FLOW AND HEAT TRANSFER IN 180-DEGREE TURN SQUARE DUCTS - EFFECTS OF TURNING CONFIGURATION AND SYSTEM ROTATION

Ten-See Wang
Computational Fluid Dynamics Branch
NASA - Marshall Space Flight Center
Marshall Space Flight Center, AL 35812

Ming-King Chyu
Department of Mechanical Engineering
Carnegie Mellon University
Pittsburgh, PA 15213

ABSTRACT

Forced flow through channels connected by sharp bends is frequently encountered in various rocket and gas turbine engines. For example, the transfer ducts, the coolant channels surround the combustion chamber, the internal cooling passage in a blade or vane, the flow path in the fuel element of a nuclear rocket engine, the flow around a pressure relieve valve piston, and the recirculated base flow of multiple engine clustered nozzles. Transport phenomena involved in such a flow passage are complex and considered to be very different from those of conventional turning flow with relatively mild radii of curvature. While previous research pertaining to this subject has been focused primarily on the experimental heat transfer, very little analytical work is directed to understanding the flowfield and energy transport in the passage. Therefore, the primary goal of this paper is to benchmark the predicted wall heat fluxes using a state-of-the-art computational fluid dynamics (CFD) formulation against those of measurement for a rectangular turn duct. Other secondary goals include studying the effects of turning configurations, e.g., the semi-circular turn, and the rounded-corner turn, and the effect of system rotation. The computed heat fluxes for the rectangular turn duct compared favorably with those of the experimental data. The results show that the flow pattern, pressure drop, and heat transfer characteristics are different among the three turning configurations, and are substantially different with system rotation. Also demonstrated in this work is that the present computational approach is quite effective and efficient and will be suitable for flow and thermal modeling in rocket and turbine engine applications.

Flow and Heat Transfer in 180-Degree Turn Square Ducts - Effects of Turning Configuration and System Rotation

Ten-See Wang

Computational Fluid Dynamics Branch

NASA-Marshall Space Flight Center

Ming-King Chyu

Department of Mechanical Engineering

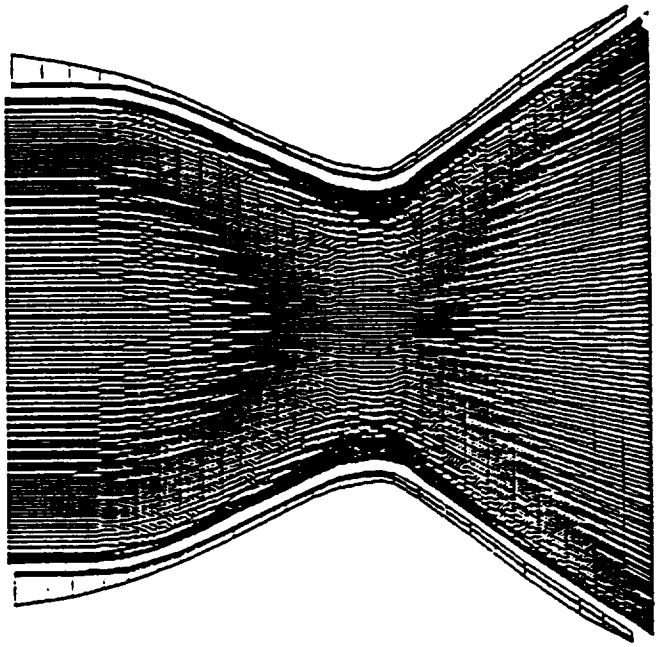
Carnegie Mellon University

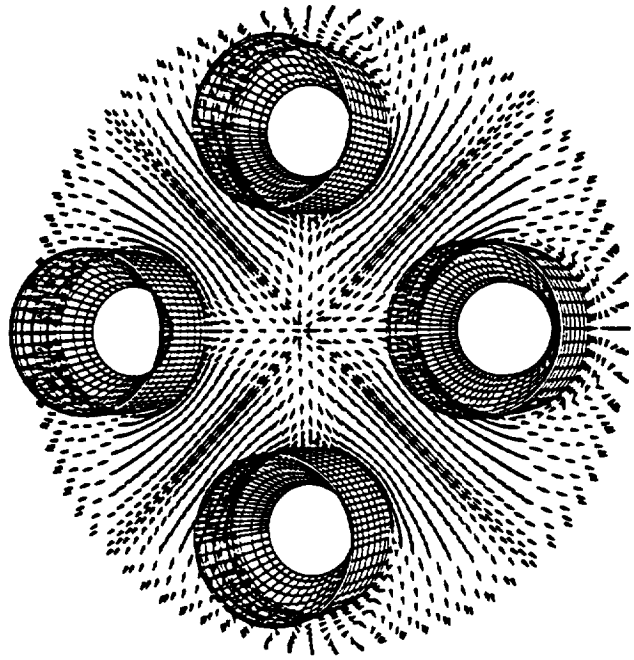
11th Workshop for CFD in Rocket Propulsion

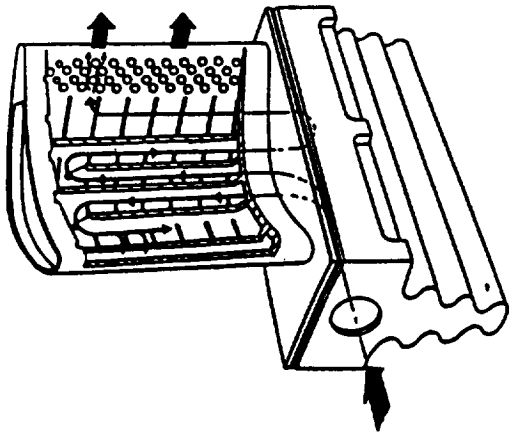
Main Sessions 5: Applications - Duct Flows

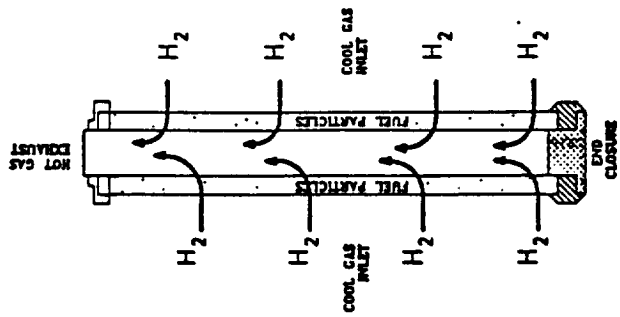
MSFC, Alabama

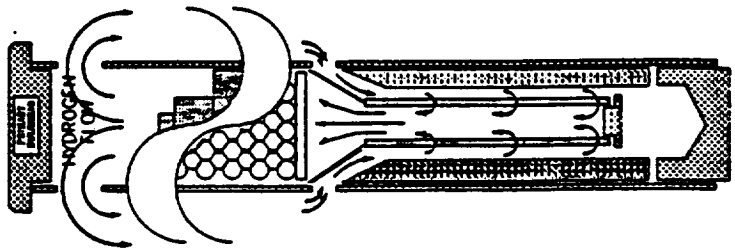
April 20, 1993








H_2
H_2
COOL GAS INLET
H_2
H_2
FUEL PARTICLES
HOT GAS EXHAUST
END CLOSURE
FUEL PARTICLES
H_2
H_2
COOL GAS INLET
H_2
H_2

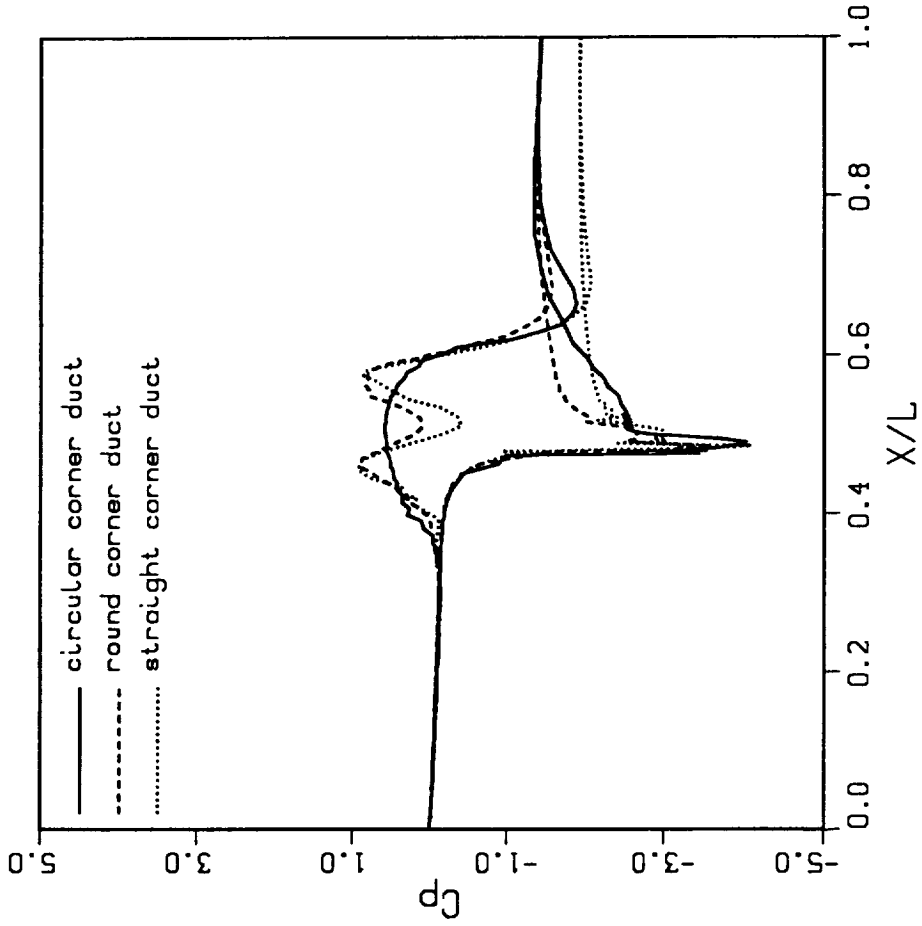

HYDROGEN FLOW

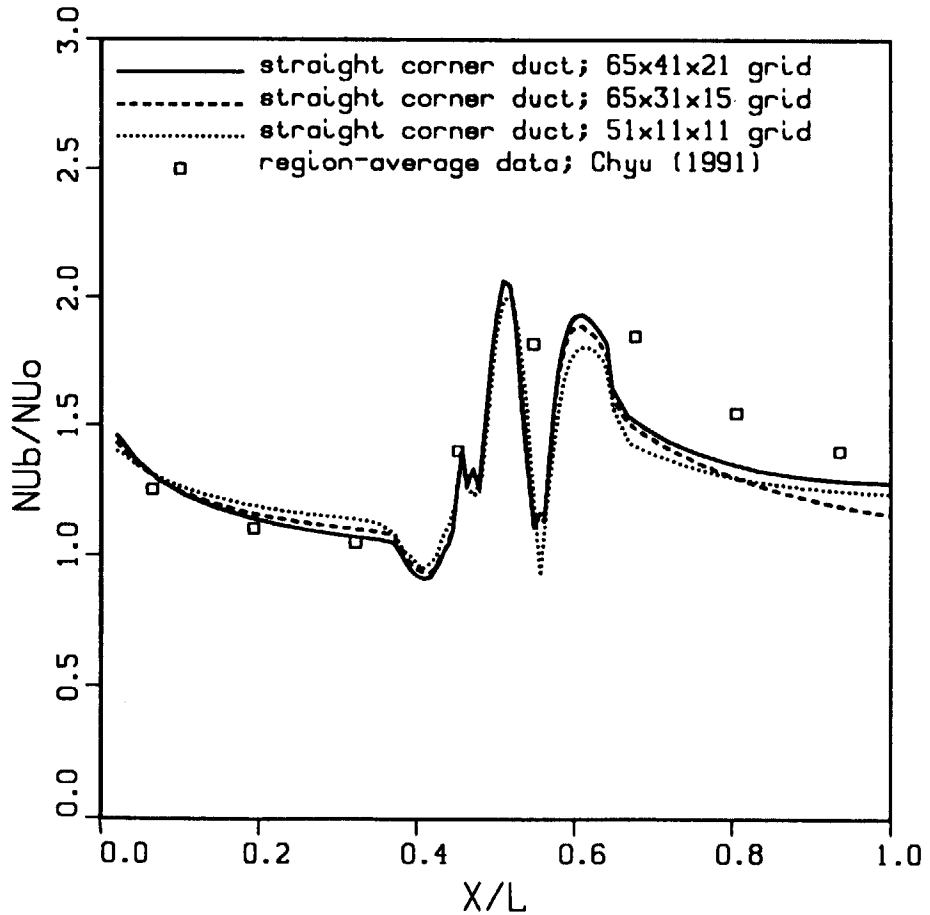
Objectives

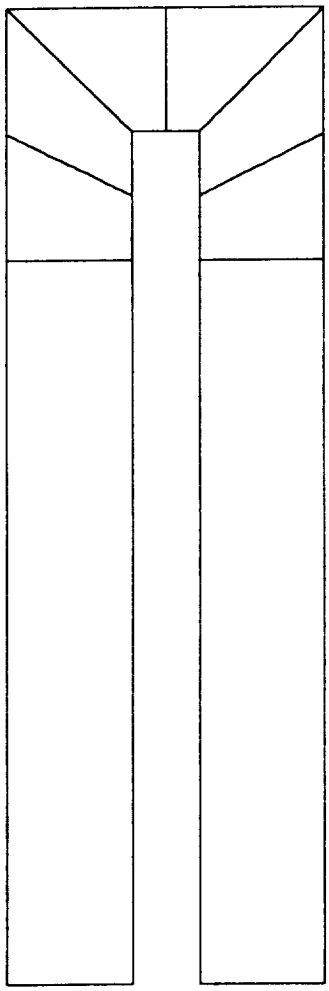
- To obtain detailed information on the turbulent flowfield and its effects on the local heat transfer

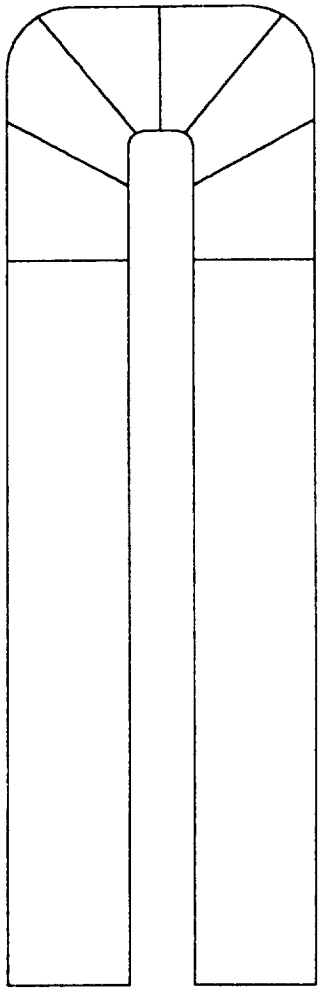
- To benchmark the CFD solution with a two-pass sharp-turned square duct experiemnt

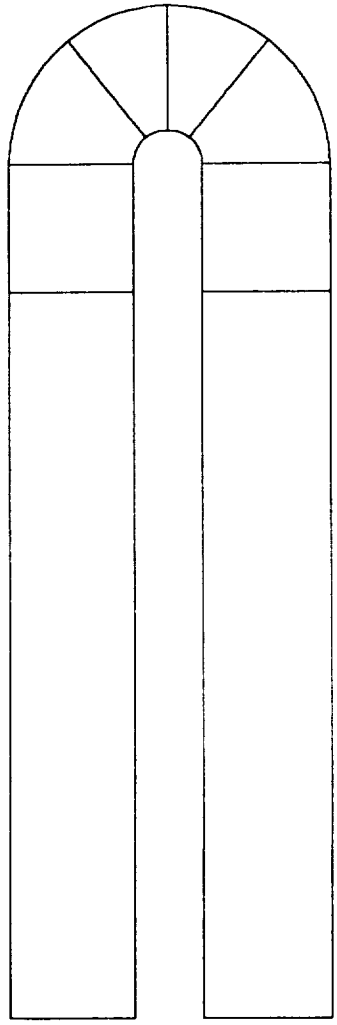
- To examine the influence of turn configuration and System rotation on the overall transport phenomena in the passage

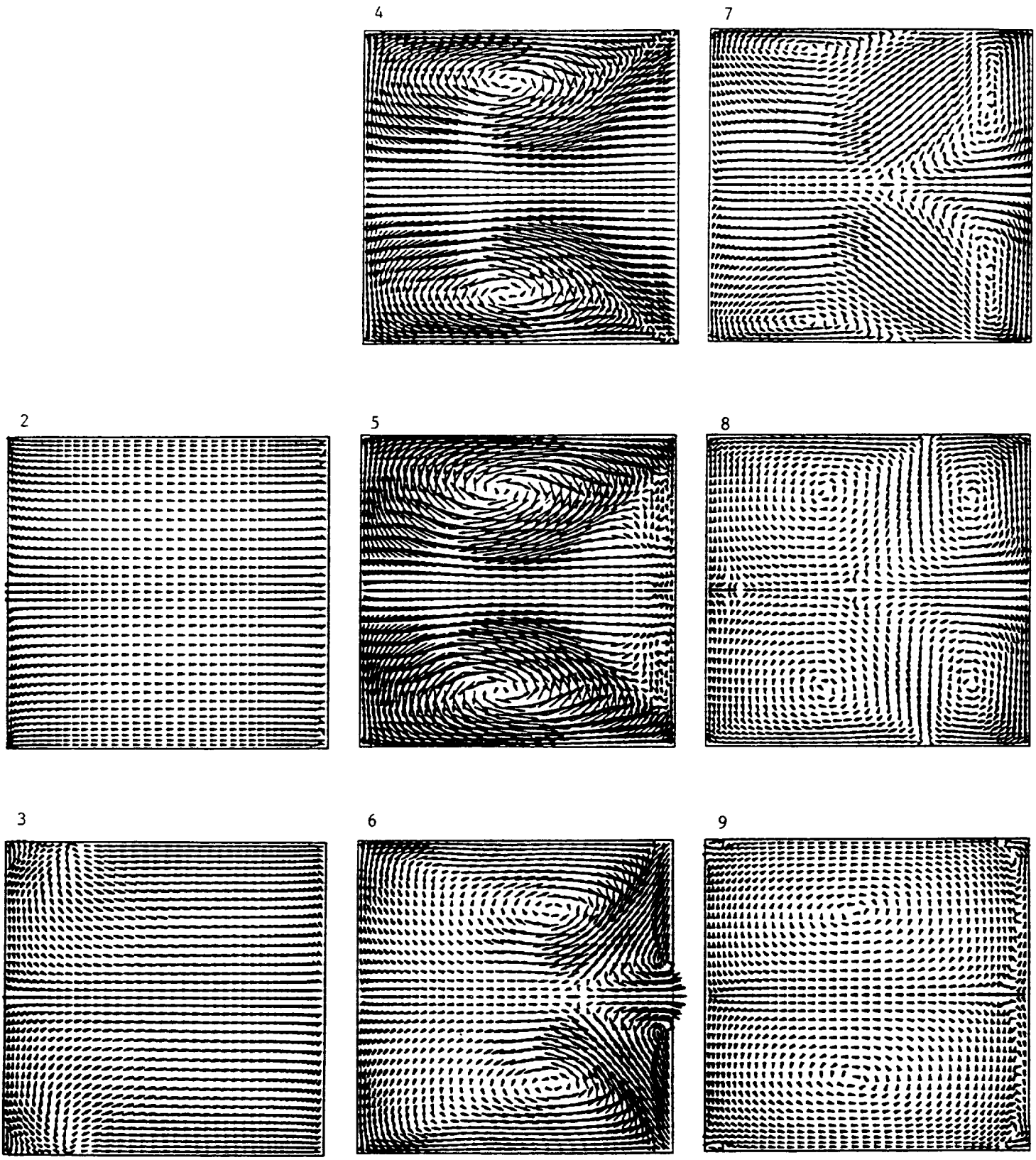

circular corner duct
round corner duct
straight corner duct
Cp
5.0
3.0
1.0
-1.0
-3.0
-5.0
0.0
0.2
0.4
0.6
0.8
1.0
X/L


straight corner duct; 65x41x21 grid
straight corner duct; 65x31x15 grid
straight corner duct; 51x11x11 grid
region-average data; Chyu (1991)
NUb/NUo
X/L
0.0
0.5
1.0
1.5
2.0
2.5
3.0
0.0
0.2
0.4
0.6
0.8
1.0

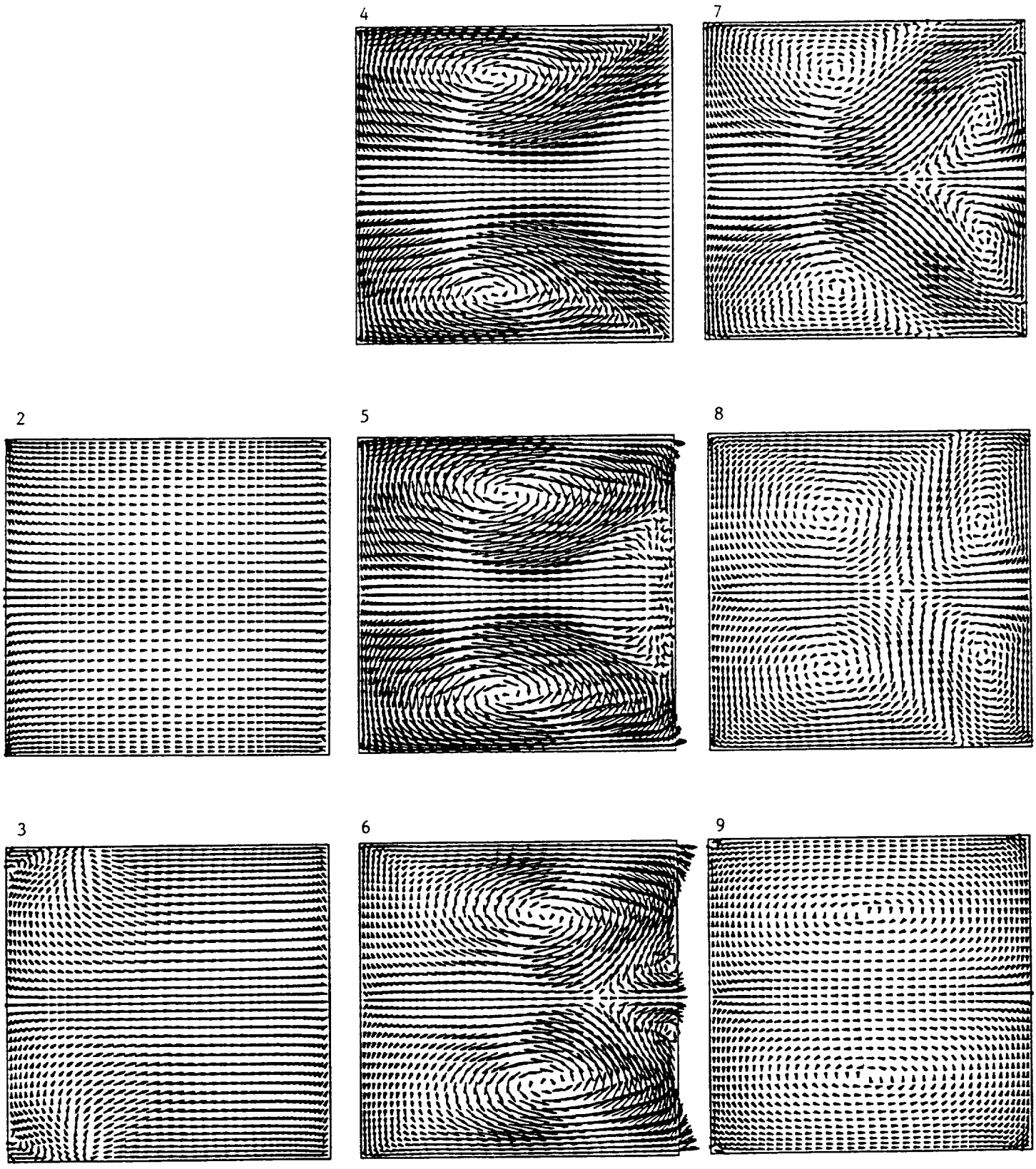




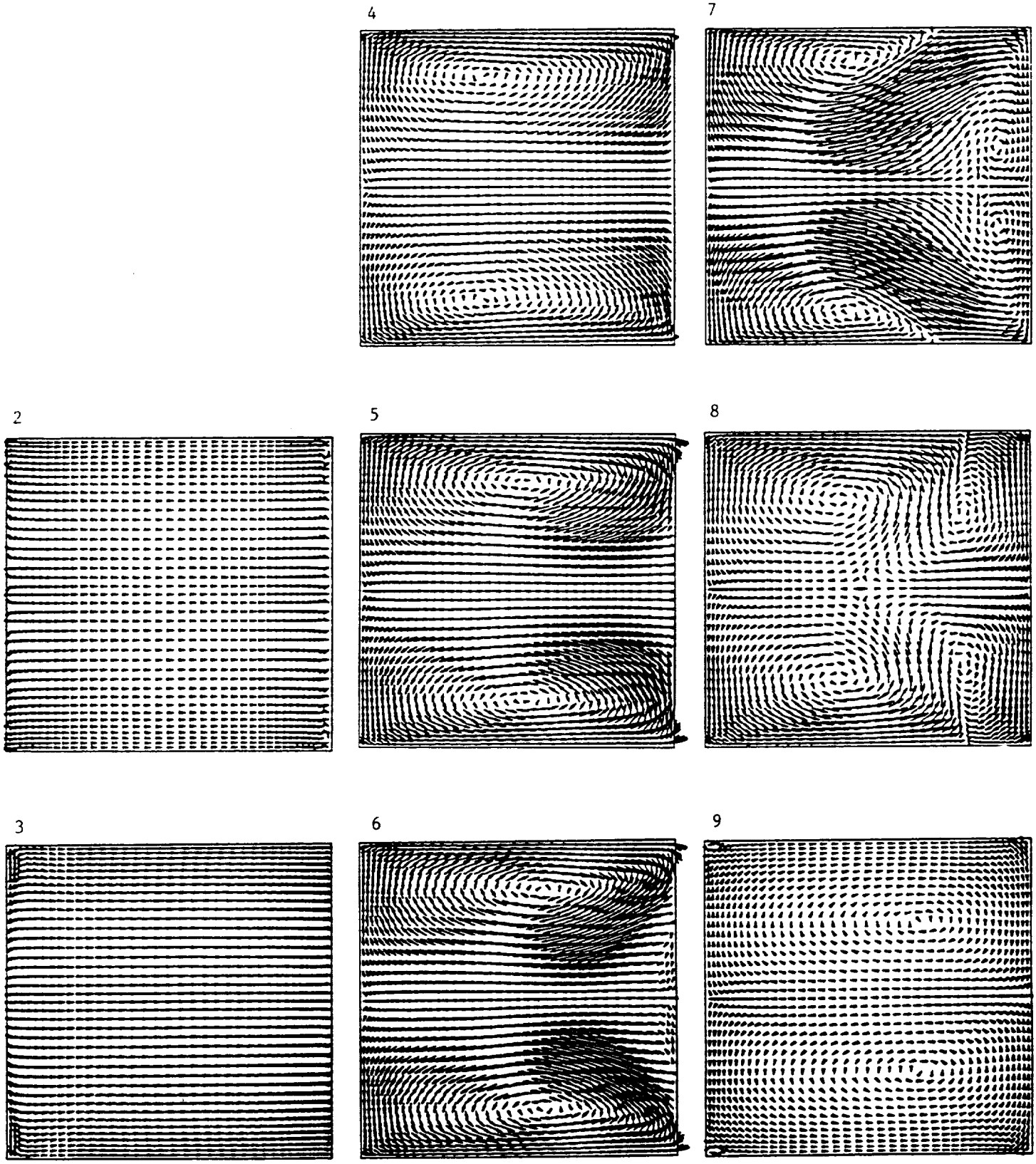




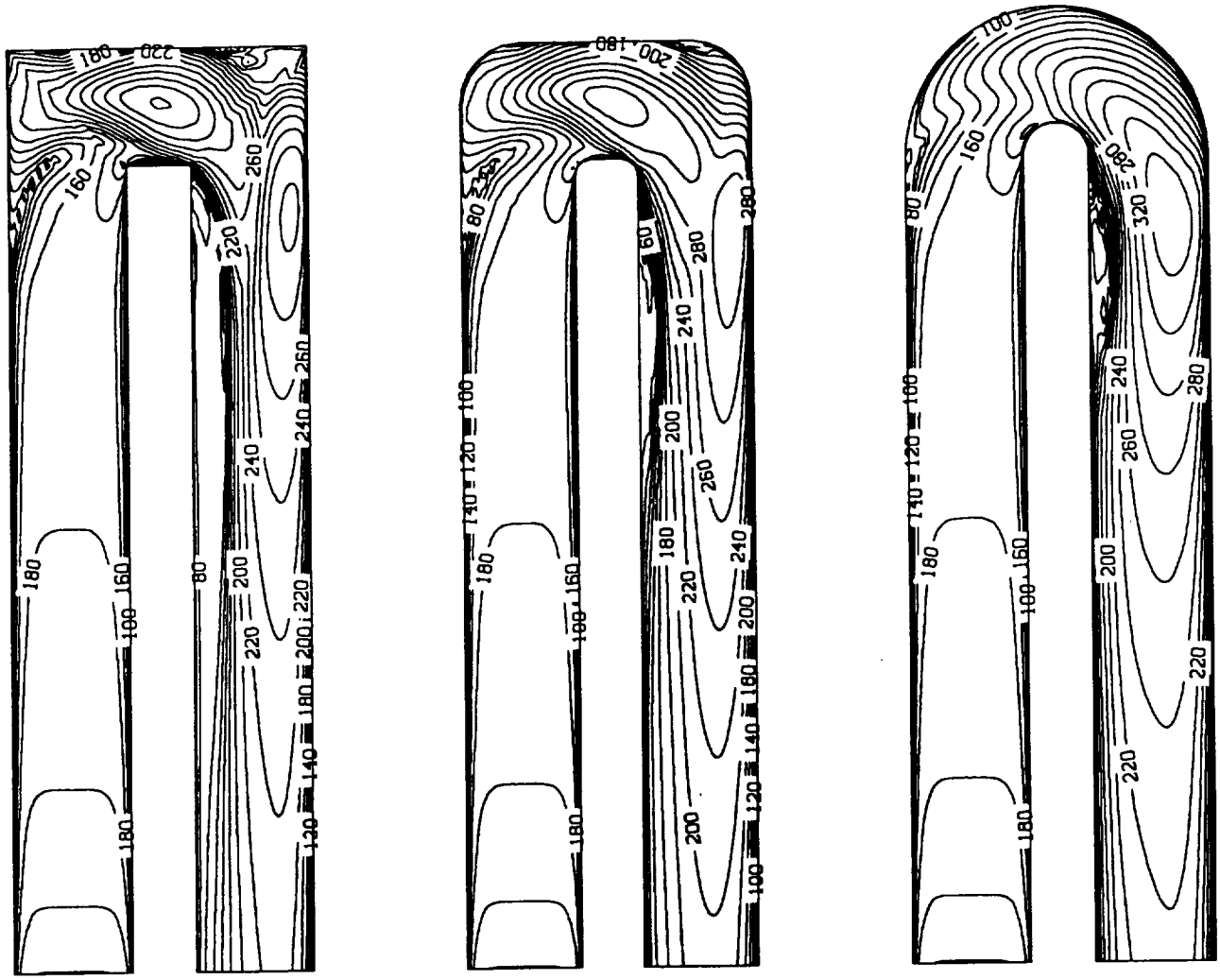
The evolution of the secondary flow pattern for a 180-degree turn straight-corner square duct

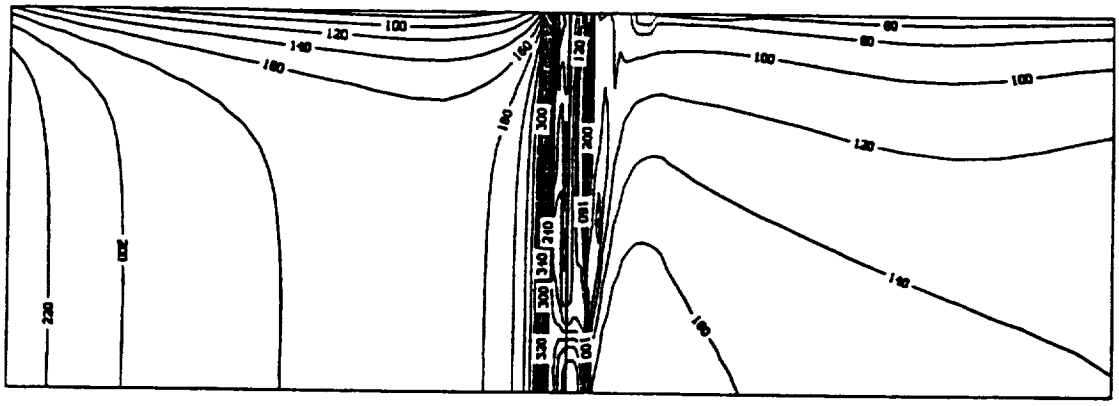


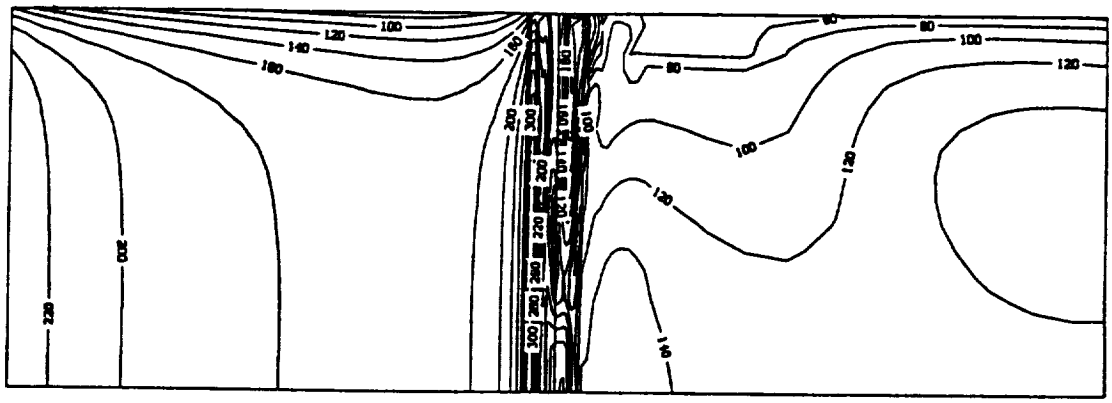
The evolution of the secondary flow pattern for a 180-degree turn round-corner square duct

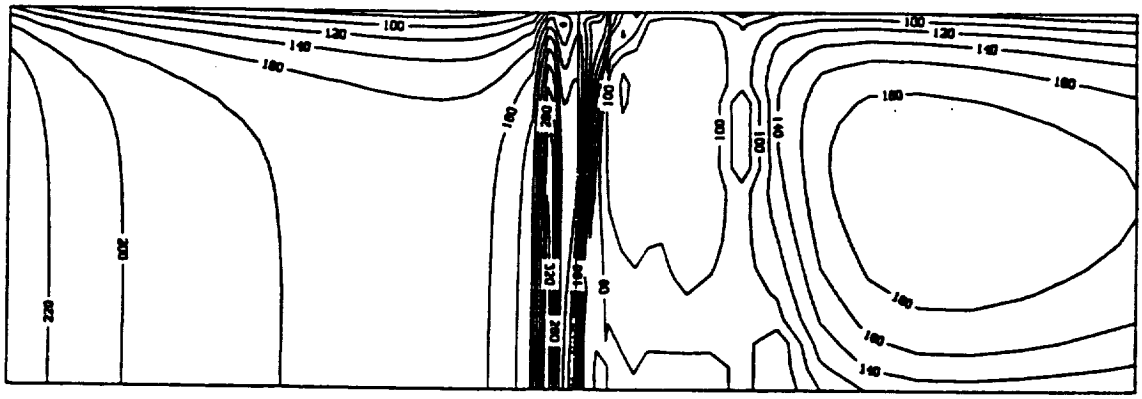


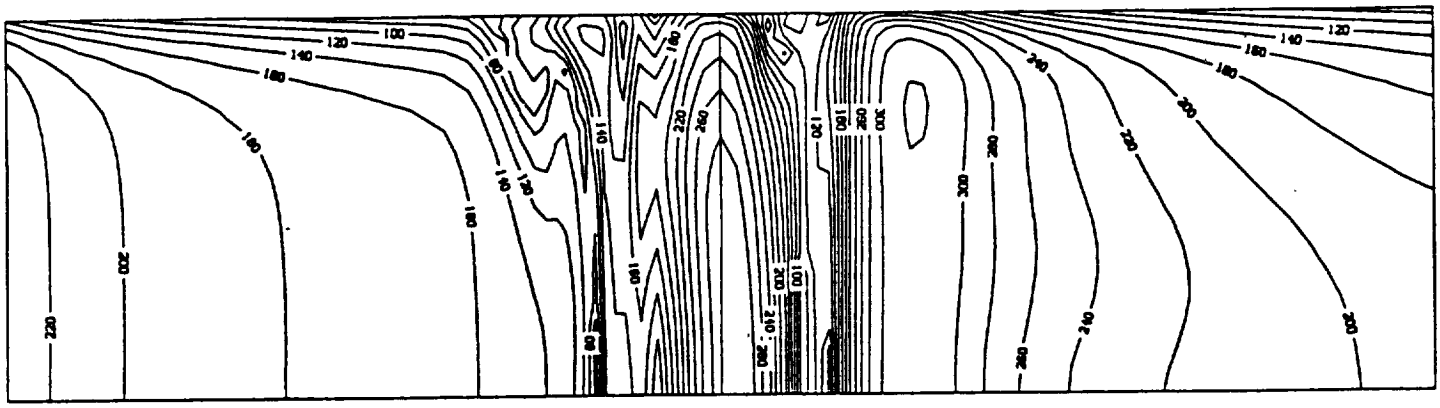
The evolution of the secondary flow pattern for a 180-degree turn circular-corner square duct

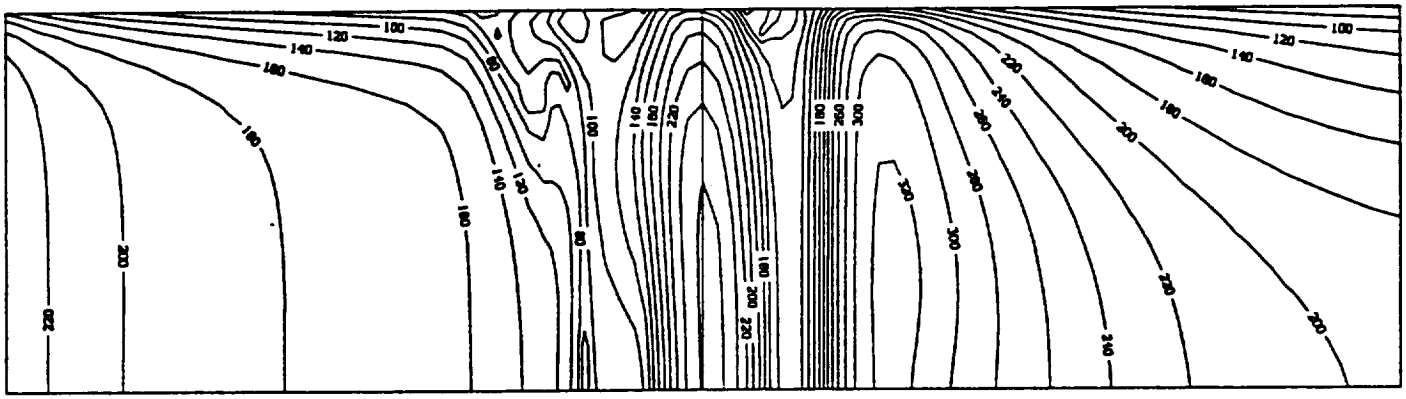


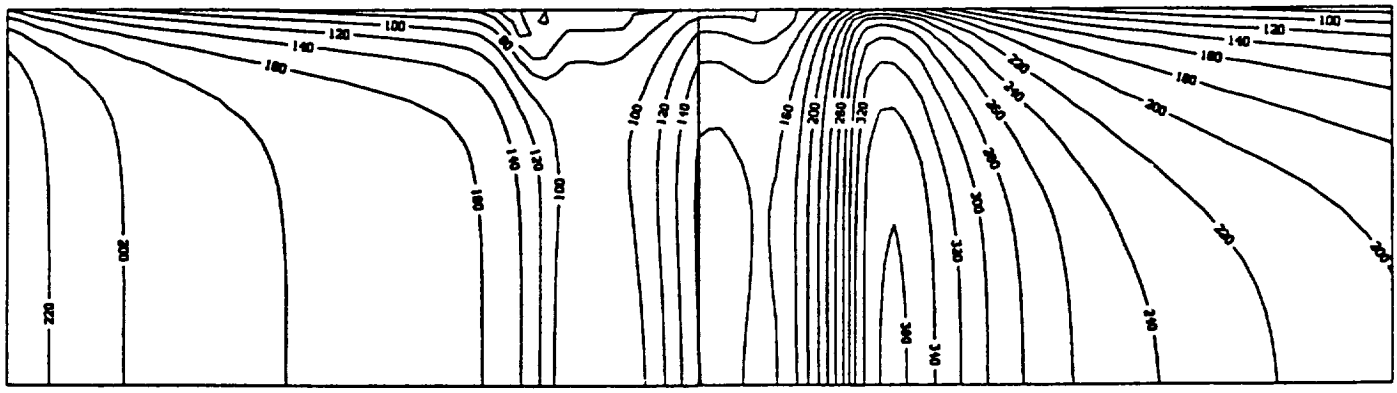


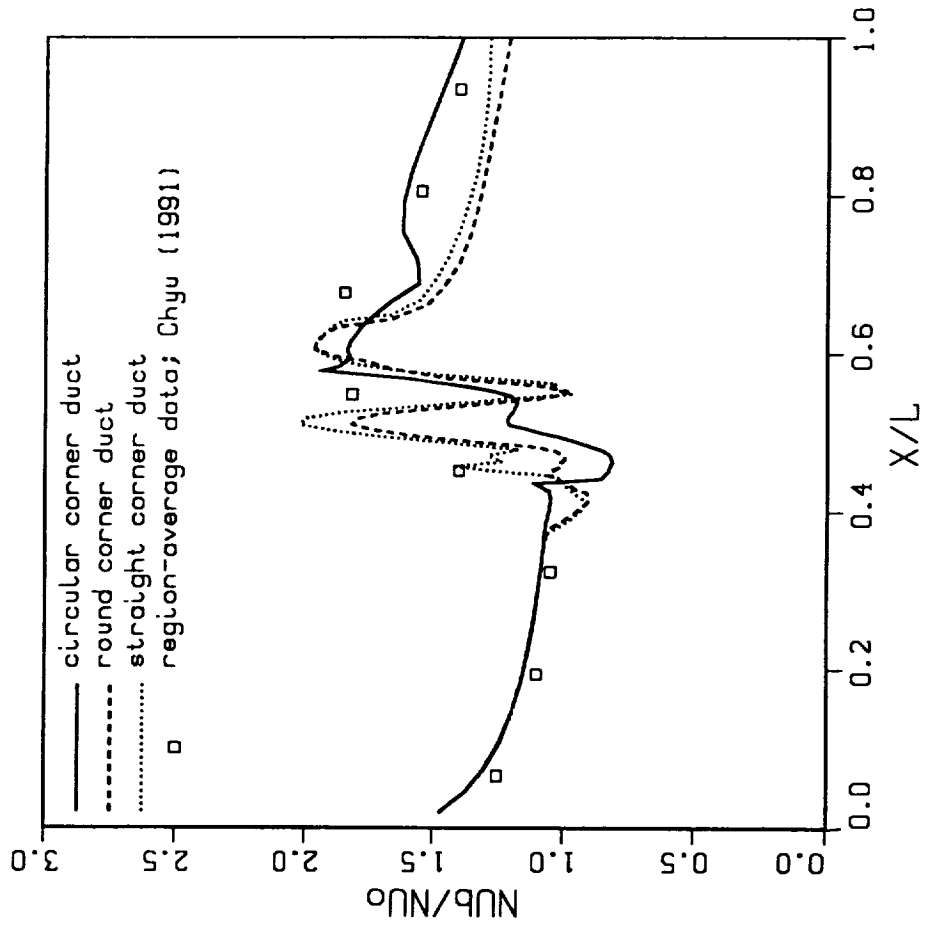


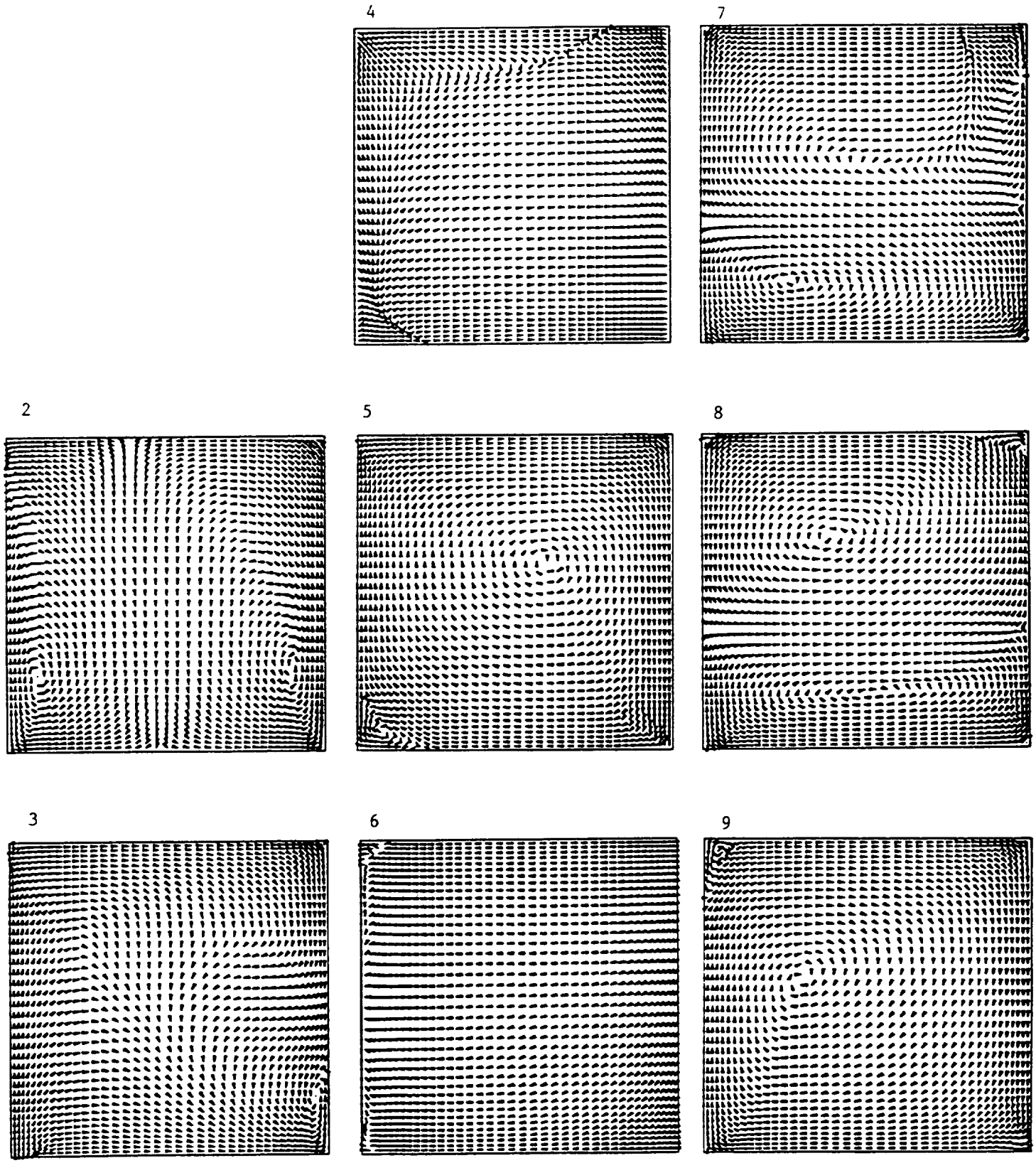




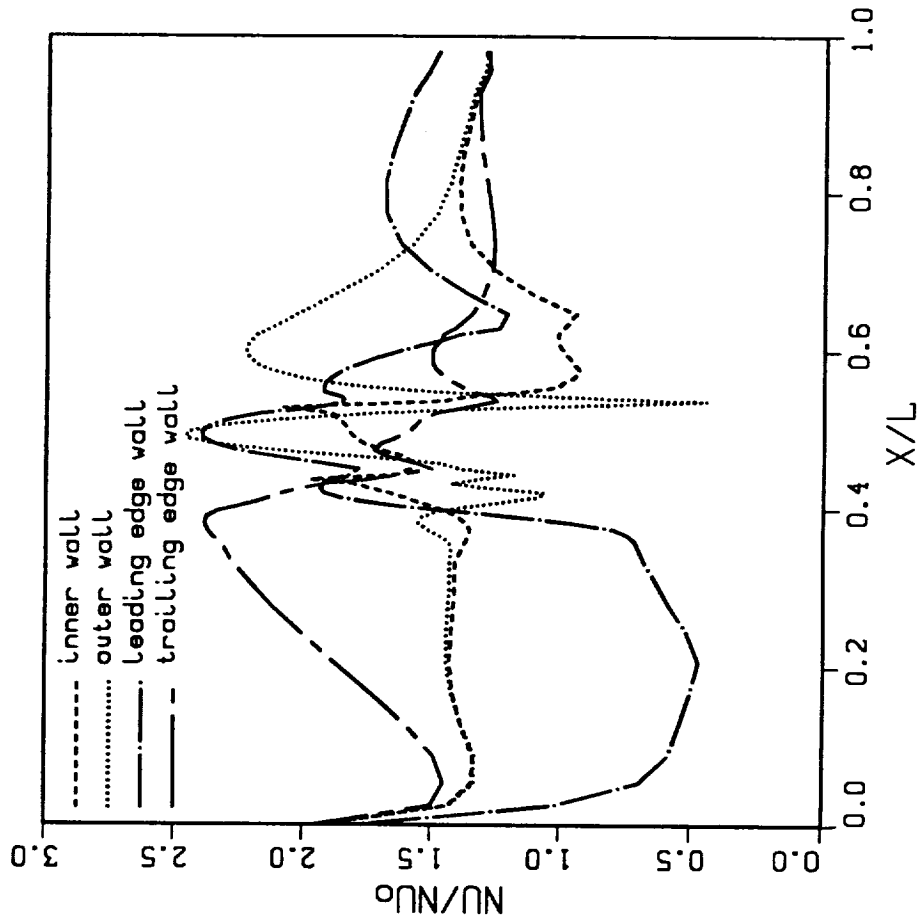


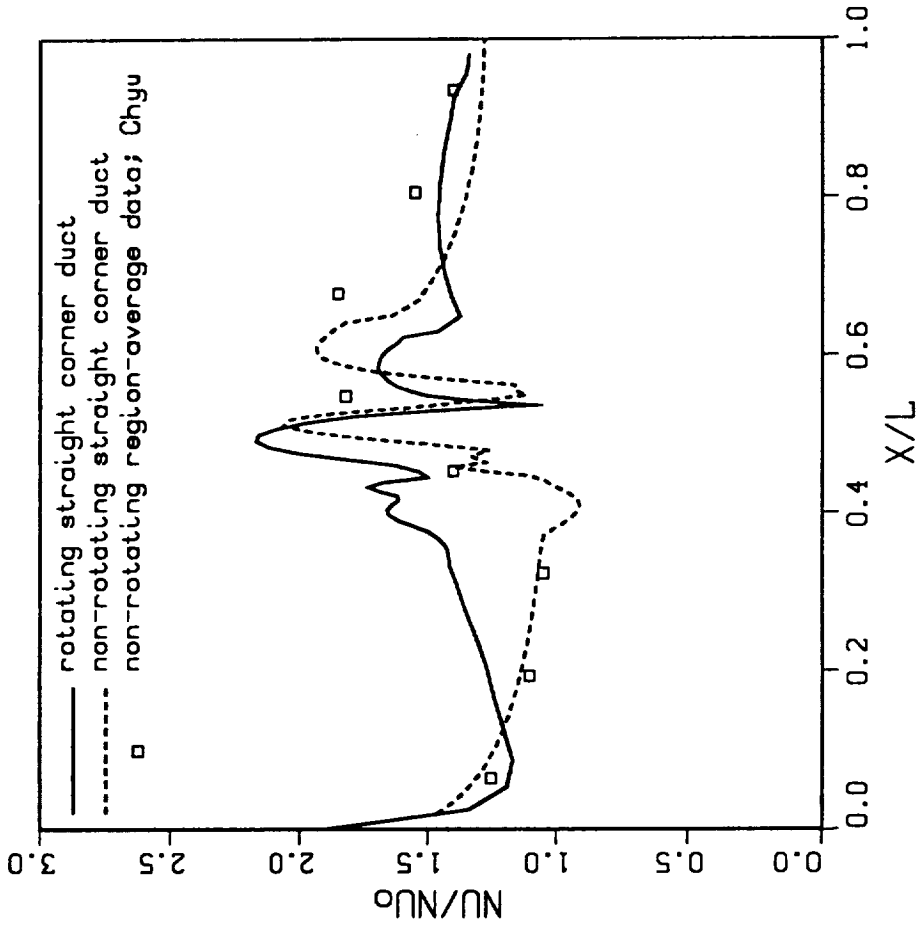



circular corner duct
round corner duct
straight corner duct
region-average data; Chyu (1991)
Nub/Nuo
X/L
0.0
0.5
1.0
1.5
2.0
2.5
3.0
0.0
0.2
0.4
0.6
0.8
1.0



The evolution of the secondary flow pattern for a 180-degree turn straight-corner square duct with rotation


inner wall
outer wall
leading edge wall
trailing edge wall
NU/NUo
X/L
0.0
0.5
1.0
1.5
2.0
2.5
3.0
0.0
0.2
0.4
0.6
0.8
1.0


rotating straight corner duct
non-rotating straight corner duct
non-rotating region-average data; Chyu
NU/NUo
X/L
0.0
0.5
1.0
1.5
2.0
2.5
3.0
0.0
0.2
0.4
0.6
0.8
1.0

The averaged total Nusselt numbers

	NUi	NUb	NUb/NUo
Circular-Corner Turn	184	210	1.3127
Round-Corner Turn	188	213	1.3321
Straight-Corner Turn	193	222	1.3837
Straight-Corner Turn with Rotation	217	250	1.5614

Conclusions

- Detailed information in flowfield, pressure drop and local heat transfer
- Favorable agreement with experimental data
- Complex features and strong turn-geometry dependence in post-turn secondary flows
- Straight-corner turn has highest turn-induced heat transfer enhancement
- Circular-corner turn has highest post-turn heat transfer
- System rotation enhances heat transfer