Community Christmas Competition V

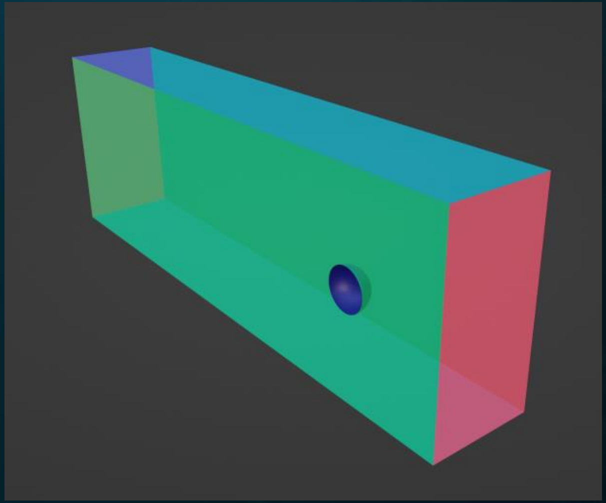
Entry by Matti Hoivala

Hosted By József Nagy

Problem

Solve the DragCoefficient of a sphere as a function of Reynolds number using OpenFoam and compare results to empirical study by NASA.

Sphere should have a diameter of 1 meter. Rest is up to the analyst.



Information

Empirical study contains test data from R_e: 10^{-2} to R_e: 10^{7}.
Such flow range contains flow domain where flow can be laminar, transitional or turbulent. Laminar or turbulent vortex shedding can also be expected.

Strategy

SimpleFoam, a steady-state solver is used due to limited computing resources. It is acknowledged that the flow will have time dependent structures which will be unresolved. Convergence problems are also expected since there should not be a steady-state solution for all Reynolds number flows.

First simulate flow with $R_e = 1$, the use the results to initialize the field for $R_e = 10$. Continue until $R_e = 1\ 000\ 000$ is reached.

Reynolds Number	Flow Type, Solver, Turbulence modeling
1	Laminar, incompressible SimpleFoam (Laminar)
10	Laminar, incompressible SimpleFoam (Laminar)
100	Laminar, incompressible SimpleFoam (Laminar)
1000	Laminar, incompressible SimpleFoam (Laminar)
10 000	Turbulent, incompressible SimpleFoam (kOmegaSST)
100 000	Turbulent, incompressible SimpleFoam (kOmegaSST)
1 000 000	Turbulent, incompressible SimpleFoam (kOmegaSST)

Computer, Software

Computer:
Lenovo Thinkpad T540p
CPU: intel i7-4700MQ @ 2.4GHz, 4 Core
GPU: NVIDIA GeForce GT 730M
Memory: 15.5 GiB
Operating System: Linux Mint 20.3 Cinnamon

Software
CFD: OpenFoam 7
Geometry: Blender 2.8
Mesh: swiftBlock & cfMesh
Residual & Drag monitoring: foamMonitor
postProcessing: ParaView 5.6.0 & Matlab R2021a

Geometry, Boundaries, Mesh, Initialization

Boundaries:

Inlet:	velocity
Outlet:	pressure
Sphere:	noSlip
Symmetry:	symmetry
Sides:	slip

Domain:
Symmetrical: Only one half is modelled.
Size: Depends on Reynolds number.
I increased the domain size as the Reynolds number grew.

Mesh:
For the first few simulations Blender plugin swiftBlock was used. Later on switched to cfMesh.
After each simulation I check that y+ value was ≤ 1.
Cartesian-, Tet- or Polyhedral mesh was used depending on which gave the best quality. Number of boundary layers Varied between 3-18

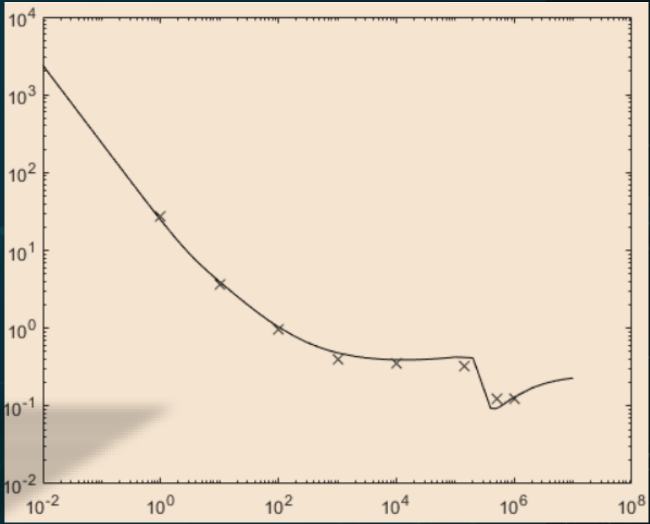
Initialization:
Each simulation was initialized with the flow field from previous simulation using mapFields.
PotentialFoam was used to Initialize the first simulation and in some rare cases where mapFields failed.

Results

Computed results followed empirical data suprisingly well. Results do now follow the strategy perfectly*. This is because I could not find a steady-state solution for Reynolds number 100 000, so I decided to try and increase the reynolds number. A steady solution was found at Reynolds number 140 000.

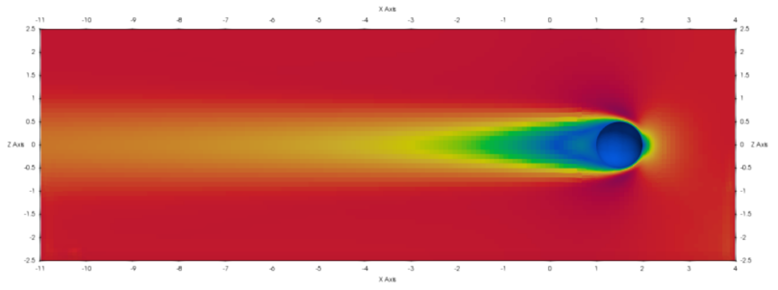
Additions to this an extra computing point was added to Reynolds number 500 000. This was merely to increase resolution near large gradient in the DragCoefficient graph.

Reynolds Number	DragCoefficient
1	26.911
10	3.699
100	0.967
1 000	0.399
10 000	0.357
140 000*	0.322
500 000*	0.122
1 000 000	0.125

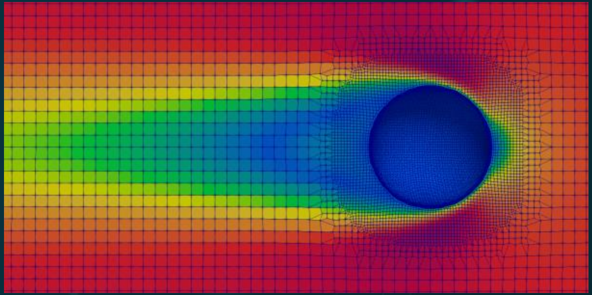


Results for Reynolds = 100

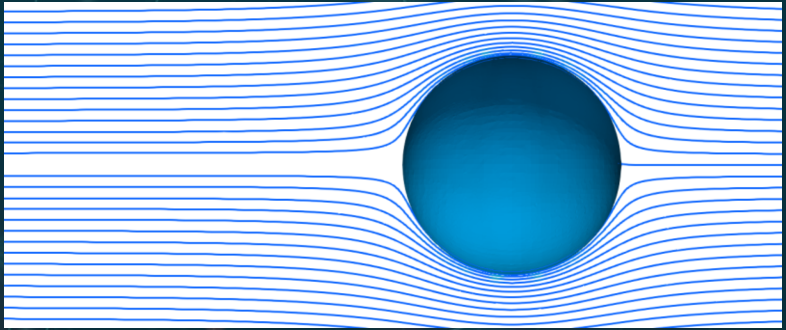
Velocity Contour



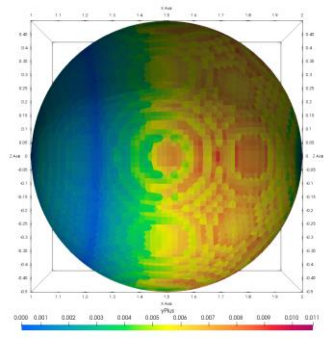
cartesianMesh



Streamlines

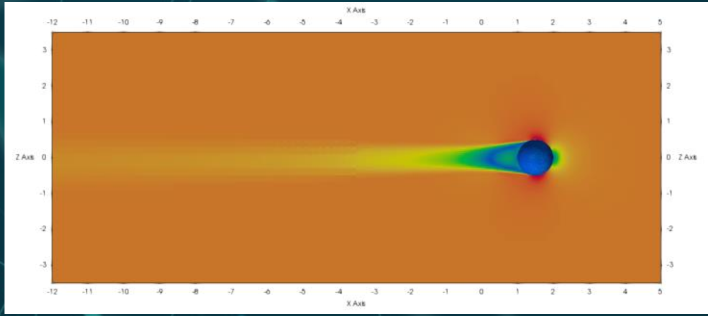


Y+ Contour
(0.000 – 0.011)

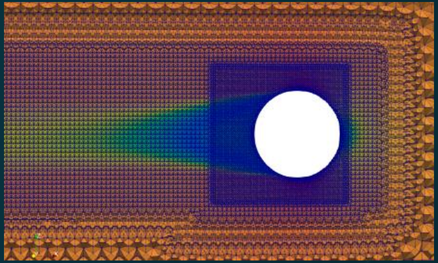


Results for Reynolds = 500 000

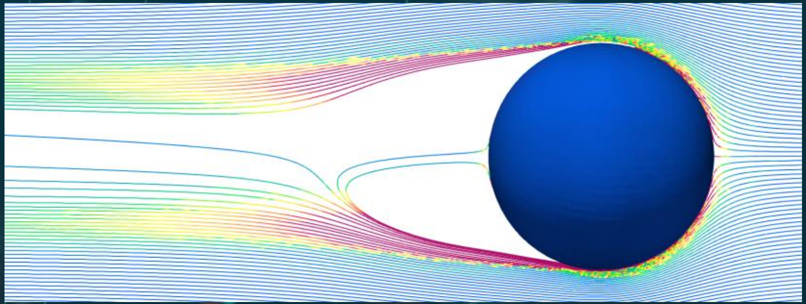
Velocity Contour



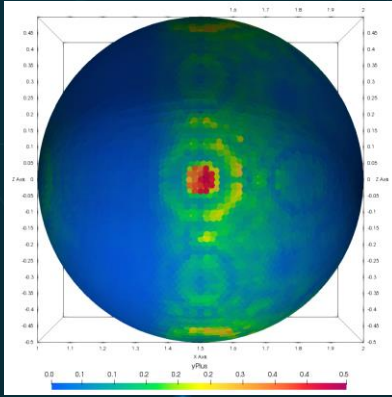
pMesh (polyhedral)



Streamlines



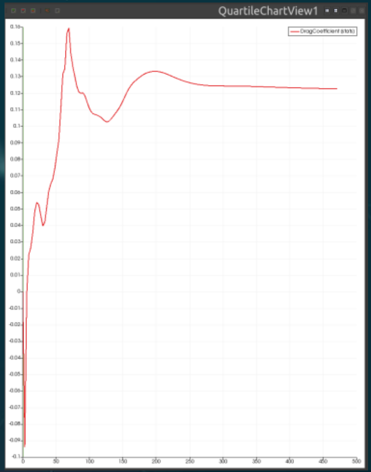
Y+ Contour
(0.0 – 0.5)



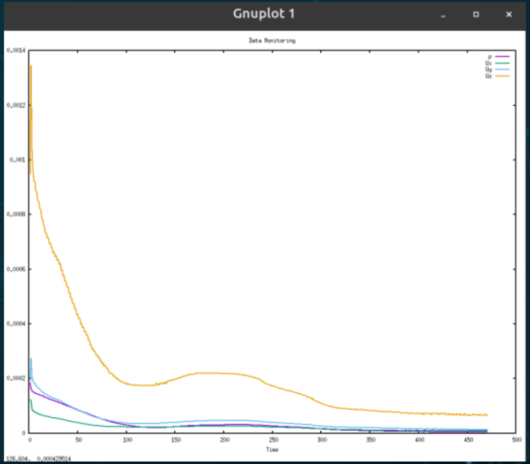
The boring stuff

To validate convergence residuals and DragCoefficient were plotted during the runs using foamMonitor. DragCoefficient was also plotted using paraview with a simple script. The result from paraview was then used as the final numerical result.

DragCoefficient monitor:



Residual monitor:



Reynolds number:

$$R_e = \frac{\rho u L}{\mu}$$

DragCoefficient:

$$C_d = \frac{2F}{\rho A u^2}$$

Thank you!

Matti Hoivala
matti.hoivala@hotmail.com
https://www.linkedin.com/in/matti-hoivala/
Application Engineer
Master's student at Tampere University