

OpenFOAM Textbook Companion
Fluid Mechanics-Yunus Cengel and John Cimbala
Example 10.9, Page No: 535

OpenFOAM version 3.0.1

Code was compiled on Ubuntu 14.04 and executed on Mac OSX.

MESH DETAILS

Mesh type : Structured Mesh using 'blockMesh' command.

Mesh stats	
points	26240
faces	51520
internal faces	25280
Cells	12800
Faces per cell	6
Boundary patches	5

The command “checkMesh” can be used to check for the mesh/grid quality and to ascertain whether there are any negative volumes present.

SOLVER DETAILS

Solver type : simpleFoam solver

RAS simulation SpalartAllmaras model

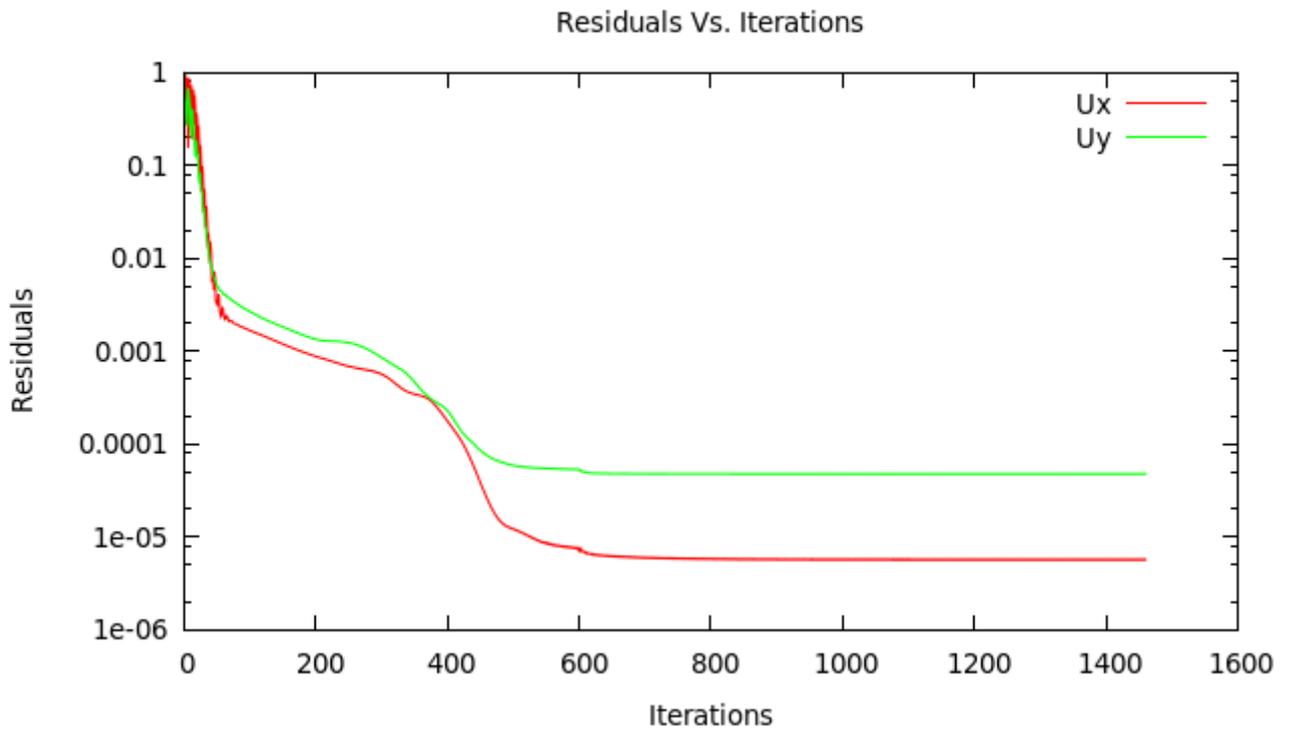
Material : Water ($\rho=999.9\text{ kg/m}^3$ and $\nu=1.519\cdot 10^{-6}$)

PROCESSING

Solution simulated for 1000 iterations

RESULTS

Residual Plot



Velocity Contour Plot

