

[Skip to content](#)



[New chat](#)

[CtrlShift0](#)



Search chats

CtrlK



[Library](#)



[Sora](#)



[GPTs](#)

Chats

[COMSOL simulation setup](#)



[COMSOL heat transfer model](#)



[COMSOL model setup](#)



[COMSOL geometry setup](#)



[Perturbation expansion terms](#)



[Metrics comparison](#)



[Clarify Tw's meaning](#)



[পিথাগোরাস প্রমাণ \(a-b\)^2 দিয়ে](#)



[চেনা অচেনা পার্থক্য](#)



[Usual metric explanation](#)



[Martingale proof explanation](#)



[বাংলাদেশের প্রথম সদস্যপদ](#)



[Itô integral variance calculation](#)



[তড়িৎ বিভব ব্যাখ্যা](#)



[ICJ arrest power explanation](#)



[Resin types and uses](#)



[আয়কর আইনজীবী সরকারি নয়](#)



[Christoffel symbols spherical coordinates](#)



[ধ্বনিবাজনা ব্যাখ্যা](#)



[আধার শব্দের অর্থ](#)



[উজান শব্দের ব্যাখ্যা](#)



[At loggerheads meaning](#)



[COMSOL app crashes solution](#)



[Zeroth-order equation derivation](#)



[Sentence analysis](#)



[Trade war impact comparison](#)



[Symmetric tensor components](#)



[Second moment calculation](#)



Titu Sarker

Free

Open sidebar

ChatGPT

ChatGPT

Share

You said:

Problem Description: Hemodynamic Simulation of Internal Carotid Artery (ICA) with Carotid Sinus (CS) Part (a): 2D Geometric Model Construction Construct a 2D axisymmetric model of the Internal Carotid Artery (ICA) with an ellipsoidal Carotid Sinus (CS) using COMSOL Multiphysics, with the following specifications: ICA dimensions: Diameter = 5.57 mm Length = 53 mm Carotid Sinus (CS) geometry: Ellipsoid with major axis = 4.6 mm, minor axis = 1.55 mm Positioned such that its center is at: 19 mm from the bottom wall (inlet) 38 mm from the top wall (outlet) Shares the same axis as the ICA walls Part (b): Governing Equations (Physics Interface) Implement the incompressible Navier-Stokes equations for hemodynamics: Continuity Equation (Mass Conservation): $\nabla \cdot \mathbf{u} = 0$ $\nabla \cdot \mathbf{u} = 0$ Momentum Equation: $\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u}$ $\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u}$ Given parameters: Blood density (ρ) = 1080 kg/m³ Dynamic viscosity (μ) = 0.00325 Pa·s Part (c): Boundary Conditions & Pulsatile Flow Simulation Inlet Boundary Condition (Pulsatile Flow): $\mathbf{u} = -U_0 \mathbf{n}$, where $u = -U_0$, where $U_0 = u_{max} \cdot f(t)$ $U_0 = u_{max} \cdot f(t)$ $f(t) = \{ 0.8 \sin(2.857 \pi t), 0 \leq t \leq 0.35 \text{ s}$ $0.6 \sin(2.22 \pi (t - 0.35)), 0.35 \leq t \leq 0.8 \text{ s}$ $f(t) = \{ 0.8 \sin(2.857 \pi t), 0.6 \sin(2.22 \pi (t - 0.35)), 0 \leq t \leq 0.35 \text{ s}$ $0.35 \leq t \leq 0.8 \text{ s}$ Peak velocity (u_{max}) = 0.5 m/s Outlet Boundary Condition: Uniform gauge pressure = 0 Pa Wall Boundary Condition: No-slip condition ($\mathbf{u} = 0$) at ICA walls Part (d): Post-Processing & Parametric Sweep Analysis Required Results: Contour Plots: Velocity magnitude distribution inside ICA Pressure distribution inside ICA Wall Shear Stress (WSS): Plot WSS along the ICA walls Velocity Profiles: Extract velocity profiles at axial positions: 8 mm, 15 mm, 19 mm (CS center), 38 mm Parametric Sweep Analysis: Repeat simulations for varying $u_{max} = 0.25, 0.45, 0.65, 0.85 \text{ m/s}$ Compare results for velocity, pressure, and WSS comsol



ChatGPT said:

Great! You want to set up a **COMSOL Multiphysics** model for simulating blood flow through the Internal Carotid Artery (ICA) with a Carotid Sinus (CS), including geometry construction, physics setup, boundary conditions, and post-processing, along with parametric sweep on peak velocity.

Here's a detailed **step-by-step workflow** to implement your problem in COMSOL:

Part (a): 2D Axisymmetric Geometric Model Construction

1. **Start COMSOL & Create Model:**
 - Open COMSOL Multiphysics.
 - Select **Model Wizard**.
 - Choose **2D Axisymmetric** space dimension.
 - Select **Laminar Flow (spf)** physics (you will set the incompressible Navier-Stokes here).
 - Click **Done**.
2. **Geometry Construction:**
 - Create the ICA artery:**
 - The artery is a cylinder with diameter 5.57 mm → radius = 5.57/2 = 2.785 mm = 0.002785 m
 - Length = 53 mm = 0.053 m
 - In axisymmetric, geometry is in r-z plane:
 - Draw a rectangle with:
 - Width (radial) = 0.002785 m
 - Height (axial length) = 0.053 m
 - Position this rectangle with its bottom left corner at (r,z) = (0,0).
 - Create Carotid Sinus (CS) Ellipsoid:**
 - Ellipsoid major axis (axial) = 4.6 mm = 0.0046 m
 - Minor axis (radial) = 1.55 mm = 0.00155 m
 - Since 2D axisymmetric, an ellipse is drawn in r-z plane.
 - Position center of ellipse:
 - Axially: 19 mm from bottom (inlet) → z_center = 0.019 m
 - Axially: 38 mm from top (outlet) → This just confirms location; total height 0.053 m - 0.019 m = 0.034 m top to ellipse center. So 19 mm from bottom is fine.
 - Radially: on same axis → center at r=0.
 - Use the **Ellipse** geometry feature:
 - Center: (r=0, z=0.019)
 - Radii: r_radius = 0.00155 m, z_radius = 0.0023 m (half of 4.6 mm major axis)
 - Combine the ellipse with artery walls:**
 - Use **Boolean Union** to combine the artery rectangle and the ellipse shape, if you want the sinus as a bulge on the artery wall.
 - Alternatively, create the artery walls as boundary curves including the sinus bulge by using **Composite Curve** or spline.
3. **Finalize Geometry:**
 - Make sure the geometry is a single domain representing the fluid region inside ICA including the sinus bulge.
 - Build and finalize the geometry.

Part (b): Physics - Incompressible Navier-Stokes Setup

1. **Select the Physics Interface:**
 - Use **Laminar Flow (spf)**, which solves the incompressible Navier-Stokes by default.
 - Check the physics settings to ensure the equations are incompressible Navier-Stokes.
2. **Define Fluid Properties:**
 - Blood density (ρ) = 1080 kg/m³ → set under **Materials**
 - Dynamic viscosity (μ) = 0.00325 Pa·s → set under **Materials**
3. **Add material:**

- Create a new material in the **Materials** node.
- Set density = 1080 kg/m³
- Dynamic viscosity = 0.00325 Pa·s

Part (c): Boundary Conditions & Pulsatile Flow

1. Inlet Boundary (Bottom):

- Boundary at z = 0 (inlet).
- Boundary Condition: **Velocity Inlet**.
- Velocity vector is axial, pointing inward.
- Define velocity as time-dependent:

$$\mathbf{u} = -U_0 \cdot \mathbf{n}, \quad U_0 = u_{max} \cdot f(t)$$

- u_{max} umax initial = 0.5 m/s.
- $f(t)$ f(t) is piecewise:
 - $f(t) = 0.8\sin(2.857\pi t)$ f(t) = 0.8 sin(2.857πt) for $0 \leq t \leq 0.35$ $0 \leq t \leq 0.35$
 - $f(t) = 0.6\sin(2.22\pi(t - 0.35))$ f(t) = 0.6 sin(2.22π(t - 0.35)) for $0.35 < t \leq 0.8$ $0.35 < t \leq 0.8$
- Implement **time-dependent velocity** expression in COMSOL:
 - Define a **piecewise function** $f(t)$ f(t) in the **Definitions** node:
 - Use **Functions > Piecewise**:
 - Interval 1: [0, 0.35], expression: 0.8*sin(2.857*pi*t)
 - Interval 2: [0.35, 0.8], expression: 0.6*sin(2.22*pi*(t-0.35))
 - Velocity magnitude at inlet boundary = -u_max * f(t) along the axial direction.
 - Since flow is downward (if z increases upward), sign may be negative as per your problem.
 - Assign velocity as (0, -u_max*f(t)) if axis is (r,z) with z vertical.

2. Outlet Boundary (Top):

- Boundary at z = 0.053 m (top).
- Apply **Pressure Outlet** boundary condition.
- Set pressure = 0 Pa (gauge pressure).

3. Wall Boundary Conditions:

- For all artery walls (including sinus boundary), apply **No-Slip** boundary condition (velocity = 0).

Part (d): Study, Post-processing & Parametric Sweep

1. Time-dependent Study:

- Set study type to **Time Dependent**.
- Time range: from 0 to 0.8 s.
- Use appropriate time step control (e.g., max step = 0.01 s) for accuracy.

2. Post-processing:

- **Contour plots**:
 - Plot velocity magnitude $\sqrt{u_r^2 + u_z^2}$ ur2 + uz2 inside domain.
 - Plot pressure distribution.
 - Plot wall shear stress (WSS) on artery walls.
 - In COMSOL, WSS can be computed as the tangential component of shear stress vector on boundaries.
 - Use **Derived Values > Boundary Integration** or **Surface** plot for WSS.
- **Velocity profiles**:
 - At axial positions: 8 mm, 15 mm, 19 mm (CS center), 38 mm.
 - Create cut lines or boundary selections at those z positions.
 - Plot velocity magnitude or axial velocity across radius at those positions.

3. Parametric Sweep:

- Define **parameter** u_max with values: 0.25, 0.45, 0.65, 0.85 m/s.
- Set up parametric sweep on u_max in the **Study** node.
- Run simulations for each u_max.
- Extract and compare velocity, pressure, and WSS results for each case.

Additional Tips:

- **Mesh:**
 - Use finer mesh near walls and sinus bulge to resolve WSS accurately.
 - Use **Boundary Layer mesh** if available.
- **Solver settings:**
 - For pulsatile flow, use **Time-dependent solver** with small enough time step.
 - Use stabilization methods if needed (under Laminar Flow physics).
- **Units:**
 - Ensure all units consistent in meters, seconds, Pa, etc.

Summary of Key COMSOL Nodes to Use:

Task	COMSOL Node/Feature
Geometry	Rectangle + Ellipse + Boolean Union
Physics	Laminar Flow (spf)
Material Properties	New Material (density, viscosity)
Boundary Conditions	Velocity Inlet (with time-dependent function), Pressure Outlet, No-slip Walls
Functions	Piecewise function for $f(t)$ f(t)
Study	Time-dependent study
Mesh	Free triangular + Boundary layer
Postprocessing	Contour plots, Line plots at cross-sections, Boundary Integration for WSS
Parametric Sweep	Study > Parametric Sweep on u_max



If you want, I can also help you draft the exact COMSOL expressions for the velocity boundary condition and piecewise function or even help prepare a step-by-step COMSOL tutorial file for this model. Just ask!



Choose FilesNo file selected

Choose FilesNo file selected

ChatGPT can make mistakes. Check important info.