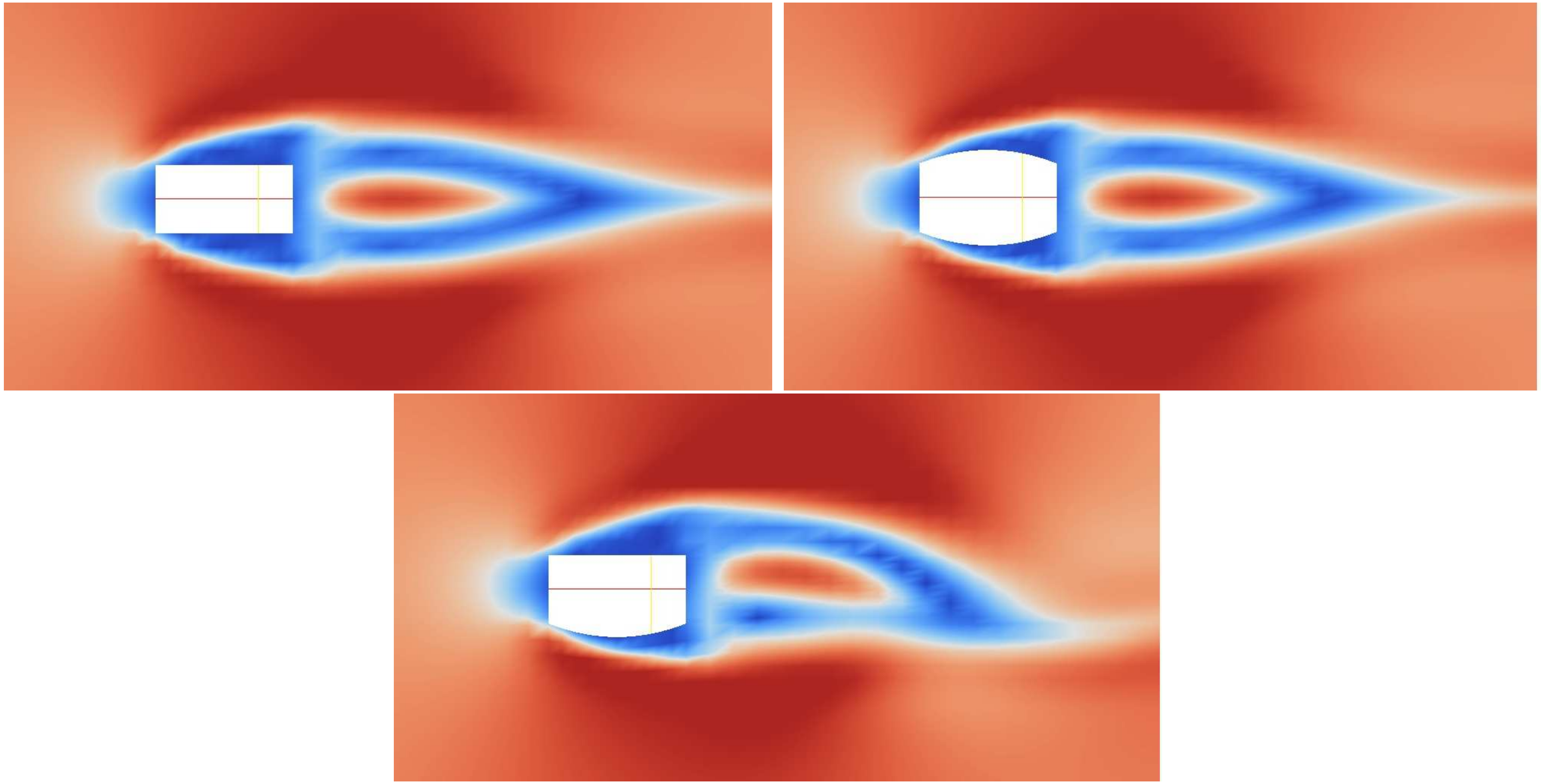
Tutorial: Point-wise deformation of mesh patches



Point-wise deformation of mesh patches

Why?

- Easily change shape of an object.
- Can be used for shape optimization.
- Active flow control with icoDyMFOAM solver.

What will be shown in this tutorial:

- How to create a library that enables point-wise deformation.
- Control the shape of a patch with a function.
- Implement periodic patch shape deformation.
- Use point-wise deformation on a squared cylinder.

Extract the library and case file:

- `tar -xvf EH_PatchDeform.tar.gz`

Creating the library (1/2)

- Start by finding something that is close to what you want.
  `$FOAM_SRC/fvMotionSolver/pointPatchFields/derived/angularOscill.../`
  The angunarOscillatingVelocity rotates patches around a defined axis.

- Copy it to your run directory
  `cp -r $FOAM_SRC/fvMotionSolver/pointPatchFields/derived/angularOscill...`
  `$FOAM_RUN/`

- Change filename, class name and function names:

```
sed -e 's/angularOscillating/libMyPolynom/g' \
libMyPolynomVelocityPointPatchVectorField.C > tmp.C
mv tmp.C libMyPolynomVelocityPointPatchVectorField.C
sed -e 's/angularOscillating/libMyPolynom/g' \
libMyPolynomVelocityPointPatchVectorField.H > tmp.H
mv tmp.H libMyPolynomVelocityPointPatchVectorField.H
```

- Edit `Make/file` to only include `libMyPolynomVelocityPointPatchVectorField.C` and change the name of the library to
  `LIB = $(FOAM_USER_LIBBIN)/libfvMotionSolvers`

Creating the library (2/2)

- Edit `Make/options` and add `-I$FOAM_SRC/fvMotionSolver/lnInclude` to `EXE\_INC`
- Quick look at the input variables that control the deformation. They are defined in the first time step in the case folder.
  `vi 0/pointMotionU`

Now on to the library: `cd ../libMyPolynomVelocity`

- The input variables are initialized in libMyPolynomVelocityPointPatchVectorField.H
  `vi libMyPolynomVelocityPointPatchVectorField.H`
- The deformation is calculated and set in libMyPolynomVelocityPointPatchVectorField.C
  `vi libMyPolynomVelocityPointPatchVectorField.C`
- After looking at the `constructor` and the `updateCoeffs()` function
  Close and compile the library
  `wmake libso`

The case

Dimensions of the cylinder are length 20 cm, width 30 cm and height 10 cm.

- Lets start by running the case, it takes a few minutes and in the meantime look at the case files
  `icoDyMFoam > log &`

- To be able to use solvers that move the mesh we need a `dinamicMeshDict` in the constant folder. Controls mesh manipulation.
  `vi constant/dynamicMeshDict`

- OpenFOAM needs to be told about the new library, that is done by adding
  `libs ("libMyPolynomVelocity.so");` at the bottom of `system/controlDict`

- Lets now look again at the `0/pointMotionU`. The constants set in CubeY and CubeYMinus describe the shape of the plane that the patch will be deformed to.

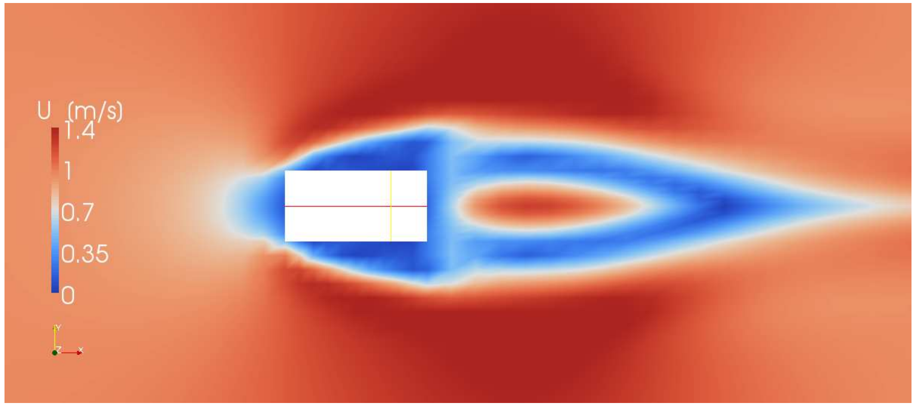
  $$z = \mathrm{X2} \cdot x^2 + \mathrm{X1} \cdot x + \mathrm{Y2} \cdot y^2 + \mathrm{Y1} \cdot y + \mathrm{Cconst}$$

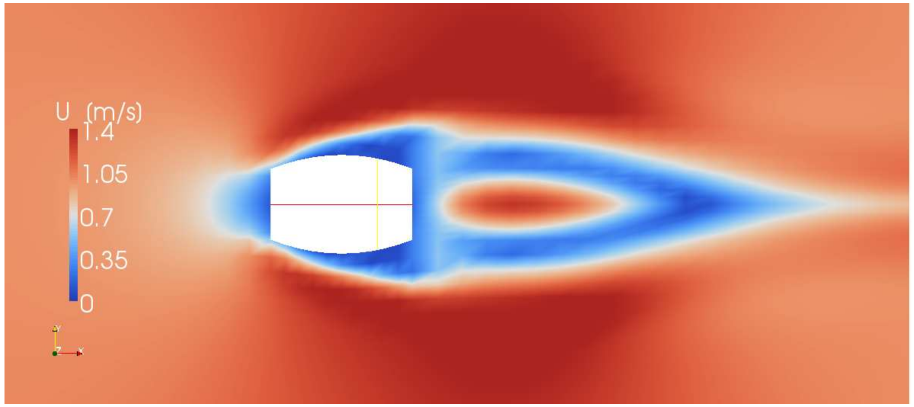
- The deformation takes 0.2 seconds in both cases but for the CubeY it is periodic and returns to origin in another 0.2 sec.

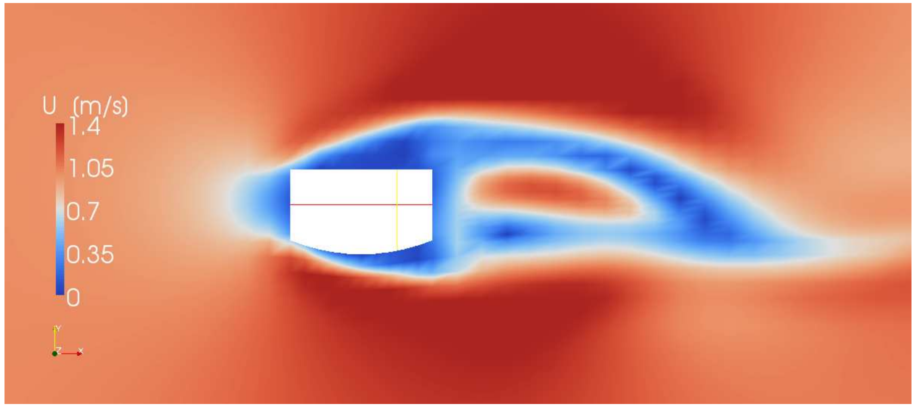
- Run paraFoam and check the results.

Results

Flow after 2.0 sec:







Thank you!

Questions?