

**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

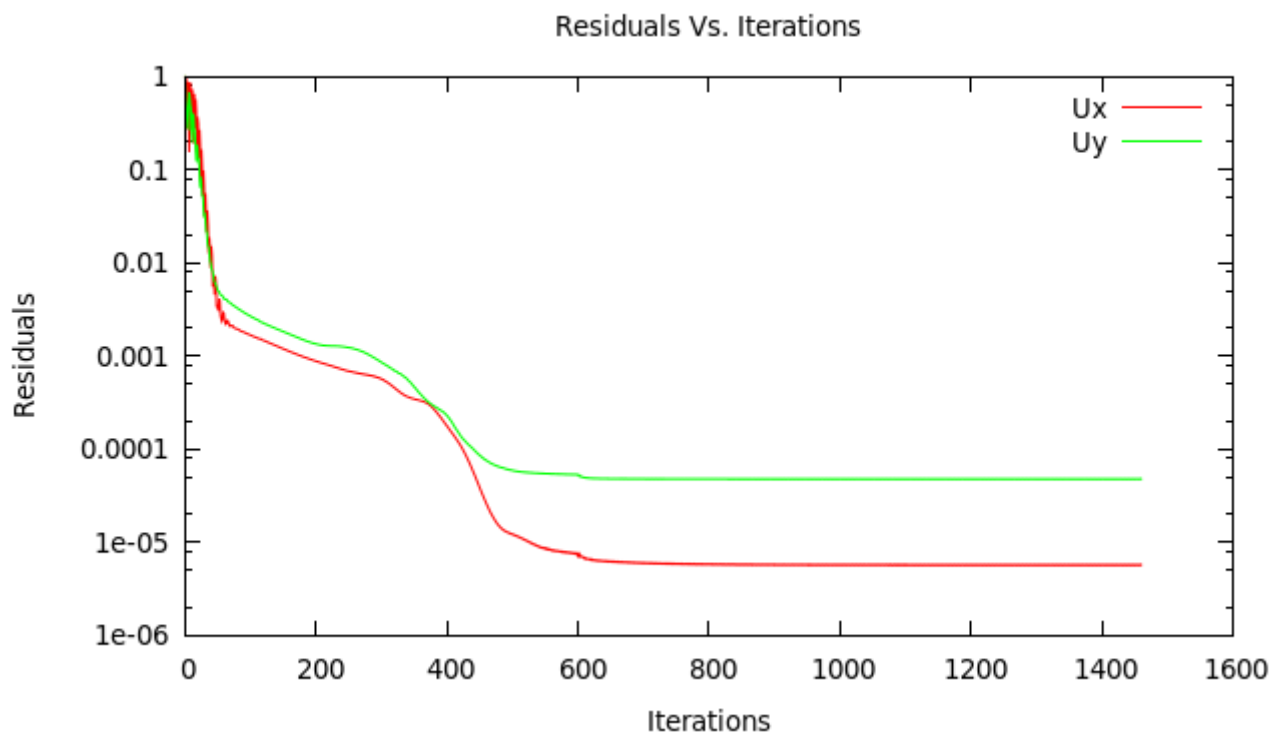
Material : Water (  $\rho=999.9\text{ kg/m}^3$  and  $\nu=1.519\cdot 10^{-6}$  )

**PROCESSING**

Solution simulated for 1000 iterations

## RESULTS

Residual Plot



Velocity Contour Plot

