Presented at the COMSOL Conference 2009 Boston

Benchmarking COMSOL Multiphysics 3.5a – CFD problems

Darrell W. Pepper

Xiuling Wang*

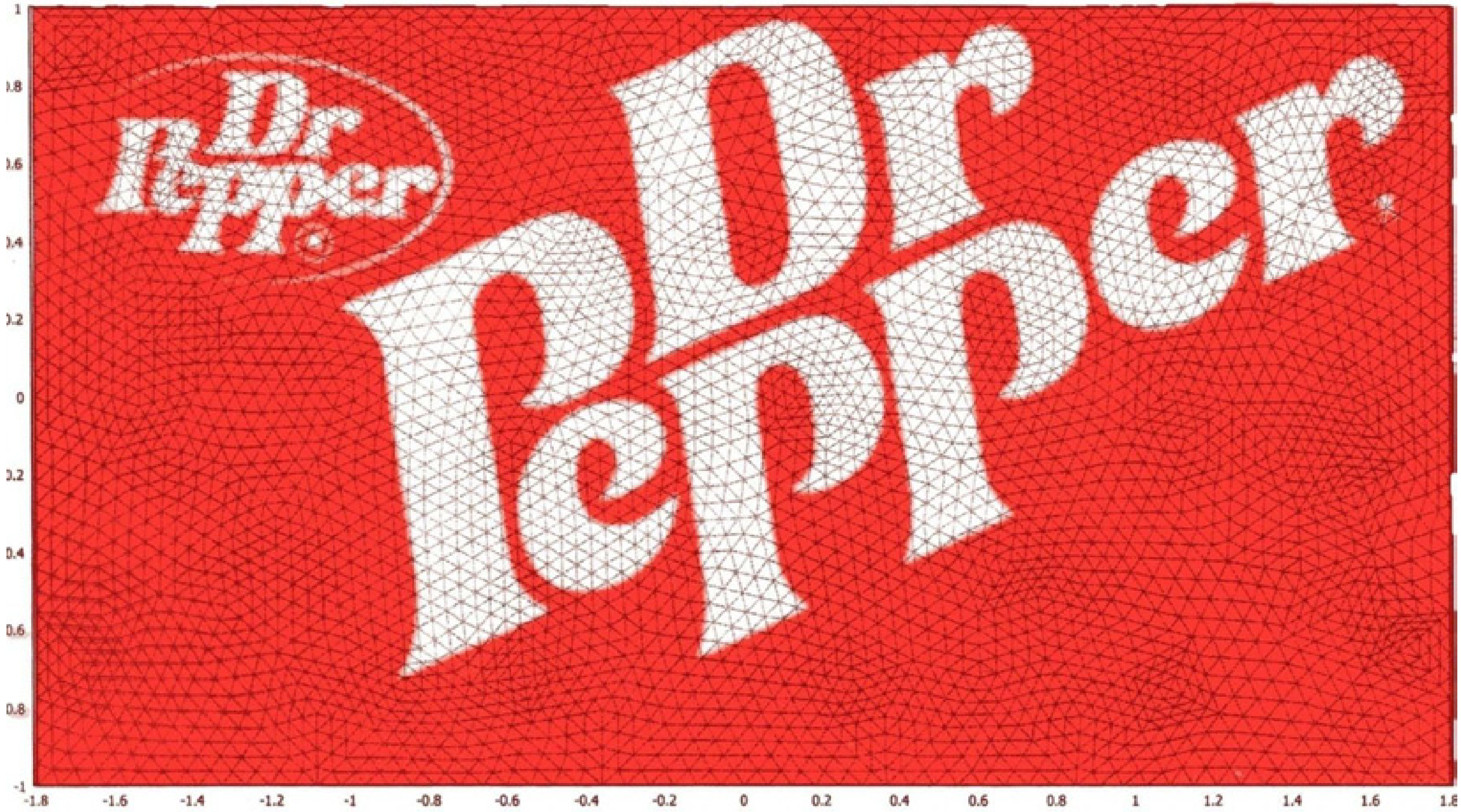
Nevada **C**enter for **A**dvanced **C**omputational **M**ethods
University of Nevada Las Vegas

*Purdue University - Calumet

Boston MA, Oct. 8-10, 2009

Xiuling Wang, PhD

Testing the mesh generator



Verification vs validation

- Verification = solving the eqns right
- Validation = solving the right eqns
- Benchmarking = validating the verification

Outline

- Introduction
- Benchmark environment and criteria
- Simulation results
  1. Flow over a 2-D circular cylinder
  2. Compressible flow in a shock tube
  3. Incompressible heated laminar flow and non-heated turbulent flow over a 2-D backward facing step
  4. 3D natural convection within an air-filled articulated cubical enclosure
- Conclusions

Introduction

- Objective
  - Compare results obtained from COMSOL Multiphysics 3.4 with those obtained from COMSOL Multiphysics 3.5a for four multiphysics problems
  - Test four CFD and CHT problems using COMSOL Multiphysics 3.5a
  - Obtain the CPU times and memory costs for solving those problems
- New features for COMSOL 3.5a – segregated solver; 32 – 64 bit; memory saving 50%

Benchmark environment and criteria

Hardware:

- Platform 1: Pentium(R) D CPU 2.80GHz, 4.0GB this configuration was used to test the first four benchmark problems.
- Platform 2: Intel ® Core ™ 2 Quad CPU Q9300 CPU 2.50GHz, 4.0GB RAM. This configuration was used for the four CFD-CHT benchmark problems.

Benchmark environment and criteria

- **Operating system**: for the first hardware platform, the operating system was 32 bit and running Windows XP; for the second hardware platform, the operating system was 64 bit running Windows Vista.

Benchmark environment and criteria

- Benchmark criteria
  - Computational accuracy (comparison difference is less than or equal to 5%)
    - Contours of key variables
    - Extreme values
    - Experimental data
  - Mesh independent study
    - Comparisons are made for results obtained for different mesh densities for a selected test problem
    - Increase in the number of elements leads to negligible differences in the solutions.

Benchmark environment and criteria-cont.

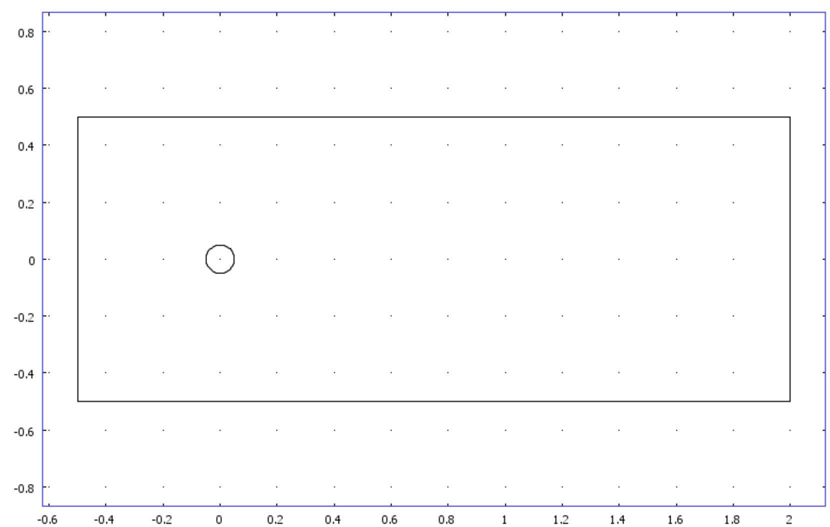
- Benchmark criteria
  - Memory
    - Provided by software package whenever possible
    - COMSOL "Mem Usage" shows the approximate memory consumption, the average memory during the entire solution procedure
  - CPU time
    - Execution times can be recorded from immediate access to the CPU time by the program or from measuring wall-clock time
    - To obtain accurate CPU time, all unnecessary processes were stopped

Comparison between 3.4 and 3.5a

Benchmark case	Software Used	Number of elements	Memory cost (MB)	CPU time (s)	Compared values
Case 1: FSI	COMSOL Multiphysics 3.4	3,407	245	1,537	$Tot_{dis-max}$:25.43µm
		6,602	267	3,342	$Tot_{dis-max}$:25.72µm
		9,728	308	5,301	$Tot_{dis-max}$:25.50µm
		14,265	349	8,475	$Tot_{dis-max}$:26.04µm
	COMSOL Multiphysics 3.5	3,372	264	240	$Tot_{dis-max}$:21.97µm
		6,221	290	522	$Tot_{dis-max}$:23.99µm
		9,918	295	719	$Tot_{dis-max}$:23.72µm
		20,545	320	2426	$Tot_{dis-max}$:25.14µm
Case 2: Actuator	COMSOL Multiphysics 3.4	5,032	220	5	$X_{dis-max}$=3.065µm
		9,635	312	11	$X_{dis-max}$=3.069µm
		15,774	520	22	$X_{dis-max}$=3.066µm
	COMSOL Multiphysics 3.5	5,032	170	3	$X_{dis-max}$=3.065µm
		10,779	360	8	$X_{dis-max}$=3.067µm
		16,893	480	22	$X_{dis-max}$=3.066µm
Case 3: Circulator	COMSOL Multiphysics 3.4	9,067	173	127	reflection, isolation and insertion loss
		19,398	376	361	
	COMSOL Multiphysics 3.5	14,089	280	103	
Case 4: Generator	COMSOL Multiphysics 3.4	38,440	303	78	B_{max}=1.225T
	COMSOL Multiphysics 3.5	32,395	190	17	B_{max}=1.257T

CFD-CHT problem 1 - Flow around a circular cylinder

- The flow around a circular cylinder has been examined over many years and is a popular CFD demonstration problem.
  - At very low Reynolds numbers, the flow is steady.
  - As the Reynolds number is increased, asymmetries and time-dependent oscillation develops in the wake region, resulting in the well-known Karman vortex street.

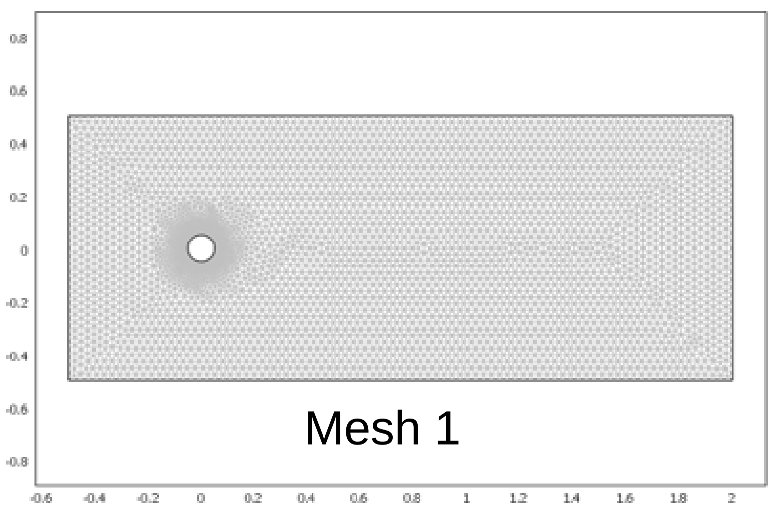


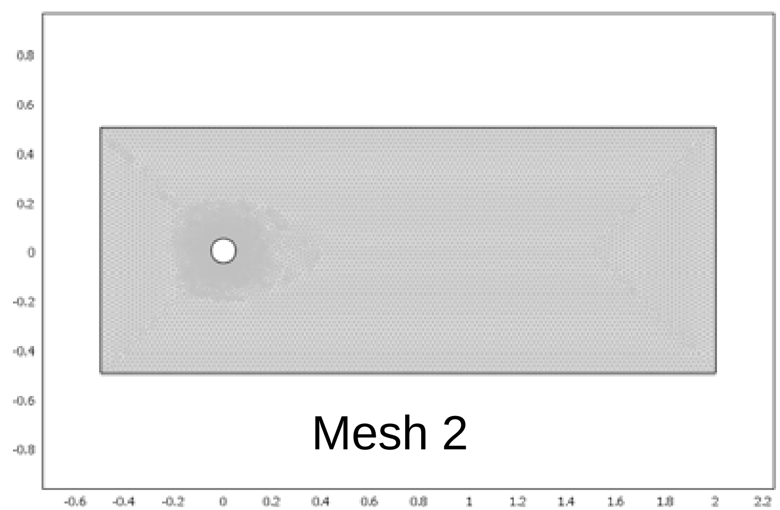
Problem configuration

CFD-CHT problem 1 - Flow around a circular cylinder –cont.

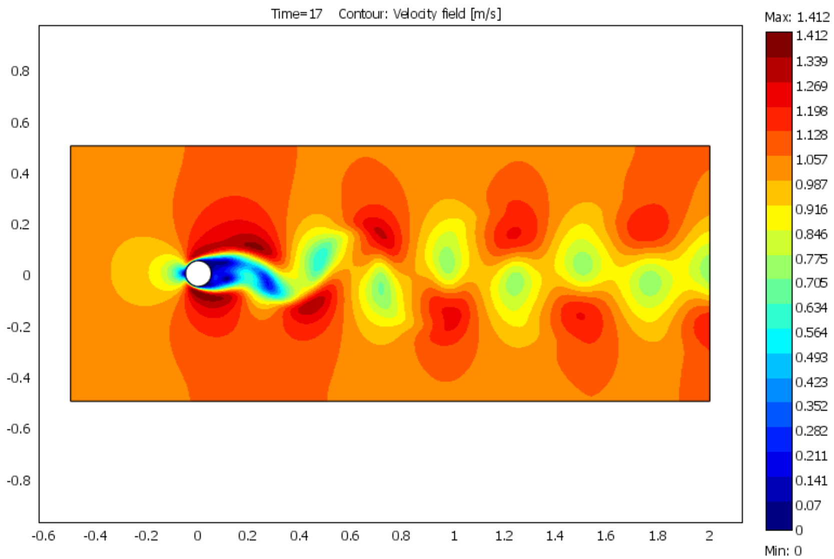
- Re = 100, results from t = 0 s to t = 17 s.
- Mesh independent study

Number of elements	Number of degrees of freedom	CPU time (s)	Memory (MB)
Mesh 1: 8,568	39,306	3,728	884
Mesh 2: 14,965	68,105	14,236	1,193

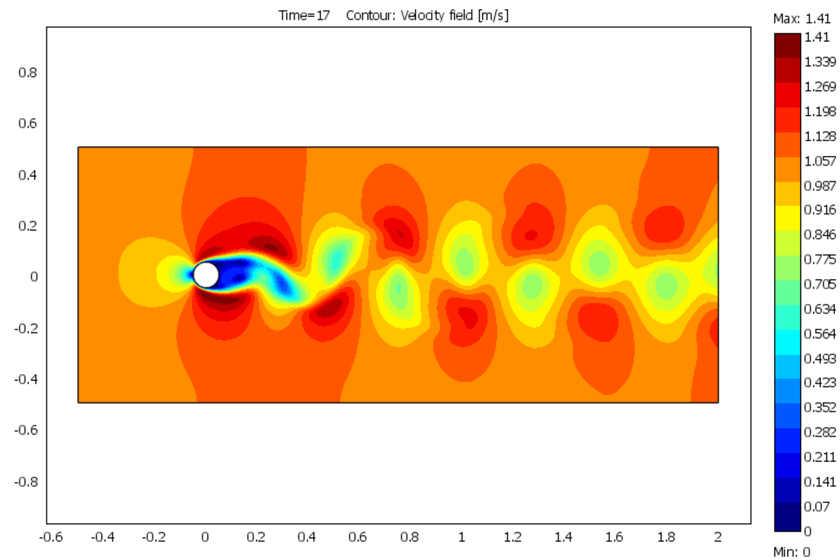

Mesh 1


Mesh 2

CFD-CHT problem 1 - Flow around a circular cylinder –cont.



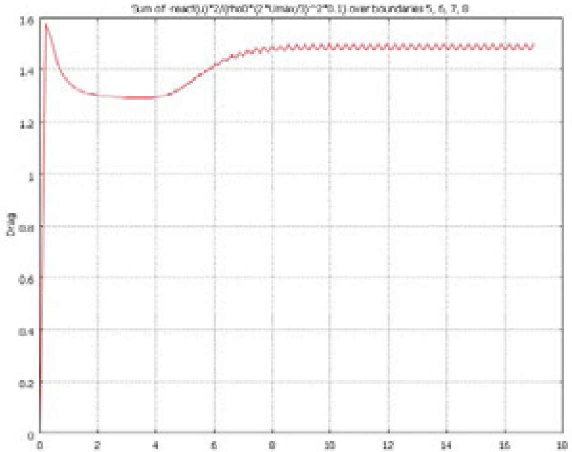
Velocity fields from mesh 1



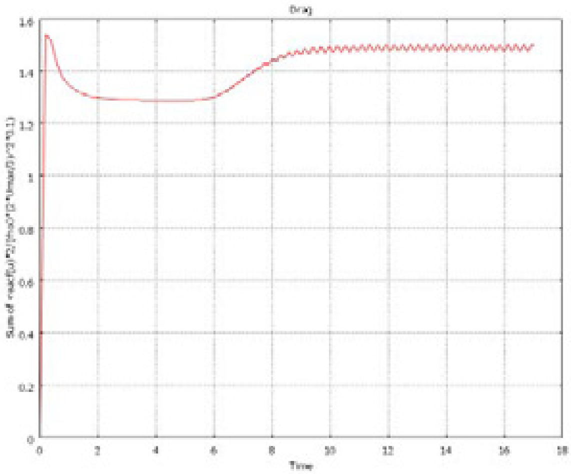
Velocity fields from mesh 2

Re = 100

CFD-CHT problem 1 - Flow around a circular cylinder –cont.



Drag coefficient from mesh 1



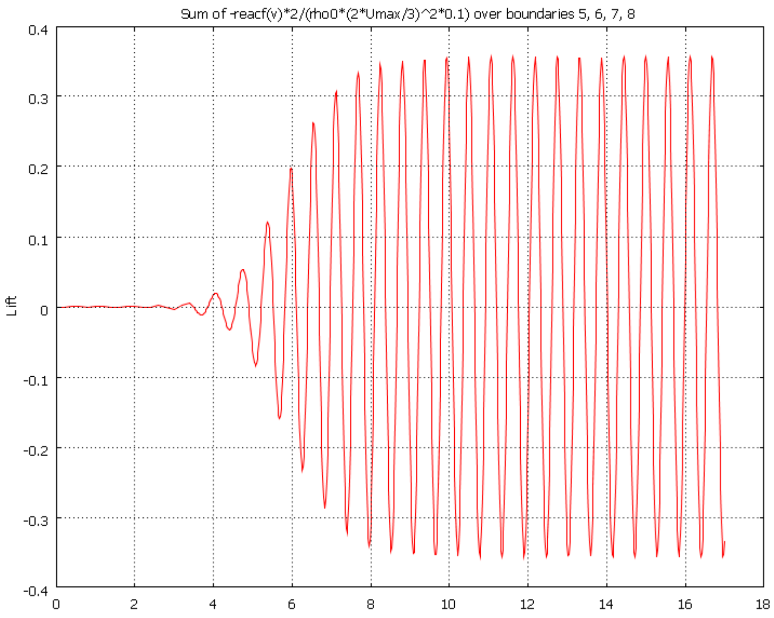
Drag coefficient from mesh 2

Comparison of drag coefficient for Re = 100 with literature data [5]

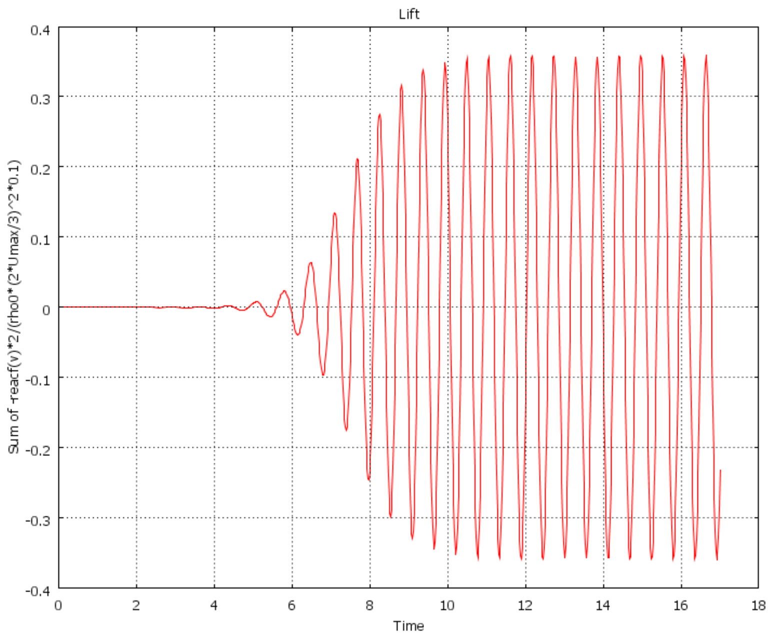
COMSOL 3.5a Mesh 1	COMSOL 3.5a Mesh 2	Numerical Results [5]
1.486	1.485	1.3353

[5] B. N. Rajani, A. Kandasamy and Sekhar Majumdar, "Numerical simulation of laminar flow past a circular cylinder", Applied Mathematical Modelling, 33, pp. 1228-1247, 2009.

CFD-CHT problem 1 - Flow around a circular cylinder –cont.



Lift coefficient from mesh 1



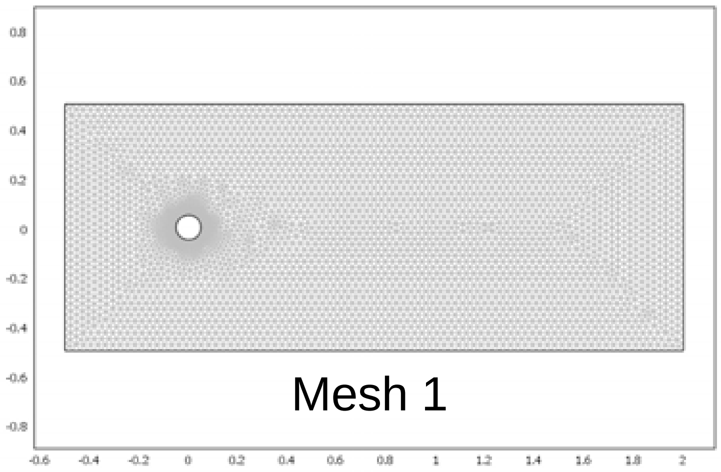
Lift coefficient from mesh 2

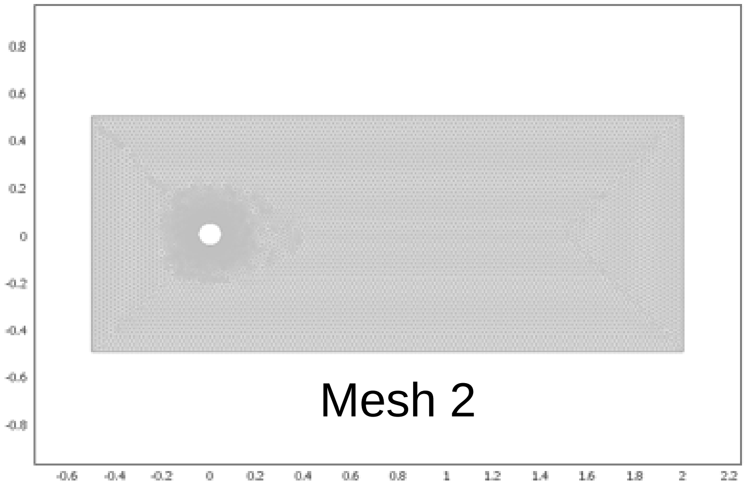
Re = 100

CFD-CHT problem 1 - Flow around a circular cylinder –cont.

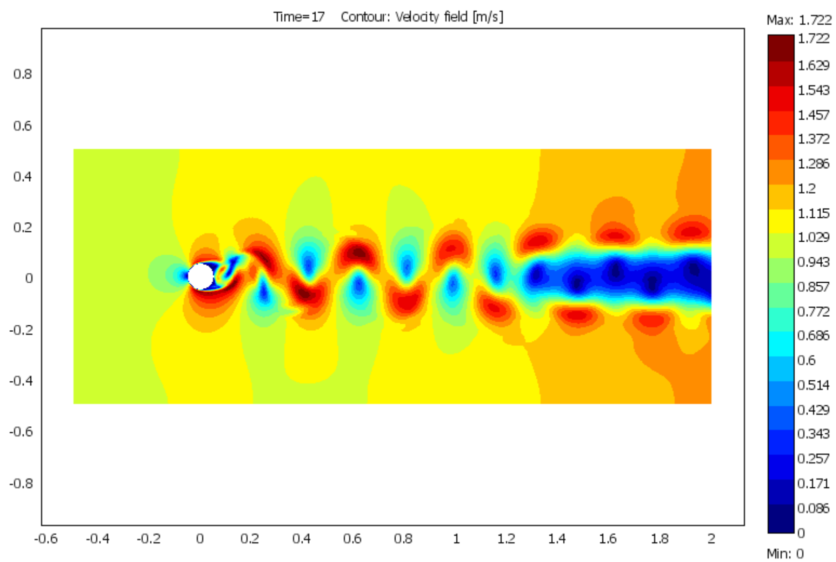
- Re = 1,000, results from t = 0 s to t = 17 s.
- Mesh independent study

Number of elements	Number of degrees of freedom	CPU time (s)	Memory (MB)
Mesh 1: 8,272	37,974	1,894	974
Mesh 2: 17,536	79,947	4,024	1,501

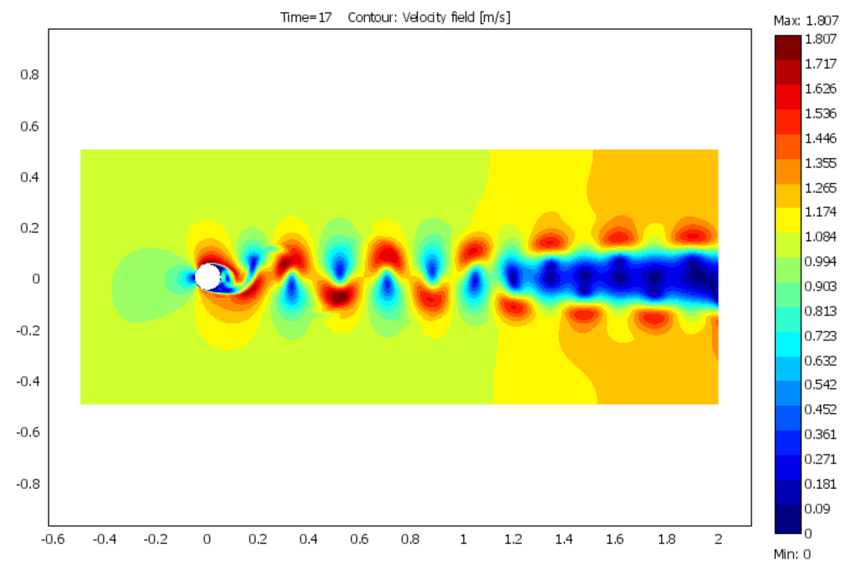




CFD-CHT problem 1 - Flow around a circular cylinder –cont.



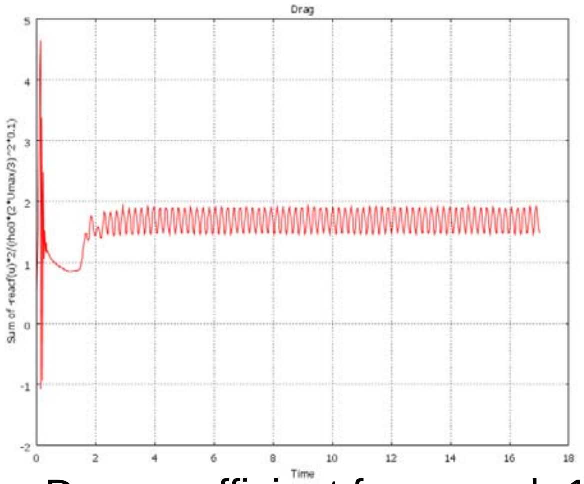
Velocity fields from mesh 1



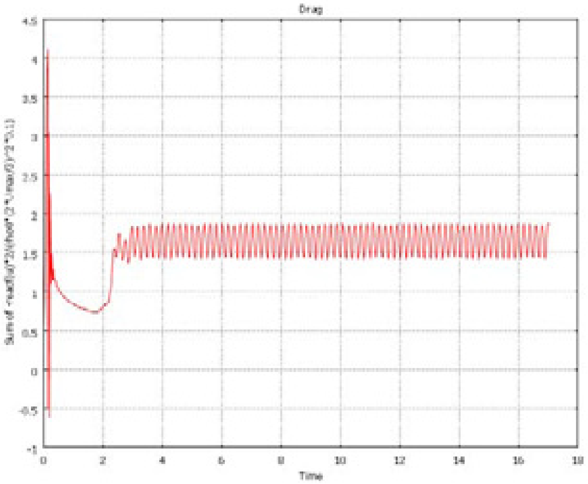
Velocity fields from mesh 2

Re = 1000

CFD-CHT problem 1 - Flow around a circular cylinder –cont.



Drag coefficient from mesh 1



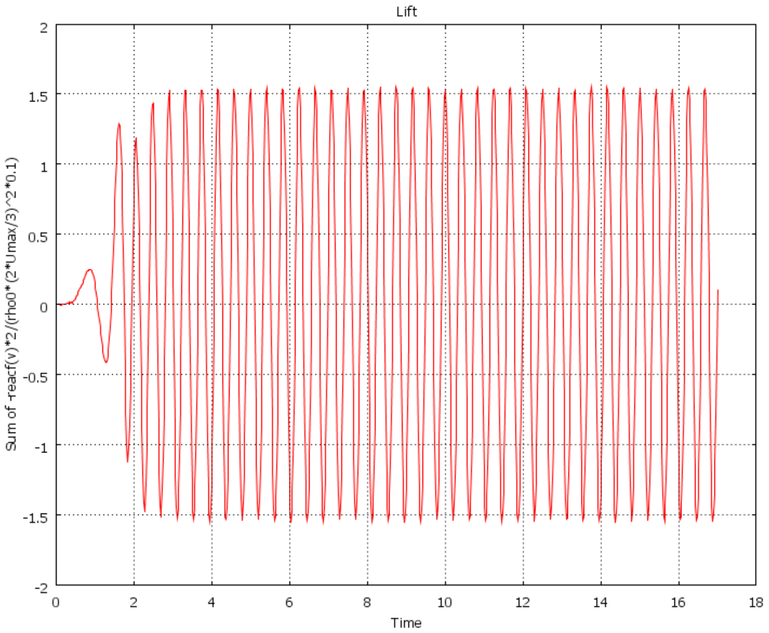
Drag coefficient from mesh 2

Comparison of drag coefficient with literature data [6]

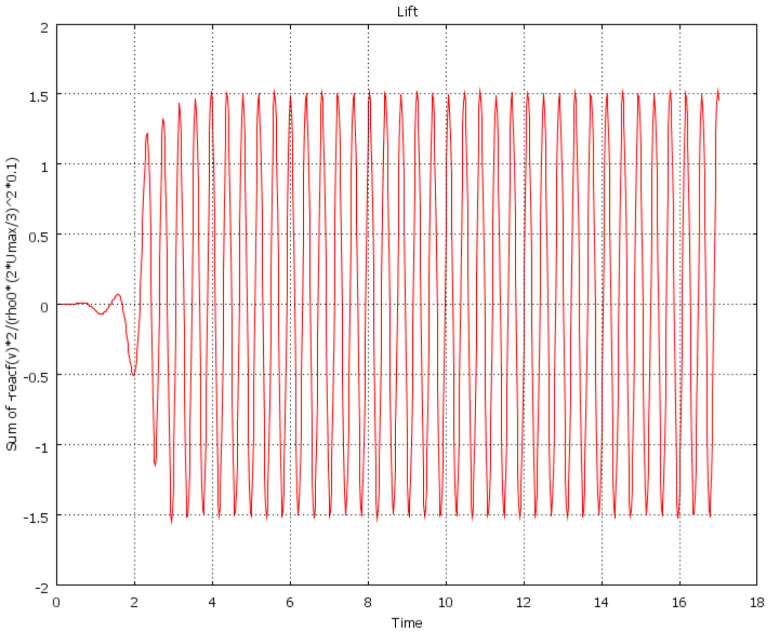
COMSOL 3.5a Mesh 1	COMSOL 3.5a Mesh 2	Numerical Results [6]
1.69	1.65	1.47

[6] G. Sod, "A survey of finite difference methods for systems of nonlinear hyperbolic conservation laws", Journal of Computational Physics, 27, pp.1-31, 1978.

CFD-CHT problem 1 - Flow around a circular cylinder –cont.

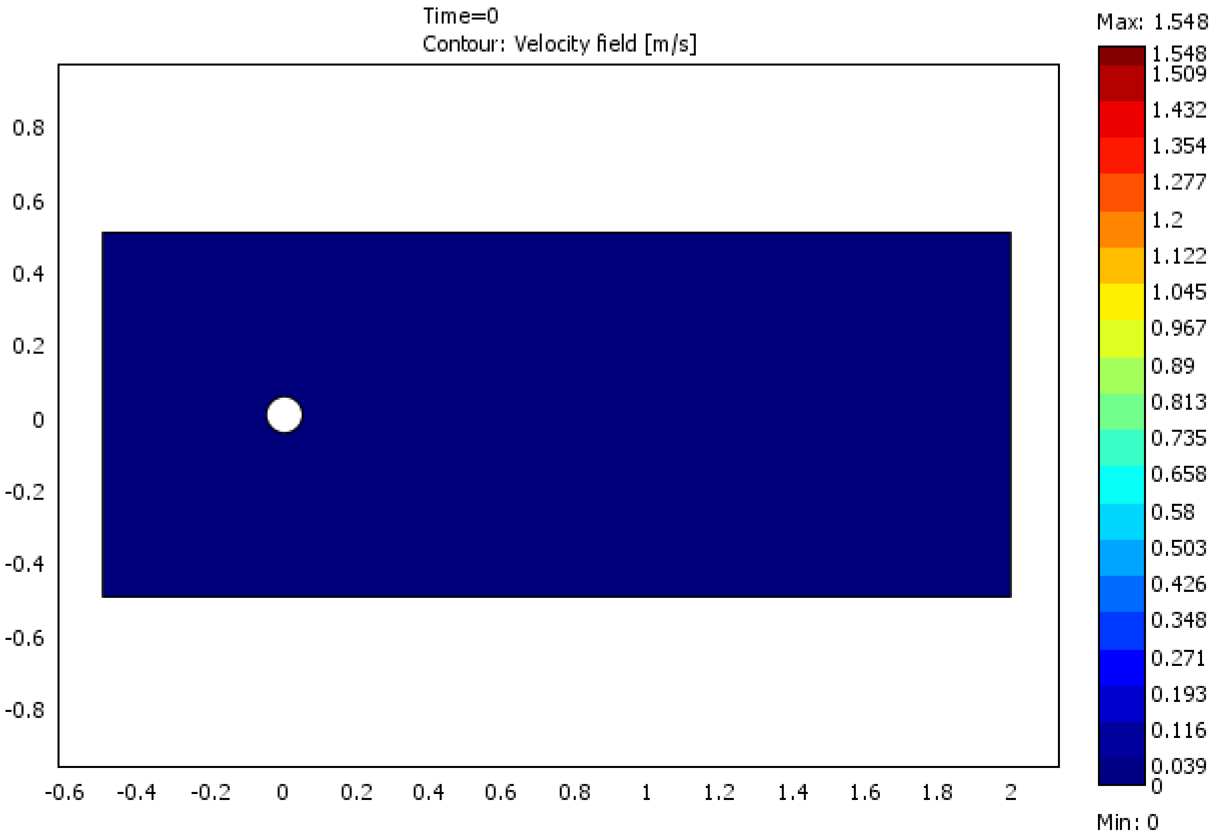


Lift coefficient from mesh 1

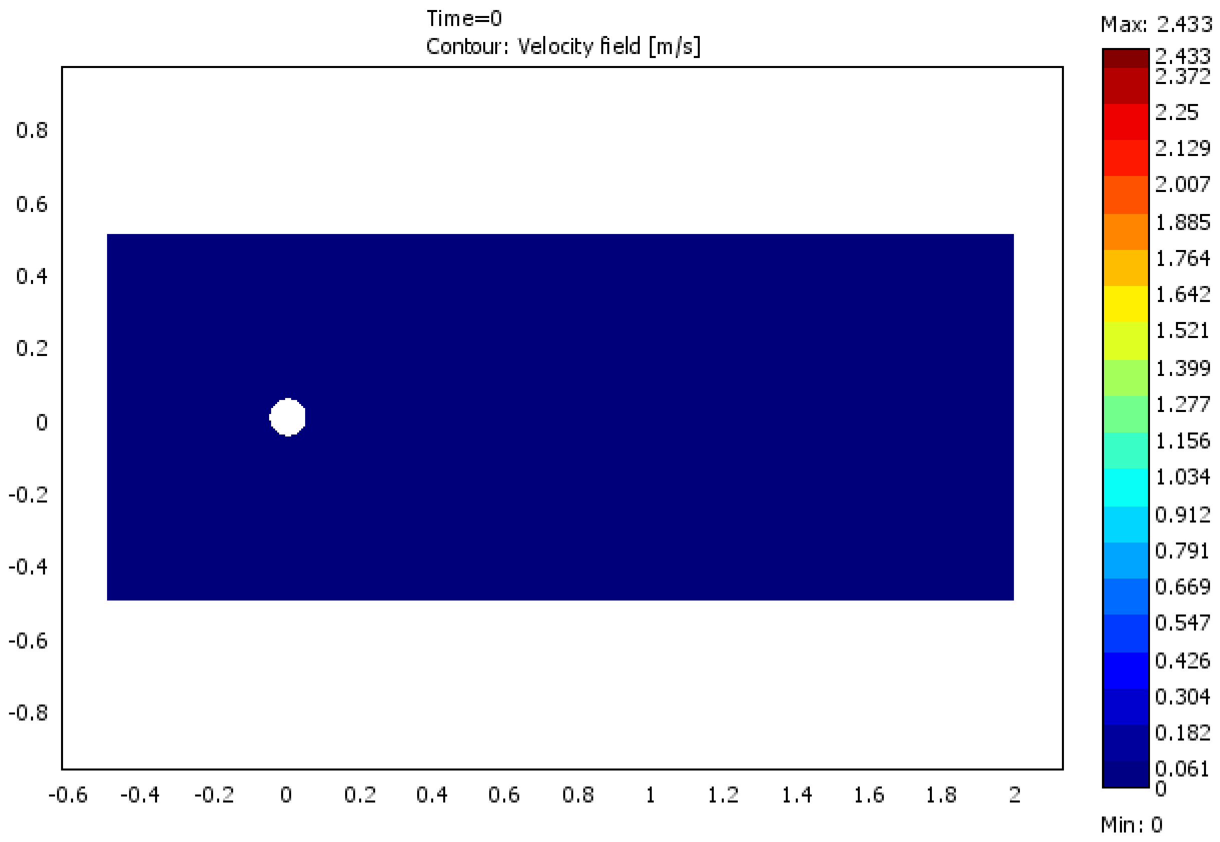


Lift coefficient from mesh 2

Flow over a cylinder Re = 100

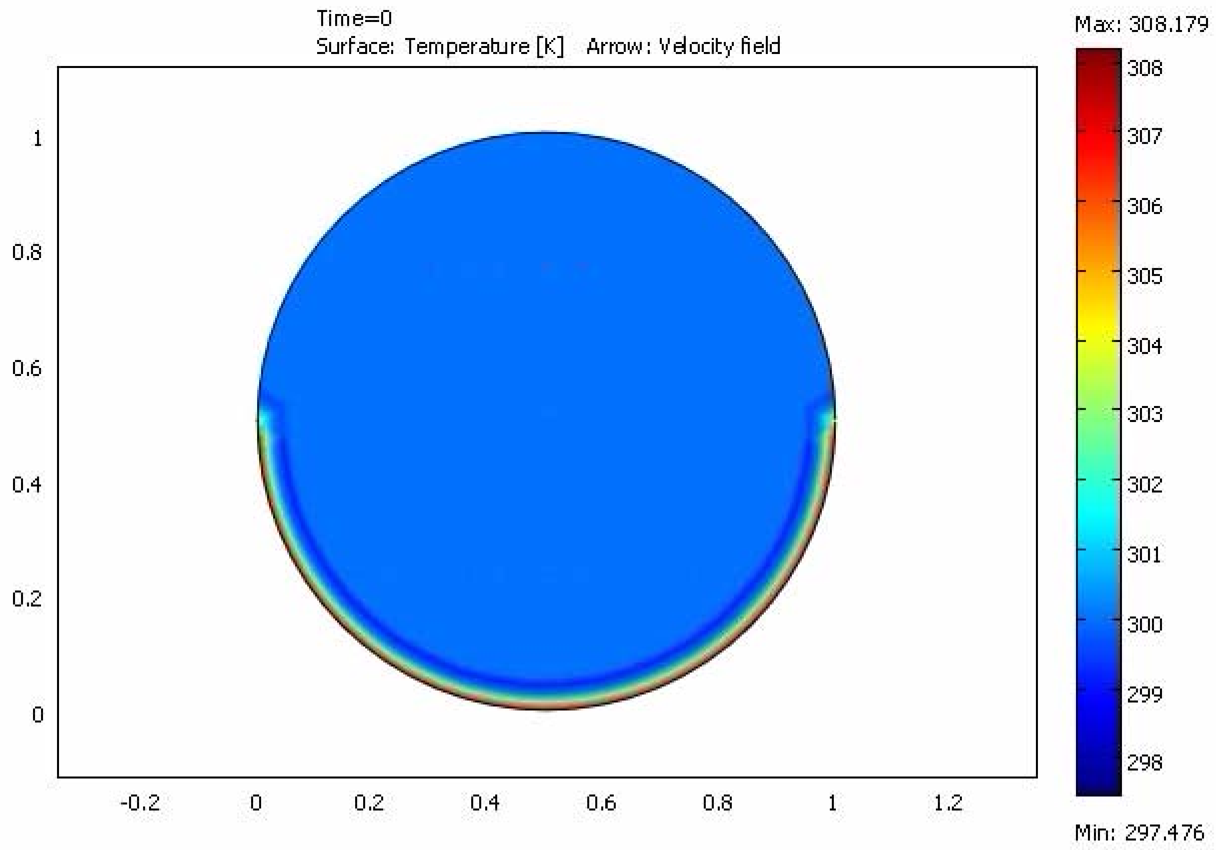


Re = 1000



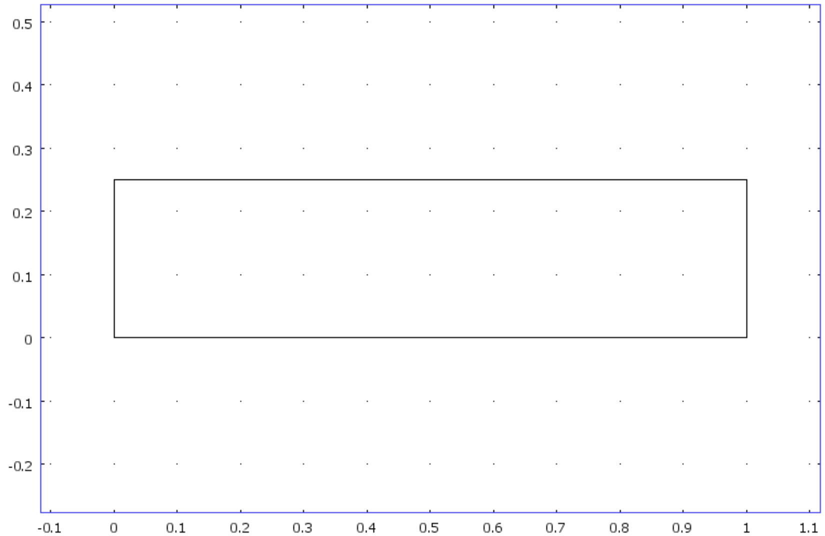
Time=0
Contour: Velocity field [m/s]
0.8
0.6
0.4
0.2
0
-0.2
-0.4
-0.6
-0.8
-0.6
-0.4
-0.2
0
0.2
0.4
0.6
0.8
1
1.2
1.4
1.6
1.8
2
Max: 2.433
2.433
2.372
2.25
2.129
2.007
1.885
1.764
1.642
1.521
1.399
1.277
1.156
1.034
0.912
0.791
0.669
0.547
0.426
0.304
0.182
0.061
0
Min: 0

Natural convection within a cylinder



CFD-CHT problem 2 - Compressible flow in a shock tube

- Shock waves arise from sudden jumps in gas properties such as temperature or pressure. They are very thin regions (~10^{-8} m) in a supersonic flow across which there is a large variation in flow properties.
- The configuration of problem is shown in the figure below, the diaphragm is located at x = 0.5.



CFD-CHT problem 2 -Compressible flow in a shock tube –cont.

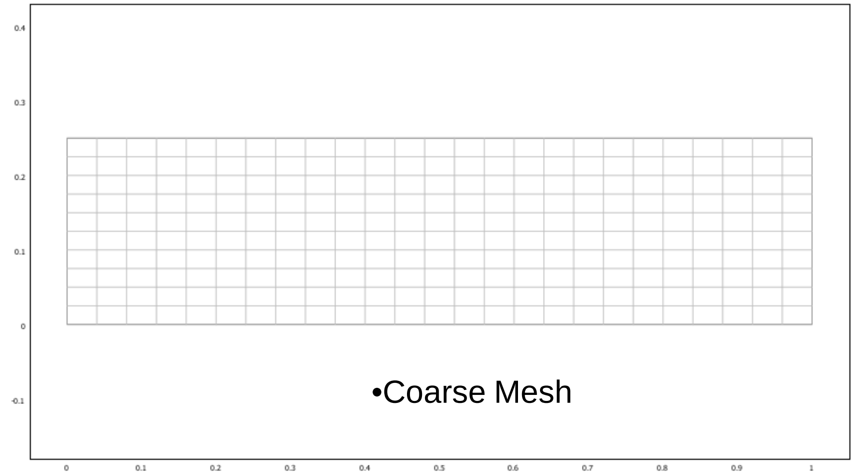
- The initial conditions for the driver section were $\rho = 8.0; P = 7.2$ and $; u = 0.0$;the initial condition for the driven section was

  $\rho = 1.0; P = 0.72; u = 0.0$
- Results were obtained and compared with analytical solutions as well as simple numerical models based on MacCormack and Roe's methods for t = 0.2.

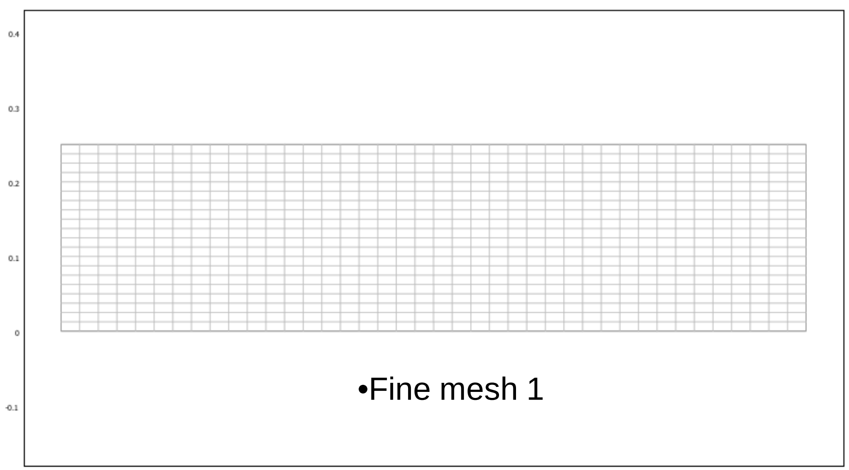
Computational meshes

Number of elements for coarse mesh	Number of degrees of freedom for coarse mesh	Number of elements for final fine mesh	Number of degrees of freedom for final fine mesh
250	3,213	800	9,963
250	3,213	4000	48,843

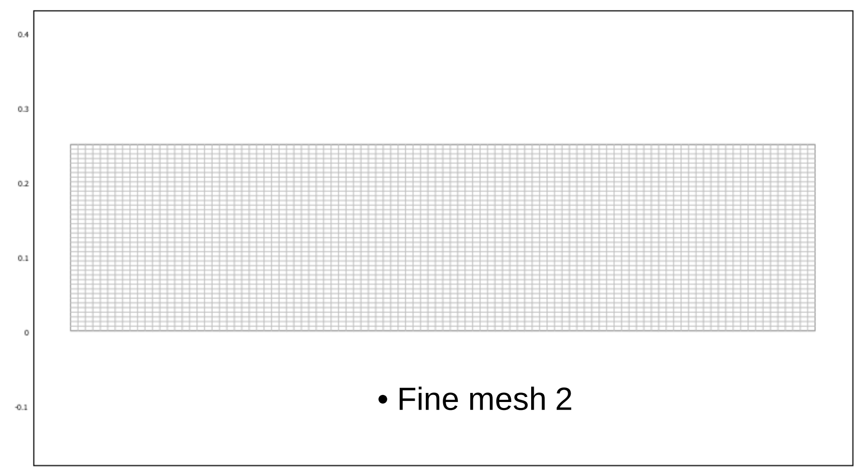
CFD-CHT problem 2 -Compressible flow in a shock tube –cont.



•Coarse Mesh

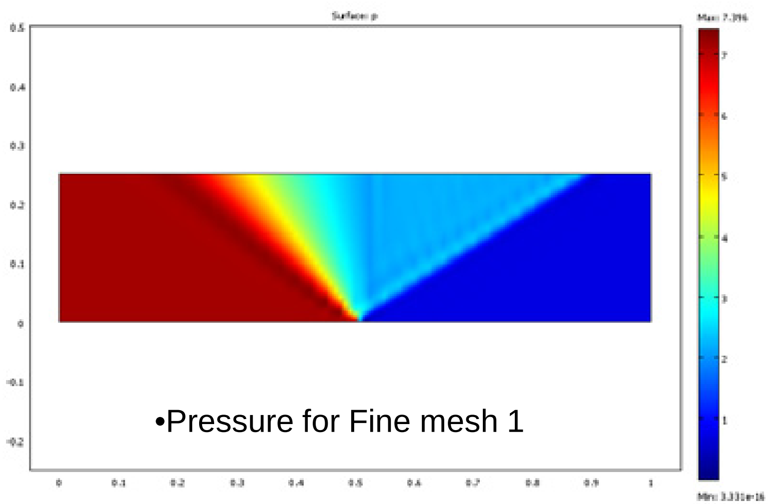


•Fine mesh 1

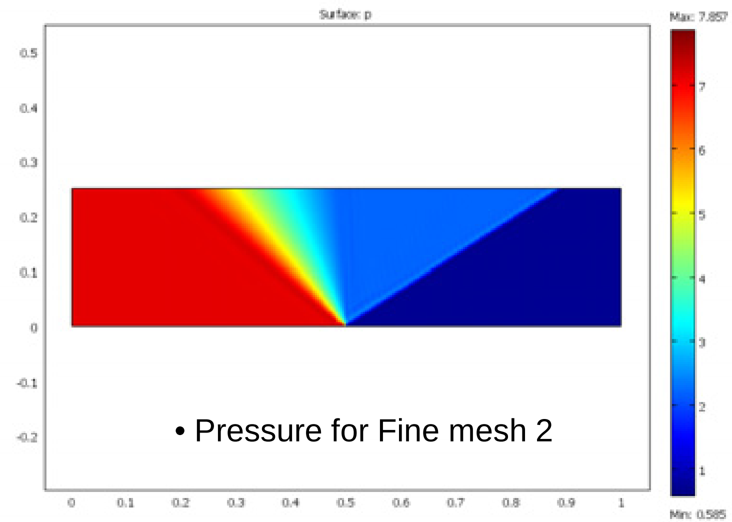


• Fine mesh 2

CFD-CHT problem 2 -Compressible flow in a shock tube –cont.

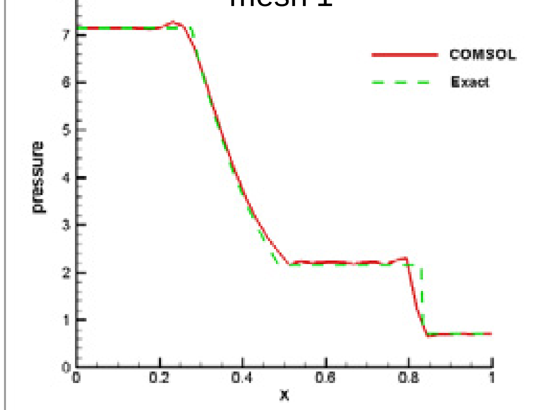


•Pressure for Fine mesh 1

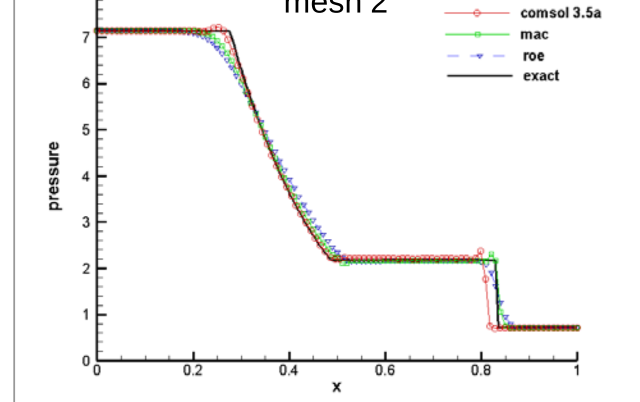


• Pressure for Fine mesh 2

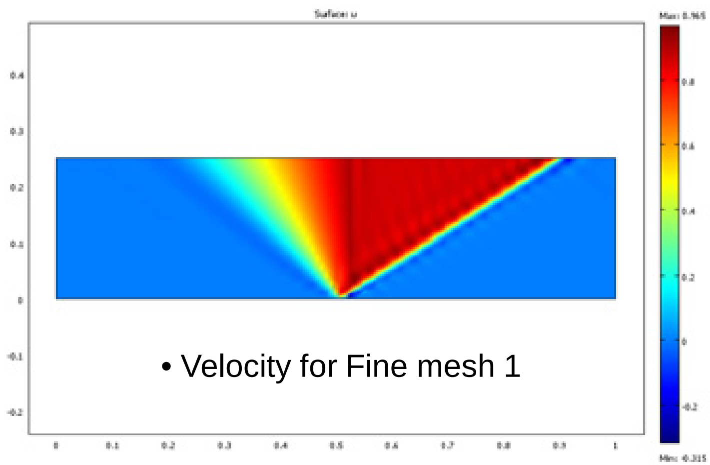
• Pressure comparison for Fine mesh 1



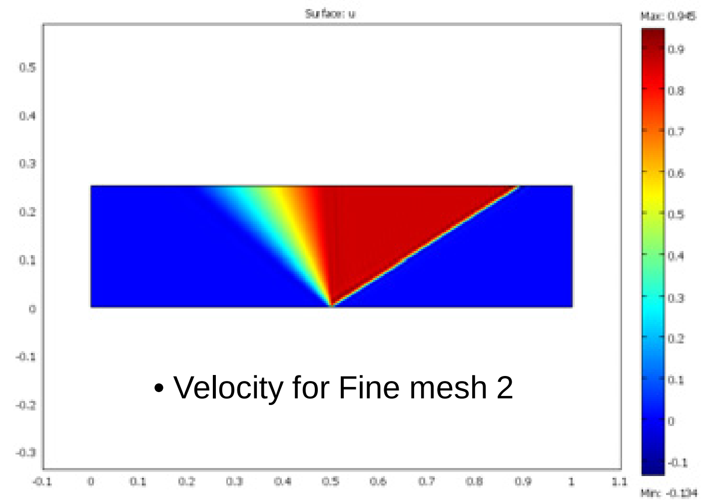
• Pressure comparison for Fine mesh 2



CFD-CHT problem 2 -Compressible flow in a shock tube –cont.

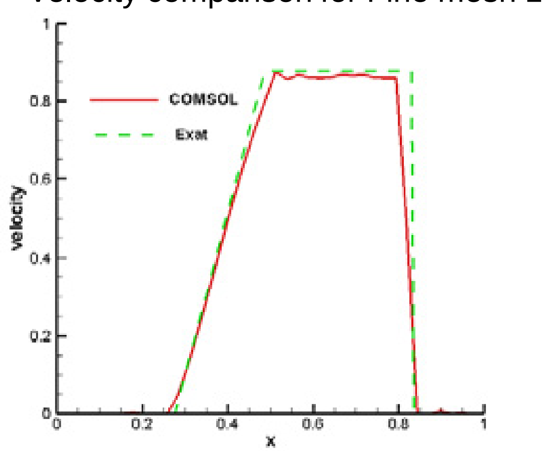


• Velocity for Fine mesh 1

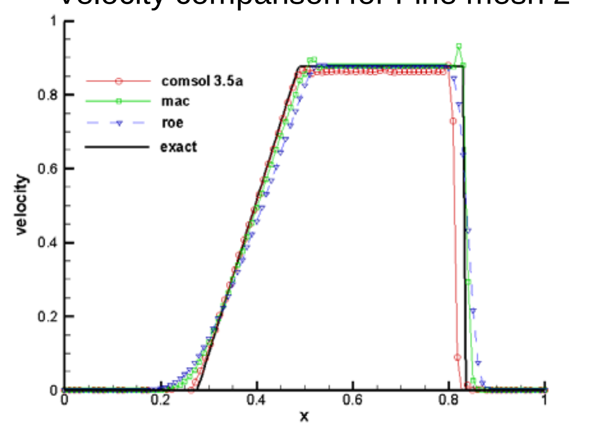


• Velocity for Fine mesh 2

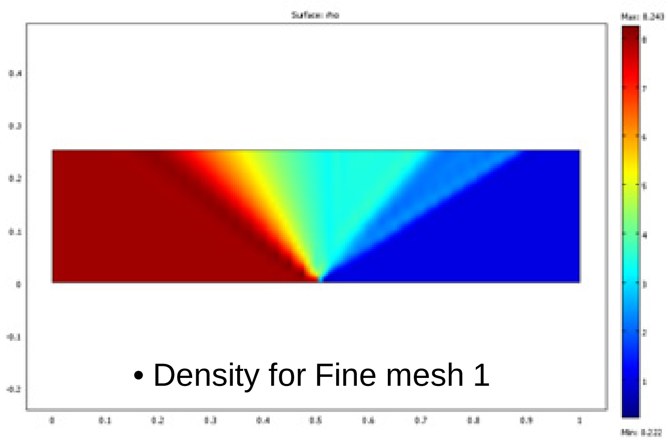
• Velocity comparison for Fine mesh 1



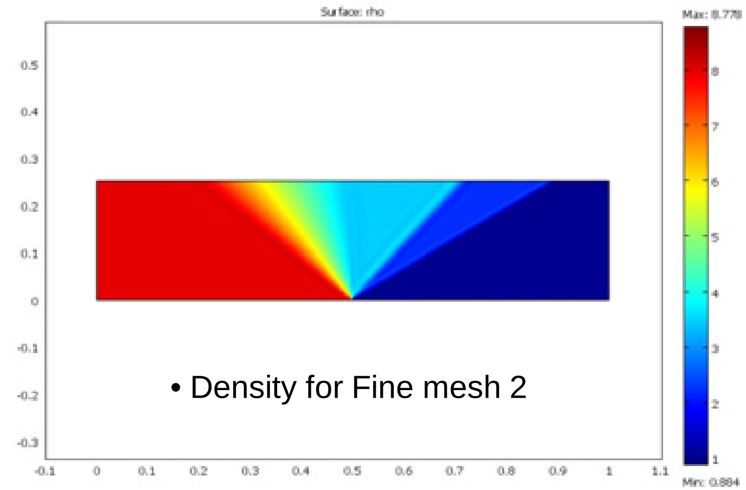
• Velocity comparison for Fine mesh 2



CFD-CHT problem 2 -Compressible flow in a shock tube –cont.

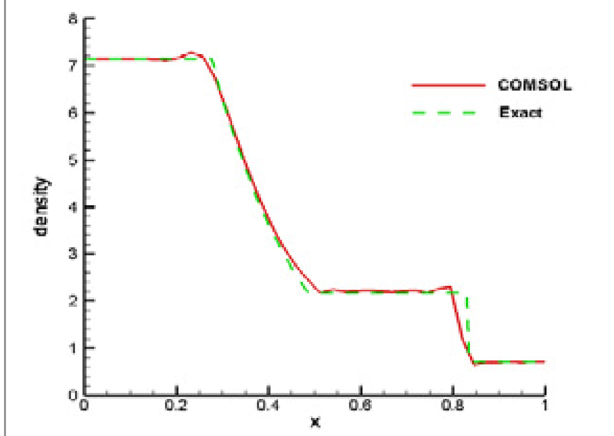


• Density for Fine mesh 1

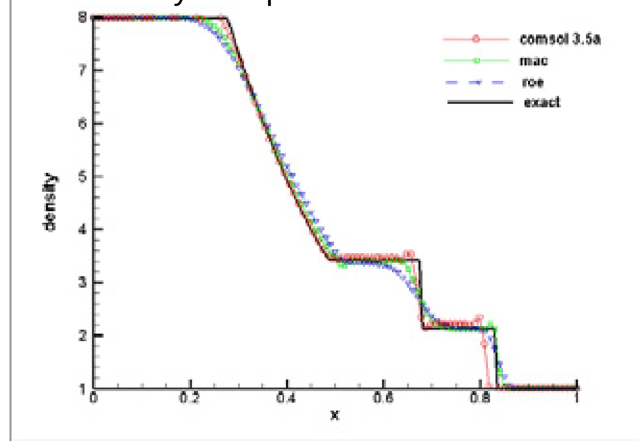


• Density for Fine mesh 2

• Density Comparison for Fine mesh 1

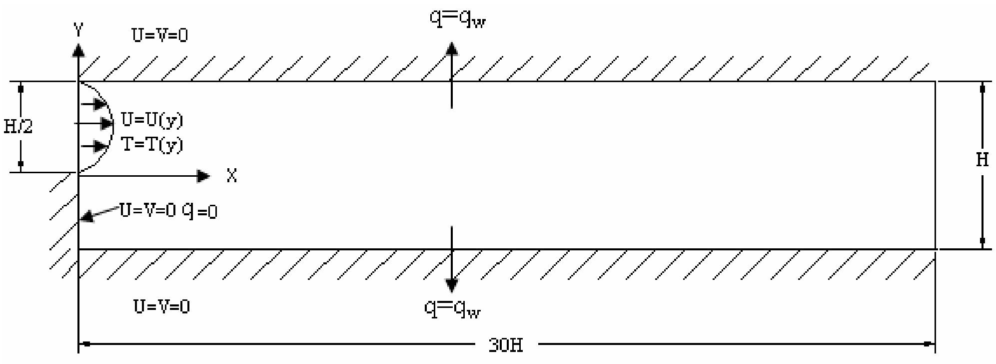


• Density Comparison for Fine mesh 2



CFD-CHT problem 3 - Flow over a backward facing step

- Incompressible flow over a backward facing step is a classic problem that has been analyzed for many years. While there are numerous fluid flow comparison studies, very few include the effects of heat transfer.
- First test case is run as Re = 800 for thermal and fluid flow; second test case is run for Re = 47,648 for fluid flow only. The configuration of problem is shown as:



CFD-CHT problem 3 - Flow over a backward facing step – cont.

- For inlet flow:

$$u(y)=\begin{cases}0,\ for\, 0\le y\le \frac{1}{2}\\ 8y(1-2y),\ for\, \frac{1}{2}<y\le 1\end{cases}\qquad v(y)=0$$

$$T(y)=\left[1-(4y-1)^2\right]\left[1-\frac{1}{5}(4y-1)^2\right]\, for\, \frac{1}{2}<y\le 1$$

$$\frac{\partial T(y)}{\partial x}=0\, for\, 0\le y<\frac{1}{2}$$

- on upper and lower walls:

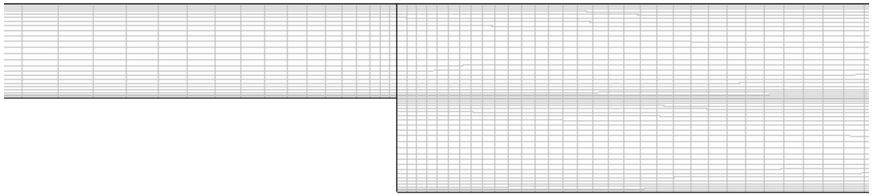
$$u(y)=v(y)=0$$

$$\nabla T\cdot\hat{n}=\frac{32}{5}$$

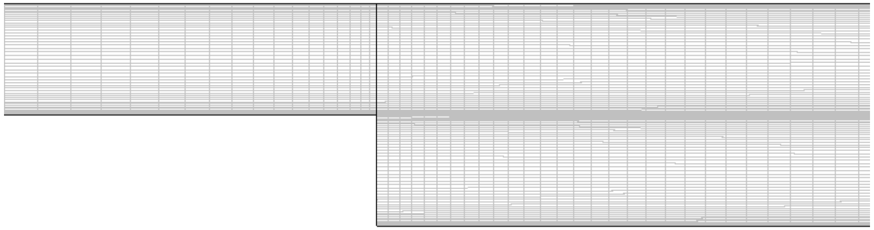
CFD-CHT problem 3 - Flow over a backward facing step – cont.

- Re = 800

Number of elements	Number of degrees of freedom	CPU time (s)	Memory (MB)
Mesh 1: 10,850	108,864	2	298
Mesh 2: 22,000	288,384	3	350



•mesh 1

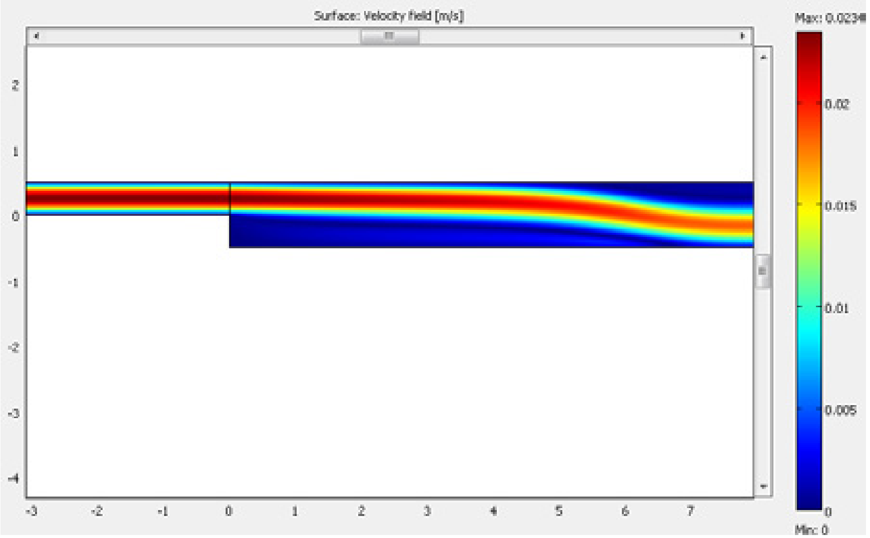


•mesh 2

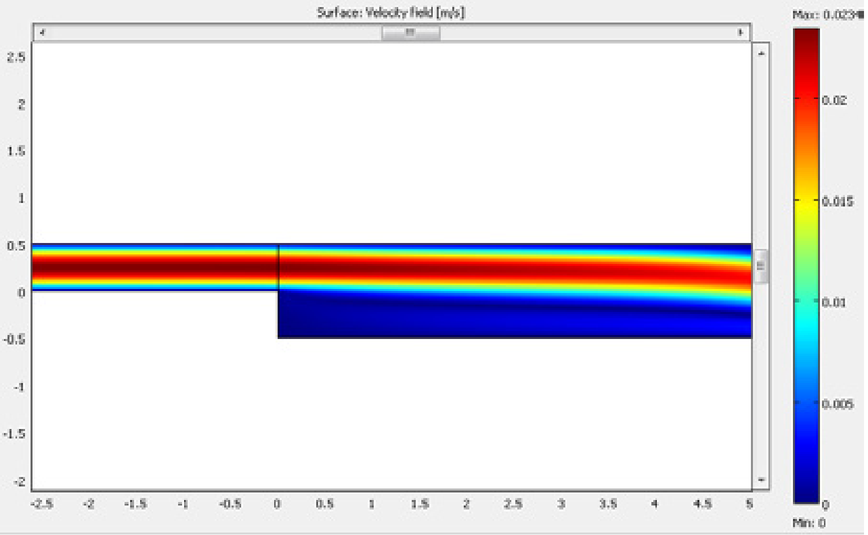
Notice the fine mesh used along the boundary and in regions close to the step

CFD-CHT problem 3 - Flow over a backward facing step – cont.

- Re = 800



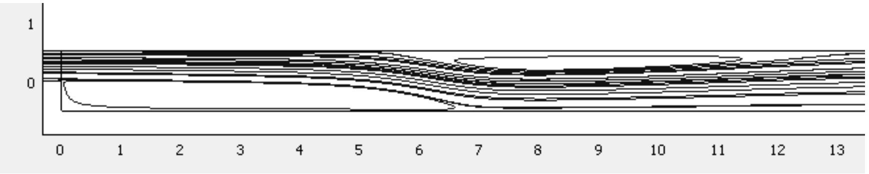
•Velocity fields from mesh 1

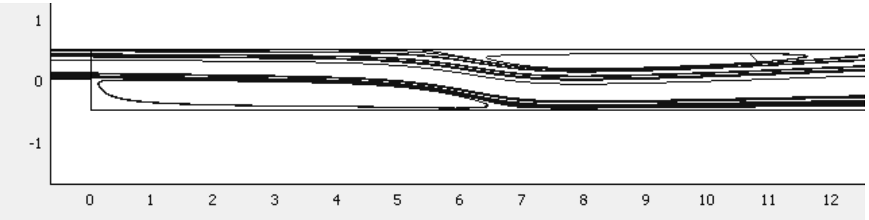


•Velocity fields from mesh 2

CFD-CHT problem 3 - Flow over a backward facing step – cont.

- Re = 800





•Streamlines from mesh 1

•Streamlines from mesh 2

Comparison of lower wall eddy sizes with literature data [12] [13]

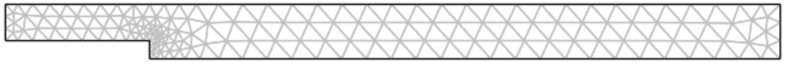
COMSOL 3.5a Mesh 1	COMSOL 3.5a Mesh 2	Gartling [12]	Wang and Pepper [13]
6.80	6.70	6.1	6.0

[12] D. K. Gartling, "A Test Problem for Outflow Boundary Conditions- Flow over a Backward-Facing Step", Int. J. Numer. Meth. Fluids, Vol. 11, pp. 953-967, 1990.
[13] X. Wang and D. W. Pepper, "Application of an *hp*-adaptive FEM for Solving Thermal Flow Problems", Journal of Thermophysics and Heat Transfer, Vol. 21, No. 1, pp.190-198, 2007.

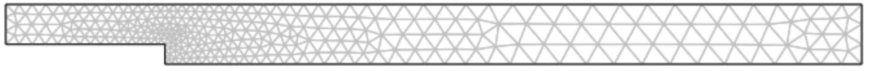
CFD-CHT problem 3 - Flow over a backward facing step – cont.

- Re = 47,648

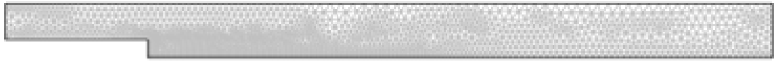
Initial Number of elements	Initial Number of degrees of freedom	Final Number of elements	Final Number of degrees of freedom	CPU time (s)	Memory (MB)
Mesh 1: 291	2,861	3,876	34,373	52	233
Mesh 2: 585	5,504	8,734	76,701	119	350



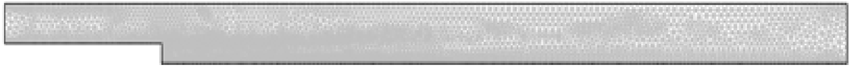
•Initial mesh 1



•Initial mesh 2



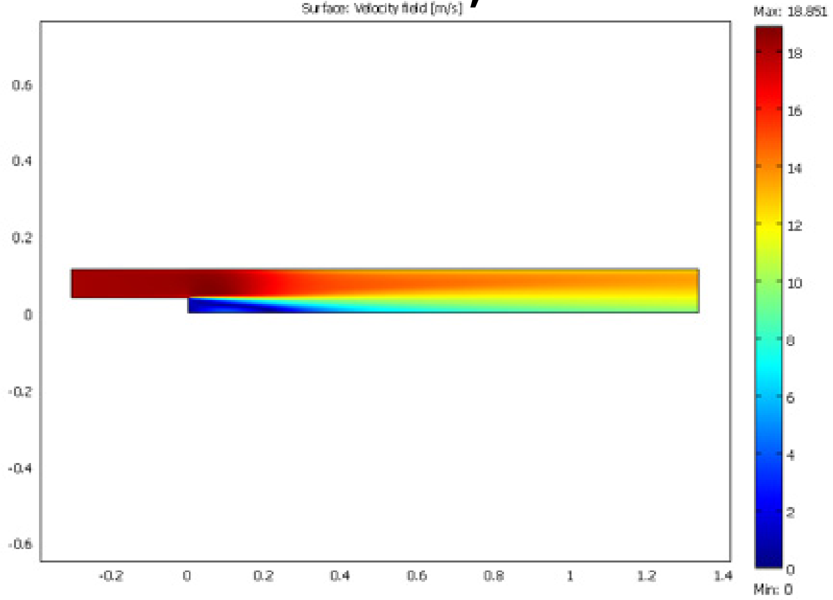
•Adapted mesh 1



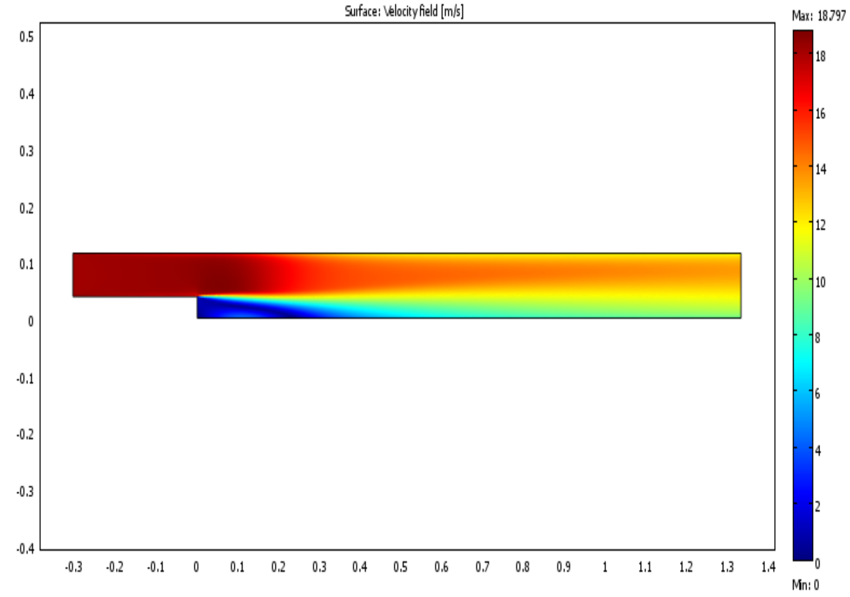
•Adapted mesh 2

CFD-CHT problem 3 - Flow over a backward facing step – cont.

- Re = 47,648



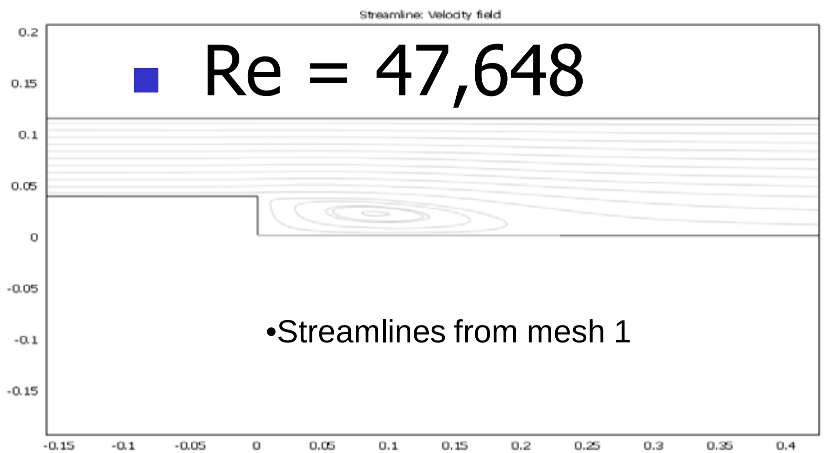
•Velocity fields from mesh 1



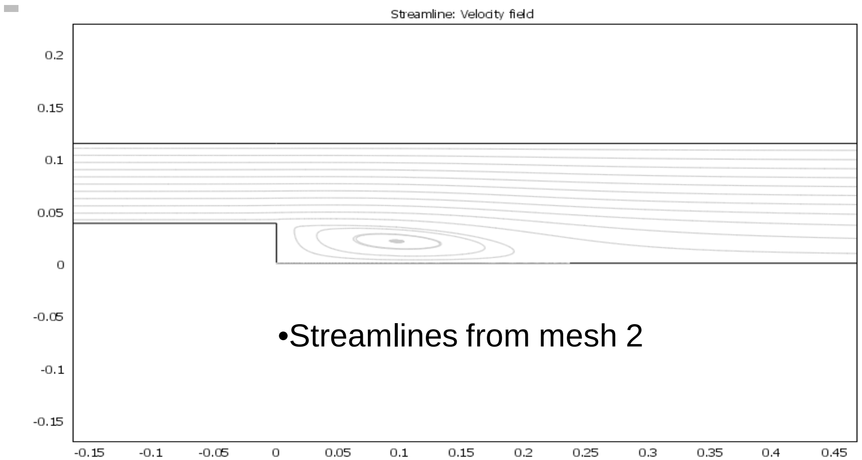
•Velocity fields from mesh 2

CFD-CHT problem 3 - Flow over a backward facing step – cont.

- Re = 47,648



•Streamlines from mesh 1



•Streamlines from mesh 2

Comparison of lower wall eddy sizes with literature data [14] [15]

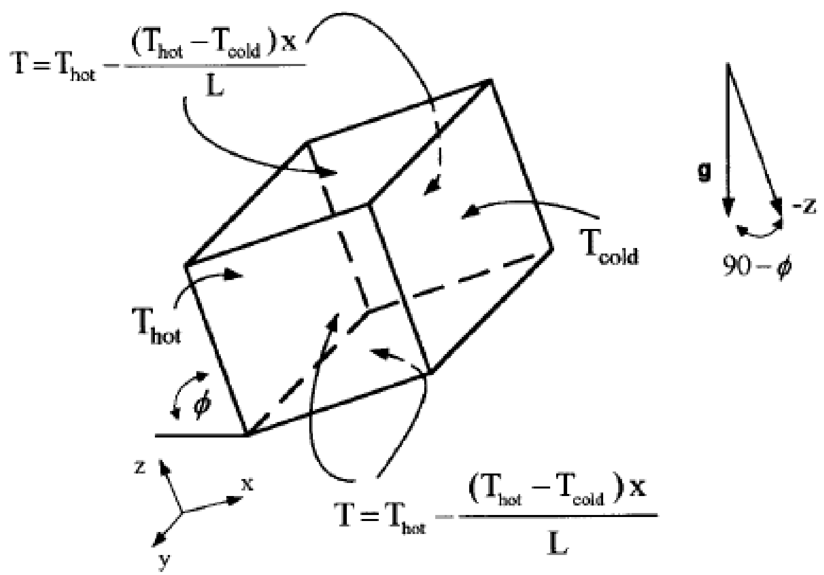
COMSOL 3.5a Mesh 1	COMSOL 3.5a Mesh 2	Experimental data	Other simulation results
6.0	6.19	7.1	6.1

[14] 1st NAFEMS Workbook of CFD Examples. Laminar and Turbulent Two-Dimensional Internal Flows, NAFEMS, 2000.

[15]Patrick J. Roache, Verification and Validation in Computational Science and Engineering, Hermosa Pub., Albuquerque, NM, 1998.

CFD-CHT problem 4 - Natural Convection within a 3-D Enclosure

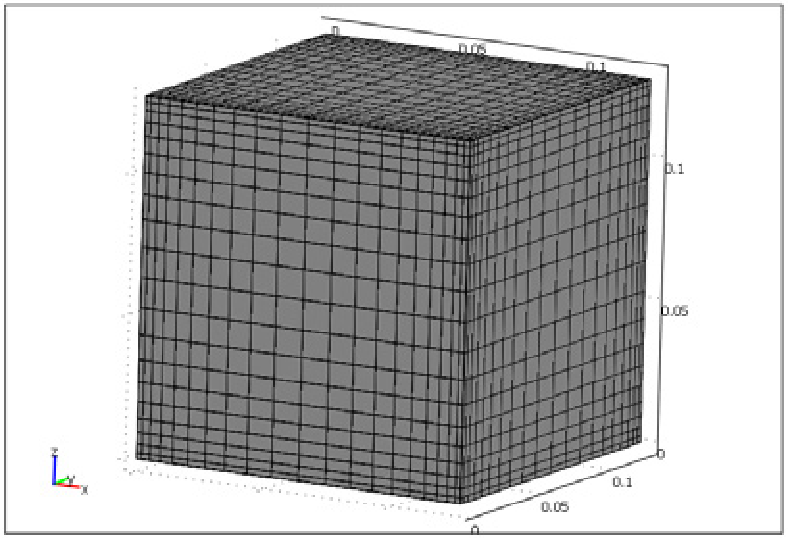
- The last CFD-CHT problem deals with natural convection within a 3-D enclosure. This problem has been studied for many decades, and was one of the earliest simulations performed numerically to examine strong fluid-heat transfer coupling.
- The following figure shows the configuration of the problem, with being set to 90^{0}C, 45^{0}C and 0^{0}C, respectively.



CFD-CHT problem 4 - Natural Convection within a 3-D Enclosure – cont.

- Case 1: φ = 90º

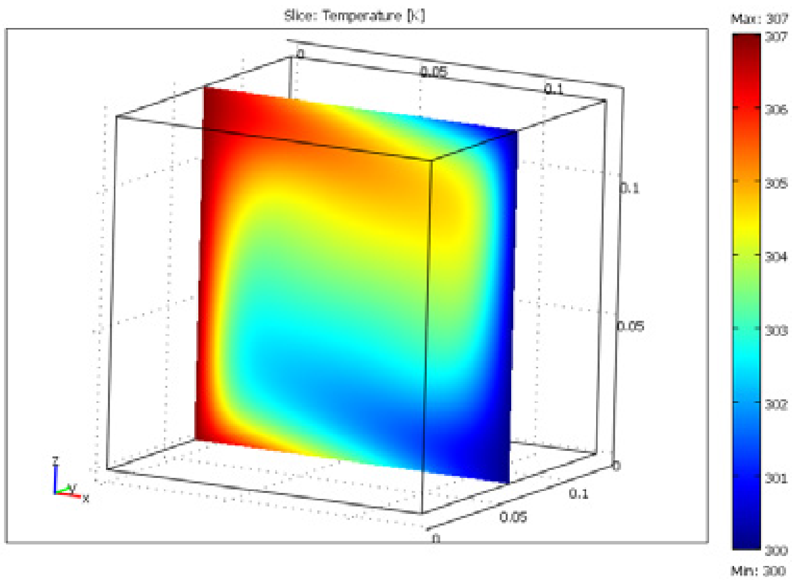
Number of elements for coarse mesh	Number of degrees of freedom for coarse mesh	Number of elements for final fine mesh	Number of degrees of freedom for final fine mesh
1,000	38,375	8,000	284,945

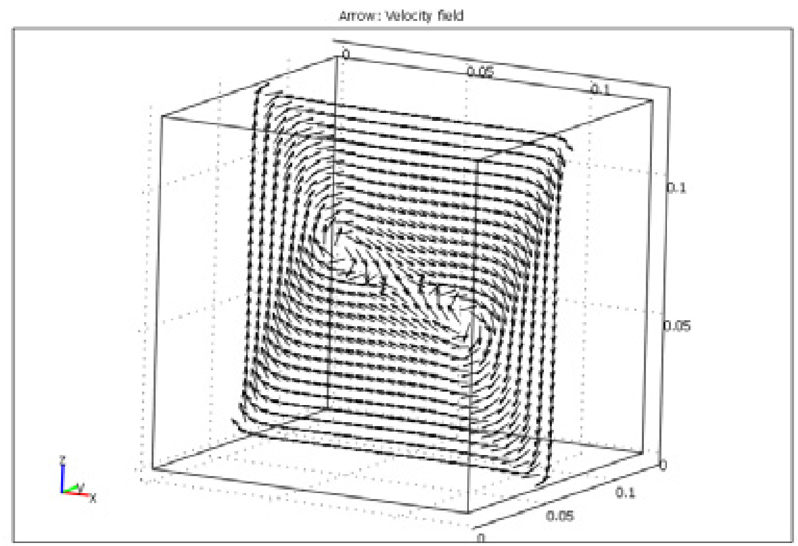


• Final Computational mesh

CFD-CHT problem 4 - Natural Convection within a 3-D enclosure – cont.

- Case 1: φ = 90° Ra = 10^5 at y = L/2





• Temperature contours

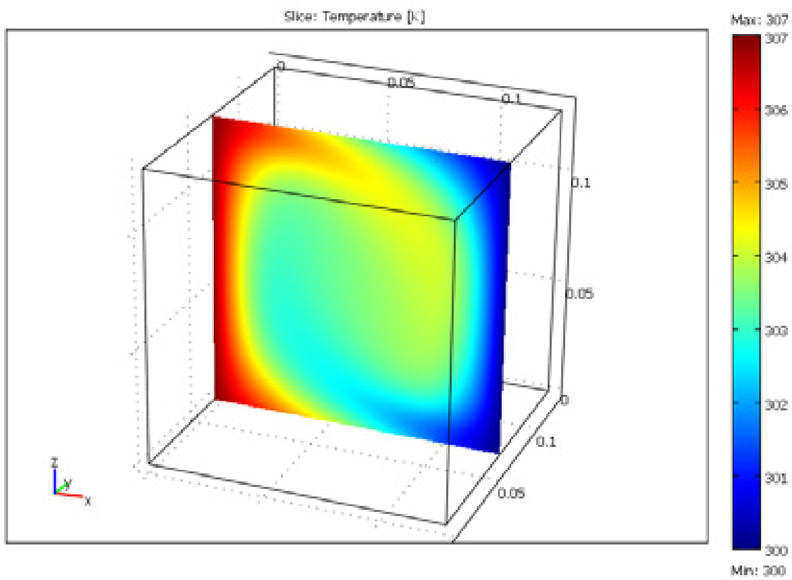
• Velocity vectors

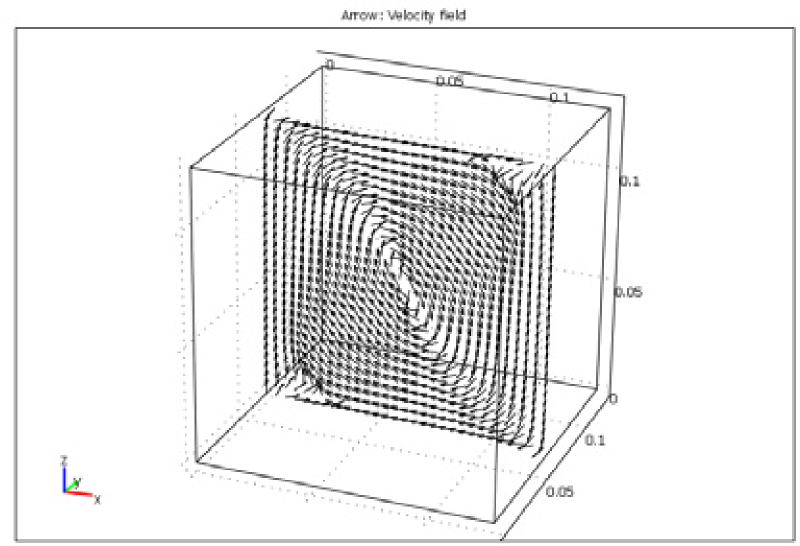
Comparison of Nu with literature data [16-19]

Results from COMSOL 3.5a	[16]	[17]	[18]	[19]
3.12	3.11	3.06-3.12	3.10	3.19-3.20

CFD-CHT problem 4 - Natural Convection within a 3-D enclosure – cont.

- Case 2: φ = 45^o Ra = 10^5 at y = L/2





• Temperature contours

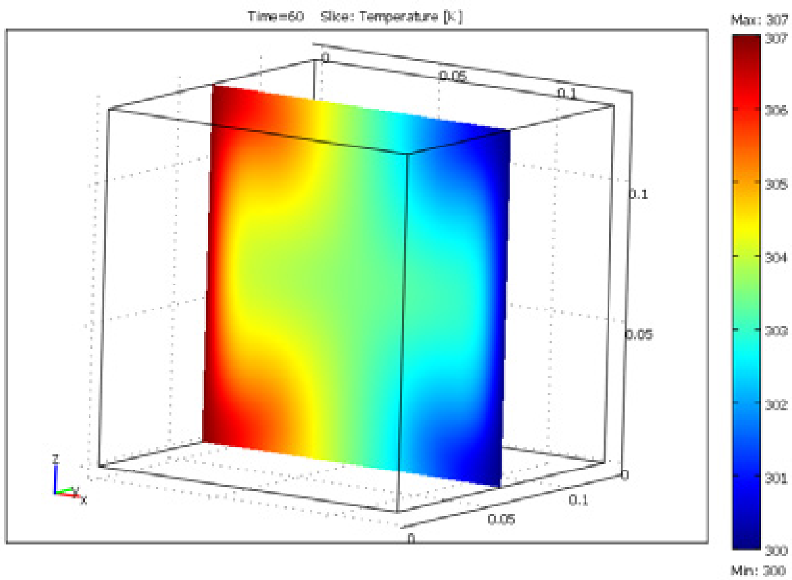
• Velocity vectors

Comparison of Nu with literature data [16-19]

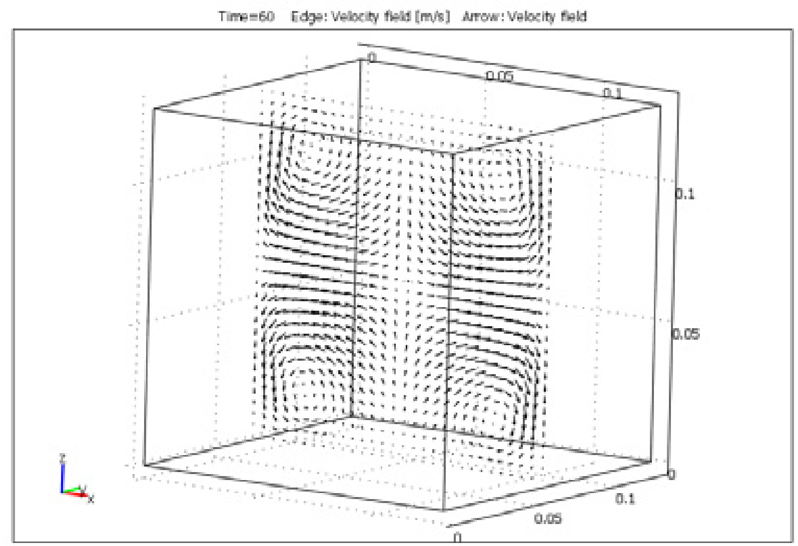
Results from COMSOL 3.5a	[16]	[17]	[18]	[19]
3.54	-	3.40-3.47	3.50	3.57-3.60

CFD-CHT problem 4 - Natural Convection within a 3-D enclosure – cont.

- Case 3: φ = 0° Ra = 10^5 at y = L/2



• Temperature contours



• Velocity vectors

Comparison of Nu with literature data [16-19]

Results from COMSOL 3.5a	[16]	[17]	[18]	[19]
2.25	3.24	3.34-3.47	2.49-3.92	3.49-4.01

CFD-CHT problem 4 - Natural Convection within a 3-D enclosure – cont.

[16] R. Bennacer, A. A. Mohamad, and I. Sezai, Transient Natural Convection in Air-Filled Cubical Cavity: Validation Exercise, ICHMT 2nd Int. Symp. on Adv. in Comput. Heat Transfer, Palm Cove, Queensland, Australia, May 20– 25, 2001.

[17] R. Mossad, Prediction of Natural Convection in an Air-Filled Cubical Cavity Using Fluent Software, ICHMT 2nd Int. Symp. on Adv. in Comput. Heat Transfer, Palm Cove, Queensland, Australia, May 20– 25, 2001.

[18] E. Krepper, CHT'01: Validation Exercise: Natural Convection in an Air-Filled Cubical Cavity, ICHMT 2nd Int. Symp. on Adv. in Comput. Heat Transfer, Palm Cove, Queensland, Australia, May 20– 25, 2001.

[19] C. Xia, J. Y. Murthy, and S. R. Mathur, Finite Volume Computations of Buoyancy- Driven Flow in a Cubical Cavity: A Benchmarking Exercise, ICHMT 2nd Int. Symp. On Adv. in Comput. Heat Transfer, Palm Cove, Queensland, Australia, May 20– 25, 2001.

Conclusions

- Comparison between running COMSOL 3.5a on 32 bit machine vs. on 64 bit machine

Comparison of flow over backward facing step Re = 800 from COMSOL 3.5a

Number of elements	CPU time (s) (32 bit machine)	CPU time (s) (64 bit machine)	Memory (MB) (32 bit machine)	Memory (MB) (64 bit machine)
Mesh 1: 10,850	3.79	2	211	298
Mesh 2: 22,000	6.958	3	303	350

Comparison of flow over backward facing step Re = 47,648 from COMSOL 3.5a

Initial Number of elements	Initial Number of degrees of freedom	Final Number of elements	Final Number of degrees of freedom	CPU time (s) (64bit)	CPU time (s) (32bit)	Memory (MB) (64bit)	Memory (MB) (32bit)
Mesh 1: 291	2,861	3,876	34,373	52	133.817	233	250
Mesh 2: 585	5,504	8,734	76,701	119	313.45	350	322

Conclusions –cont.

Benchmark case	Number of elements	Number of degrees of freedom	CPU time (s)	Memory cost (MB)	Compared values	
					COMSOL results	Literature data
Case 1-a: Flow over circular cylinder Re = 100	8,568	39,306	3,728	884	C_d = 1.486	C_d = 1.3353 see [5]
	14,965	68,105	14,236	1,193	C_d = 1.485	
Case 1-b: Flow over circular cylinder Re = 1000	8,272	37,974	1,894	974	C_d = 1.69	C_d = 1.47 see [6]
	17,536	79,947	4,024	1,501	C_d = 1.65	
Case 2: Compressible flow in a shock tube	800	9,963	Multi-grid scheme has been applied		Pressure, velocity and density are compared with analytical solution (Fig. 33, 34, 35)	
	4000	48,843				
Case 3-a: Flow over a backward facing step Re = 800	10,850	108,864	2	298	$L_{loweddy}$ = 6.8	$L_{loweddy}$ = 6.1 [12]; $L_{loweddy}$ = 6.0 [13]
	22,000	288,384	3	350	$L_{loweddy}$ = 6.7	
Case 3-b: Flow over a backward facing step Re = 47,648	3,876	34,373	52	233	$L_{loweddy}$ = 6.0	$L_{loweddy}$ = 7.1 [14]; $L_{loweddy}$ = 6.1 [15]
	8,734	76,701	119	350	$L_{loweddy}$ = 6.19	
Case 4-a: Natural convection within a 3D enclosure φ = 90º	8,000	284,945	Multi-grid scheme has been applied		Nu = 3.12	3.10 [18]
Case 4-b: Natural convection within a 3D enclosure φ = 45º	8,000	284,945	Multi-grid scheme has been applied		Nu = 3.54	3.50 [18]
Case 4-c: Natural convection within a 3D enclosure φ = 0º	8,000	284,945	Multi-grid scheme has been applied		Nu = 2.25	2.49-3.92 [18]

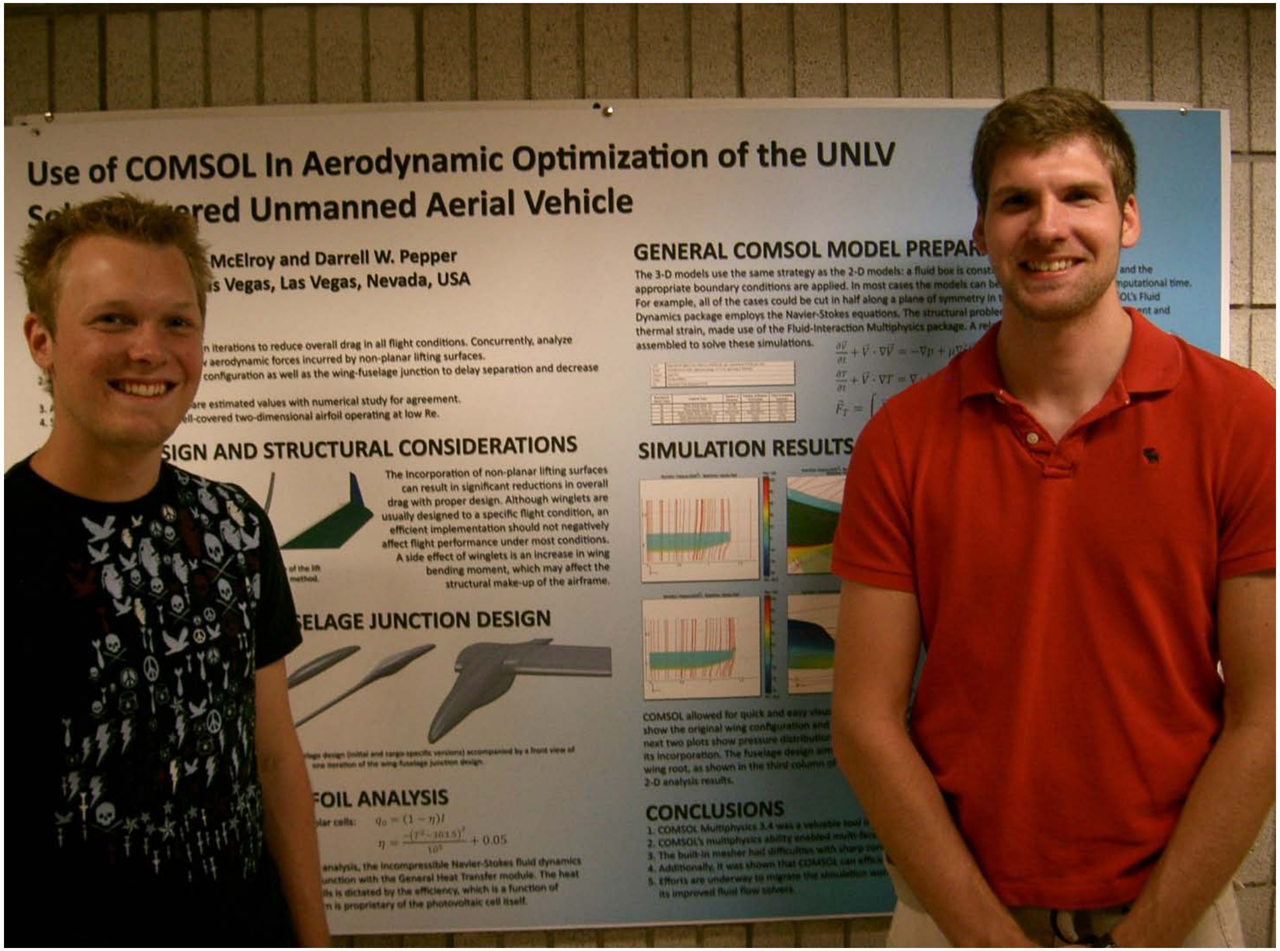
Comparison between XXXXXX, COMSOL and Literature Data

Benchmark case	Number of cells	CPU time (s)		Compared values	
			XXXXXX	COMSOL	Literature
Case 1-a: Flow over circular cylinder Re = 100	16,689 14,955	2,245 3,728 14,236	C_d = 1.479	C_d = 1.485	C_d = 1.3353 see [5]
Case 1-b: Flow over circular cylinder Re = 1000	16,689 17,536	2,425 1,894 4,024	C_d = 1.55	C_d = 1.65	C_d = 1.47 see [6]
Case 2: Compressible flow in a shock tube	40 in x-dir 200 in x-dir		Density, velocity and pressure are compared with analytical solutions (Fig. 9, 10, 11)		
Case 3-a: Flow over a backward facing step Re = 800	12,000	Final report	$L_{loweddy}$ = 6.82	$L_{loweddy}$ = 6.70	$L_{loweddy}$ = 6.1 [12]; $L_{loweddy}$ = 6.0 [13]
Case 3-b: Flow over a backward facing step Re = 47,648	9,600	Final report	$L_{loweddy}$ = 7.0	$L_{loweddy}$ = 6.19	$L_{loweddy}$ = 7.1 [14]; $L_{loweddy}$ = 6.1 [15]
Case 4-a: Natural convection within a 3D enclosure φ = 90º	32,000	Final report	Nu = 3.10	Nu = 3.12	3.10 [18]
Case 4-b: Natural convection within a 3D enclosure φ = 45º	32,000	Final report	Nu = 3.43	Nu = 3.54	3.50 [18]
Case 4-c: Natural convection within a 3D enclosure φ = 0º	32,000	Final report	Nu = 3.43	Nu = 2.25	2.49-3.92 [18]

The Future

- **COMSOL 4**
  - Compare results obtained from COMSOL Multiphysics 4 with those obtained from COMSOL Multiphysics 3.5a and other data
  - Obtain the CPU times and memory costs
  - Try parallel version on Cray CX1
- What’s coming: multiscale, multiphysics, stochastic modeling
- Advances in h-p adaptation; meshless methods

UAV - 2008


Use of COMSOL In Aerodynamic Optimization of the UNLV
ered Unmanned Aerial Vehicle
McElroy and Darrell W. Pepper
s Vegas, Las Vegas, Nevada, USA
GENERAL COMSOL MODEL PREPAR
SIGN AND STRUCTURAL CONSIDERATIONS
The incorporation of non-planar lifting surfaces can result in significant reductions in overall drag with proper design. Although winglets are usually designed to a specific flight condition, an efficient implementation should not negatively affect flight performance under most conditions. A side effect of winglets is an increase in wing bending moment, which may affect the structural make-up of the airframe.
SIMULATION RESULTS
ELAGE JUNCTION DESIGN
FOIL ANALYSIS
CONCLUSIONS

SolidWorks - COMSOL

The Flight of COMSOL- I


38

Contacts

Darrell W. Pepper
NCACM
University of Nevada Las Vegas
dwpepper@nscee.edu

Xiuling Wang
Department of Mechanical Engineering
Purdue University – Calumet
wangx@calumet.purdue.edu