Cardiovascular diseases are very common today and there are many difficulties for specialists in curing them. There are a lot of difficulties in diagnostics. Due to modern technology, today’s medical instruments can give much more raw data that can be used for computational modeling, and that kind of modeling is very common nowadays. The 3D model of a particular organ is the base for computational simulation.

Computational simulating can be drastically sped up by parallel computing using a GRID infrastructure – thus time need for calculation that would last for a week (even on good computers) is reduced to a few hours on few computing nodes.

Computer simulations of blood flow through the cardiovascular organs give today’s scientists a powerful advantage. They can provide detailed information about circulation of the fluid – velocities, pressure and shear stress even in the zones that are very difficult to inspect. Thus a future medical intervention can be planned using information provided by simulation.

The motivation for this work was the aorta modeling in corresponding image format (.stl) that our research center recently received from the University Clinical Center in Heidelberg from a multislice CT scanner. In this particular case from raw .stl data (from CT scanner) using Gmsh [1], and TetGen [2] as well our own software tools (STL Toolz, Schvorova) we created a 3D 8-node (brick) mesh model which is input for our solver PAKF [3]. Due to a huge number of finite elements (about 160,000) the calculations were done on GRID.

Figure 1. Raw .stl from CT scanner
The boundary is provided in .stl format, which defines a set of triangles on the object boundary. It is obvious that the object boundary that is given in raw .stl (Figure 1) is not adequate for there are many objects that are sufficient. Some of them are independent and some of them are attached to the main object (aorta in this particular case).

The first step in creating an adequate boundary is removing sufficient objects that are not attached to the main object. This procedure was done using Gmsh.

In the second step, (step i, Figure 4) sufficient objects that are attached to the main (aorta) were cut. This procedure was not as easy as the first one because these objects were actually parts of the main. We created a software (STL Toolz) which as input takes .stl surface boundary with hole (where the particular object was cut) and as output returns .stl patch automatically (step ii, Figure 4). There is also a module of software which works with two holes where the aorta is connected with two branches, and other parts with three or more branches could be reduced to a two branch variant.

After this the arterial branches were shortened and the boundary of the model was finished. Patching was very fine so that we could maintain the smoothness of the boundary (Figure 2).

![Figure 2. Part of the boundary before and after the intervention, triangular structure is visible and nevertheless it is almost impossible to identify the patch.](image)

MESH

The next part of the work was meshing where we used TetGen (step iii, Figure 4). As input for TetGen we used .stl file and result was a tetrahedral mesh with about 12,000 nodes. The number of nodes in the mesh was drastically boosted with the number of boundary nodes (because of the quality of given boundary – density of the nodes on boundary).

In the next step (step iv, Figure 4) re-meshing is done using our application 8chvorova. In this procedure input mesh information (nodes, edges, elements) is taken from TetGen as well as inlet/outlet data from the aorta with arteries by STL Toolz. These data have information about edges that were "alone" when the particular part of the main boundary was cut (and the whole was patched). The result of this step is standard .DAT input file for our solver. This file contains data concerning mesh, boundary and initial conditions, material constants, prescribed velocities.

The mesh created from TetGen is re-meshed into bricks as shown in the Figure 3.

![Figure 3. Re-mesh: 1 tetrahedra into 4 bricks](image)

It is probably obvious from Figure 4 that classes of the gravity centers are respectively for 1D, 2D and

![Figure 4. Diagram of the whole process](image)
3D objects (line segments, triangles and tetrahedron). That is very important because the new nodes in the mesh of tetrahedrons have to be placed correctly. For the first and second class of new nodes algorithm guarantee that nodes that are same for neighbor tetrahedrons will be the same (the same nodes on the same position). The number of nodes was drastically increased in this re-meshing due to splitting of tetrahedra, so in this particular example the number is increased from 12,000 nodes in the tetra-mesh to 205,000 in brick mesh. In this re-meshing procedure the model was also resized adequately.

SOLVING

The final step in the whole process was calculation on GRID. This was done on our GRID site AE:GI04-KG on 8 computational nodes, and time used for calculation was about 12 hours.

In Figure 5 (wall shear stress) and Figure 6 (velocity) is shown resulting step in pick of systole.

CONCLUSION

The main idea for whole this work is its practical medical application in clinics. Preparation of raw data from CT scanner lasts about 2 to 3 days, modeling and meshing up to 6 hours and solving up to 24 hours, so we think that this approach is not far from being implemented into real clinical practice.

REFERENCES


IZVOD

SOFTVERSKI ALATI ZA AUTOMATSKO GENERIRANJE MREŽE KONAČNIH ELEMENATA I PRIMENA BIOMEHANIČKIХ PRORAČUNA U MEDICINI

(Naučni rad)

Danko Z. Milašinović1, Miloš R. Ivanović1, Nenad D. Filipović1,2,3, Miloš R. Kojić1,3
1Centar za naučna istraživanja SANU i Univerzitet u Kragujevcu
2Mašinski fakultet, Kragujevac
3Harvard School of Public Health, Boston, USA


 Ključne reči: Kardiovaskularni sistem • Aorta • Protok krvi • Metoda konačnih elemenata •

Key words: Cardiovascular system • Aorta • Blood flow • Finite elements •

D. MILAŠINOVIC, et al.: SOFTWARE TOOLS FOR AUTOMATIC...