N95- 23634

S19-34

On the Accuracy of CFD-Based Pressure Drop Predictions for Right-Angle Ducts

p. 13

A. Brankovic
Pratt & Whitney, Florida

The predictive capability of computational fluid dynamics (CFD) codes for turbulent flow through curved ducts is of significant importance to the design and performance analysis of modern rocket engine flowpaths. Code calibration and validation studies for this class of flow are desireable to estimate the performance margin and operating range of components designed using Navier-Stokes methods. Parametric experimental studies such as that of Weske (NACA ARR W-39) provided a wealth of performance data for the design of single- and compound elbow configurations with various cross-sections, curvature and aspect ratios at varying Reynolds numbers. In that work, the majority of data is presented in the form of loss coefficients, characterizing pressure losses due to duct curvature, and including losses due to wall friction. Using measured friction coefficients, losses of equivalent straight lengths of duct are subtracted, resulting in performance curves useful for design computations. These data are currently used in a CFD-based parametric study covering a broad range of operating conditions. Of particular interest for the accuracy of CFD predictions are the effects on pressure loss due to inlet boundary layer thickness (dependent on upstream development length), and the wall treatment for the turbulence equations (conventional wall functions vs. wall integration using a two-layer model). The experimental data are reassessed in the form of an error analysis, and are compared with CFD predictions for 18 computational cases. Grid-independence, grid spacing, and convergence requirements of the cases are discussed. Conclusions regarding the relative importance of the parametric variables will be presented.

ON THE ACCURACY OF CFD−BASED PRESSURE DROP PREDICTIONS FOR RIGHT−ANGLE DUCTS

by

Andreja Brankovic

Pratt & Whitney, Florida

Presented at

Workshop for CFD Applications

in Rocket Propulsion

NASA Marshall Space Flight Center

April 20−22, 1993

INTRODUCTION / OBJECTIVE

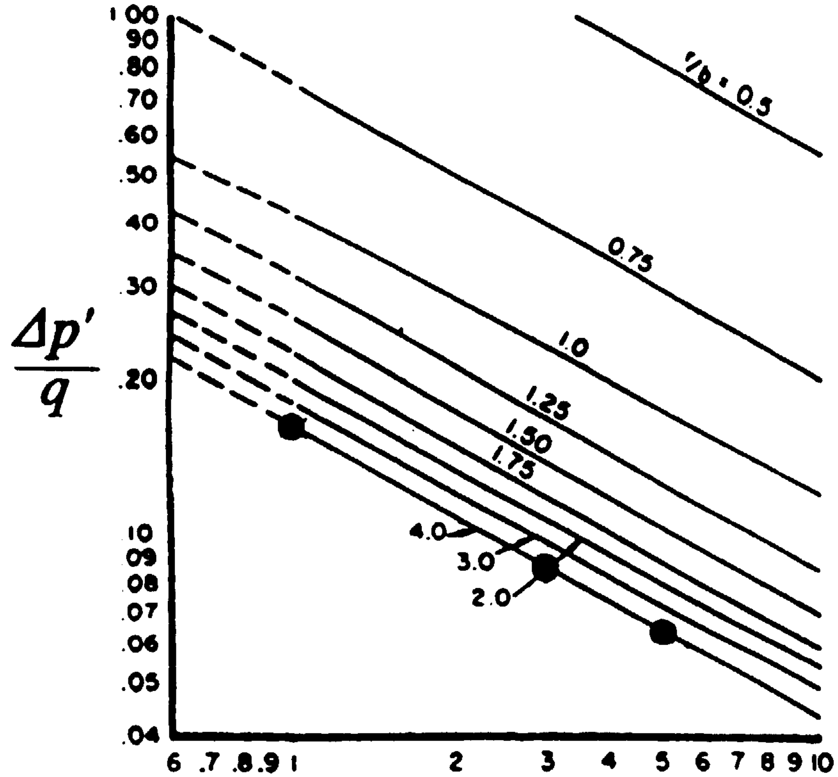
- Physics of flow in a right–angle ducts has much in common to that in a rocket engine flowpaths, i.e., large pressure loss, secondary flows, possible separation.

- Experimental pressure loss data have long been used for design of high performance single and compound elbow duct systems.

- Data from these early measurement programs, obtained over a wide range of operating conditions, provide an excellent source for CFD target data suitable for studies of B.C.'s, grid resolution, wall functions vs. integration to wall, etc.

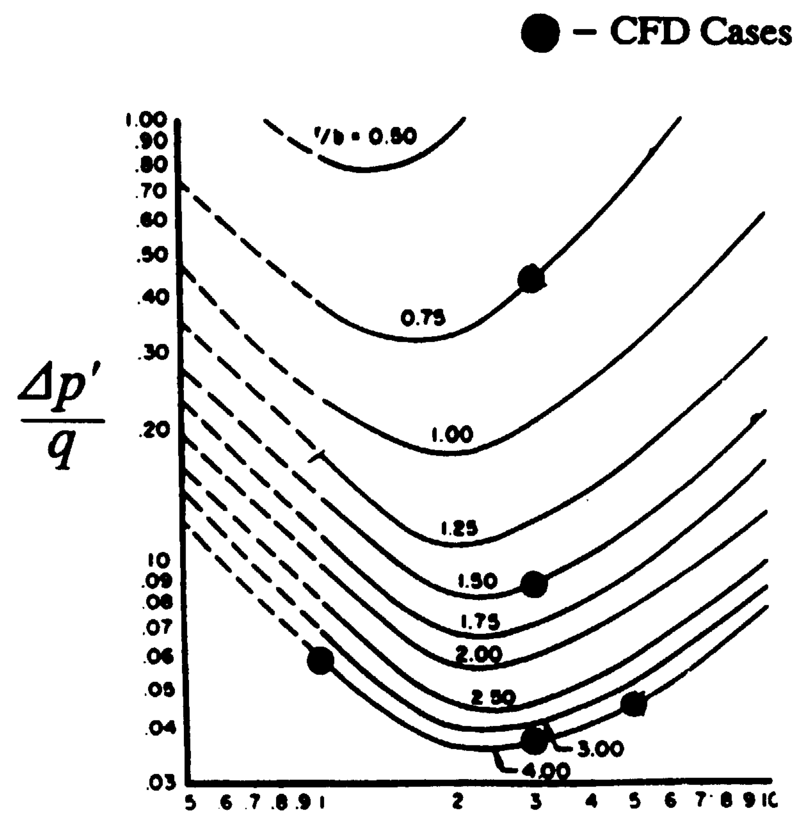
- Present CFD test matrix designed to investigate experimentally–observed trends of pressure losses in right–angle ducts, due to varying Re #, duct aspect and curvature ratio.

EXPERIMENTAL DATASETS



Aspect Ratio = a/b

$R_e = 10^5$

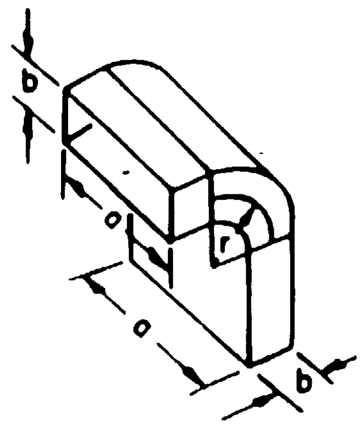


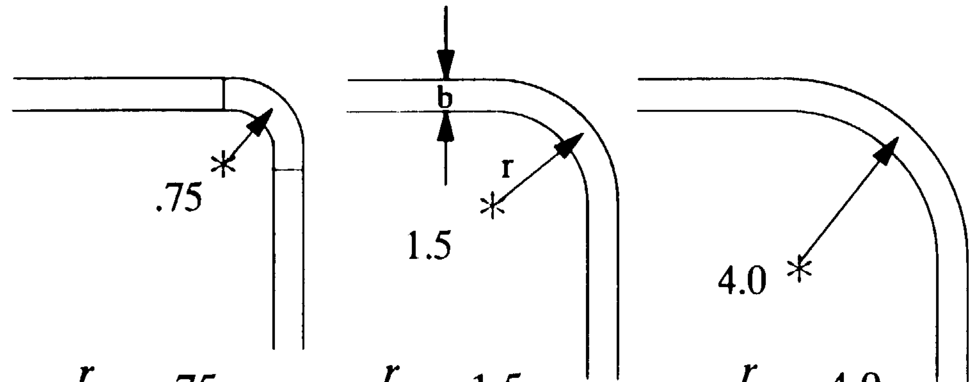
Aspect Ratio = a/b

$R_e = 6 X 10^5$

Reference: Weske, J.R., "Pressure Loss in Ducts with Compound Elbows", NACA Wartime Report W–39 February 1943.

EXPERIMENTAL GEOMETRY

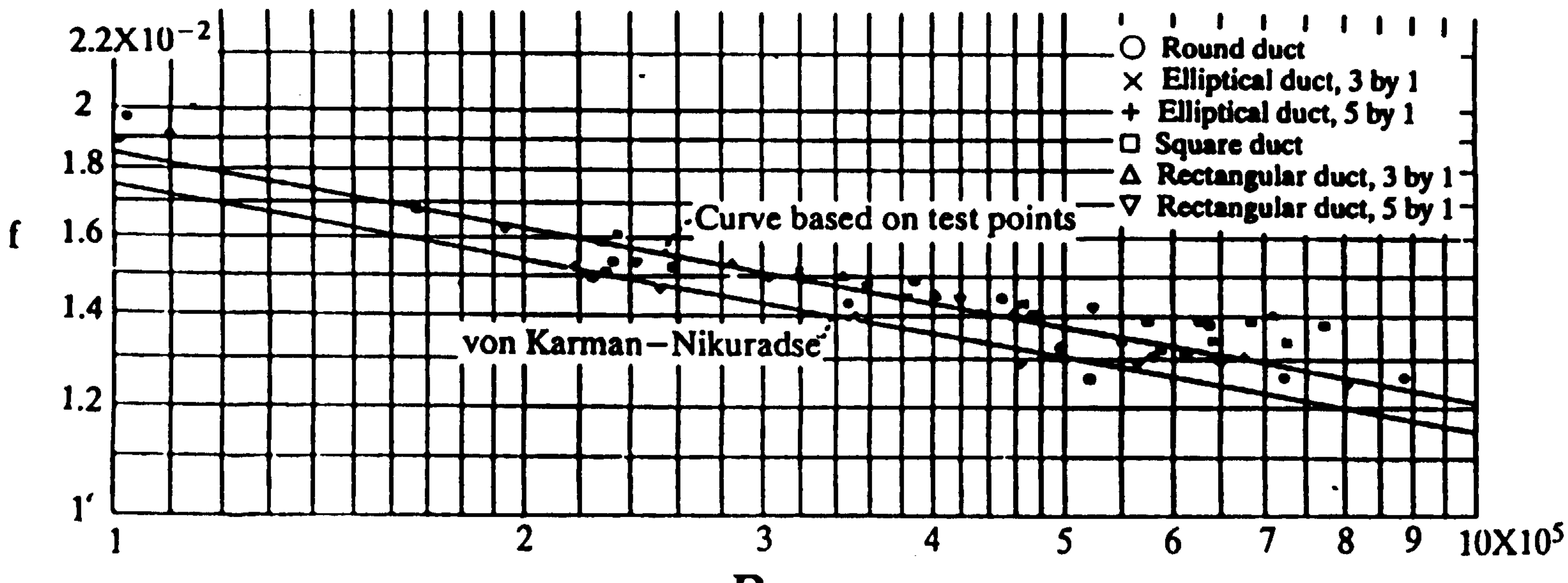




$\frac{r}{b} = .75$ $\frac{r}{b} = 1.5$ $\frac{r}{b} = 4.0$

		Aspect Ratio (a/b)	Curvature Ratio (r/b)	Dimensions (a x b)	Hydraulic Diameter (D_H)	Mean Radius (R)
$R_e = 10^5$						
	1.	1:1	4.0	5.125″ x 5.125″	5.125″	20.75″
	2.	3:1	4.0	9.1875″ x 3.06″	4.5909″	12.25″
	3.	5:1	4.0	11.875″ x 2.375″	3.9583″	9.5″
$R_e = 6 \times 10^5$						
	4.	1:1	4.0	5.125″ x 5.125″	5.125″	20.75″
	5.	3:1	4.0	9.1875″ x 3.06″	4.5909″	12.25″
	6.	3:1	1.5	9.1875″ x 3.06″	4.5909″	4.6″
	7.	3:1	0.75	9.1875″ x 3.06″	4.5909″	2.3″
	8.	5:1	4.0	11.875″ x 2.375″	3.9583″	9.5″

DATA REDUCTION METHOD



$$\frac{\Delta P'}{q} = \frac{\Delta P}{q} - f\left[\frac{L_1 + L_2 + L_c}{D_H}\right]$$

where: $\frac{\Delta P'}{q}$ = *pressure coefficient*

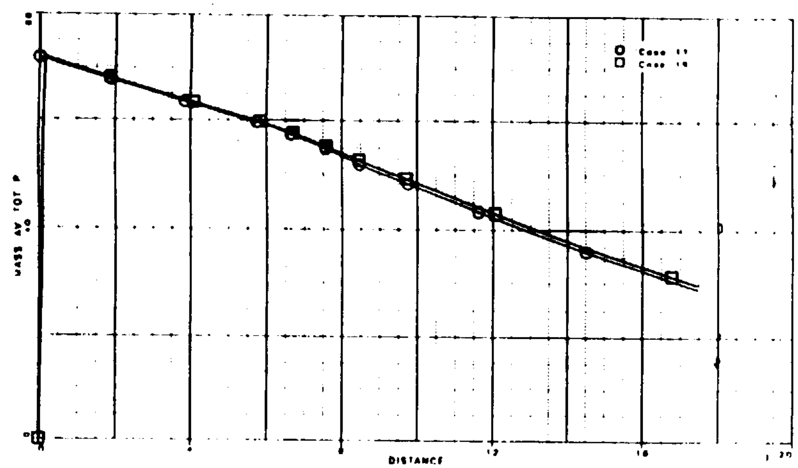
$q = .5\varrho Q^2$

L_1, L_2, L_c = *upstream, downstream, curve lengths*

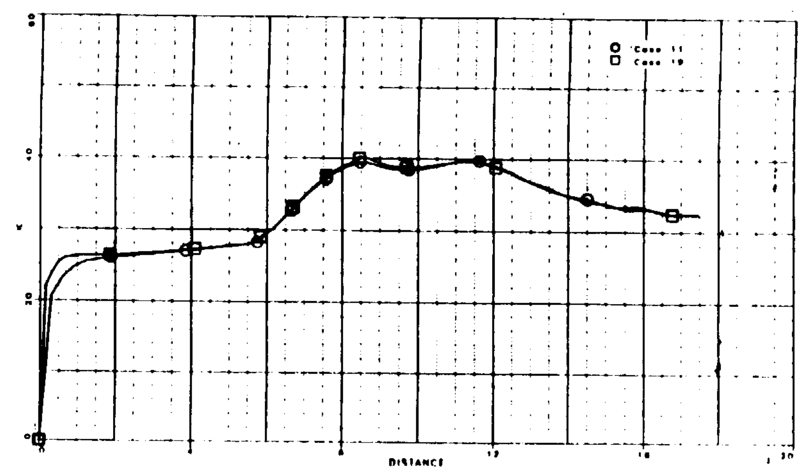
f = *friction factor*

CFD NUMERICS

Grid Independence



Total Pressure



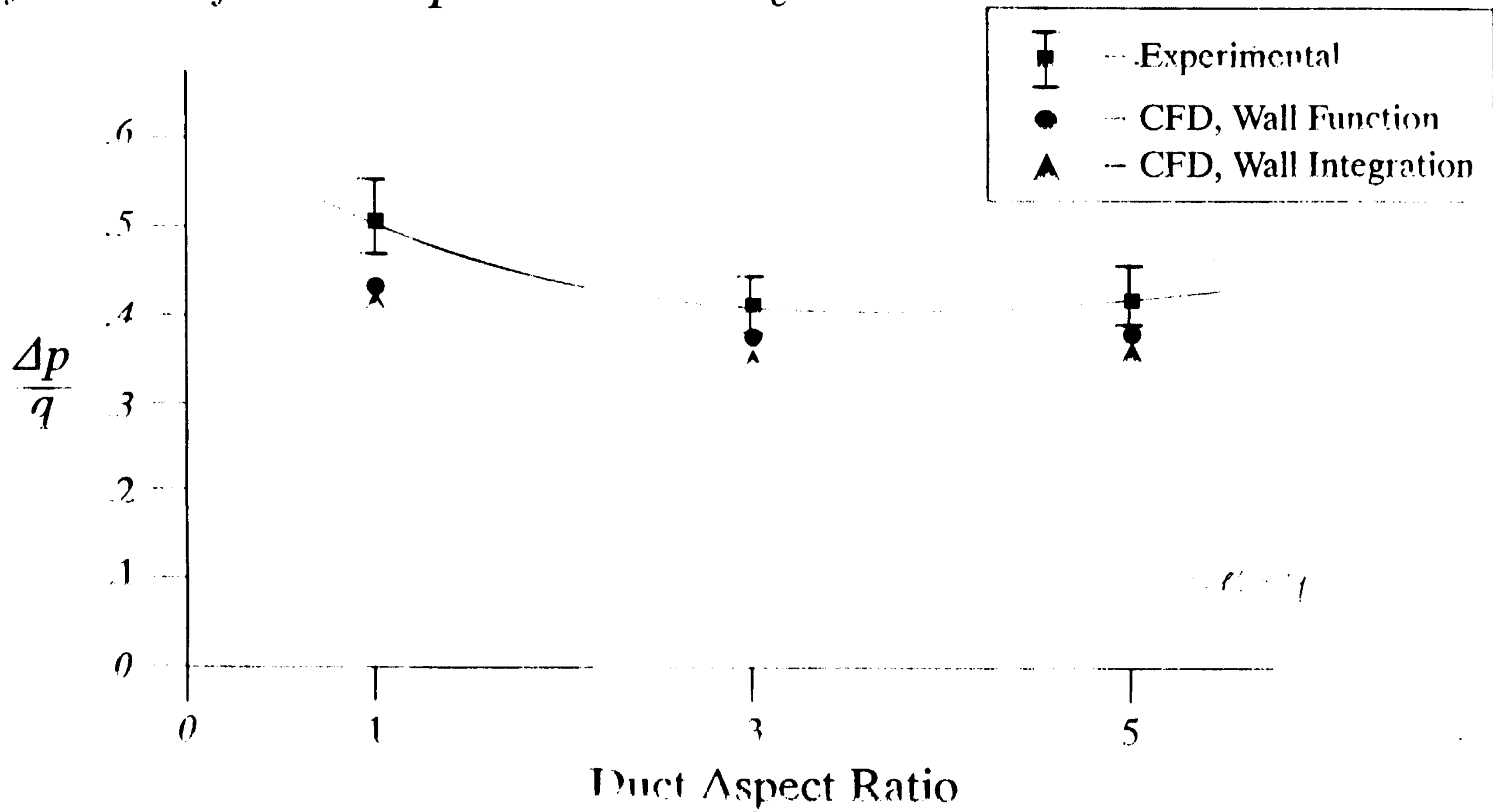
Turbulent Kinetic Energy

Coarse Mesh: 59 x 40 x 30 = 70,800 Points
Fine Mesh: 92 x 80 x 60 = 441,600 Points

Similar results obtained for wall function and wall integration models

PREDICTED vs. MEASURED RESULTS

Influence of Duct Aspect Ratio at $R_e = 10^5$

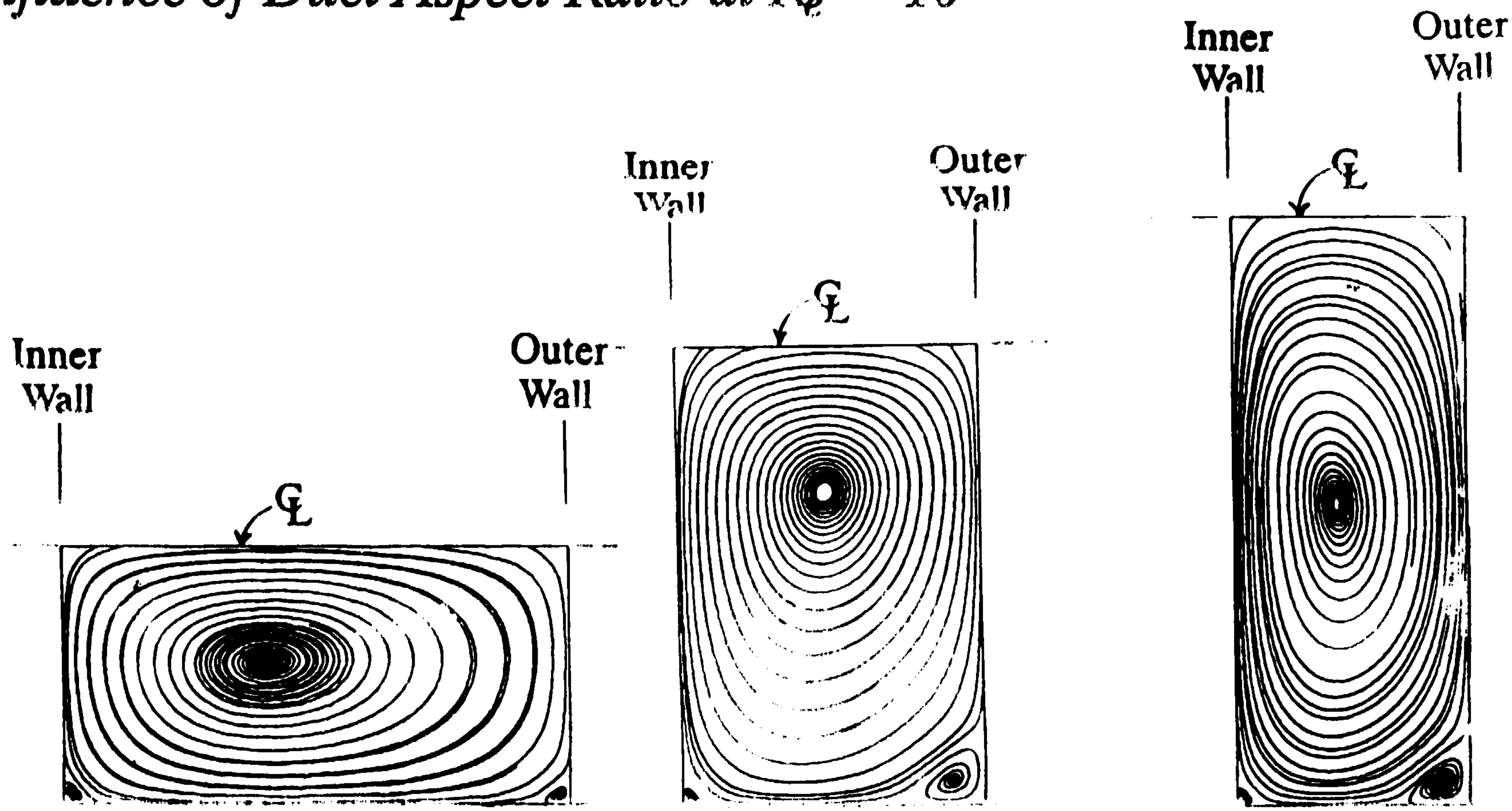


Data shows distinct minimum when plotted as

$$\frac{\Delta p}{q} \quad \text{instead of} \quad \frac{\Delta p'}{q}$$

COMPUTED FLOW STRUCTURE

Influence of Duct Aspect Ratio at $R_e = 10^5$



AR = 1:1
CR = 4.0

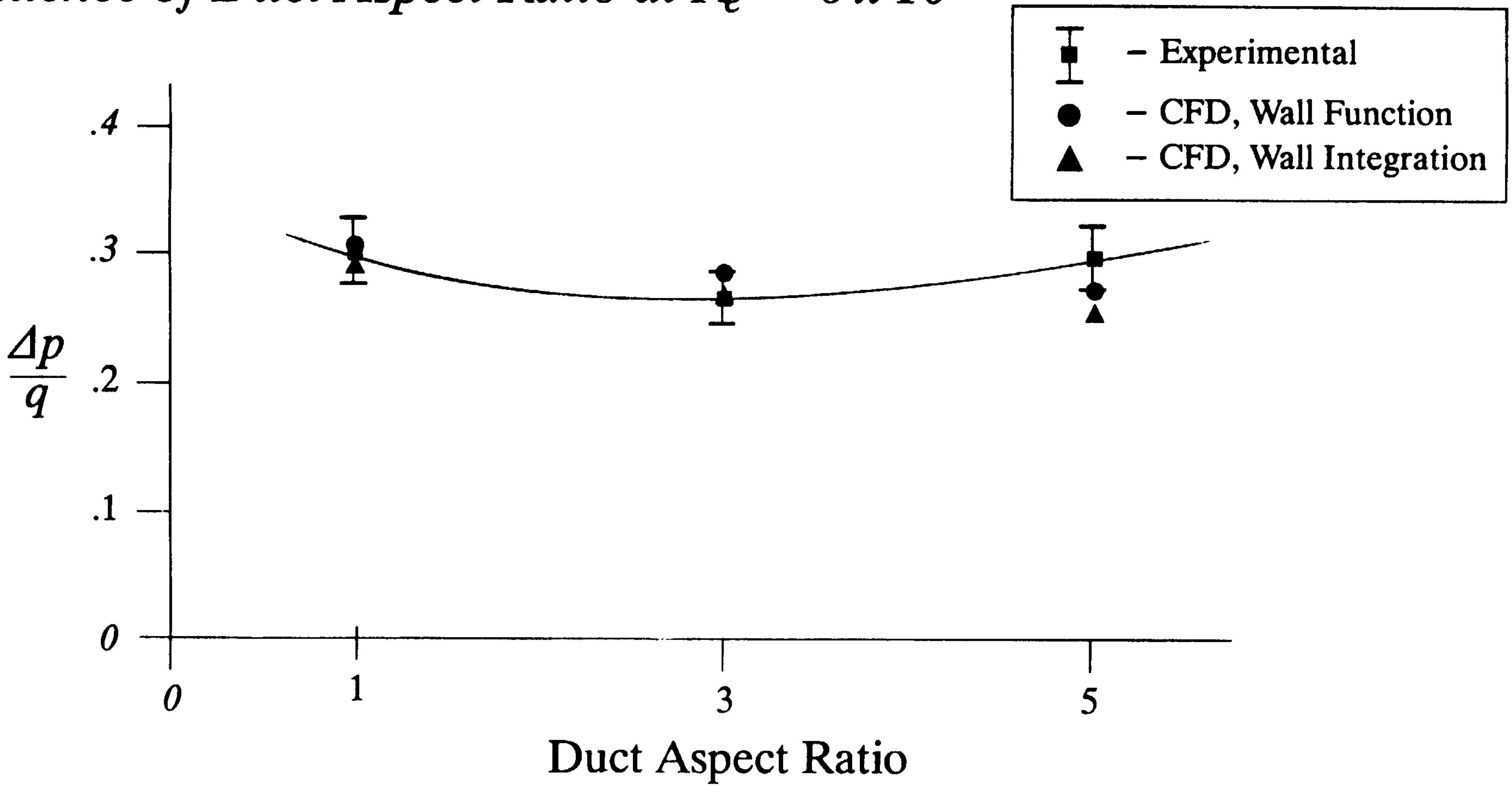
AR = 3:1
CR = 4.0

AR = 5:1
CR = 4.0

Shown at 48″ downstream of elbow exit plane

PREDICTED vs. MEASURED RESULTS

Influence of Duct Aspect Ratio at $R_e = 6 \times 10^5$

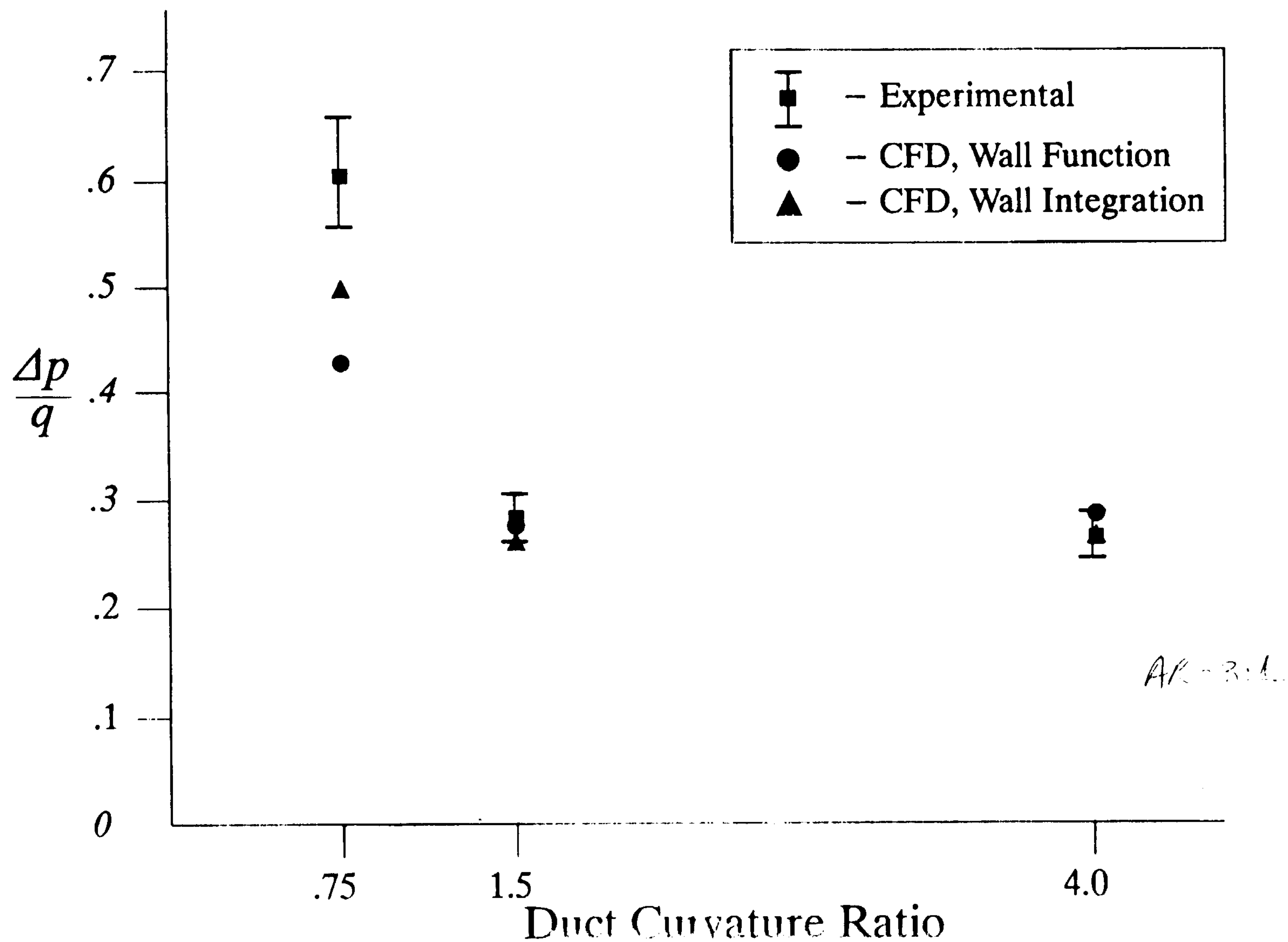


Minimum in data not as pronounced when plotted as

$$\frac{\Delta p}{q} \text{ instead of } \frac{\Delta p'}{q}$$

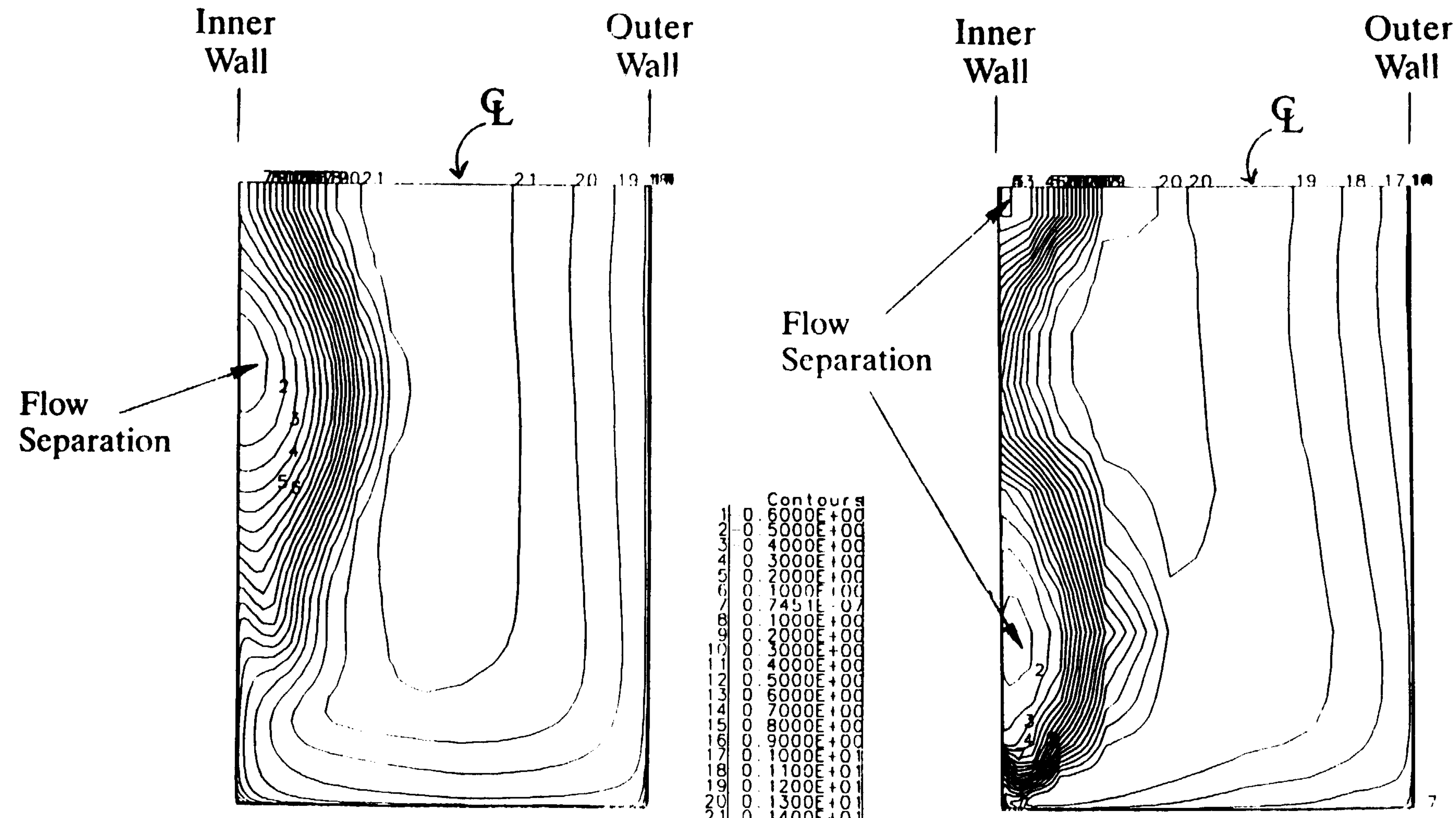
PREDICTED vs. MEASURED RESULTS

Influence of Duct Curvature Ratio at $R_e = 6 \times 10^5$



FLOW SEPARATION DETAILS

Turbulence Model Influences Shape, Location Of Separation Bubble



(a) Wall Function

(b) Wall Integration

Details of separated flow region for CR = 0.75, AR = 3:1, $R_e = 6 \times 10^5$.

CONCLUSIONS

- Details of experimental setup, uncertainty estimates, data reduction method critical to appropriate modeling of cases.

- CFD predictions usually within 15% of data for pressure drop across elbow; larger error associated with low curvature elbow due to flow separation.

- Wall function $k-\varepsilon$ model as good or better than wall integration model for loss calculations.

- Reynolds number effects shown in data are apparent, not real; variation of pressure loss due to duct aspect and curvature ratios has been captured by CFD model.