

# FLUID FLOW SIMULATIONS FOR BIOMEDICAL APPLICATIONS

THE UNIVERSITY OF TENNESSEE AT KNOXVILLE

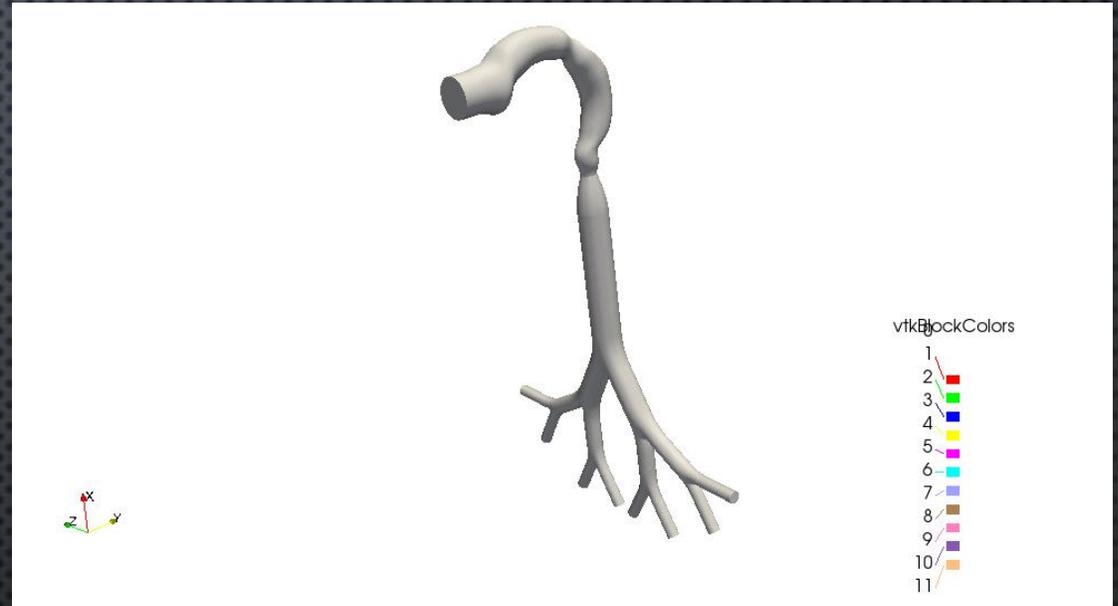
SAMUEL SCRUGGS

MENTOR: KWAI WONG



# ABSTRACT

- WHEN IT COMES TO SIMULATING FLUID FLOW, WHAT PROGRAM AND PROCESS WILL DELIVER THE MOST REALISTIC RESULT FOR COMMON GEOMETRIES? HOW CAN THESE BENCHMARK RESULTS BE APPLIED TO REAL LIFE SITUATIONS SUCH AS PARTICLE FLOW THROUGH HUMAN AIRWAYS?



- THROUGH COMPUTATIONAL FLUID DYNAMICS (CFD), FLUID FLOW IN VARIOUS GEOMETRIES CAN BE PREDICTED. MORE SPECIFICALLY, AIRFLOW THROUGH THE HUMAN AIRWAYS CAN BE SIMULATED WITH VARIOUS PROGRAMS TO HELP CALCULATE AND DISPLAY THE FLOW. BY ANALYZING THE RESULTS OF THESE CALCULATIONS, CERTAIN CONDITIONS APPLIED TO THE GEOMETRY CAN BE ALTERED IN ORDER TO ACHIEVE A MORE ACCURATE PREDICTION OF THE FLOW RESULT. WITH THE MULTIPLE PROGRAMS OUT THERE TO SIMULATE THIS FLOW, SOME PROGRAMS PRODUCE A BETTER RESULT THAN OTHERS FOR DIFFERENT SITUATIONS, BUT A LARGE PORTION OF THE RESULT ACCURACY INVOLVES HUMAN INPUT.

# PURPOSE

- THROUGH VARIOUS SIMULATIONS THAT CAN BE RUN WITH THESE PROGRAMS THE OVERALL RESULT THAT IS SOUGHT BY THIS PROJECT IS TO STUDY THE EFFECT OF PARTICLE FLOW THROUGH THE HUMAN AIRWAYS.
- ONE EXAMPLE OF SUCH PARTICLE FLOW IS THE FLOW THROUGH INHALATION.
- USING THESE SIMULATIONS CAN HELP TO SHOW THE FLOW OF THESE PARTICLES THROUGH THE AIRWAYS AND HOW THEY AFFECT THEIR SURROUNDINGS. ONE OF THESE EFFECTS, FOR EXAMPLE, BEING HOW VELOCITY AND PRESSURE DISTRIBUTES THE PARTICLE FLOW.

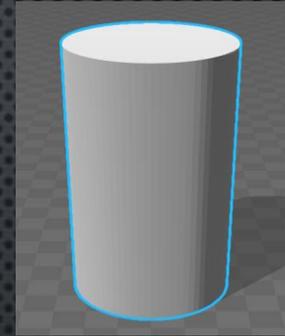
# PROCESS

- THE PROCESS TO RUN THESE SIMULATIONS CAN BE BROKEN DOWN INTO FOUR MAIN STEPS:
  - 1.) GEOMETRY CONSTRUCTION: ANY OBJECT CAN BE CONSTRUCTED IN ORDER TO HAVE IT RUN THROUGH THE PROCESS OF A FLUID SIMULATION.
  - 2.) PRE-PROCESSING/MESHING: THIS STEP INVOLVES BREAKING DOWN THE GEOMETRY INTO SMALLER SHAPES IN ORDER TO BE PROCESSED.
  - 3.) SOLVING: POINTS OF THE MESHED GEOMETRY CAN THEN BE USED WHEN IT COMES TO CALCULATING FLUID FLOW THROUGHOUT THE OBJECT USING NAVIER-STOKES EQUATIONS.
  - 4.) POST-PROCESSING: RESULTS CAN BE VIEWED AND ANALYZED IN THIS STEP. ACCURACY OF THE RESULTS CAN BE EXAMINED IN ORDER TO REFINE THE PROCESS FOR A MORE REALISTIC OUTCOME SHOULD THAT BE NECESSARY.

# GEOMETRY CONSTRUCTION

- WITH THE USE OF ANY PREFERRED SOFTWARE, GEOMETRIES CAN BE CONSTRUCTED IN ORDER TO BE MESHED FOR A FLUID FLOW SIMULATION.

- IN THIS CASE, A STRAIGHT, 3-DIMENSIONAL PIPE AND AN ELBOW PIPE WERE BOTH CONSTRUCTED (TEST CASES) IN ORDER TO SIMULATE FLUID FLOW IN COMMON GEOMETRIES.



- AFTER THIS TASK IS COMPLETED, MORE COMPLICATED GEOMETRIES, SUCH AS THE ONE USED IN THIS PROJECT, CAN BE CONSTRUCTED IN ORDER TO BE ANALYZED USING BENCHMARKS FROM THE PREVIOUS RESULTS.

# GMSH PROGRAM

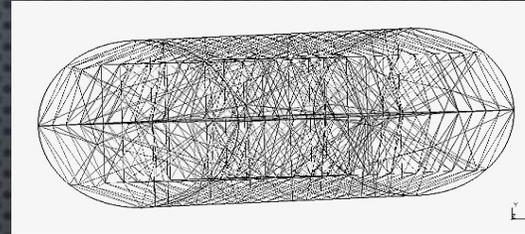
- DOWNLOADED FOR WINDOWS, GMSH BREAKS UP THE GEOMETRY INTO MULTIPLE OPTIONS SUCH AS TRIANGULAR AND TETRAHEDRAL MESHES
- THE PROGRAM IS FREE AND CAN BE ACQUIRED ONLINE FOR NOT ONLY WINDOWS BUT LINUX AND MAC OS X AS WELL
- IT IS DOWNLOADED FROM [GMSH.INFO](http://GMSH.INFO)
- CERTAIN FILES ARE ACCEPTED BY THE PROGRAM. WHEN IT COMES TO THE GEOMETRY, STL FILES AND INVENTOR FILES CAN BE USED, IN THIS CASE STL FILES WERE USED.
- GMSH PRODUCES A .MSH FILE THAT CAN BE FORMATTED AND READ BY OTHER PROGRAMS FOR PROCESSING OR VIEWING

# MESHING

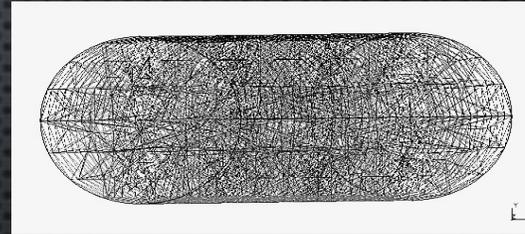
- IN ORDER TO GET THE MOST ACCURATE REPRESENTATION OF FLUID FLOW, THE MESH OF THE GEOMETRY IS IMPORTANT. A MESH WITH MORE POINTS ALLOWS FOR MORE DATA TO BE CALCULATED AND ANALYZED.

MESH (B) IS A BETTER MESH BECAUSE IT HAS MORE POINTS FOR CALCULATIONS. IT IS A MORE REFINED MESH.

A)



B)



- IN THE CASE OF A HOLLOW TUBE WITH WATER FLOWING THROUGH IT, A MORE REALISTIC RESULT WILL COME FROM MESH (B) DUE TO IT BEING MORE DETAILED.
- THE PROGRAM USED IN THIS CASE TO HELP CREATE THE MESH IS GMSH. MULTIPLE OTHER PROGRAMS SUCH AS CUBIT AND ANSYS FLUENT, MESH PROCESSORS, CAN BE USED TO HELP CREATE GEOMETRY MESHES. IN THE SPECIFIC EXAMPLE BEING USED FOR THIS PROJECT, HUMAN AIRWAYS, THE MESH WAS PRODUCED WITH FLUENT.

# BOUNDARY CONDITIONS

- CONDITIONS CAN BE ESTABLISHED FOR THE MESH BEFORE UTILIZING THE SOLVER. THIS HELPS TO SET UP WHAT TYPE OF OBJECT THE FLUID ACTUALLY ENCOUNTERS.
- FOR EXAMPLE: AN ORDINARY PIPE HAS AN INLET ESTABLISHED AT ONE END, WALLS THAT SURROUND THE OPEN AREA, AND AN OUTLET AT THE OPPOSITE END. ALONG WITH THE INSIDE BEING SET AS EMPTY, THESE BOUNDARY CONDITIONS ARE ESTABLISHED WITH THE MESH IN ORDER FOR THE SOLVER PROGRAM TO READ.
- SIMULATION THROUGH THE AIRWAYS IS A SIMILAR SET UP. ESSENTIALLY AIR IS CARRIED FROM THE MOUTH TO THE LUNGS THROUGH WHAT HAS BEEN MODELLED AS A PIPE. BOUNDARY CONDITIONS ARE THEN SET UP FOR THESE “PIPES” TO ESTABLISH WHERE THE PARTICLES ENTER, WHERE THEY TRAVEL TO, AND WHERE THEY ULTIMATELY EXIT.

# INITIAL CONDITIONS

- THESE CRITERIA CAN BE INPUT IN EITHER THE MESHING PROCESS OR IN THE SOLVER
- IN THIS CASE THEY WERE INPUT IN THE SOLVER AND INCLUDE INITIAL VELOCITY, PRESSURE, AND TIME THE SIMULATION WOULD RUN
- OPENFOAM IS THE SOLVER USED FOR THIS PROJECT. THROUGH THIS PROGRAM MANY INITIAL VARIABLES CAN BE SPECIFIED. TO NAME A FEW:
  - VELOCITY
  - KINEMATIC VISCOSITY
  - PRESSURE
  - CONSTANT/CHANGING VARIABLES

# OPENFOAM PROGRAM

- OPENFOAM IS THE SOLVER USED TO PERFORM DIFFERENT SIMULATIONS IN THIS PROJECT.
- THE PROGRAM IS AN OPEN SOURCE CODE THAT CAN BE ACQUIRED ONLINE AND RUN ON EITHER A LINUX OR WINDOWS SYSTEM.
- DOWNLOADED FOR A LINUX SYSTEM IN THIS CASE, OPENFOAM USES SPECIFIC COMMANDS IN ORDER TO CONVERT MESHES INTO A FORMAT THAT CAN BE SOLVED BY THE PROGRAM.
- AFTER THE SOLVING PROCESS, THE RESULT IS FORMATTED IN A WAY THAT CAN BE VIEWED IN A POST-PROCESSING PROGRAM.

# SOLVER

THE EQUATIONS USED TO SOLVE FLUID FLOW THROUGH GEOMETRIES ARE THE INCOMPRESSIBLE NAVIER-STOKES EQUATIONS.

$$\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + \rho g_x$$
$$\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + \rho g_y$$
$$\rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + \rho g_z$$

$$\text{Continuity: } \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0.$$

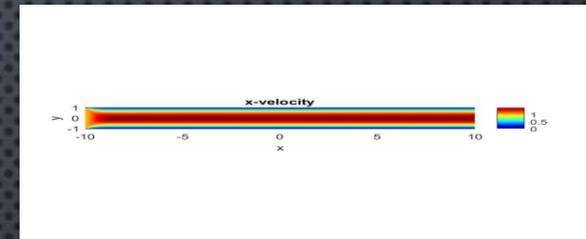
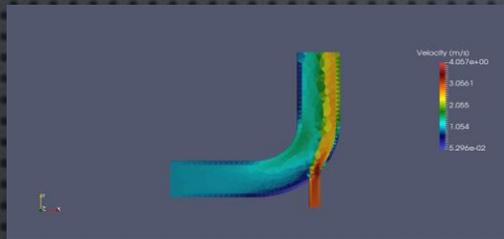
- THESE VARIOUS EQUATIONS ARE USED BY SOLVER PROGRAMS TO CALCULATE THE RESULT OF THE FLUID FLOW.
- THE MAIN CONCEPT OF THESE EQUATIONS INVOLVES CONSERVATION OF MASS, MOMENTUM, AND ENERGY. HOWEVER THE FLUID ENTERS THE GEOMETRY, THAT STATE MUST BE PRESERVED WHEN IT EXITS.
- FOR CERTAIN SITUATIONS, THERE ARE DIFFERENT INITIAL CONDITIONS THAT ARE MADE. VELOCITY OF THE FLOW CAN CHANGE, DENSITY OF THE FLUID, AND EVEN THE MATERIAL OF THE GEOMETRY ITSELF ARE JUST A FEW OF THE POSSIBLE VARIABLES THAT COULD CHANGE THE OUTCOME.
- THESE EQUATIONS WILL ALSO BE USED WHEN IT COMES TO SOLVING FLUID FLOW THROUGHOUT THE AIRWAYS.

# SOLVING PROCESS

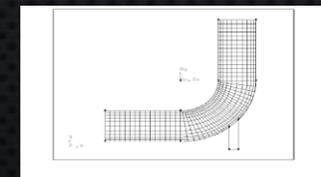
- THE PROCESS OF SOLVING THE SIMULATION INVOLVES A PROGRAM WRITTEN IN OPENFOAM CALLED ICOFOAM.
- THIS SOLVER USES THE INCOMPRESSIBLE NAVIER-STOKES EQUATIONS TO SIMULATE THE PARTICLE FLOW THROUGH THE GEOMETRIES.
- INITIAL AND BOUNDARY CONDITIONS ARE READ IN THE TRANSPORT PROPERTY AND VELOCITY, PRESSURE LIBRARIES BY ICOFOAM IN ORDER TO SOLVE THE FORMULAS FOR THE PROPER GEOMETRY.
- ONCE THE ICOFOAM COMMAND IS RUN, THE SOLVER WILL RUN THROUGH THE DESIGNATED AMOUNT OF STEPS FOR THE TIME FRAME THAT IT HAS BEEN GIVEN.
- THE SOLVER WRITES THE RESULTS TO A FILE, USUALLY A .VTK FILE, THAT IS THEN READABLE BY PARAVIEW.
- THE COMMAND PARAVIEW IS THEN RUN IN ORDER TO OPEN A WINDOW THAT ALLOWS THE USER TO VIEW THE RESULTS OF THE SIMULATION.

# POST-PROCESSING

- THERE ARE SEVERAL POST-PROCESSING UNITS USED TO VIEW THE RESULTS OF THE SOLVER AND IN THIS CASE THE PROGRAM PARAVIEW WAS USED.



- IN THE PHOTO ABOVE, THE RESULT OF WATER FLOWING THROUGH A PIPE IS SHOWN. THE EFFECTS OF FRICTION FROM THE WALLS AND THE SPEED OF THE WATER AT THE BEGINNING IS SHOWN THROUGH THE DIFFERENT COLORS. BOTH AN ELBOW PIPE AND STRAIGHT PIPE ARE SHOWN.
- THE SAME SITUATION CAN BE MADE WHEN SIMULATING PARTICLE FLOW THROUGH THE AIRWAYS. TAKING FROM THE BENCHMARK RESULTS OF FLUID FLOW, THIS EXPERIENCE CAN BE APPLIED TO THE RESULTS FROM FUTURE TESTS.
- THROUGH THIS KNOWLEDGE INACCURATE RESULTS WILL BE RECALCULATED WITH MORE BETTER CONDITIONS IN ORDER TO CREATE A MORE REASONABLE OUTCOME.



# SUMMARY OF HOW TO

- THE GEOMETRY'S FILE MUST BE FORMATTED PROPERLY IN ORDER FOR EACH PROGRAM TO READ IT AND PERFORM ITS TASK:
  - 1) GEOMETRY AND MESH CREATED USING THE PROGRAMS SUCH AS GMSH OR ANSYS FLUENT. BOUNDARY CONDITIONS ARE DEFINED IN THE MESH
  - 2) USING SPECIFIC COMMANDS, THE MESH FILE CAN BE CONVERTED FROM THE MESHING PROGRAM TO OPENFOAM
  - 3) OPENFOAM CAN THEN BE USED TO PERFORM SPECIFIC CALCULATIONS FOR THE FLUID FLOW WITH DIFFERENT SPECIFIED INITIAL CONDITIONS
  - 4) THE RESULT OF THESE CALCULATIONS ARE THEN READ IN PARAVIEW TO BE ANALYZED
  - 5) THROUGH BENCHMARK TESTS AND PREVIOUS KNOWLEDGE OF FLUID FLOW, THE REASONABILITY OF THE RESULTS CAN BE DETERMINED BY VIEWING THE OUTCOME IN PARAVIEW
  - 6) MISTAKES ARE THEN CORRECTED BY REFINING THE MESH, BOUNDARY CONDITIONS, AND INITIAL CONDITIONS AND THEN RUNNING THE GEOMETRY BACK THROUGH THE DIFFERENT PROGRAMS

# FUTURE RESEARCH

- THE RESEARCH TO CONTINUE INVOLVES ANALYZING PARTICLE FLOW FROM THROUGH THE AIRWAYS.
- THIS PARTICLE FLOW CAN BE ANALYZED USING THESE SAME PROGRAMS IN ORDER TO SIMULATE FLUID FLOW THROUGH THE BODY, AND THE EFFECTS OF THE INJECTION OF THESE PARTICLES CAN ALSO BE ANALYZED.
- IN THE SAME WAY THAT FLUID FLOW IS ANALYZED THROUGH A TUBE, THESE METHODS CAN BE USED TO HELP DETERMINE THE FLOW OF THESE PARTICLES IN THE BODY.
- THERE CAN ALSO BE SIMULATIONS PERFORMED TO HELP SHOW WHERE HEAT TRANSFERS THROUGHOUT THE AIRWAYS. THIS CAN BE PAIRED WITH FLUID FLOW SIMULATIONS TO DETERMINE WHERE THE MOST AFFECTED AREAS ARE BY THESE CERTAIN PARTICLES.

# REFERENCES & ACKNOWLEDGEMENTS

- [WWW.SOURCEFORGE.NET](http://WWW.SOURCEFORGE.NET)
- [USERS.UGENT.BE/~MVBELLEG/FLUG-12-0.PDF](http://USERS.UGENT.BE/~MVBELLEG/FLUG-12-0.PDF)
- [HTTPS://CUBIT.SANDIA.GOV](https://CUBIT.SANDIA.GOV)
- [WWW.OPENFOAM.ORG](http://WWW.OPENFOAM.ORG)
- [WWW.PARAVIEW.ORG](http://WWW.PARAVIEW.ORG)
- [GMSH.INFO](http://GMSH.INFO)

THIS PROJECT IS SUPPORTED BY THE NATIONAL SCIENCE FOUNDATION