CFD Modelling and Validation of a Cone meter

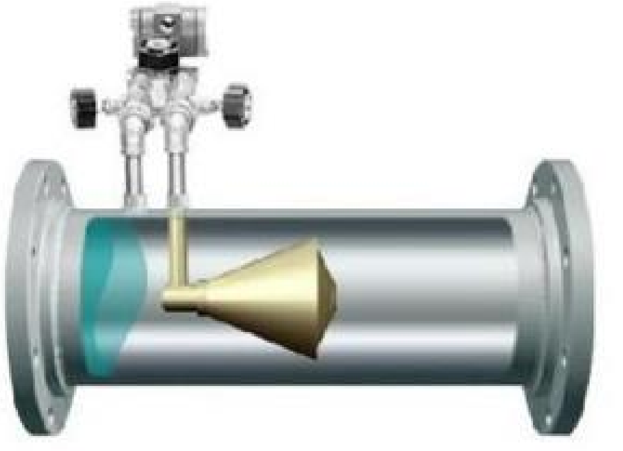
C Haswell[1], C Doyle, W Dempster[1], R Stevens[2]

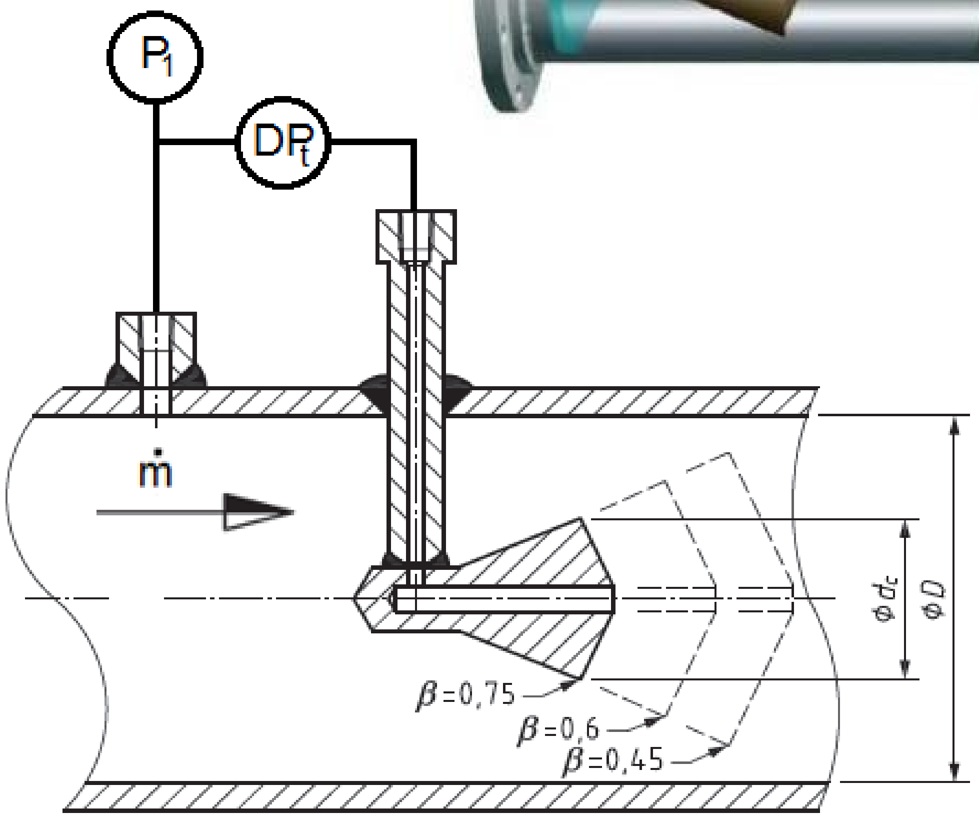
[1] Department of Mechanical and Aerospace Engineering
University of Strathclyde, UK

[2] CEESI Colorado Engineering Experiment Station Inc, USA

Cone Meter Background

University of Strathclyde Engineering





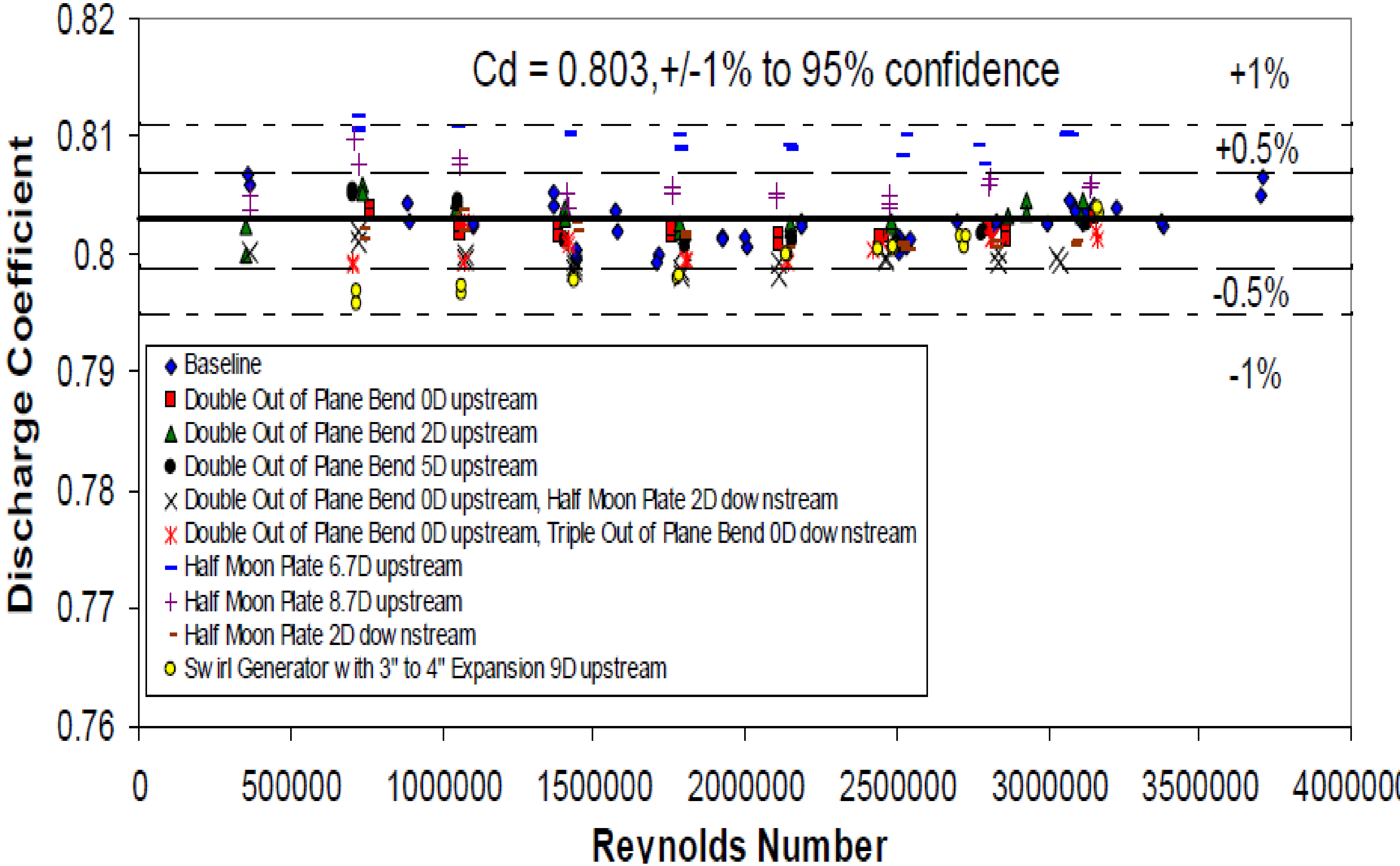
$$m_{actual} = C_d E A_t \varepsilon \sqrt{2\rho \Delta P}$$

$$E = \frac{1}{\sqrt{1-\beta^4}} \qquad \beta = \sqrt{1 - \left(\frac{d_c}{D}\right)^2}$$

$$\varepsilon = 1 - \left(0.649 + \left(0.696\beta^4\right)\right)\left(\frac{\Delta P}{\gamma P_1}\right)$$

$$C_d = f(Re) = f\left(\frac{\rho \bar{U} D}{\mu}\right)$$

Cone Meter Characteristics



Presentation Objectives

- **Can you calibrate a Cone meter using CFD ?**

  -Investigate CFD prediction of Cone C_d accuracy of RANS based approaches.

  -Investigate Reynolds number effects

Methodology

- Use the FLUENT 16 CFD code
- Compare directly with CEESI Meter Cone Test data

Previous Validation Studies

- **Singh–** limited Reynolds number range- RANS-kε models ≈ 4% accuracy

  R. Singh, S. Singh and V. Seshadri, “Study on the effect of vertex angle and upstream swirl on the performance of a V Cone Flow meter using CFD,” *Flow Measurement and Instrumentation,* no. 20, pp. 69-74, 2009

- **Zhu –** RANS-kε (≈ 7%) and LES (≈ 1.54%)

  W. Zhu, J. Meng and Z. Yin, “3-D Numerical Simulation and Analysis on V-cone flowmeter based on LES method,” *Applied Mechanics and Materials,* Vols. 88-89, pp. 408-412, 2011.

Flow Model

- **Reynolds Averaged Navier-Stokes (RANS)** – time independent
- **Geometry** – Three dimensional (symmetric half model)
- **Fluid**- Air with ideal gas equation
- **Solver**- compressible density based solver
- **Turbulence Model-** RNG k-ε Model

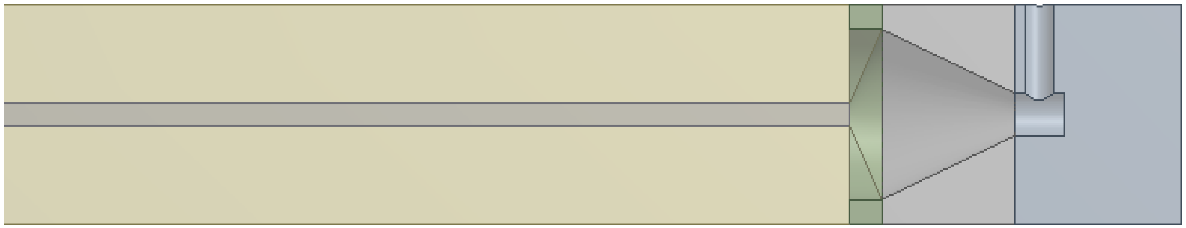
2.5% variation between k-ε models (standard, k-*w*, RNG k-*w*, SS k-w) with RNG- k-ε most accurate

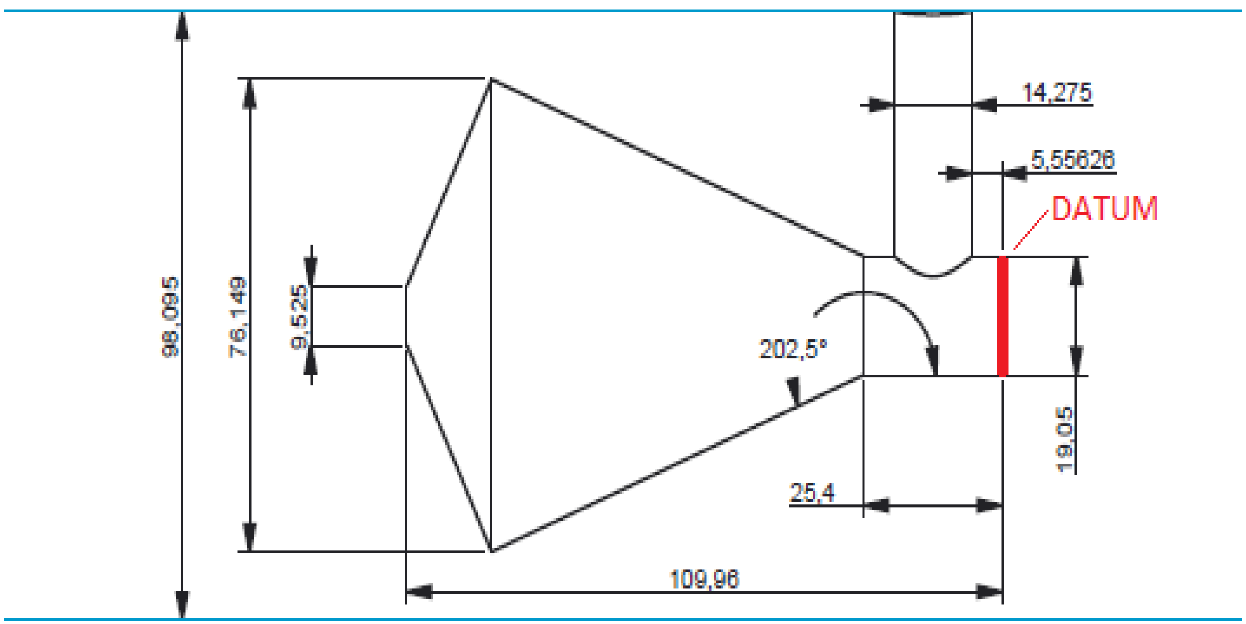
- **Boundary Conditions** : Fixed inlet flow and pressure outlet
- **Wall boundary-** Standard Wall Functions

2.% variation between wall models (standard, scalable, Enhanced) with standard being most accurate

Geometry and Mesh Representations

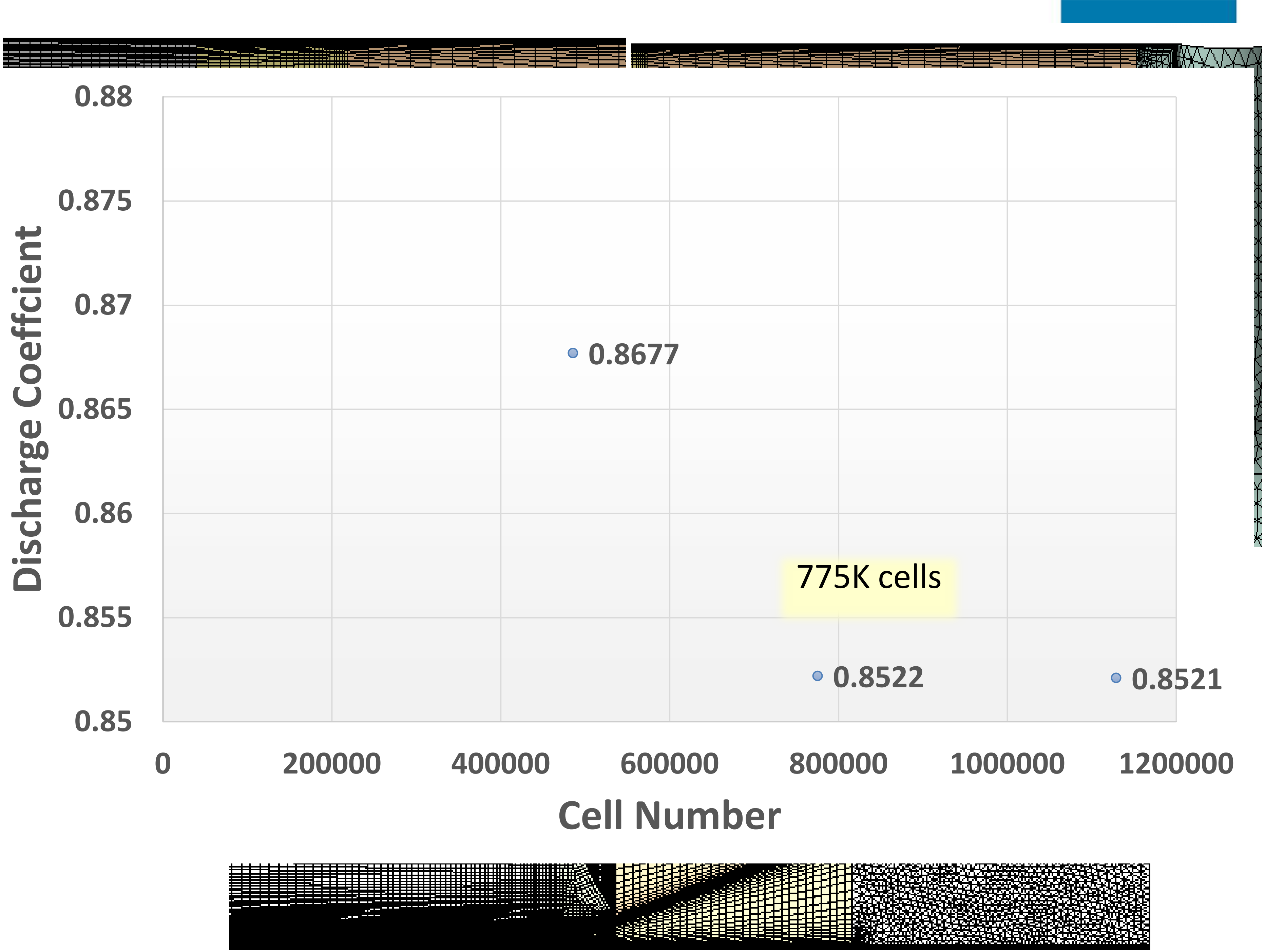
University of Strathclyde Engineering





Key Dimensions

D_{pipe}	98 mm
D_{cone}	76.5 mm
ß	0.63
Length	110 mm

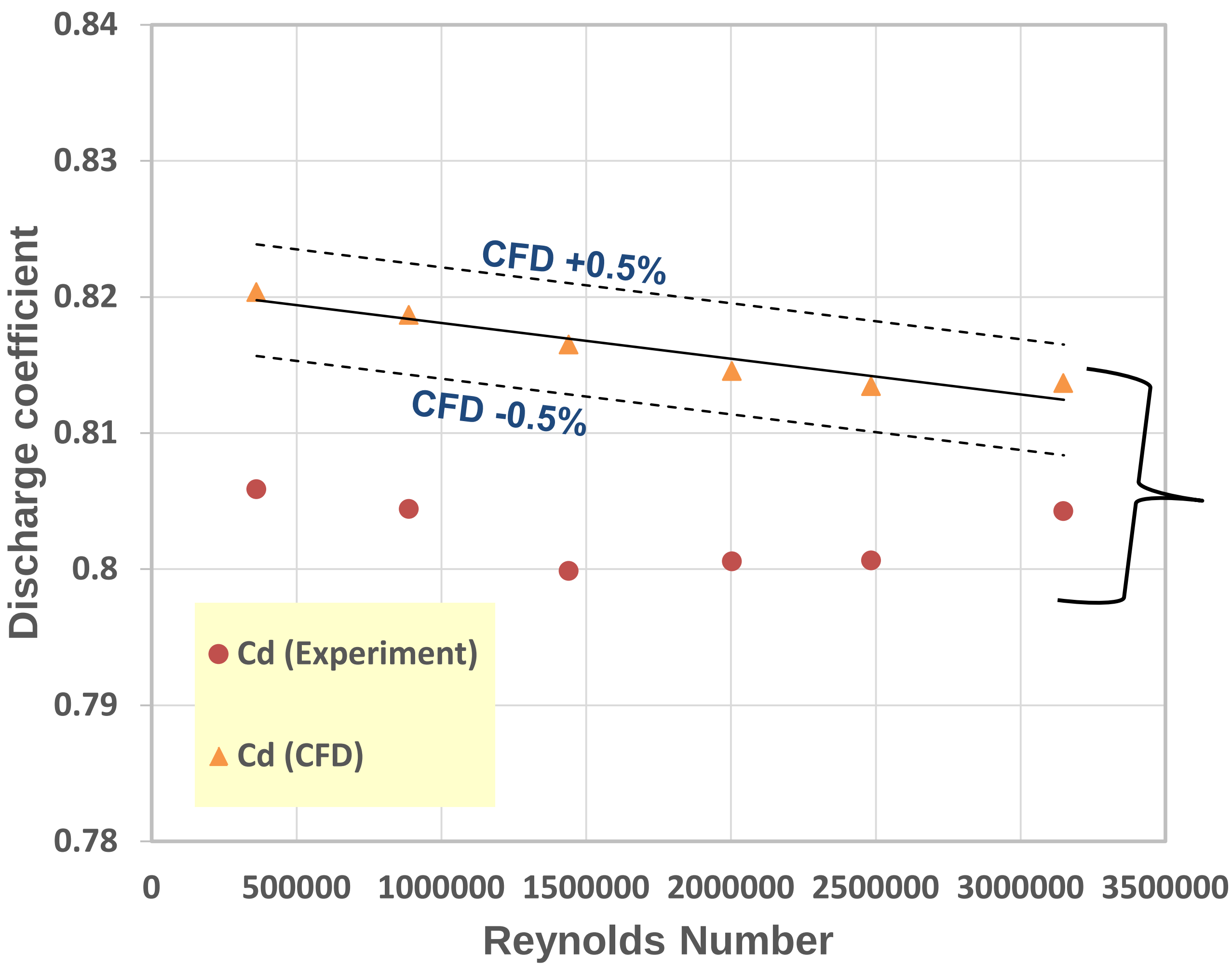


0.88
0.875
0.87
0.865
0.86
0.855
0.85
Discharge Coeffcient
0.8677
775K cells
0.8522
0.8521
0
200000
400000
600000
800000
1000000
1200000
Cell Number

TEST CONDITIONS

Reynold's Number	Inlet Pressure	Inlet Density	Mass Flow	Inlet Speed
	Pa	kg/m^3	kg/s	m/s
362390	1725965	20.632	0.517	3.314
888490	1727688	20.685	1.265	8.095
1441300	1727500	20.595	2.059	13.226
2004000	1729343	20.336	2.891	18.814
2484600	1730860	20.337	3.587	23.339
3148900	1726240	20.765	4.470	28.482

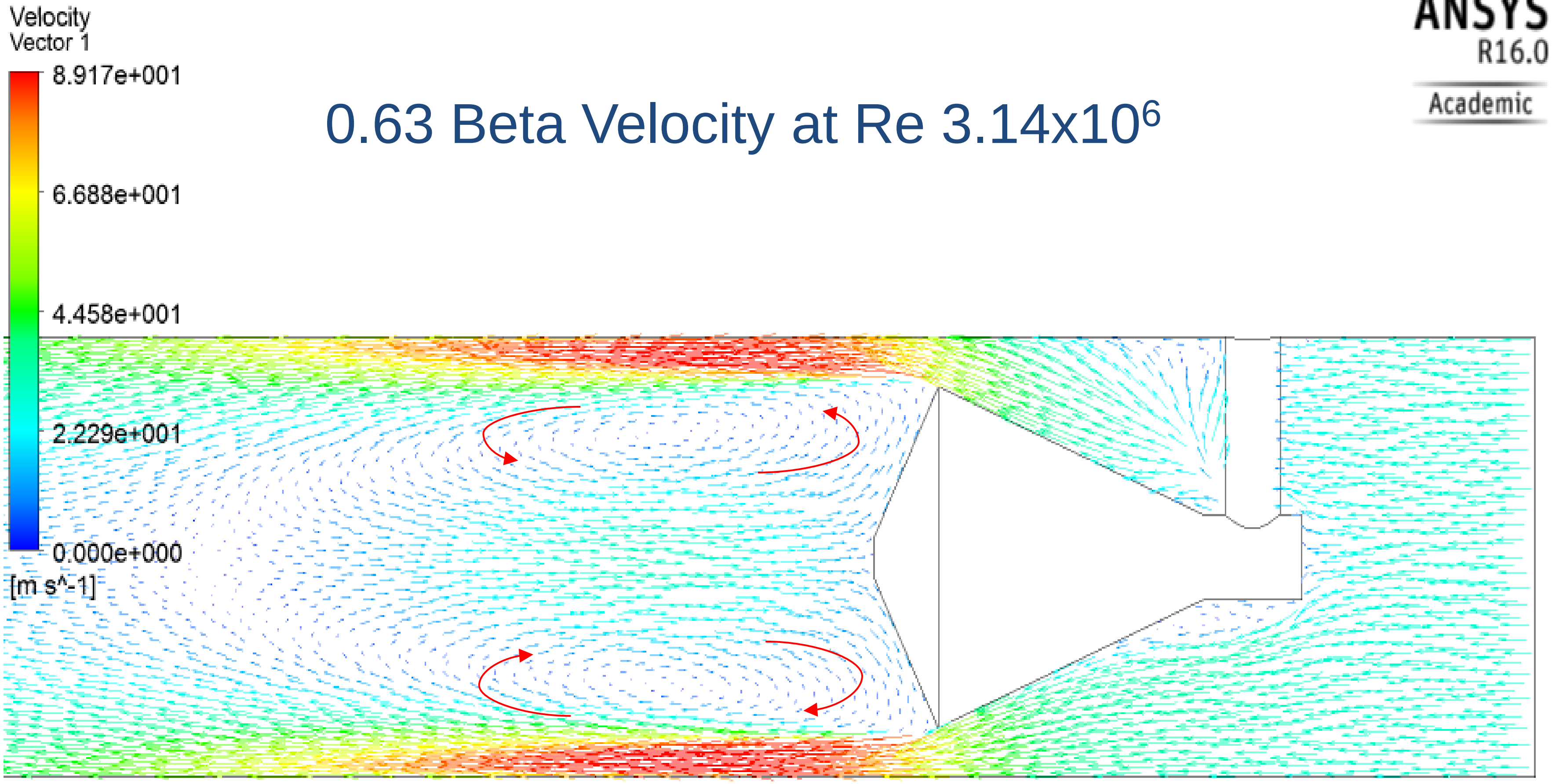
RESULTS



Reynolds Number [E+05]	Discharge Coefficient	CD error(%)
3.62	0.8203	1.79
8.89	0.8187	1.77
14.41	0.8165	2.08
20.04	0.8145	1.74
24.85	0.8134	1.60
31.49	0.8137	1.17

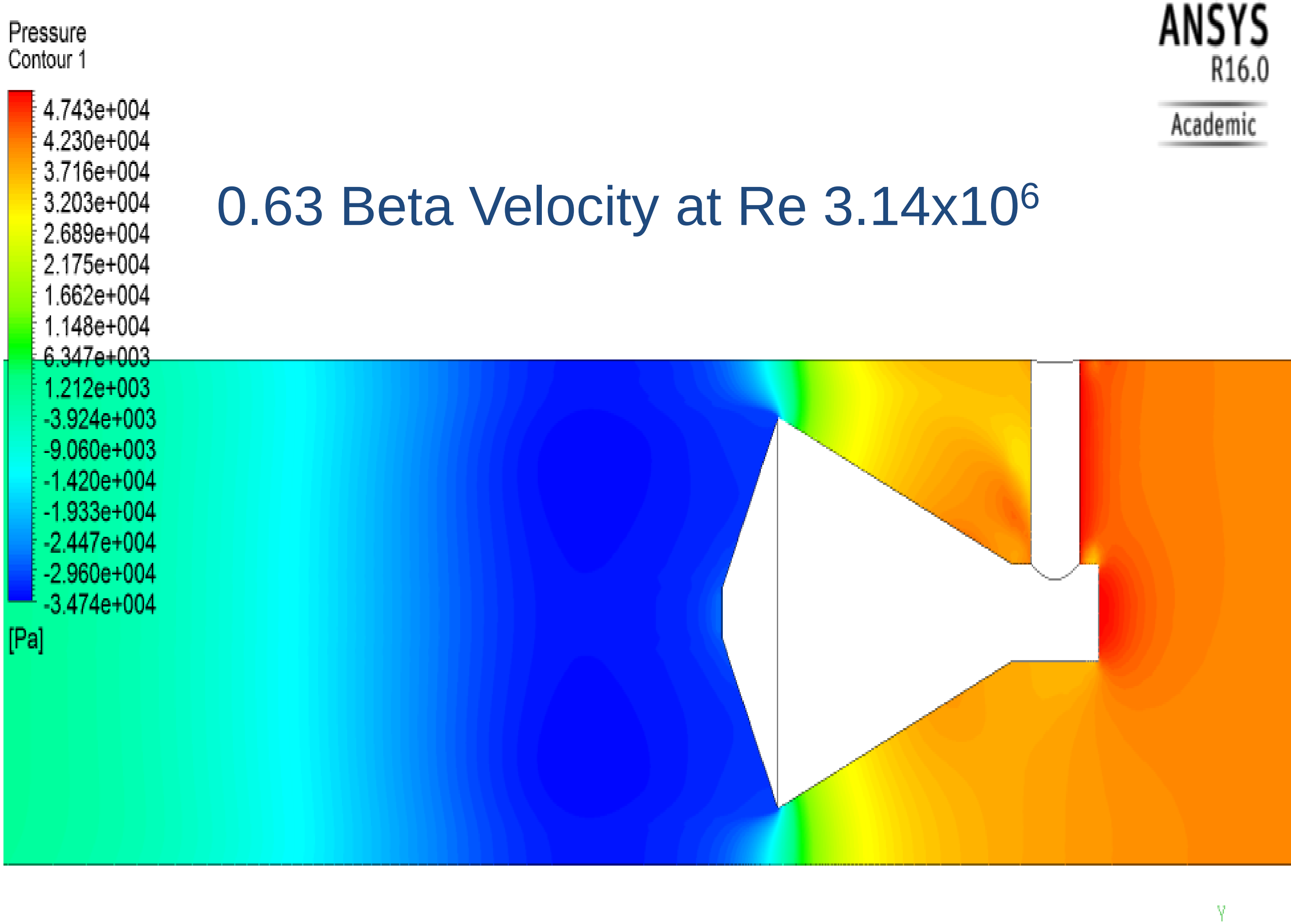
DETAILED RESULTS - VELOCITY

0.63 Beta Velocity at Re 3.14×10^6



DETAILED RESULTS - Pressure

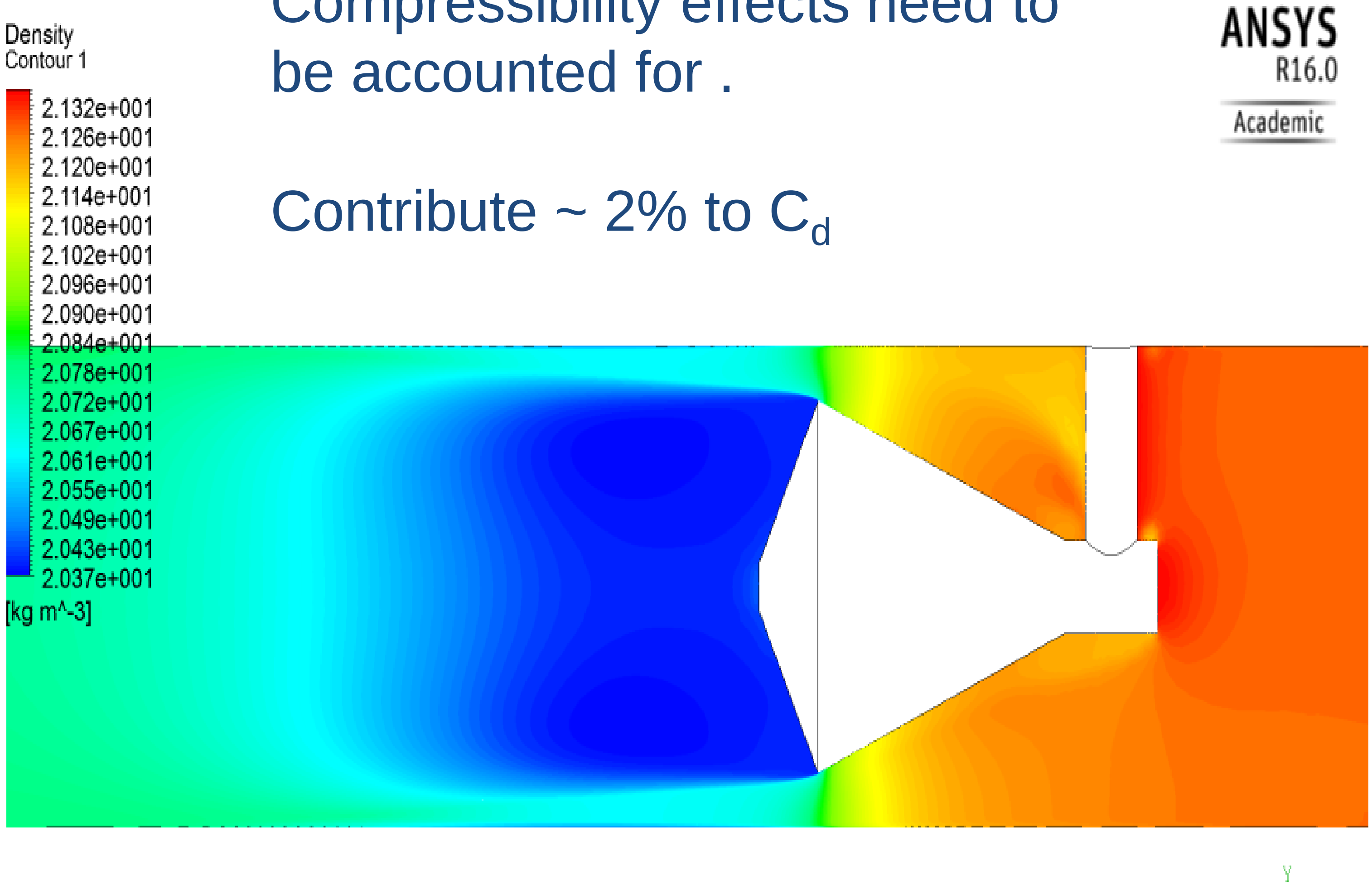
0.63 Beta Velocity at Re 3.14×10^6



Points to note - compressibility

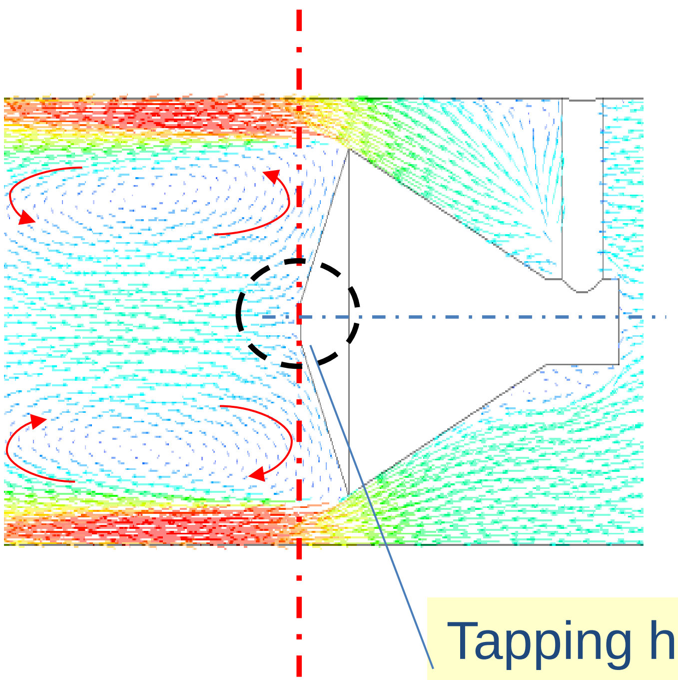
Compressibility effects need to be accounted for .

Contribute ~ 2% to C_d



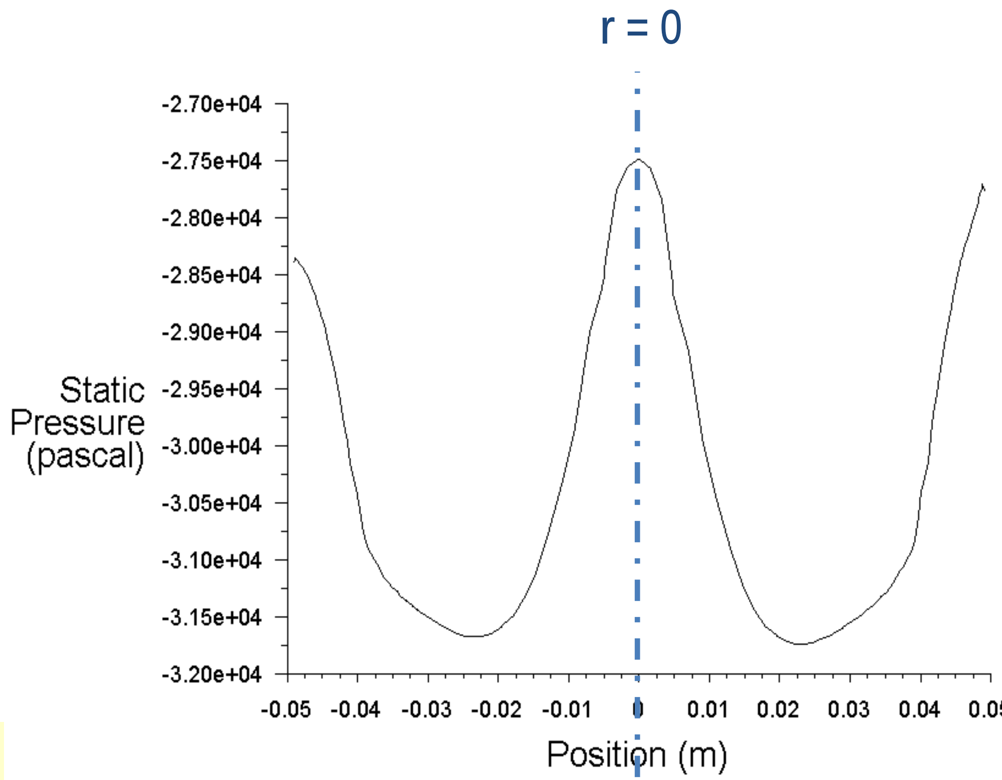
Points to note-

Dowstream pressure profile /tappings



r = 0

Tapping holes not modelled – may need to be



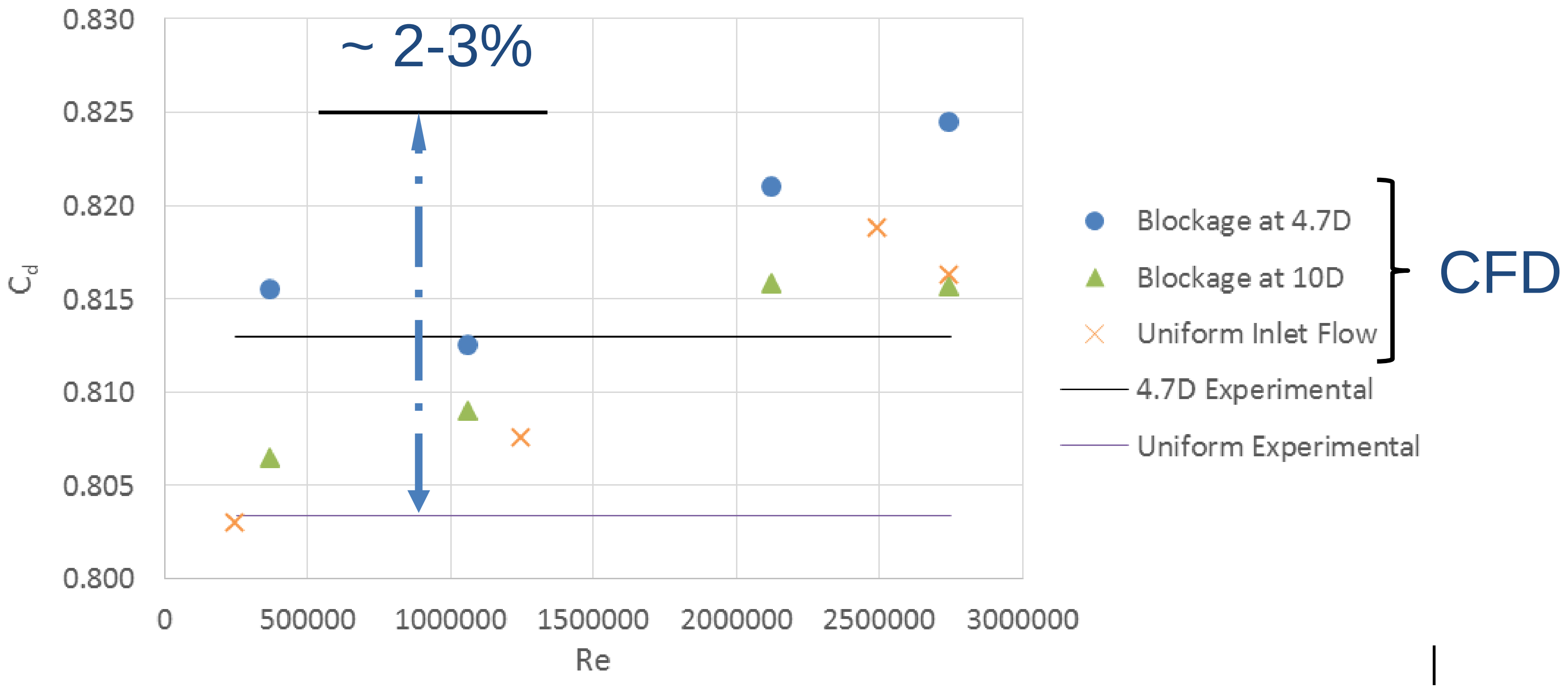
Pressure variation across plane

Conclusions

- Steady RANS with RNG k-e models can predict C_d coefficients for Cones Meters to < 2%
- The dependency of the Cd on Reynolds Number can be predicted reasonably correctly.
- It is speculated that additional accuracy may be achieved by considering detailed modelling of downstream pressure tapping

Final note

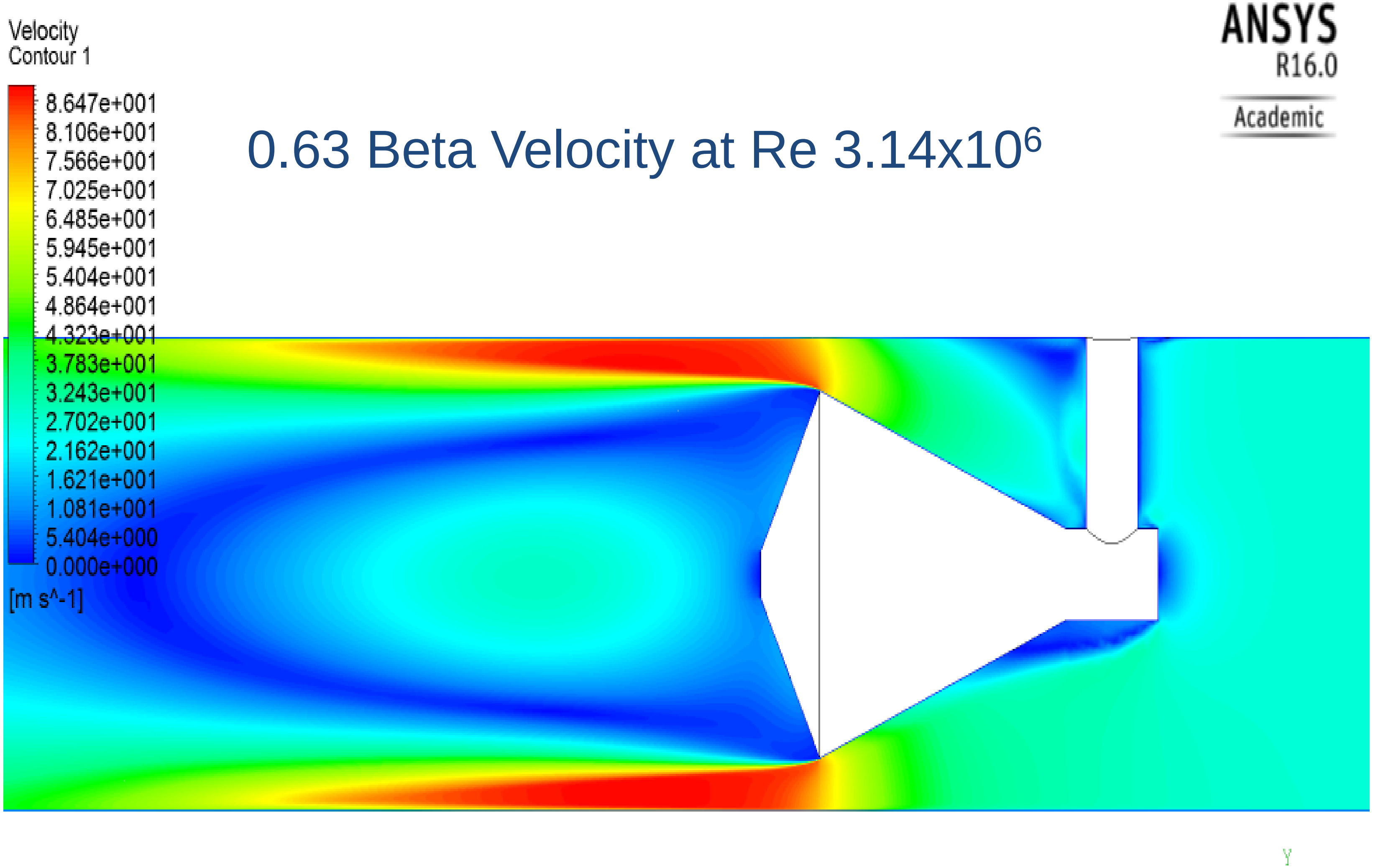
Can CFD predict C_d for non uniform upstream flows As accurately as for uniform inlet conditions ?



Preliminary results – say yes

DETAILED RESULTS

0.63 Beta Velocity at Re 3.14x10^6



ABSTRACT

Cone meters , a differential pressure flow meter device have been found to be resilient to upstream flow disturbances and hence more suitable for complex pipe geometries than traditional differential meters such orifice plates and venturi nozzles.

In this paper the computational fluid dynamic modelling of a Cone meter is studied and the ability to accurately predict the flow meter discharge coefficients is presented. The open literature suggests that prediction can be achieved to 4-5 % using RANS based two equation turbulence models and that greater accuracy to < 2% require using LES techniques. In this paper a 4” Cone , beta ratio 0.63 is studied over a Reynolds number range of 3.6×10^5 – 3.14×10^6 in a pressurised gas flow line.

This paper will show that by appropriate meshing and accounting for compressibility that RANs based approaches can achieve an accuracy of less than 2%, establishing CFD as a suitable vehicle for flow meter design.