

Modelowanie i symulacja zagadnień biomedycznych

Tomasz Stręk

Institute of Applied Mechanics, Poznan University of Technology

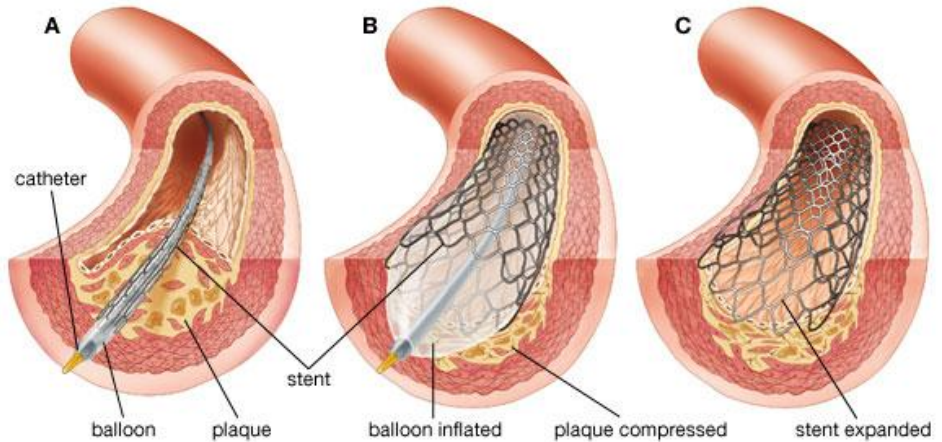
ul. Jana Pawła II 24, 60-965 Poznan, Poland

Room 438

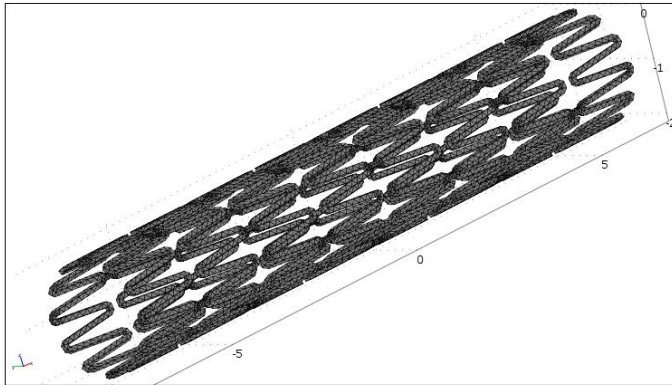
www.strek.strefa.pl

DATE: 2015.05.11

MEDICAL STENT – STRUCTURE MECHANICS ANALYSIS



GEOMETRY MODEL & MESH

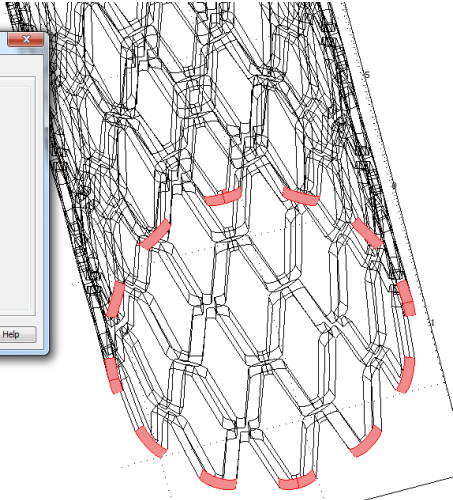
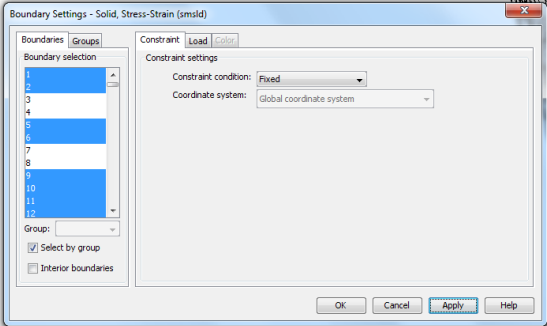


MATERIAL PROPERTIES

The image shows a screenshot of a finite element software interface. The main window displays a mesh of a red, wavy structure. Two dialog boxes are open over the mesh:

- Constants:** A table with columns for Name, Expression, Value, and Description. It contains two entries: μ with value 1e2 and ν with value 303.
- Subdomain Settings - Solid, Stress-Strain (smid):** A dialog box for defining material properties. It includes tabs for Material, Constraint, Load, Damping, Initial Stress and Strain, Init, Element, and Color. The Material settings are as follows:
 - Library material: TI Grade 1 (UNS R30250)
 - Material model: Isotropic
 - Coordinate system: Global coordinate system
 - Use mixed U-P Formulation (nearly incompressible material):
 - Quantity: Value, Expression, Unit, Description
 - E: $E[1, K][Pa]$, Pa, Young's modulus
 - ν : $\nu[1, K]$, -, Poisson's ratio
 - α : $\alpha[1, K][1, K]$, 1/K, Thermal expansion coeff.
 - ρ : $\rho[1, K][kg, m^{-3}]$, kg/m³, Density

BOUNDARY CONDITIONS – TYPE 1- FIXED



BOUNDARY CONDITIONS – TYPE 2 - LOAD

The screenshot displays the COMSOL Multiphysics interface. The main window shows a 3D model of a structure with a red mesh. A dialog box titled "Boundary Settings - Solid, Stress-Strain (simul)" is open, showing the "Load" tab. The "Type of load" is set to "Distributed load". The "Coordinate system" is set to "Tangent and normal coord. sys. (t_n, n)". The "Quantity" table is as follows:

Quantity	Value/Expression	Unit	Description
F_{ts}	0	N/m^2	Face load (force/area) (t - s).
F_{tn}	0	N/m^2	Face load (force/area) (t - n).
F_{ns}	0	N/m^2	Face load (force/area) (n - s).

The dialog box also includes a "Boundary selection" list on the left, a "Group" dropdown, and checkboxes for "Select by group" and "Interior boundaries". The main window shows a 3D model of a structure with a red mesh. The structure is a long, thin, curved beam with a repeating pattern of loops. The mesh is composed of small, rectangular elements. The structure is oriented along the x-axis, with the z-axis pointing upwards and the y-axis pointing out of the page. The structure is supported at the bottom by a fixed boundary condition. The top surface of the structure is highlighted in red, indicating the location of the distributed load. The dialog box is positioned over the top-left corner of the 3D model.

BLOOD VESSELS - FLUID DYNAMICS ANALYSIS

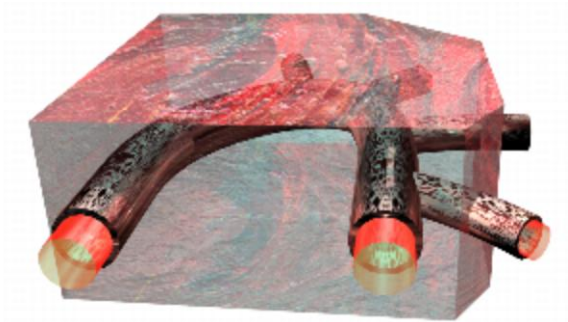
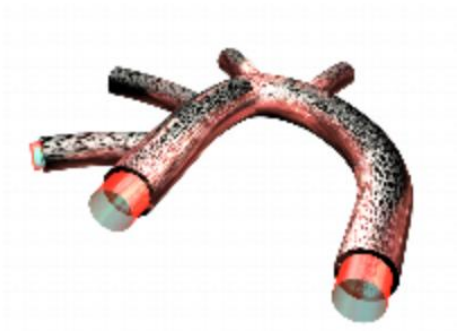
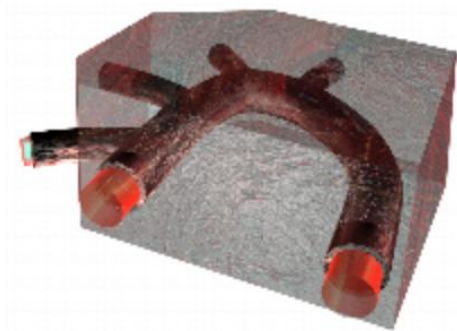


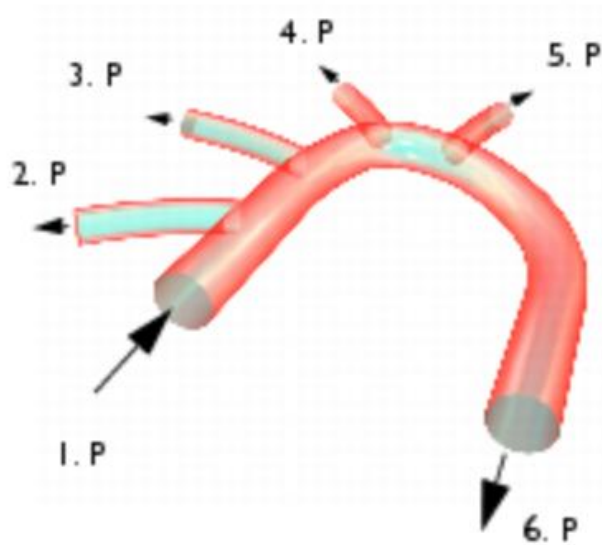
Figure 5-1: The model domain consists of part of the aorta, its branches, and the surrounding tissue.



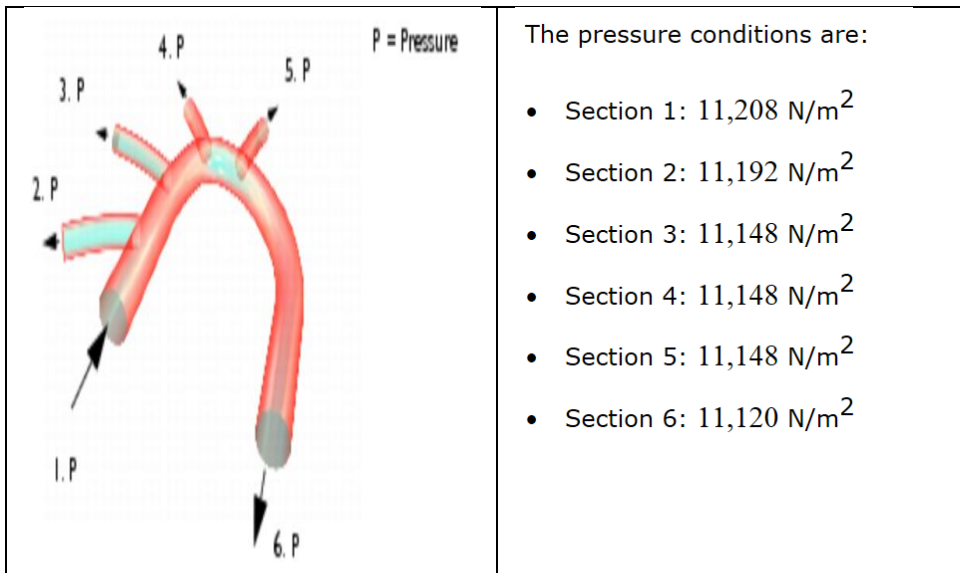
MATERIALS

The model in this discussion uses the following material properties:

- Blood
 - density = 1060 kg/m^3
 - dynamic viscosity = 0.005 Ns/m^2



P = Pressure



$$f(t) = \begin{cases} \sin \pi t & 0 \leq t \leq \frac{1}{2} \text{ s} \\ \frac{3}{2} - \frac{1}{2} \cdot \cos\left(2\pi\left(t - \frac{1}{2}\right)\right) & \frac{1}{2} \text{ s} \leq t \leq \frac{3}{2} \text{ s} \end{cases}$$

SCALAR EXPRESSION

f (t<=0.5)*sin(pi*t[1/s])+(t>0.5)*(1.5-0.5*cos(-2*pi*(0.5-t[1/s])))

NUMERICAL RESULTS

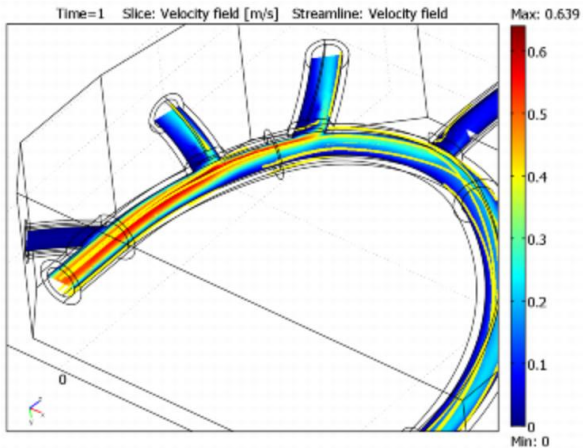


Figure 5-4: Velocity field color slice and flow lines in the aorta and its ramification (branching).