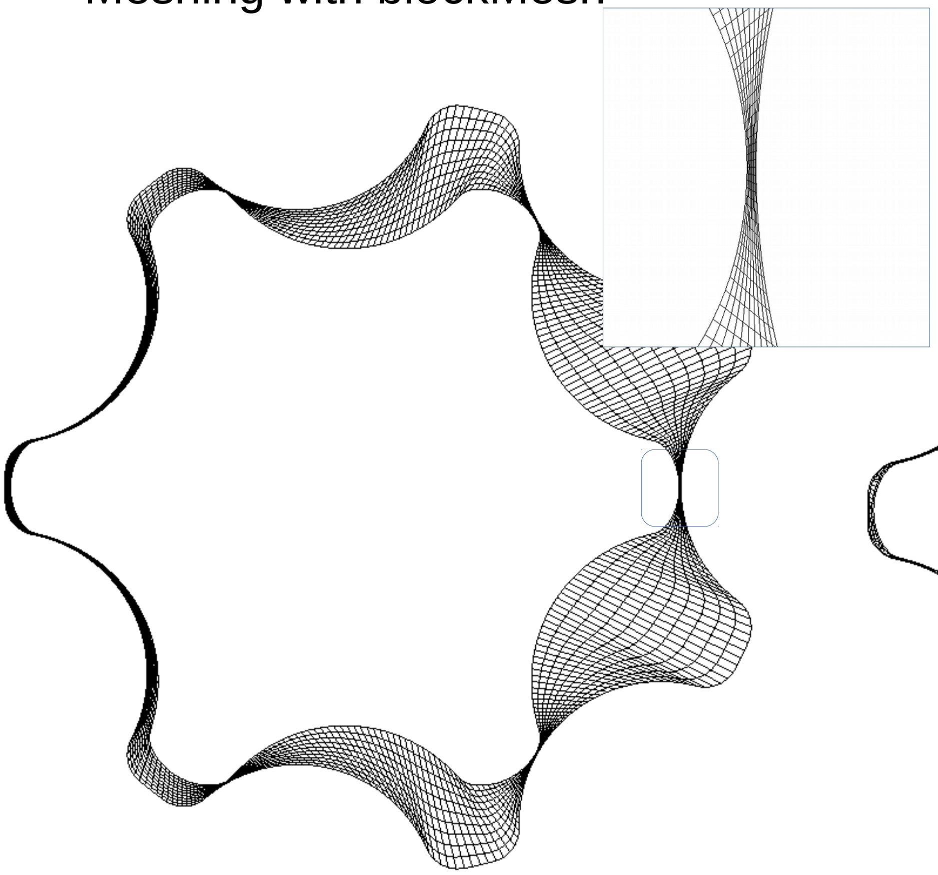
CFD. Meshing the domain. The inter-teeth domain

Several options have been considered for the inter-teeth fluid domain

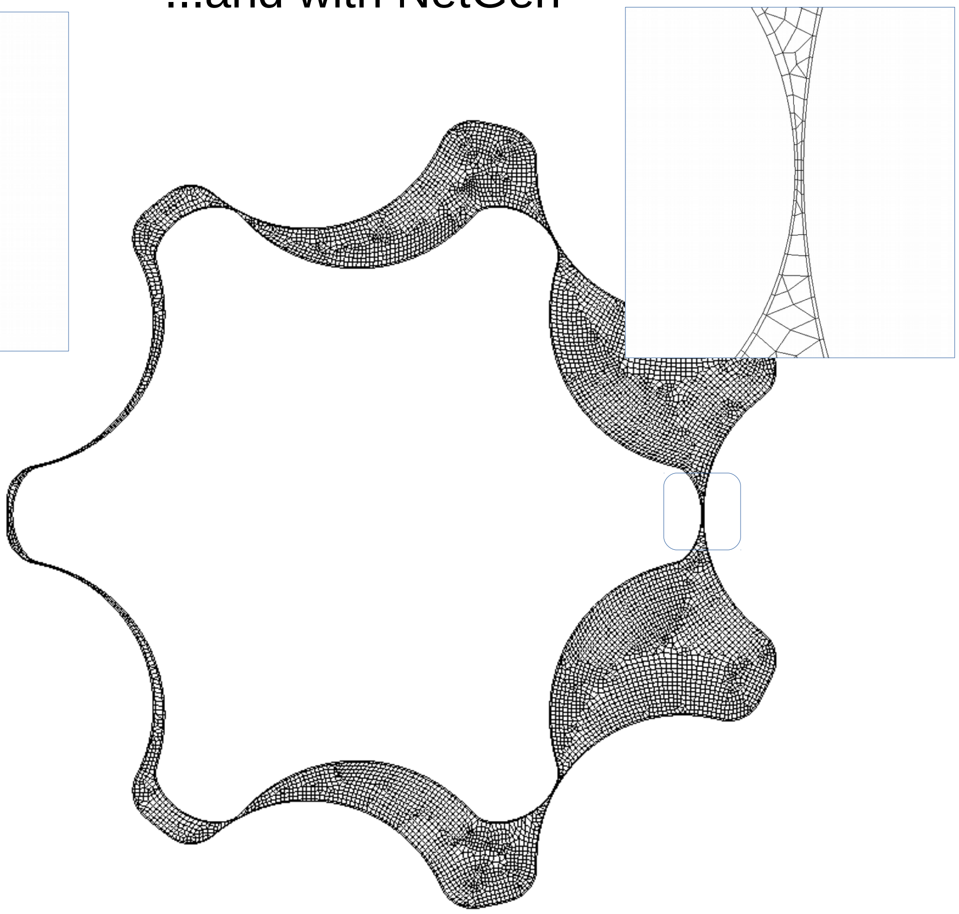
- *Meshing with snappyHexMesh* (like in the pump case).
  - An STL surface is needed.
  - Slow, and quality is not that good
- *Meshing with NetGEN* (Salome)
  - Fast and simple, if it is done with a python script
- *Meshing with blockMesh* (like in the clearance disk)
  - Fast
  - Possible with the help of a python tool available in github: blockmeshdicthelper

CFD. Meshing the domain. The inter-teeth domain

Meshing with blockMesh

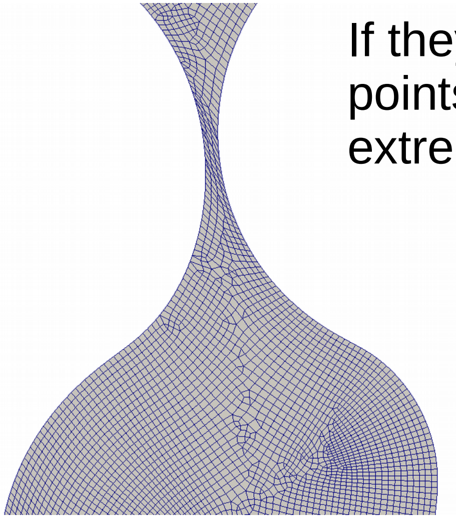


...and with NetGen



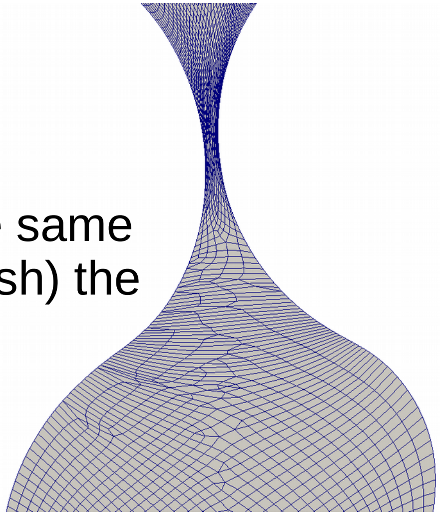
CFD. The Dynamic Mesh

Problem: How to move the mesh points in the profiles?



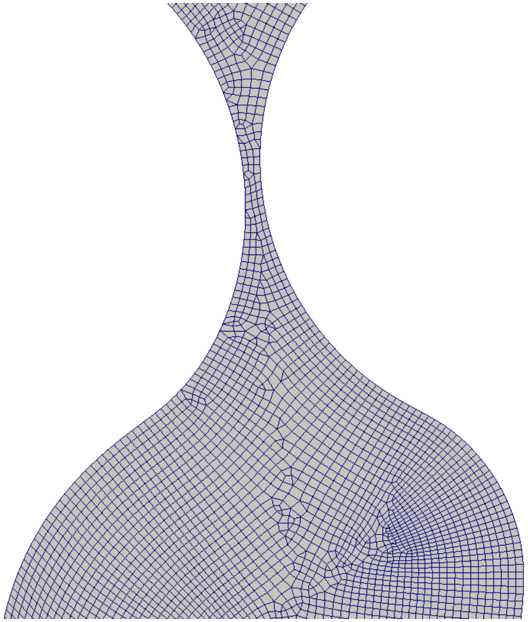
If they move with the geometry points, the mesh becomes extremely distorted

If they remains in the same position (slipping mesh) the grading is lost.



CFD. The Dynamic Mesh

The solution is to define a partial slip.
Mesh is moving with a different velocity from the geometry points
This angular velocity (mesh velocity) has some value between the velocity of inner profile and of outer profile.



It can be, for example, the average of both velocities

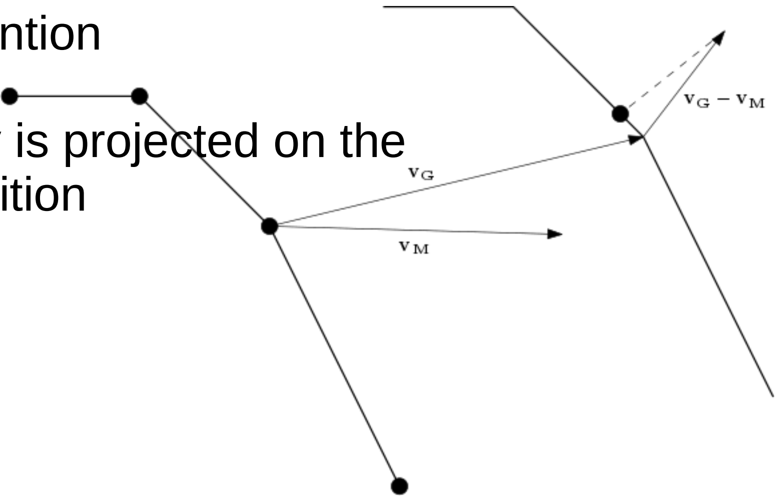
… but this only works for certain positions

For proper mesh motion, velocity has to be calculated with contact point motion as a function of angular position

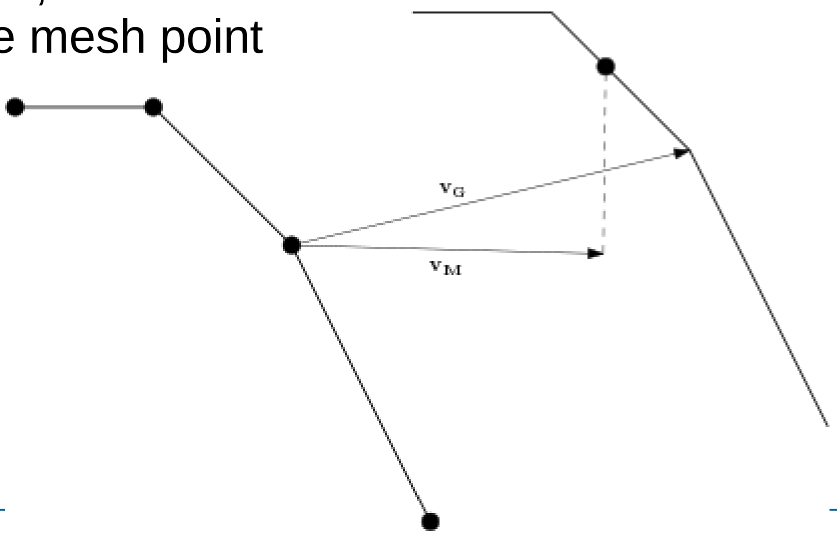
CFD. The Dynamic Mesh

Slipping mesh requires also special attention

In usual slip method the relative velocity is projected on the surface to estimate new mesh point position



This works (fairly) when velocity is not very normal to the surface
In this case, it is more convenient to project the mesh point



CFD. The Dynamic Mesh

Finally, mesh motion should be smoothed in order to avoid cells distortion that can eventually degenerate to negative volumes.

Unfortunately there is no motion solver with smoothing in standard OpenFOAM distribution.

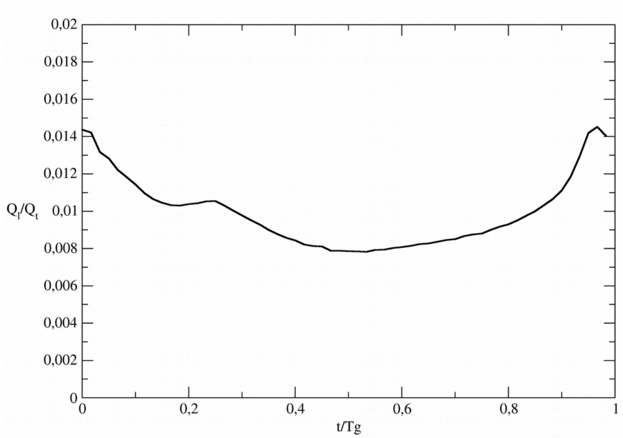
Mesquite library is an option but it only works for tetrahedral meshes.

Another option that is being explored is the contribution to the development of hexMeshSmoother library.

It still requires some coding...

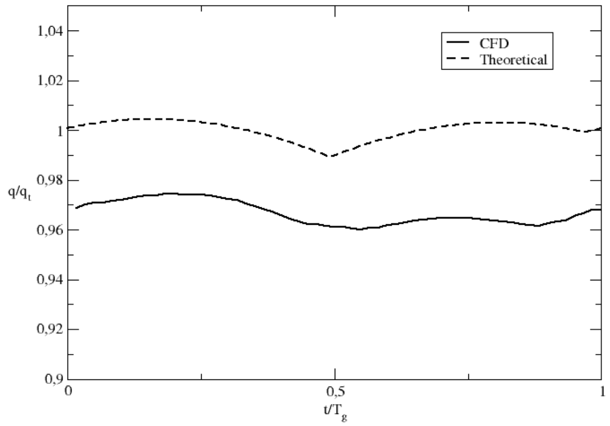
CFD. Some results

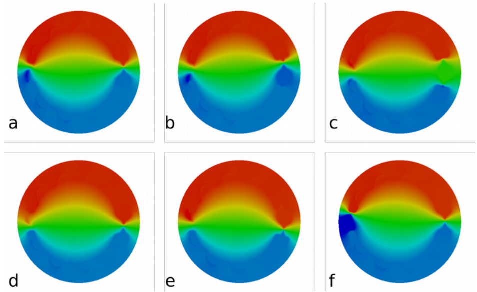
- So far, results with smooth teeth can be presented (Z=11)
- 5.25 Mcells mesh. 50 microns clearance disk in a HPC cluster using 64 cores



Leakage flow rate in clearance disk

Instantaneous flow ripple (and theoretical computation with geroLab)





Pressure distribution on clearance disk

Conclusions

- A simulation of a gerotor pump is presented, with OpenFOAM, based on three fluid domains (case, clearance disk and inter-teeth) coupled with ACMI
- Dynamic mesh in inter-teeth domain is not straightforward. It requires
  - Proper quality meshing (blockMesh)
  - Proper definition of mesh velocity
  - Proper implementation of slipping mesh
  - Proper smoothing process for cells in order to avoid negative volumes
  - Some results with smooth teeth have been presented

Thank you for your attention!

Contact: Robert Castilla
robert.castilla@upc.edu
Also in forum of cfd-online, user rcastilla