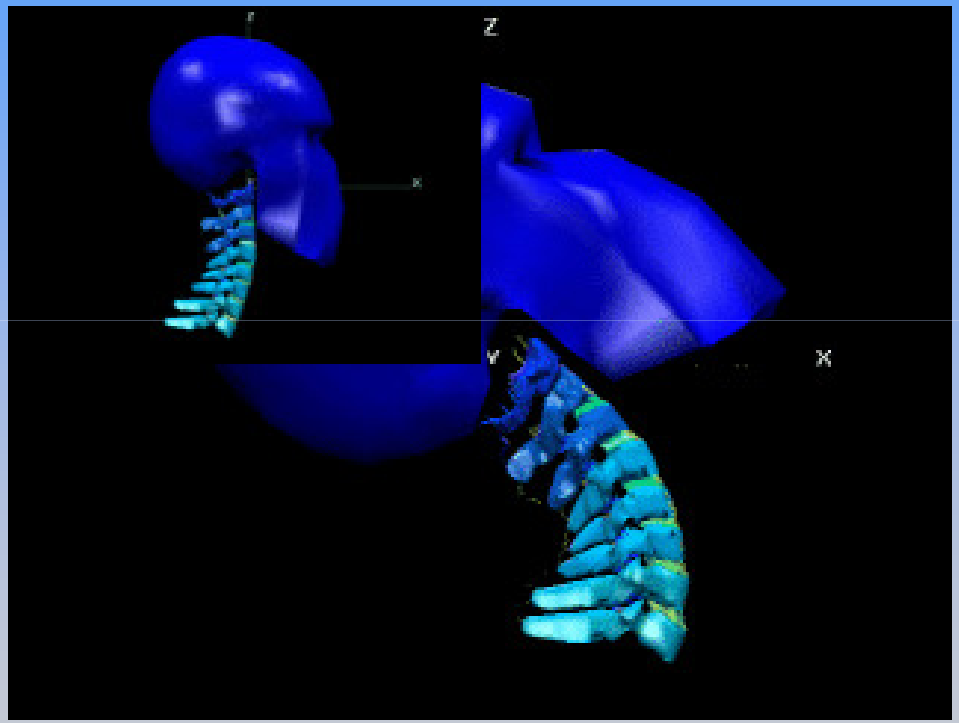
Application of OpenFOAM for Whiplash-like Motion Modeling

-- A Pilot Study

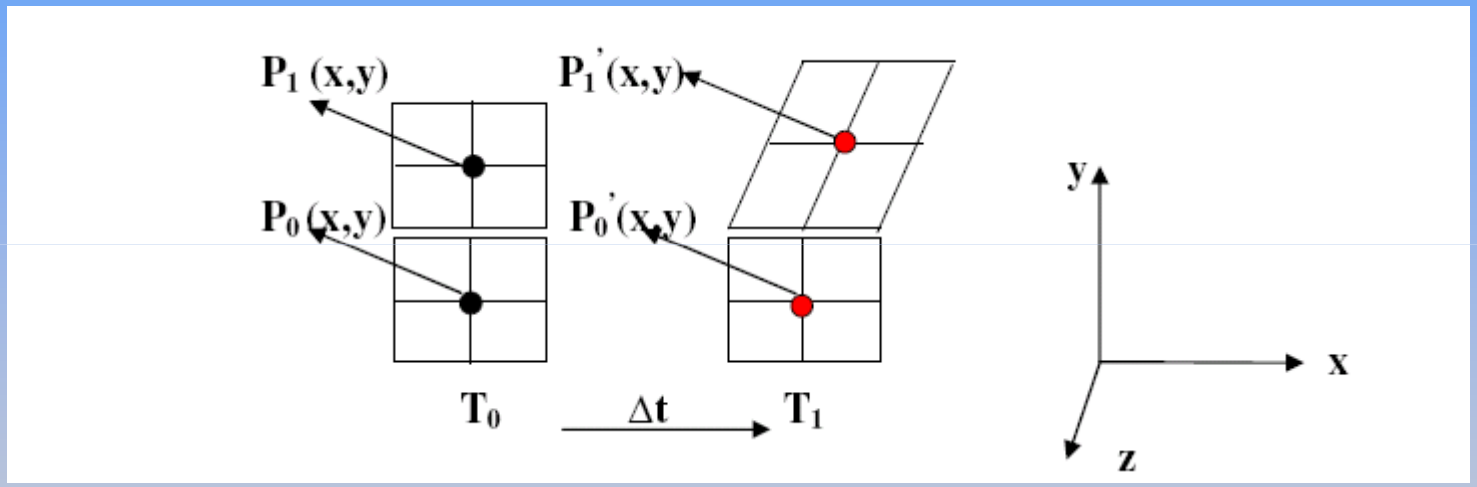
Student: Junfeng Yang
Teacher: Håkan Nilsson

Chalmers University of Technology
Gothenburg, Sweden

Theoretical Background and motivation



Methodology



OpenFOAM Application

- OpenFOAM-1.5-dev
- **icoDyMFoam application**

Copy icoDyMFoam directory into own computer

1: cp –r $FOAM_TUTORIAL/icoDyMFoam/movingConeTopo/ $FOAM_RUN

2: cd $FOAM_RUN/movingConeTopo

3: blockMesh . .

4: icoDyMFoam . .

The case will run. The paraFoam could check the mesh motion.

Modify the directory into whiplash application

**1: cp –r $FOAM_SRC/dynamicFvMesh/dynamicFvMesh \
$WM_PROJECT_USER_DIR/dynamicWhiplashFvMesh**

The library was copied and the names were changed to the dynamicWhiplashFvMesh. Copied were also the Make folder containing the files : files and options

Modify the directory into whiplash application

**1: cp –r $FOAM_SRC/topoChangerFvMesh/Make/ \
$WM_PROJECT_USER_DIR/dynamicWhiplashFvMesh**

The files were rewritten into the following line order to only compile the dynamicWhiplashFvMeshlibrary.

Modify the directory into whiplash application

1: dynamicWhiplashFvMesh.C
2: LIB = $(FOAM_USER_LIBBIN)/ libmyDynamicFvMesh

In the options file, it should contain following line:

```
1: -I$(LIB_SRC)/dynamicFvMesh/lnInclude \
```

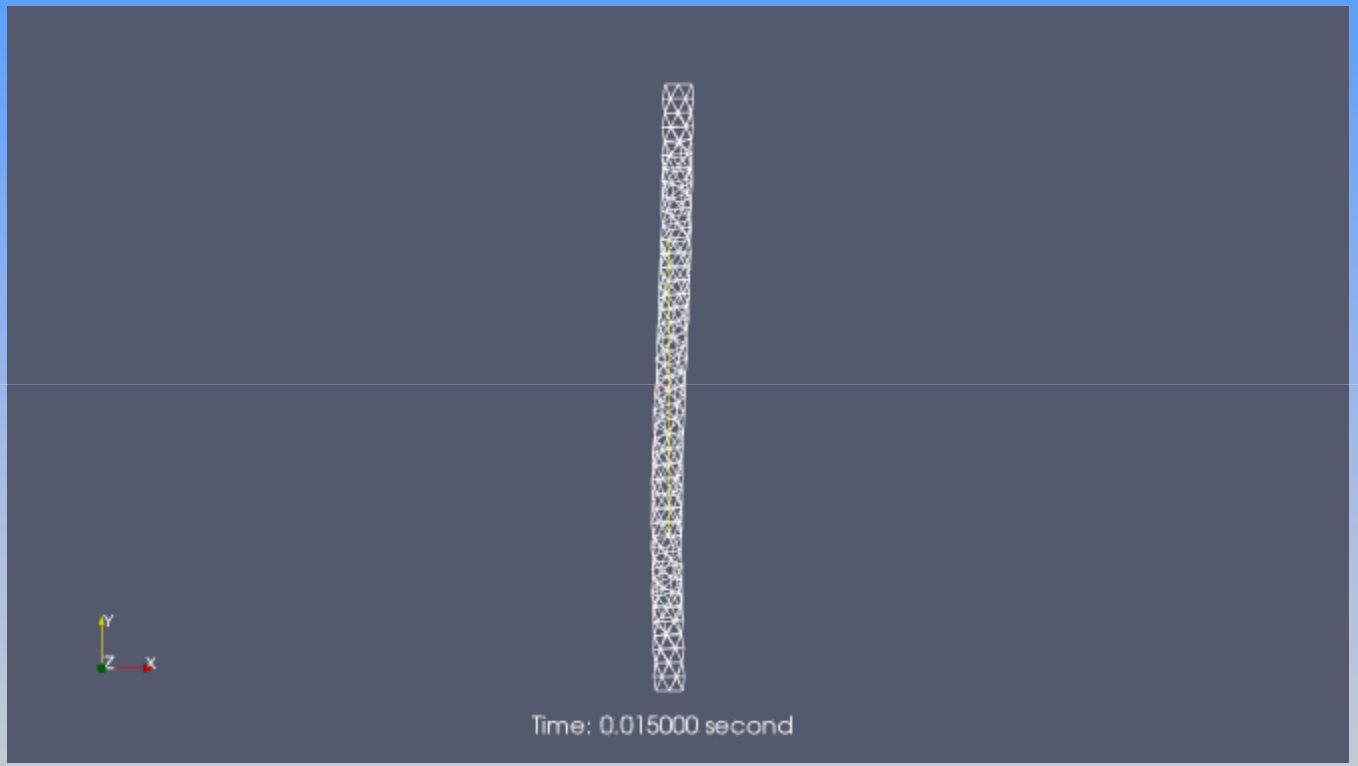
If it does not, please add the above line into the options file like :

```
1: EXE_INC = \
2:     -I$(LIB_SRC)/dynamicFvMesh/lnInclude
```

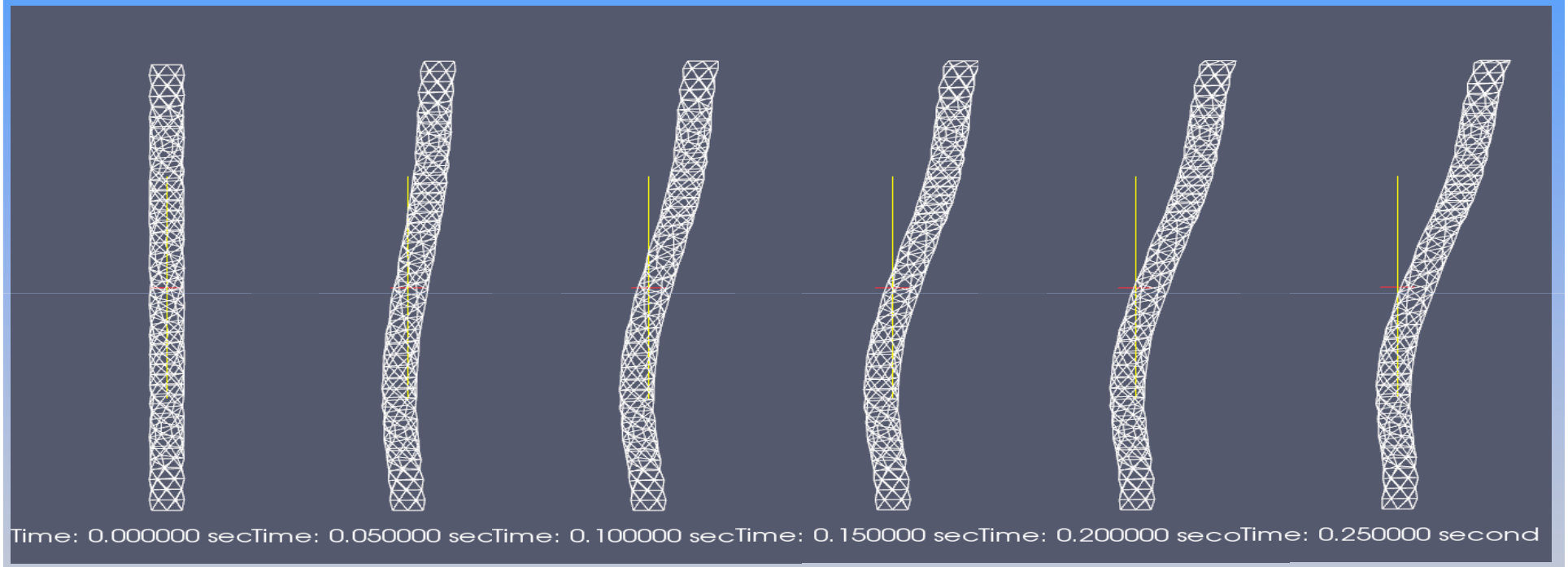
The copied dynamicFvmesh can now be adopted for the new functionality. To compile the changes made in the library, the command:

```
1: wmake libso
```

Results



Results


Time: 0.000000 secTime: 0.050000 secTime: 0.100000 secTime: 0.150000 secTime: 0.200000 secoTime: 0.250000 second

Future Work

- Compressible fluid or flexible wall
- Adaptive grid technology
- Weak-contrain pipe
- Fluid properties, blood
- Boundary conditions: inlet and outlet

Suggestion & Question!