# Blood flow in cerebral arteries – automated way from Computed Tomography to ANSYS Fluent

**JAKUB BODYS**

**JAKUB PORAJ**

**MACIEJ KRYŚ**

Authors: Jakub Bodys (jakub.bodys@gmail.com), Jakub Poraj(jakubporaj@gmail.com), Maciej Kryś (mkrys@mesco.com.pl), MESCO, Poland

## Abstract

With the constant growth of computer simulation significance in science and engineering, many new fields are gaining access to these powerful tools. One of these new disciplines is medicine. Human body provides many fascinating areas that could be researched from completely different angle and could gain all the benefits that computer simulation offers. For example blood flow in human arteries can be studied using Computational Fluid Dynamics. Researchers of cerebrovascular disorders can get an insight view on physical phenomena of blood flow and study risk factors of embolism or cerebral aneurysm.

Main issue in using computer simulation in medical research is the complexity and uniqueness of geometry that needs to be handled. After all, human body is one of the most sophisticated engineering systems created by nature. In this paper, a workflow for creating a numerical mesh for CFD simulation purposes is shown. Application shown in the example focus on cerebral arteries blood flow simulation. Numerical mesh is generated based on CT scan of patient's head, using freeware tools Slicer3D and AutoIt3 as well as commercial software ANSYS Fluent Meshing 15.0.

KEYWORDS: **Computational Fluid Dynamics (CFD)**, **blood flow**, **meshing**, **Computed Tomography Scan (CT scan)**

## Introduction

Numerical simulation supports medical world in caring for human health. From artificial heart to displacements of mandible bone parts and airflow in lungs, state of the art computer solutions allow us to take a closer look at what happens in our bodies.

Today's engineering software gives us a possibility to manufacture highly efficient medical devices, that can save human life or greatly improve medical diagnostics.

Blood flow irregularities can cause many cerebrovascular disorders. For instance, thrombus may occur in areas where low velocity of blood flow is observed. These blood cloths may jam the arteries (embolism) and can even lead to heart attack. Furthermore, high wall shear stress induced by blood flow can damage artery walls and create cerebral aneurysm [1]. Cerebral aneurysm is a very dangerous cerebrovascular disorder which in case of rupturing will cause stroke. Statistic says that about 2% of population may have cerebral aneurysm [2] which makes it very serious issue.

Medical environment may find it very useful to use modern Computational Fluid Dynamics (in short CFD) tools to analyse blood flow in cerebral blood arteries of patients. It could greatly improve our knowledge about formation and preventing cerebrovascular disorders. The biggest challenge in using CFD tools in such application is handling the arteries geometry. Solving problems with non-human made design is generally more sophisticated due to non-regular shapes of objects. Our bodies consist of irregular, rugged parts and it is crucial to precisely prepare numerical model to receive valuable outcome. Another issue is that medical personnel that could make use of such tools may not have sufficient experience in CFD modelling field. Therefore, there should be some tools available that would deal with model preparations in fast and mostly automatic way. Finally, calculations will require to take into account real blood and heart action parameters, such as pressure, flow ratio etc.

In this paper, a workflow for creating a numerical mesh for such applications is shown using freeware tools Slicer3D, AutoIt3 and commercial software for CFD analysis ANSYS Fluent 15.0 with its meshing module.

## Creating the geometry

To obtain most accurate geometry for a specific patient a CT scan can be used (fig. 1). Tomography output files contain series of 2D pictures of patient's head. Simple open-source tool – Slicer3D (http://www.slicer.org/) convert the DICOMM libraries (CT scan output libraries) to STL file.
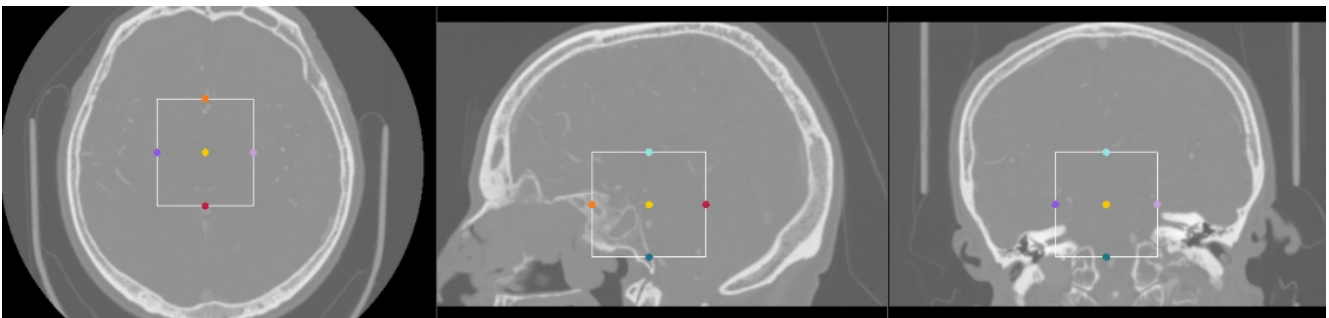


Fig. 1. CT scan processed in Slicer 3D

First step to extract the 3d geometry is to narrow the region of interest - it makes easier to handle the geometry when it is focused on particular part of the head. Then,

we can use tools that are based on the grayscale thresholds of CT scans. This step creates some first difficulties. Since the bone parts have the same grayscale level as the arteries, the geometry is often a mixture of arteries and bones, glued together. This issue will be addressed in next steps.
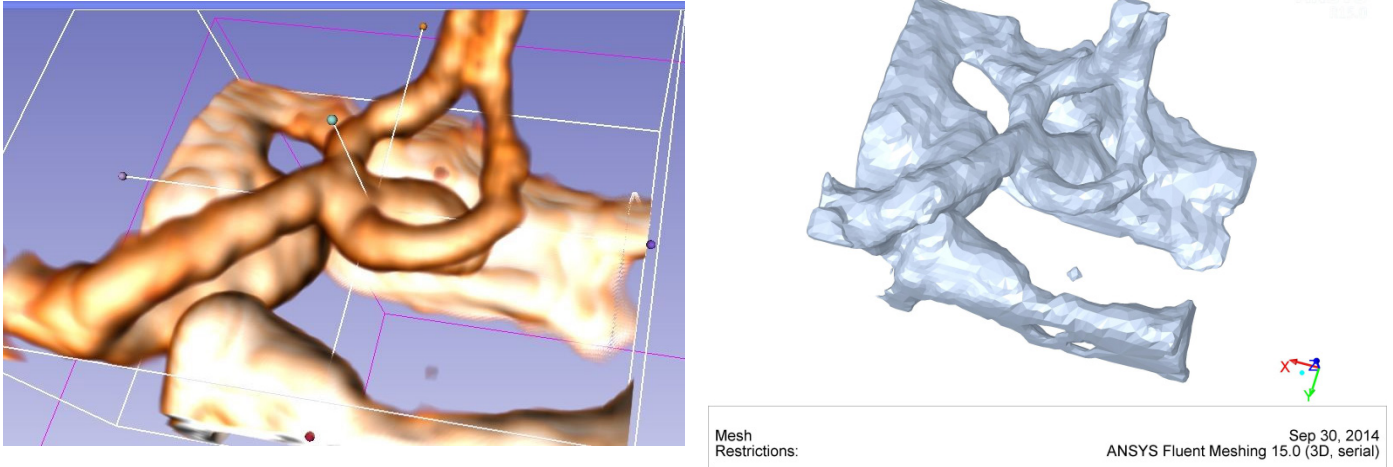


*Fig. 2. On the left model visualization in Slicer 3d. On the right imported STL model in Ansys Fluent*

Extracted STL file can now be transferred to Fluent Meshing (fig. 2). The process of preparing the mesh includes four basic steps: separating the arteries from the bones, repairing the artery surface, creating boundary conditions and finally, generating the volumetric mesh. This workflow was included in Fluent Meshing with journals (scripts automatically executed by the application), and those are controlled by AutoIT3 software (www.autoitscript.com/site/). All of those scripts perform actions that normally user would have to perform by himself. This helps to reduce time and user input significantly. AutoIT3 allows user to choose between manual or automatic execution of mesh operation. It allows to control the stages of each process and check the quality after each step.

## Separating bones from arteries

As mentioned before imported model includes both arteries and bones. Unwanted elements are removed from model with separating macros which tracks the artery surface by connection angle between each cell. More complicated geometries have to be separated manually.

## Repairing the boundary surface

During the separation process boundary surface of artery will be ruptured in the areas where it used to be connected to the bone. All holes should be repaired before proceeding. Holes closure is performed in automatic manner and is very quick. Next step is improving the surface mesh quality (fig. 3).
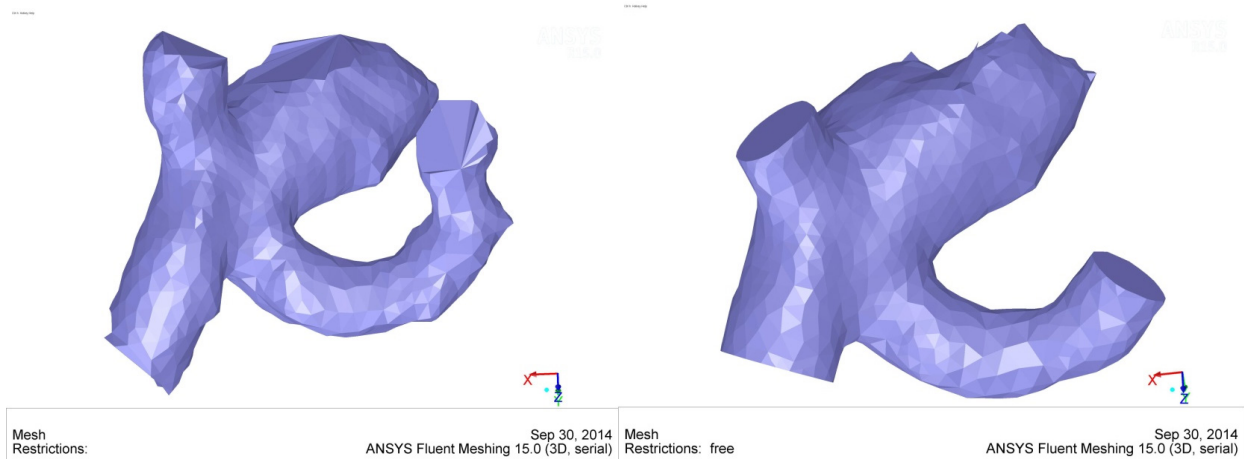
*Fig. 3. On the left: separated artery mesh with holes closed; on the right after first smoothing operation*

Surface mesh consists of many triangle elements that are sewed together, creating a continuous surface. Better the quality of each triangle the better will the inside mesh be, and more accurate results from the simulation [3]. The element quality is measured by skewness parameter which, in short, says how well-shaped is the triangle. Improving the mesh quality phase is done in two step process. First the algorithm selects the low quality elements for smoothing. These filtered elements are then split into smaller pieces and their nodes are moved around, so their shape is improved. Second part includes global refining the shape of artery in terms of all elements quality - it makes more natural, smoothed surface (fig. 4).

## Boundary condition preparation

Creating particular inlet/outlet zones is simple. User is required to point onto artery where boundary condition have to be. Scripts allow to make up to 10 inlets/outlets matched to edge of boundary edge. Default name for inlet/outlet may be later changed preferably.
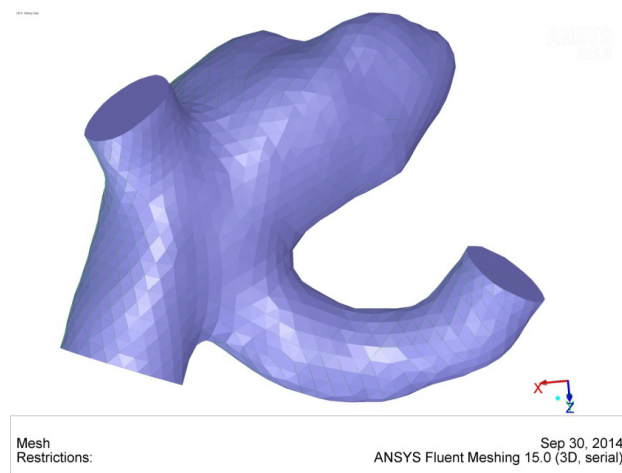


*Fig. 4. Mesh after global smoothing and boundary conditions formation*

## Generating the volumetric mesh

Volumetric mesh can be created automatically or with step-by-step method (for complex geometries and quality control). Mesh is built upon boundary mesh prepared in previous steps. First the prism layers are built (crucial for boundary layer computation) and then, the remaining space is filled with tetrahedral elements (fig. 5). Last macro sorts every zone and changes its names to more user friendly.
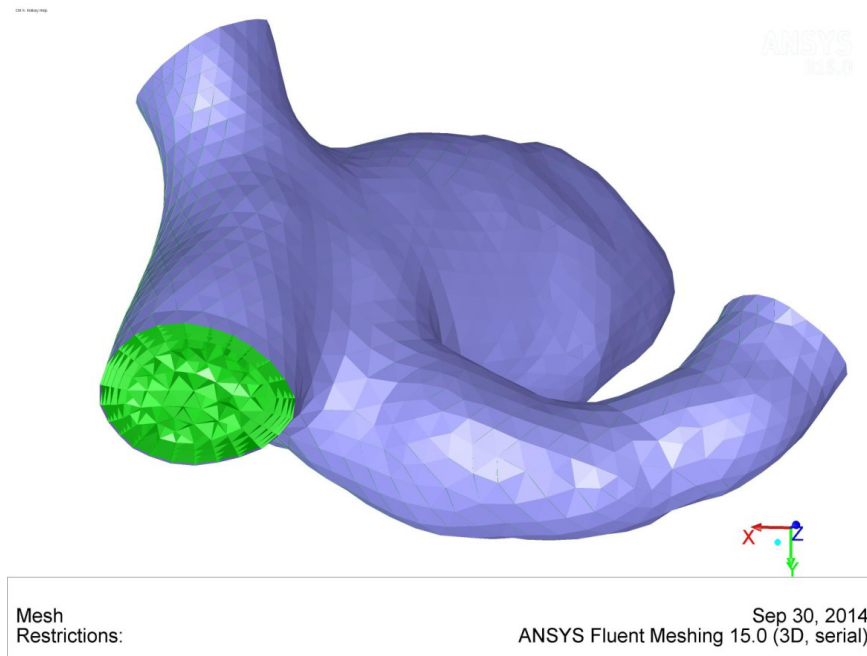


*Fig. 5. Final mesh, ready for the simulation*

## What we get from 2d pictures?

Starting from series of 2D pictures we created high quality mesh ready to use in simulation using Ansys Fluent (fig. 6). The main advantage of such approach is significant reduction of time and human resources needed to work at the geometries. One case processed with presented workflow takes about 45 minutes to complete. Thanks to the time savings it is possible to process many cases in relatively small amount of time. And with the ease of use such workflow can be used by CFD not-experienced personnel.
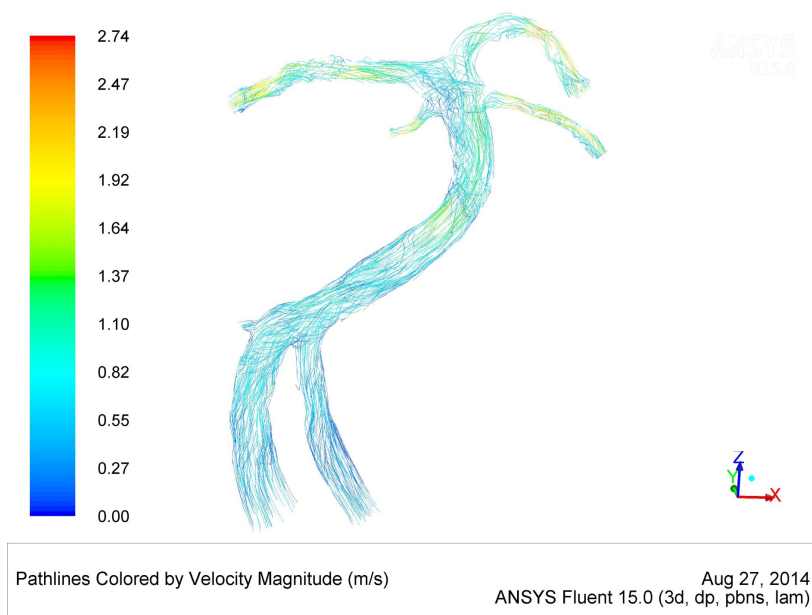
*Fig. 6. Flow visualization through a cerebral arteries*

## References:

Papaioannou, Stefandis: Vascular Wall Shear Stress: Basic Principles and Methods. Hjc Hellenic Journal of Cardiology 46: 9-15, 2005

Ajiboye, Chalouhi, Starke , Zanaty, Bell, "Unruptured Cerebral Aneurysms: Evaluation and Management," The Scientific World Journal, vol. 2015

ANSYS® Fluent, Release 15.0, Help System, Fluent User's Guide, ANSYS Inc.