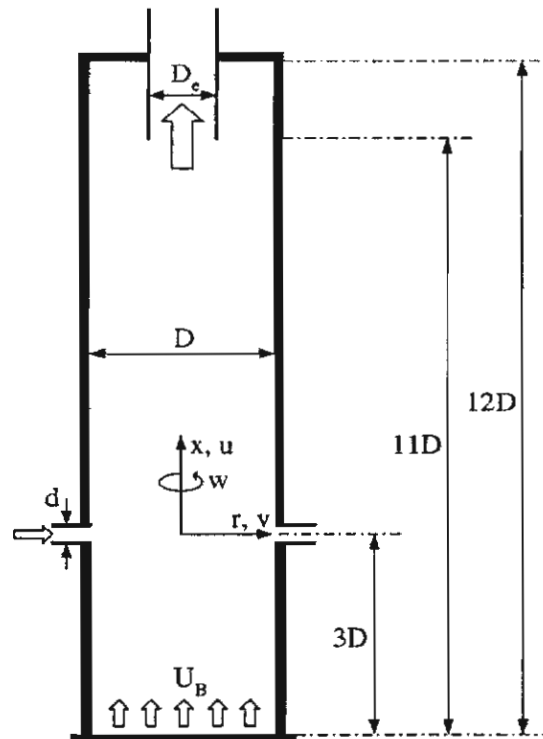
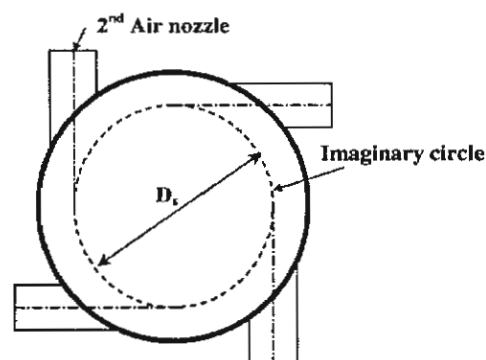


การทดลองของ Lin *et. al.* (1998) ไม่ได้บอกค่าความเร็วอากาศปฐมภูมิกับอากาศทุติยภูมิที่ทางเข้าทั้งสอง ดังนั้นเงื่อนไขทางเข้าสำหรับความเร็วในแนวสัมผัส (w) หาได้โดยการประมาณนอกช่วง (extrapolated) จากค่าความเร็วของผลการทดลอง ส่วนความเร็วในแนวแกน



รูปที่ 6.30 ลักษณะเตาฟลูอิดไดซ์เบดแบบฉีดอากาศทุติยภูมิด้านข้าง (VFBC) ของ Lin *et. al.* (1998) และ computational domain



รูปที่ 6.31 Imaginary circle และ การจัดวาง secondary air injection nozzles

(U_B) และความเร็วในแนวรัศมี (V_j) หาได้จากความสัมพันธ์ระหว่างอัตราการไหลของอากาศปฐมภูมิกับอากาศทุติยภูมิ ซึ่งหาได้จากสมการดังนี้

$$U_B = Q_r \times \frac{4w}{D_s} \times \frac{d_0^2}{D}$$

$$V_j = w \frac{\sqrt{D_s^2 + D^2}}{D_s}$$

$$l = \frac{d_o^2}{\sqrt{D_s^2 + D^2}}$$

โดยที่ Q_r เป็นอัตราส่วนการฉีด ($Q_r = Q_B/Q_j$, Q_B และ Q_j เป็นอัตราการไหลอากาศปฐมภูมิและอากาศทุติยภูมิตามลำดับ), U_B เป็นความเร็วในแนวแกน, V_j ความเร็วในแนวรัศมี, l เป็นขนาดของช่องฉีดที่ใช้ในการคำนวณ ในการทำนายจะกำหนดให้ $D = 190$ mm, $d_o = 19$ mm, $D_s = 0.095$ mm

ในการจำลองการไหลนี้ได้ใช้เงื่อนไขของ axisymmetry ในการคำนวณ เพื่อศึกษาเปรียบเทียบการใช้ $k-\epsilon$ model และ ASM ที่ใช้ numerical differencing scheme 4 แบบ คือ upwind, hybrid, QUICK และ SOU schemes ซึ่งผลลัพธ์ที่ได้จะนำไปเปรียบเทียบกับข้อมูลที่ได้จากการทดลองของ Lin *et. al.* (1998) ที่ตำแหน่ง $x = 4D, 5D, 6D, 7D, 8D, 9D$ และ $10D$

การศึกษาความเป็นอิสระของกริด (grid independent) ต่อผลการทำนายความเร็วตามแนวแกนและตามแนวสัมผัส ซึ่งทำการคำนวณที่จำนวนกริดเท่ากับ 60×25 , 70×25 , 80×30 , 90×35 และ 100×40 จากการวิเคราะห์พบว่าจำนวนกริดมีผลต่อการทำนายการไหลพอสมควร แต่ค่าที่ได้ก็ไม่แตกต่างกันมากดังนั้นในการคำนวณจะเลือกใช้กริดขนาด 80×30 ก็เพียงพอ

การทำนายการไหลในกรณี $Q_r = 2$ และ $D_s/D = 1.0$ เมื่อทำการประมาณนอกช่วงความเร็วฉีดในแนวสัมผัสจากผลการทดลองจะได้ $w = 9$ m/s และคำนวณความเร็วที่ทางเข้าของอากาศปฐมภูมิ, $U_B = 0.8$ m/s และความเร็วการฉีดอากาศทุติยภูมิในแนวรัศมี, $V_j = 4.3587$ m/s ส่วนขนาดของช่องฉีด, $l = 4.359$ mm

6.5.1 การทำนายการไหลโดย $k-\epsilon$ model

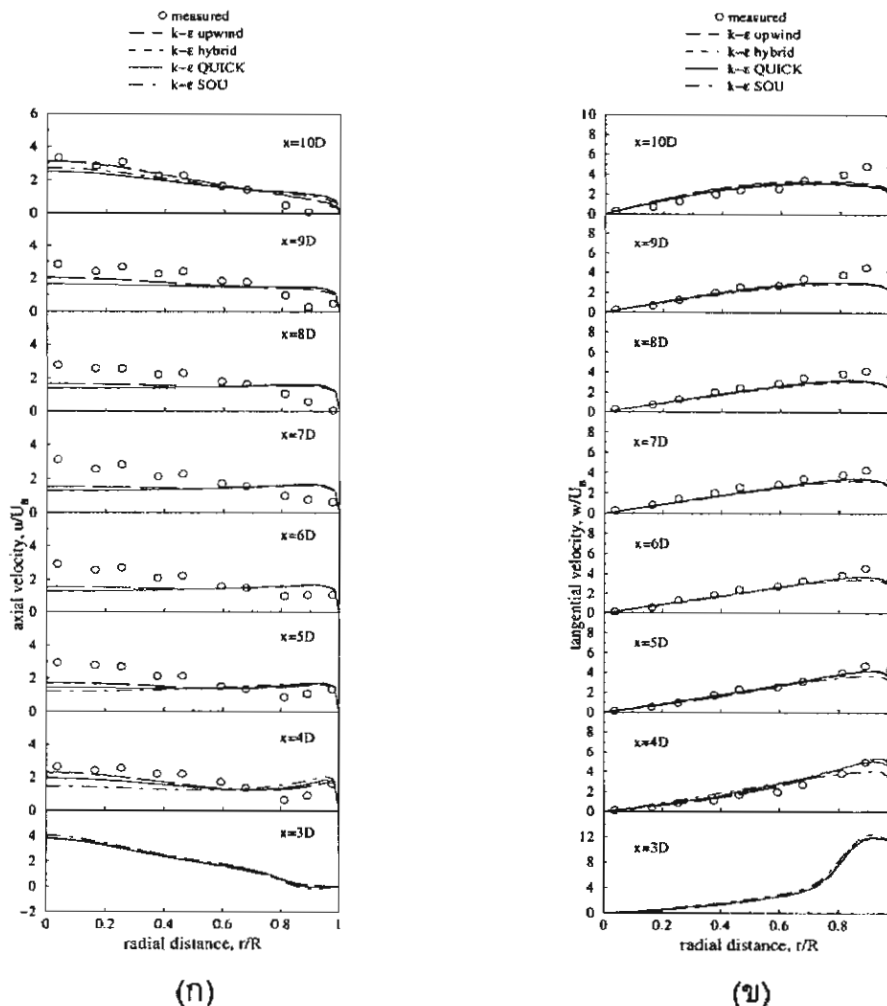
จากการจำลองการไหลโดย $k-\epsilon$ model ร่วมกับ scheme ต่างๆ ผลลัพธ์ที่ได้ถูกแสดงในรูปของความเร็วตามแนวแกนและความเร็วตามแนวสัมผัสรวมถึง contour plot และ vector plot

ผลของความเร็วตามแนวแกนและความเร็วตามแนวสัมผัส ได้แสดงในรูปที่ 6.32(ก) และ 6.32(ข) ซึ่งจากรูปแสดงให้เห็นการเปลี่ยนแปลงของความเร็วจากระยะกึ่งกลางถึงผนัง รูปที่ 6.3(ก) แสดงการเปลี่ยนแปลงของความเร็วตามแนวแกน จากรูปพบว่าในตำแหน่ง $x = 3D$ ถึง $8D$ ค่าที่คำนวณได้ทั้ง 4 schemes พบว่าค่าตอบที่ได้จาก upwind และ hybrid schemes สามารถทำนายได้ใกล้เคียงกับข้อมูลจากการทดลองมากกว่า QUICK และ SOU schemes โดยเฉพาะในช่วง r/R ตั้งแต่ 0 ถึง 0.6

การเปลี่ยนแปลงของความเร็วตามแนวสัมผัสแสดงในรูป 6.32(ข) จากรูปการทำนายของทั้ง 4 schemes มีลักษณะเดียวกับการทำนายในแนวแกน แต่ค่าที่ได้จาก upwind และ hybrid schemes จะทำนายได้ดีในตำแหน่ง $x = 4D$ ถึง $7D$ ส่วนตำแหน่งที่เหลือ QUICK และ SOU schemes ทำนายได้ต่ำกว่า ซึ่งจากภาพรวมทั้งหมดเห็นว่าทุก scheme ทำนายค่าได้ใกล้เคียงกับข้อมูลจากการทดลอง

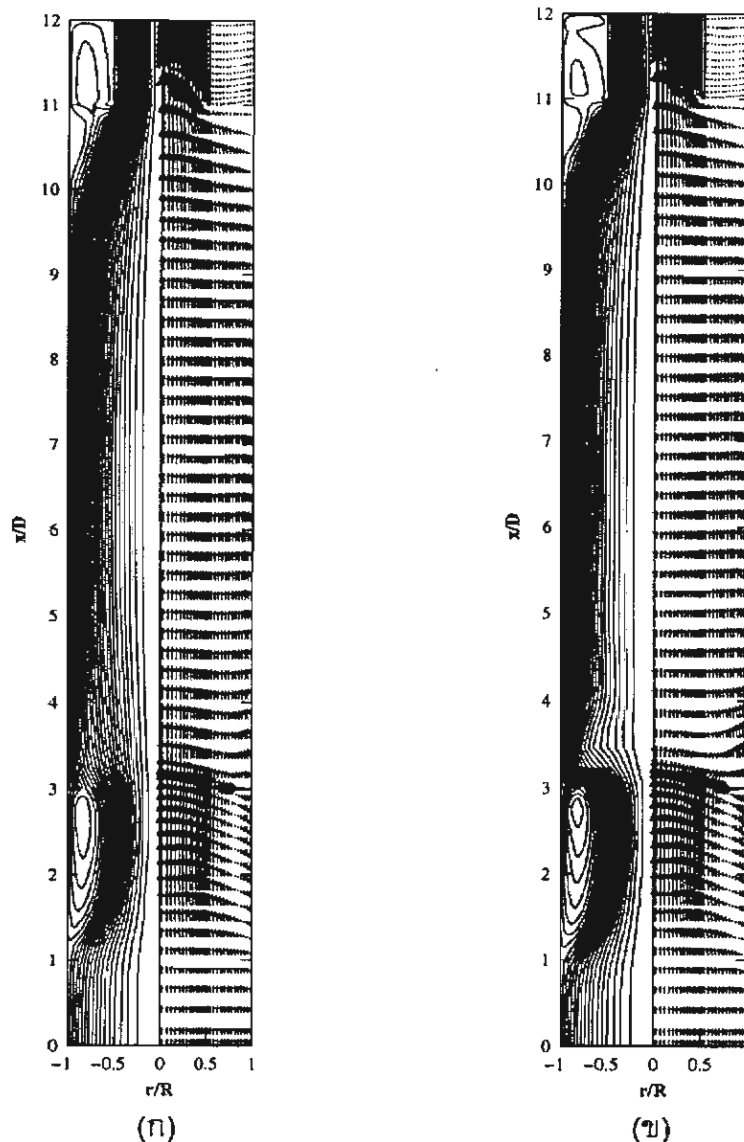
Contour plot ของ stream function และ vector plot ที่ทำนายโดย upwind และ SOU schemes ได้แสดงในรูปที่ 6.33(ก) และ 6.33(ข) ตามลำดับ จาก contour plot ของรูปที่ 6.33

(ก) สังเกตว่ามี recirculation เกิดขึ้นเลย 3 ตำแหน่งด้วยกัน ที่บริเวณด้านล่างของช่อง 2 ตำแหน่ง และปลายทางออกที่อยู่ระหว่างปล่องกับผนังอีก 1 ตำแหน่ง ขณะที่ contour plot ของ รูปที่ 6.33(ข) จะสังเกตเห็น recirculation ได้ถึง 6 ตำแหน่งด้วยกัน



รูปที่ 6.32 การเปลี่ยนแปลงของความเร็วที่ทำนายโดย $k-\epsilon$ model (ก) ความเร็วตามแนวแกน (ข) ความเร็วตามแนวสัมผัส

จาก vector plot รูปที่ 6.33(ก) สามารถสังเกตเห็น recirculation วงใหญ่ด้านล่างของช่องฉีดที่พบใน contour plot ชัดเจนยิ่งขึ้นโดยเกิดขึ้นที่บริเวณ $x/D = 1.8$ ถึง 3 และ $r/R = 0.7$ ถึง 1.0 และ recirculation วงเล็กที่ตำแหน่ง $x/D = 2.9$ และ $r/R = 0.95$ ส่วน vector plot ที่ด้านปลายทางออกบริเวณปล่องกับผนังเดามีขนาดเล็กมาก ใน vector plot ของรูปที่ 6.33(ข) สังเกตเห็นการเปลี่ยนแปลงขนาดและทิศทางของความเร็วได้อย่างชัดเจนการจำลองการไหลโดย $k-\epsilon$ model ซึ่งทำการเปรียบเทียบประสิทธิภาพของ scheme ต่างๆ อาจสรุปได้ว่า upwind และ hybrid schemes ทำนายได้ดีใกล้เคียงกับผลการทดลองที่สุดบนความเร็วทั้งในแนวแกนและแนวสัมผัส



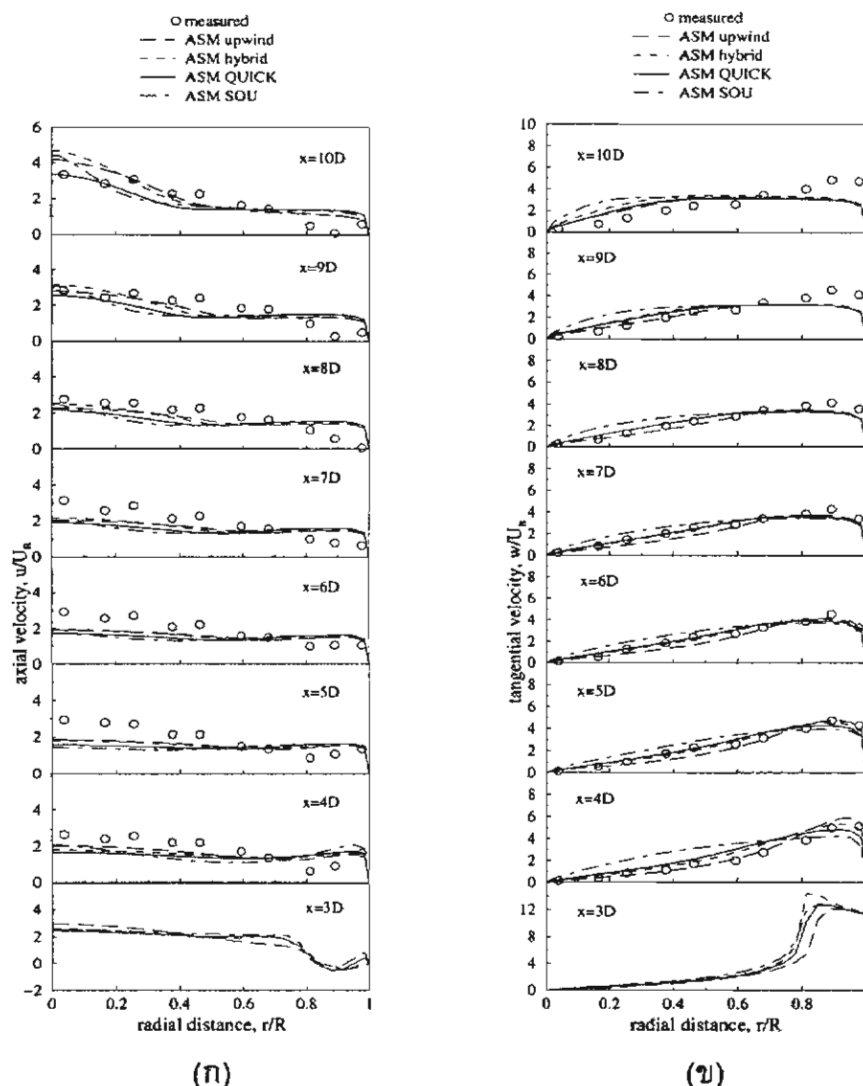
รูปที่ 6.33 Contour plot ของ stream function กับ vector plot ของความเร็ว ทำนายโดย $k - \varepsilon$ model ที่ใช้ (ก) upwind scheme และ (ข) SOU scheme

6.5.2 การทำนายการไหลโดย Algebraic Stress model (ASM)

การจำลองโดยใช้ ASM ร่วมกับ scheme ต่างๆ ได้นำมาแสดงในรูปของความเร็วตามแนวแกน, ความเร็วตามแนวสัมผัส, contour plot และ vector plot

รูปที่แสดงให้เห็นการเปลี่ยนแปลงของความเร็วจากระยะกึ่งกลางถึงผนังของความเร็วตามแนวแกนและความเร็วตามแนวสัมผัสในรูปที่ 6.34(ก) และ 6.34(ข) ตามลำดับ

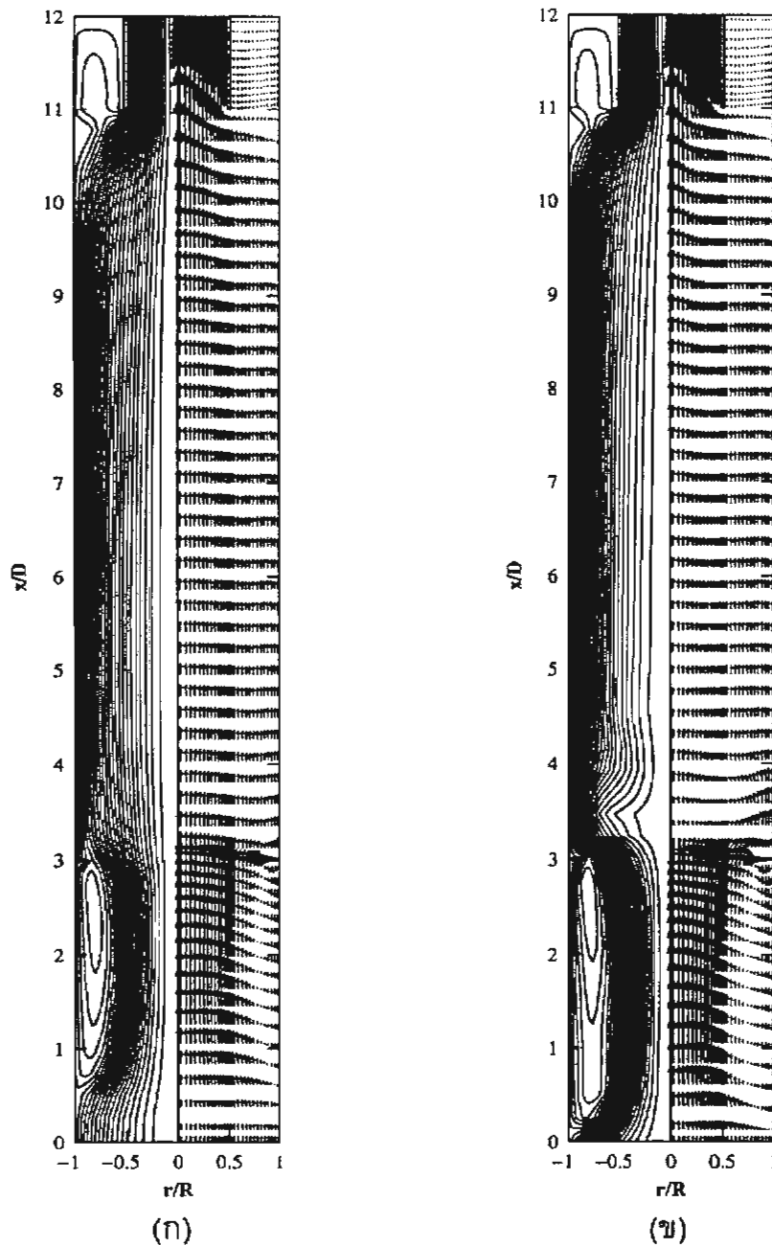
รูปที่ 6.34(ก) ซึ่งแสดงการเปลี่ยนแปลงของความเร็วตามแนวแกน พบว่าในตำแหน่ง $x = 3D$ ถึง $8D$ ค่าที่คำนวณได้จาก SOU scheme มีค่าที่ใกล้เคียงกับผลการทดลองมากกว่าโดยเฉพาะในตำแหน่งที่ใกล้กับช่องฉีด ในบริเวณ r/R ประมาณ 0 ถึง 0.5 ส่วนความเร็วในแนวแกนสัมผัสในรูปที่ 6.34(ข) พบว่า upwind และ hybrid schemes ทำนายได้ใกล้เคียงกว่า SOU scheme อย่างชัดเจนในช่วง r/R ประมาณ 0 ถึง 0.6



รูปที่ 6.34 การเปลี่ยนแปลงของความเร็วที่ทำนายโดย ASM (ก) ความเร็วตามแนวแกน (ข) ความเร็วตามแนวสัมผัส

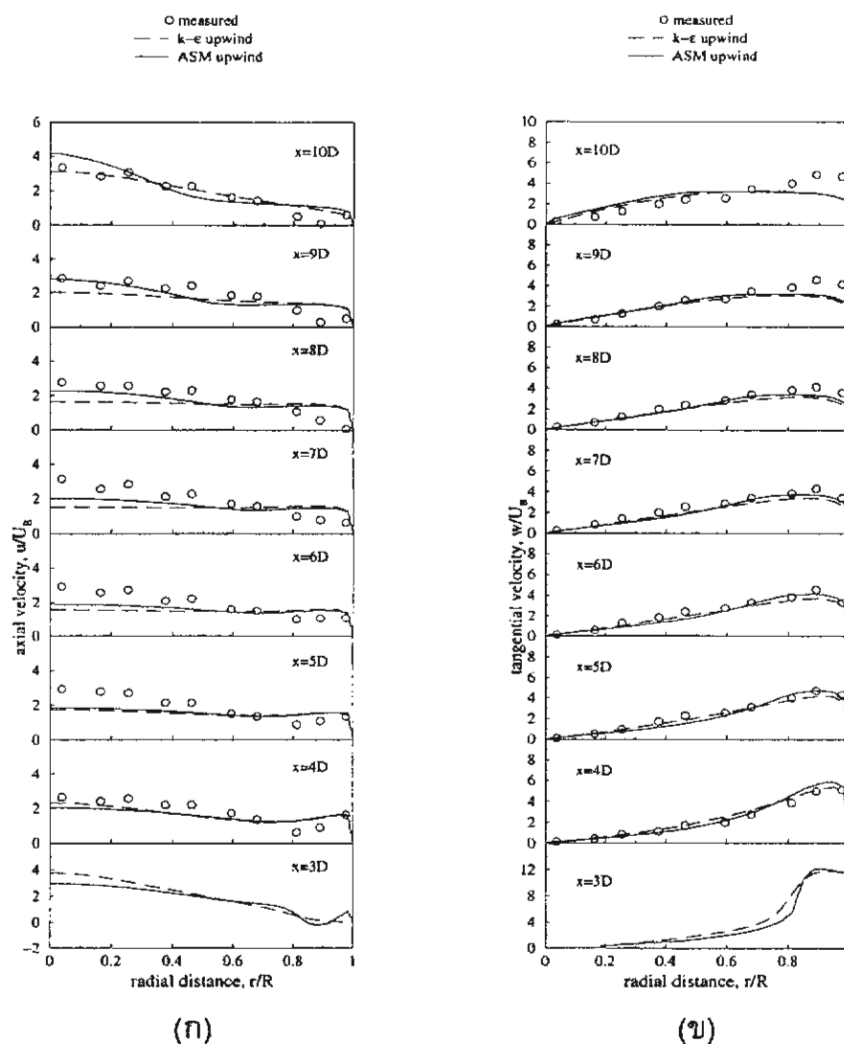
Contour plot และ vector plot ที่ทำนายโดย upwind และ SOU schemes ได้แสดงในรูปที่ 6.35(ก) และ 6.35(ข) ตามลำดับ จาก contour plot ของรูปทั้งสอง สังเกตว่า การทำนายโดย SOU scheme จะเกิด recirculation ที่ตำแหน่งได้และเหนือช่องฉีด ในขณะที่ผลจาก upwind กลับไม่แสดงให้เห็น recirculation ที่ตำแหน่งดังกล่าว จาก vector plot รูปที่ 6.35(ก) ไม่สามารถเห็นการไหลย้อนกลับซึ่งก็ยืนยัน ผลที่พบใน contour plot ชัดเจนยิ่งขึ้น ส่วนใน vector plot ของรูปที่ 6.35(ข) จะสังเกตการไหลย้อนกลับของ vector ความเร็วได้อย่างชัดเจนที่ตำแหน่งได้ช่องฉีด

การจำลองการไหลโดย ASM ซึ่งทำการเปรียบเทียบความสามารถของ scheme ต่างๆ คือ upwind, hybrid, QUICK และ SOU สรุปได้ว่า upwind และ hybrid scheme ให้ผลการทำนายความเร็วทั้งในแนวแกนและแนวสัมผัสได้ใกล้เคียงกับผลการทดลอง



รูปที่ 6.35 Contour plot ของ stream function กับ vector plot ของความเร็ว ทำนายโดย ASM ที่ใช้ (ก) upwind scheme และ (ข) SOU scheme

รูป 6.36(ก) และ 6.36(ข) เป็นรูปที่เปรียบเทียบผลการทำนายโดย $k-\varepsilon$ model กับ ASM ของความเร็วในแนวแกนและความเร็วในแนวสัมผัสตามลำดับ จากรูปที่ 6.36(ก) ซึ่งเป็นความเร็วในแนวแกนที่ใช้ upwind scheme ในการคำนวณของทั้ง 2 models พบว่า ค่าที่ทำนายโดย ASM นั้นใกล้เคียงกับผลการทดลองมากกว่า $k-\varepsilon$ model ส่วนการเปรียบเทียบ model ในการทำนายความเร็วในแนวสัมผัสในรูปที่ 6.36(ข) ซึ่งใช้ upwind scheme มีผลการเปรียบเทียบเหมือนกับในแนวแกนคือ ASM ทำนายใกล้เคียงกับผลการทดลองมากกว่า $k-\varepsilon$ model



รูปที่ 6.36 กราฟเปรียบเทียบการทำนายระหว่าง $k-\epsilon$ model กับ ASM (ก) ความเร็วตามแนวแกน (ข) ความเร็วตามแนวสัมผัส

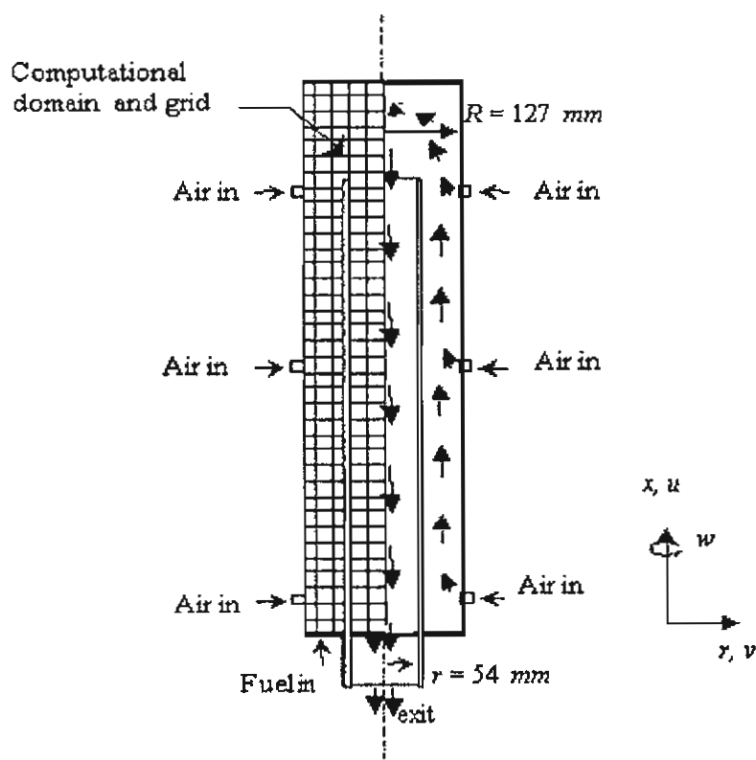
6.6 เตาเผาออร์เทคของ Nieh and Zhang (1992)^{****}

เตาเผาออร์เทค (vortex combustor, VC) เป็นเตาที่ถูกพัฒนาขึ้นมาโดยที่ใช้เทคนิคของไหลแบบหมุนวนความเร็วสูงช่วยในการเผาไหม้ การไหลแบบหมุนวนความเร็วสูงได้ถูกนำมาใช้ในงานทางด้านอุตสาหกรรมเป็นจำนวนมาก เช่น เตาเผา หัวเผา เครื่องยนต์สันดาปภายใน เครื่องยนต์กังหันก๊าซ และอุปกรณ์แลกเปลี่ยนความร้อนต่างๆ ในระบบการเผาไหม้นั้น การหมุนวนได้ถูกนำมาใช้เพื่อเพิ่มความเสถียรของเปลวไฟ เพิ่มการผสมคลุกเคล้าของเชื้อเพลิงและอากาศ และช่วยเพิ่มการถ่ายเทความร้อน นอกเหนือจากช่วยเพิ่มเวลาอยู่ในเตาของเชื้อเพลิง (fuel residence time) และช่วยลดมลพิษที่เกิดจากการเผาไหม้นี้ เพราะว่าการหมุนวนช่วยเหนี่ยว

^{****} ได้นำเสนอใน Regional Conference on Energy Technology Towards a Clean Environment เชียงใหม่ และ จะนำเสนอใน Intersociety Energy Conversion Engineering Conference , USA.

ให้เกิดการไหลวนไปมาในบริเวณกึ่งกลางของเตา (central recirculation zone) เป็นผลให้เกิดความปั่นป่วนอย่างรุนแรงขึ้นในแต่ละชั้นระหว่างการไหลวนไปมาและการไหลของชั้นภายนอก ซึ่งช่วยให้เปลวไฟในเตาเสถียรยิ่งขึ้น โครงสร้างของความปั่นป่วนในการไหลแบบหมุนวนความเร็วสูงนี้โดยทั่วไปมีความสัมพันธ์ที่ซับซ้อนมาก การไหลวนไปมา (recirculation) อันเป็นผลจากการหมุนวนสามารถนำไปสู่ การเพิ่มประสิทธิภาพการสันดาป ลดมลภาวะที่ปล่อยออก (NO_x , เขม่า, สารคาร์บอนที่เผาไหม้ไม่หมด) ลดความยาวของเปลวเพลิงและเป็นผลให้เตาเผามีขนาดเล็กลง เพิ่มความเสถียรของการสันดาป ซึ่งเตาเวอร์เทคนี้ถูกออกแบบมาเพื่อแก้ปัญหาต่างที่เกิดขึ้นในเตาที่กล่าวในข้างต้น โดยเตาเวอร์เทคนั้นจะรวมข้อดีของเตาไซโคลอน, เตาถ่านหินผง (pulverized coal), ห้องสันดาปของเครื่องยนต์กังหันก๊าซ และเตา fluidized bed เข้าด้วยกันเพื่อแก้ปัญหาในเรื่องของ slag, เขม่า, ชีเก้ รวมถึงการเพิ่มเวลาในการเผาไหม้เพื่อการเผาไหม้ที่ดีขึ้น และที่แน่นอนย่อมนำไปสู่ขนาดที่เล็กลงกว่าเตาที่กล่าวมาแล้ว

การไหลแบบหมุนวนอย่างรุนแรงในเตาเผาเวอร์เทคถูกนำมาพิจารณาในงานวิจัยนี้ โดยมีโครงสร้างของเตาเผาดังแสดงในรูปที่ 6.37 ซึ่งประกอบไปด้วยท่อตรงกลางที่มีขนาดเส้นผ่าศูนย์กลางเท่ากับ 0.11 m โดยมีศูนย์กลางร่วมกับห้องเผาไหม้ทรงกระบอกที่มีความสูง 0.66 m และมีขนาดเส้นผ่าศูนย์กลางเท่ากับ 0.25 m ขนาดความสูงของท่อตรงกลางที่วางในห้องเผาไหม้เท่ากับ 0.51 m อากาศถูกฉีดเข้าไปในห้องเผาไหม้ในแนวสัมผัสผ่านหัวฉีด 3 ชุดและไหลออกผ่านท่อตรงกลาง ค่า swirl number ที่วัดได้เท่ากับ 13.9 ข้อมูลจากการทดลองวัดที่ตำแหน่ง $x = 0.051, 0.112, 0.173, 0.234, 0.295, 0.358, 0.42, 0.483, 0.544$, และ 0.605 m. ข้อกำหนดของเตาเผาได้สรุปไว้ในตารางที่ 1 ด้านล่าง



รูปที่ 6.37 การจัดการไหล,โดเมนและกริดในการคำนวณสำหรับเตาเผาเวอร์เทค

ตารางที่ 6.1 โครงสร้างและเงื่อนไขการไหลสำหรับเตาเผาเวอร์เทค

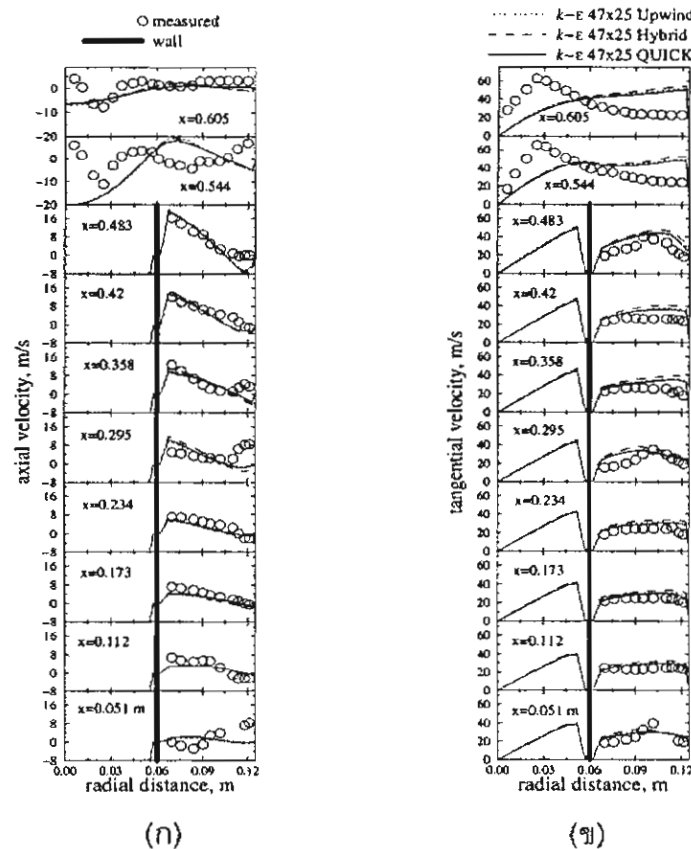
Parameter	Magnitude		
Vortex chamber geometry			
chamber inner dia., <i>m</i>		0.25	
chamber height, <i>m</i>		0.66	
centre tube outer dia., <i>m</i>		0.13	
centre tube inner dia., <i>m</i>		0.11	
centre tube height, <i>m</i>		0.51	
Multiple air injection			
nozzle height distribution, <i>m</i>	0.051	0.295	0.483
flow rate distribution, <i>m</i> ³ / <i>h</i>	226	226	226
tangential velocity, <i>m/s</i>	24.6	24.6	24.6
air temperature, K	300	300	300

สำหรับการจำลองการไหลในเตาเผาเวอร์เทค เชื้อเพลิงก๊าซมีเทน (CH_4) ถูกใช้แทนแก๊สเพื่อทำให้การจำลองง่ายขึ้น เชื้อเพลิงก๊าซจะถูกฉีดใต้เตาเผาไปยังด้านล่างของเตา เพื่อที่จะศึกษาอิทธิพลทางเข้าของการฉีดเชื้อเพลิงต่อคุณสมบัติการไหลที่มีการเผาไหม้เมื่อฉีดอากาศลงที่ จึงได้ทำการแบ่งประเภทของการฉีดเชื้อเพลิงเป็น 2 แบบ คือการฉีดเชื้อเพลิงในแนวแกนและการฉีดเชื้อเพลิงทำมุม (30 องศาจากแนวสัมผัส)

6.6.1 การทำนายการไหลในเตาเผาที่ไม่มีปฏิกิริยาเผาไหม้

อิทธิพลของจำนวนกริดต่อรูปร่างความเร็วที่ทำนายได้ถูกตรวจสอบโดยการเปรียบเทียบเมื่อใช้จำนวนกริดขนาดต่างกัน พบว่าจำนวนกริดขนาด 40×20 หรือจำนวนกริดที่ละเอียดกว่า เพียงพอที่จะรับรองได้ว่าค่าจากการจำลองการไหลไม่ได้รับอิทธิพลจากจำนวนกริดที่ใช้ การกระจายความเร็วในแนวแกนและแนวสัมผัสแสดงในรูปที่ 6.38(ก) และ 6.38(ข) ตามลำดับ จากการใช้ schemes ทั้งสามเมื่อเปรียบเทียบกับผลการทดลองพบว่ามีค่าที่ทำนายได้เหมือนกัน ซึ่งแสดงว่าทุก schemes สามารถใช้ได้ในการคำนวณ ซึ่งต่อไปจะใช้เพียง upwind scheme ในการคำนวณถัดไป ตามรูปที่ผ่านมาแสดงให้เห็นว่าผลจากการทำนายโดย $k-\varepsilon$ model มีความแตกต่างจากผลการทดลองโดยเฉพาะบริเวณด้านบนของเตา ตัวอย่างเช่น $k-\varepsilon$ model ไม่สามารถทำนาย แถบของ central toroidal recirculation และการเคลื่อนที่แบบ combined forced and free vortex ในส่วนบนของเตาเผา (ที่ $x = 0.544$ และ $0.605 m$) อย่าง

ไว้ก็ตามในการวิจัยนี้ ยังคงใช้ $k-\varepsilon$ model ในการทำนายแนวโน้มของลักษณะการไหลที่มีการเผาไหม้ในเตาเผาเวอร์เทคเนื่องจากความไม่ยุ่งยากในการใช้นั่นเอง



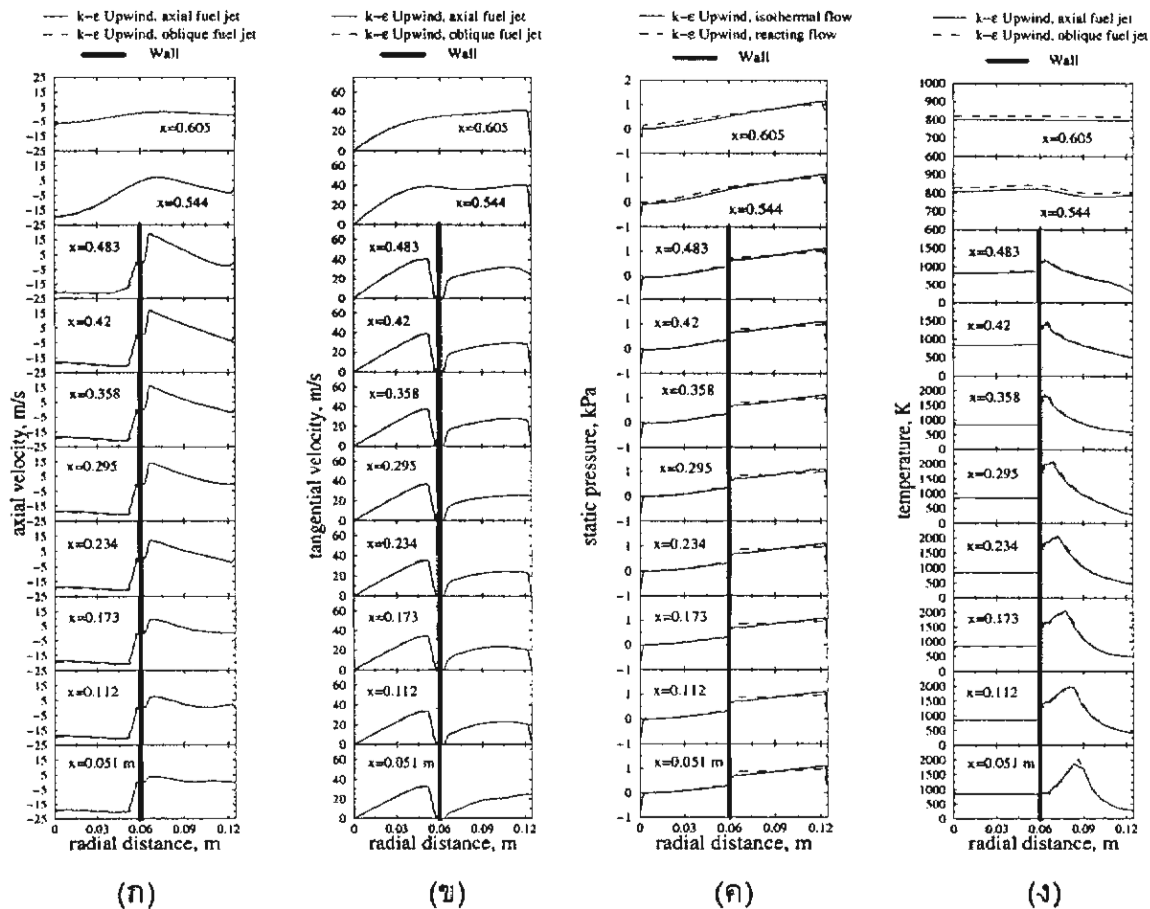
รูปที่ 6.38 เปรียบเทียบผลจากการทดลองกับผลจากการทำนายโดยใช้ schemes ที่แตกต่างกัน
(ก) ความเร็วตามแนวแกนและ (ข) ความเร็วตามแนวสัมผัส

6.6.2 การทำนายการไหลในเตาเผาที่มีปฏิกิริยาเผาไหม้

การศึกษาอิทธิพลของทางเข้าในการฉีดเชื้อเพลิงต่อสนามการไหลและอุณหภูมิ โดยมีการฉีดเชื้อเพลิง 2 แบบคือแบบแรกทำได้โดยการวางหัวฉีดเชื้อเพลิงขนานกับแนวแกนของเตาเผา เรียกว่า การฉีดเชื้อเพลิงตามแนวแกน ส่วนแบบที่สองทำได้โดยการวางแนวของหัวฉีดทำมุม 30 องศา กับเส้นสัมผัสที่ทางเข้าของเชื้อเพลิง รูปร่างของความเร็วตามแนวแกน, ความเร็วตามแนวสัมผัส, ความดันสถิตยและอุณหภูมิที่คำนวณได้สำหรับการฉีดเชื้อเพลิงทั้ง 2 แบบถูกแสดงในรูปที่ 6.39(ก), 6.39(ข) และ 6.39(ค) ตามลำดับ สังเกตว่าการฉีดเชื้อเพลิงทั้งสองแบบไม่มีผลต่อคุณสมบัติการไหลเฉลี่ย ตัวอย่างเช่นการกระจายความเร็วและความดันดังจะเห็นได้จากรูปที่ 6.39(ก), 6.39(ข) และ 6.39(ค)

เมื่อเปรียบเทียบกับแบบจำลองที่ไม่มีการเผาไหม้ ความเร็วตามแนวแกนที่ทางออกในกรณีนี้มีค่าสูงกว่า (ประมาณ 3 เท่า) สำหรับรูปร่างของอุณหภูมิพบว่าการฉีดเชื้อเพลิงแบบทำมุมทำให้อุณหภูมิด้านบนของเตาสูงขึ้น ขณะที่อุณหภูมิมิบริเวณวงแหวนมีค่าเหมือนกันดังแสดงในรูปที่

3d ซึ่งแสดงให้เห็นว่าเมื่อต้องการอุณหภูมิภายในเตาที่มีความสม่ำเสมอ รูปแบบทางเข้าของเชื้อเพลิงควรที่จะฉีดไปในเตาในแนวสัมผัส

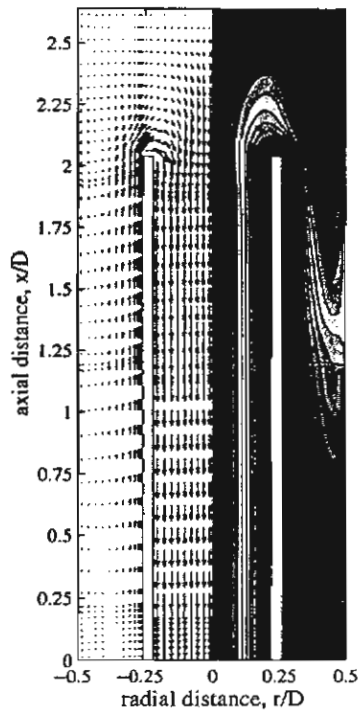


รูปที่ 6.39 รูปร่างในแนวรัศมีของความเร็ว (ก) ตามแนวแกนและ (ข) แนวสัมผัส (ค) ความดันสถิตยและ (ง) อุณหภูมิที่คำนวณโดยใช้การฉีดเชื้อเพลิงที่แตกต่างกัน

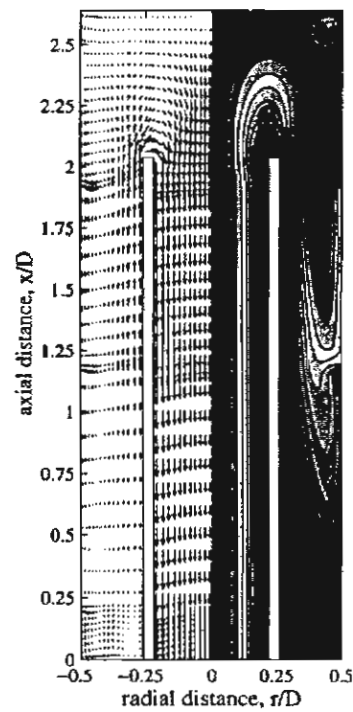
Streamlines และ vectors ของความเร็วแสดงในรูปที่ 6.40(ก) ขณะที่รูปที่ 6.40(ข) เป็นกรณีที่ไม่มีการเผาไหม้เพื่อเปรียบเทียบกัน ในรูปที่ 6.40(ก) และ 6.40(ข) มีอยู่ด้วยกันสองซีกคือซีกด้านซ้ายเป็น vector ของความเร็ว โดยที่ซีกด้านขวาเป็น streamlines สำหรับ vector plots ของความเร็วสังเกตว่ารูปแบบการไหลเหมือนกันทั้งสองแบบ รูปร่างความเร็วในท่อตรงกลางที่ทางออกในกรณีมีการเผาไหม้ค่อนข้างสม่ำเสมอ ความเร็วมีความหนาแน่นในบริเวณใกล้กับผนังท่อตรงกลางดังแสดงในรูปที่ 6.40(ก) และ 6.40(ข) สำหรับ streamlines จะเห็นการหมุนวนที่มีขนาดใหญ่และความรุนแรงกว่าเมื่อสังเกตในการไหลที่มีการเผาไหม้

Contour plots ของอุณหภูมิและความดันแสดงในรูปที่ 6.40(ค) และ 6.40(ง) ตามลำดับ เช่นเดียวกันในรูปซีกด้านซ้ายเป็นกรณีฉีดเชื้อเพลิงตามแนวแกนขณะที่ซีกด้านขวาเป็นกรณีฉีดเชื้อเพลิงทำมุม ใน contour plot ของอุณหภูมิพบว่าการฉีดเชื้อเพลิงทั้งสองแบบให้ผลที่แทบเหมือนกันแต่การฉีดเชื้อเพลิงทำมุมอุณหภูมิบริเวณแกนกลางเกือบจะสม่ำเสมอกันดังเห็นได้ในรูปที่

6.40(ค) จาก Contour ของความดันสถิตยในรูปที่ 6.40(ง) จะเห็นได้อย่างชัดเจนว่าผลที่ได้มีความเหมือนกัน



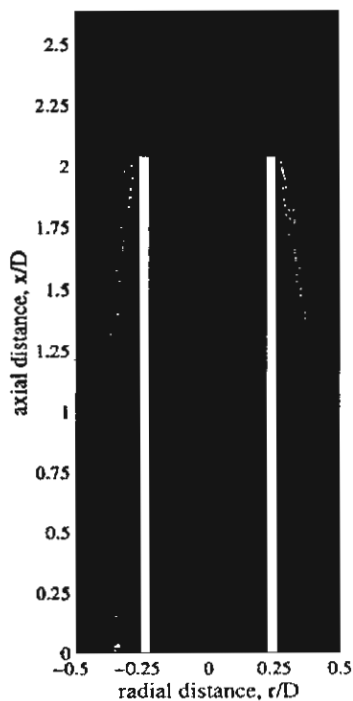
(ก)



(ข)

ฉีดตามแนวแกน

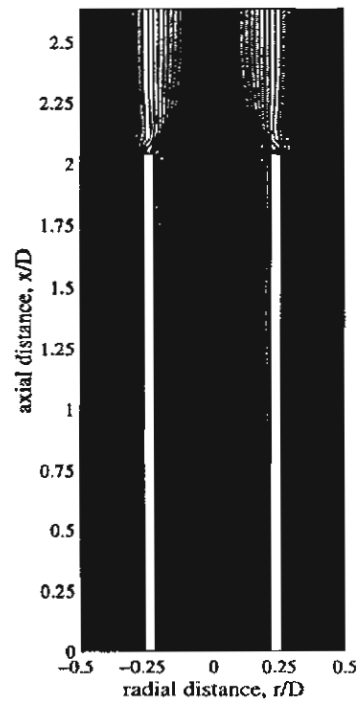
ฉีดทำมุม



(ค)

ฉีดตามแนวแกน

ฉีดทำมุม



(ง)

รูปที่ 6.40 vector และ streamlines ของความเร็ว (ก) การไหลที่มีปฏิกิริยาการเผาไหม้ (ข) การไหลที่อุณหภูมิคงที่; contour plots ของ (ค) อุณหภูมิและ (ง) ความดันที่ทำนายได้

6.7 สรุป

ผลการจำลองการไหลในเตาเผาซึ่งเป็นการเปรียบเทียบระหว่าง $k-\varepsilon$ model กับ ASM ทั้งหมดสามารถสรุปได้ดังต่อไปนี้

1. Recirculation มีผลต่อจำนวนกริดที่น้อยที่สุดในการคำนวณอย่างมาก
2. ASM สามารถทำนายการไหลได้ดีกว่า $k-\varepsilon$ model โดยเห็นอย่างชัดเจนบนความเร็วตามแนวสัมผัส รวมถึงการใช้ ASM ทำให้ลักษณะรูปร่างของความเร็วตามแนวแกน และความเร็วตามแนวรัศมีที่ทำนายได้เปลี่ยนไปจาก $k-\varepsilon$ model เล็กน้อย
3. ลักษณะของการไหลนั้นขึ้นอยู่กับรูปร่างของเตาเป็นอย่างมากกว่าผลของ swirl number (S) หรือการฉีดอากาศ
4. ผลของ numerical differencing scheme ที่ใช้นั้นมีค่อนข้างสูงต่อการไหลที่มี recirculation รวมถึงการใช้ ASM ต่อการไหลที่มี recirculation ควรเลือกใช้ scheme ที่ใช้การประมาณเป็นเส้นตรง (first-order scheme)

สำหรับผลการทำนายการไหลภายในเตาฟลูอิดไดซ์เบดแบบฉีดอากาศทุติยภูมิด้านข้าง ผลจากการเปรียบเทียบ model ที่ใช้พบว่า ASM ทำนายได้ดีกว่า $k-\varepsilon$ model และจากผลการทำนายโดยใช้ numerical differencing scheme ต่างๆ พบว่า upwind และ hybrid schemes ประเมินค่าได้ใกล้เคียงกับผลการทดลองมากกว่า QUICK และ SOU schemes ทั้งในแนวแกนและแนวสัมผัส

ผลการทำนายที่มีความผิดพลาดส่วนหนึ่งน่าจะเป็นผลมาจากการไหลจริงที่เกิดขึ้นเป็นการไหลแบบ 3 มิติ ซึ่งในการทำนายการไหลในเตาฟลูอิดไดซ์เบดแบบฉีดอากาศทุติยภูมิด้านข้างนี้ พิจารณาการไหลแค่ 2 มิติแบบสมมาตรในแนวแกน (axisymmetric)

การคำนวณเชิงตัวเลขที่ทำนายการไหลปั่นป่วนแบบหมุนอย่างรุนแรงและลักษณะการเผาไหม้ในเตาเผาออร์เทคโดยใช้ $k-\varepsilon$ model ผลการคำนวณสำหรับการไหลที่ไม่การทำปฏิกิริยาถูกเปรียบเทียบกับค่าที่ได้จากการทดลอง ผลของจากการวิเคราะห์สามารถสรุปได้ดังต่อไปนี้

สำหรับการไหลในเตาเผาที่ไม่มีปฏิกิริยาเผาไหม้ การไหลของก๊าซในเตาเผาจะเป็นการไหลปั่นป่วนซึ่งมีการหมุนอย่างรุนแรง, เกิด recirculation และ เป็นการไหลแบบ nonisotropic โดยอยู่ภายใต้อิทธิพลของความเร็วตามแนวสัมผัสเป็นส่วนใหญ่

สำหรับการไหลในเตาเผาที่มีปฏิกิริยาเผาไหม้ การคำนวณแสดงให้เห็นว่ารูปแบบการไหลเหมือนกับแบบจำลองการไหลในเตาเผาที่มีปฏิกิริยาเผาไหม้แต่ขนาดความเร็ว, แถบ recirculation มีมากกว่า

บทที่ 7

สรุปผลงานวิจัยและข้อเสนอแนะ

7.1 สรุปผลงานวิจัย

จากการจำลองการไหลปั่นป่วนแบบหมุนวนในเตาเผาทั้งหมดโดยใช้ $k-\varepsilon$ model และ ASM ร่วมกับ numerical differencing scheme ทั้งหมด 4 ชนิด คือ upwind, hybrid, QUICK และ SOU พบว่าการจำลองการไหลมีความสอดคล้องกับข้อมูลจากการทดลองเป็นอย่างดี โดย ASM นั้นสามารถทำนายได้ดีกว่า $k-\varepsilon$ model อย่างเห็นได้ชัด แต่การใช้ ASM นั้นพบว่าจำเป็นจะต้องมีการเลือกใช้ numerical differencing scheme ให้เหมาะสมโดยเฉพาะบนการไหลที่มี recirculation ควรเลือกใช้ numerical differencing scheme ที่ใช้การประมาณแบบเส้นตรงจะมีความเหมาะสมที่สุด รวมถึงลักษณะของการไหลมีผลจากรูปร่างของเตามากกว่า swirl number (S) หรือการฉีดอากาศ

ผลการทำนายการไหลภายในเตาฟลูอิดไดซ์เบดแบบฉีดอากาศทุติยภูมิด้านข้างในแนวสัมผัสโดยเปรียบเทียบระหว่าง $k-\varepsilon$ model กับ ASM ที่ $Q_r = 2$ และ $D_s/D = 1.0$, ผลจากการเปรียบเทียบ model ที่ใช้พบว่า ASM ทำนายได้ดีกว่า $k-\varepsilon$ model และผลของ scheme ที่ใช้พบว่า upwind และ hybrid schemes ประเมินค่าความเร็วในแนวแกนและแนวสัมผัสได้ใกล้เคียงมากกว่า QUICK และ SOU schemes

ส่วนการทำนายการไหลในเตาเผาเอร์เทคโดยใช้ $k-\varepsilon$ model สำหรับการไหลในเตาเผาที่ไม่มีปฏิกิริยาเผาไหม้ การไหลของก๊าซในเตาเผาจะเป็นการไหลปั่นป่วนซึ่งมีการหมุนวนอย่างรุนแรง, เกิด recirculation และ เป็นการไหลแบบ nonisotropic สำหรับการไหลในเตาเผาที่มีปฏิกิริยาเผาไหม้ ผลจากการทำนายแสดงให้เห็นว่ารูปแบบการไหลเหมือนกับการทำนายการไหลในเตาเผาที่มีปฏิกิริยาเผาไหม้แต่ขนาดความเร็ว, แถบ recirculation จะมากกว่า

7.2 ข้อเสนอแนะในการศึกษาวิจัยต่อไป

1. ควรทำการศึกษาเปรียบเทียบรูปร่างของเตาเผาแบบต่างๆต่อลักษณะการไหล
2. ควรมีลองการใช้ Reynolds stress model ทำนายการไหลในลักษณะเดียวกัน เพื่อทำการเปรียบเทียบรวมถึงศึกษาพฤติกรรมที่เกิดขึ้น
3. ควรมีการพัฒนา model ของการเผาไหม้ต่อจากงานวิจัยนี้จาก 2 มิติ เป็น 3 มิติ เพื่อให้งานสมบูรณ์มากขึ้นต่อไป

บทที่ 8

ผลที่ได้จากโครงการวิจัย

โครงการวิจัยนี้มีเป้าหมายหลัก คือ ต้องการศึกษาและเข้าใจอย่างลึกซึ้งเกี่ยวกับพฤติกรรมการไหลในเตาเผาที่ใช้กันในภาคอุตสาหกรรม โดยใช้แบบจำลองทางคณิตศาสตร์มาช่วย อย่างไรก็ตามเนื่องจากข้อจำกัดบางประการทำให้ไม่สามารถจำลองการไหลได้เต็มรูปแบบ คือ สามมิติ จึงใช้ข้อสมมติฐานของการไหลแบบสมมาตรมาช่วยทำให้ลดรูปจาก 3 มิติ เป็น 2 มิติ ได้ เป็นการลดความยุ่งยากได้พอสมควร แต่ก็สูญเสียความแม่นยำบางส่วนไป

ผลจากโครงการวิจัยนี้สามารถจำแนกได้ดังต่อไปนี้

1. ผลงานตีพิมพ์ในวารสารวิชาการนานาชาติ

ปัจจุบันยังไม่มีงานตีพิมพ์ แต่กำลังร่างอยู่ คาดว่าจะมีการตีพิมพ์ในช่วงประมาณ 6 เดือนข้างหน้า เพราะกำลังเก็บข้อมูลเพิ่มเติมบางส่วนเพื่อความสมบูรณ์ของบทความ โดยจะตีพิมพ์ในวารสารนานาชาติดังต่อไปนี้

1.1 JSME International J. of Fluids and Thermal Engineering

1.2 ASEAN Journal on Science & Technology for Development

2. การนำผลงานวิจัยไปใช้ประโยชน์

ถึงแม้ว่างานวิจัยนี้จะไม่สามารถนำไปประยุกต์ใช้ในเชิงพาณิชย์ได้ แต่ก็มีประโยชน์อย่างมากในเชิงวิชาการ เพราะสามารถสร้างนักวิจัยใหม่ ในระดับปริญญาโท ได้ถึง 4 คน และกำลังจะจบปริญญาโทอีก 2 คน นอกจากนี้ยังสร้างความร่วมมือกับนักวิจัยในต่างประเทศ เพื่อแลกเปลี่ยนความรู้ทางวิชาการ (ดูภาคผนวก ข)

3. ผลงานตีพิมพ์ (ทั้งหมดอยู่ในภาคผนวก ก)

3.1 การเสนอผลงานในที่ประชุมวิชาการระดับนานาชาติ จำนวน 5 บทความ ก็คือ

3.1.1 Pongjet Promvong, "Simulation of a Turbulent Channel Flow with Injection Through a Slit", The Sixth Asian International Conference on Fluid Machinery, 18-21 July, 2000, Johor Bahru, Malaysia. (หน้า ก1)

3.1.2 P. Promvong, L. Vongsarnpigoon and K. Silapabanleng "A Low Emission Annular Vortex Combustion Firing Rice Husk Fuel: Part I– Numerical Simulation", The First Regional Conference on Energy Technology Towards a Clean Environment, 1-2 December 2000, Chiang Mai, Thailand. (หน้า ก9)

3.1.3 P. Promvong, L. Vongsarnpigoon and N. piriyarungroj "A Low Emission Annular Vortex Combustion Firing Rice Husk Fuel: Part II–Experimental Investigation", The First Regional Conference on Energy Technology

Towards a Clean Environment, 1-2 December 2000, Chiang Mai, Thailand. (หน้า ก16)

- 3.1.4 P. Promvong, L. Vongsarnpigoon and K. silapabanleng "Computation of Isothermal Confined Swirling Flow in a Combustor", 4th. SEA Regional Conference On Higher Engineering Education Network (RECHEEN 2000), 15-17 November 2000, Kuala Lumpur, Malaysia. (หน้า ก22)
- 3.1.5 P. Promvong and K. silapabanleng "Experimental Study of Combustion Characteristics in a Rice Husk Fired Vortex Combustor", Proceedings of IECEC'01 36th. Intersociety Energy Conversion Engineering Conference, July 29-August 2, 2000, Savannah, Georgia (หน้า ก29)

3.2 การเสนอผลงานในที่ประชุมวิชาการระดับชาติ จำนวน 5 บทความ

- 3.2.1 Pongjet Promvong and Kulthon Silapabanleng, "Simulation of Turbulent Flow through a Circular Orifice" Proceeding of 13th National Mechanical Engineering Conference, December 1999, Vol. 2, pp. 172-176.(หน้า ก36)
- 3.2.2 P. Promvong, L. Vongsarnpigoon and K. silapabanleng "Numerical Simulation of Isothermal Swirling Turbulent Flow in a Combustor", The Fifth Annual National Symposium on Computational Science and Engineering, 19-20 June 2001, Bangkok Convention Center (BCC), Central Plaza. (หน้า ก42)
- 3.2.3 พงษ์เจต พรหมวงศ์ และ ปริญญารวมภักดีกุล, "การจำลองการไหลปั่นป่วนแบบหมุนวนในเตาเผา" สัมมนาวิชาการวิศวกรรมเครื่องกลแห่งประเทศไทย ครั้งที่ 13, 2-3 ธันวาคม พ.ศ. 2542, เล่มที่ 1, หน้า 160-165 (หน้า ก52)
- 3.2.4 พงษ์เจต พรหมวงศ์ ชินรักษ์ เรียรพงษ์ และ สมชาย ศรีพัฒนะพิพัฒน์ "การศึกษาเชิงตัวเลขของการไหลแบบปั่นป่วนในช่องที่มีการฉีดด้านข้าง" สัมมนาวิชาการวิศวกรรมเครื่องกลแห่งประเทศไทยครั้งที่ 14, 2-3 พฤศจิกายน 2543, หน้า 66-72 (หน้า ก59)
- 3.2.5 พงษ์เจต พรหมวงศ์ ปริญญารวมภักดีกุล และ กุลธร ศิลปบรรเลง "การคำนวณการไหลหมุนวนแบบอุณหภูมิกึ่งที่ในเตาเผาโดย Algebraic Reynolds Stress Model" สัมมนาวิชาการวิศวกรรมเครื่องกลแห่งประเทศไทยครั้งที่ 14, 2-3 พฤศจิกายน 2543, หน้า 47-52 (หน้า ก67)

3.3 วารสารวิชาการในประเทศ จำนวน 4 บทความ

- 3.3.1 พงษ์เจต พรหมวงศ์ และ ยุทธนา พลอยฉาย "การทำนายการไหลแบบปั่นป่วนในช่องขนานที่มีคريب" วารสารพระจอมเกล้าลาดกระบังปีที่ 8 ฉบับที่ 3, ธันวาคม 2543, หน้า 1-6 (หน้า ก74)
- 3.3.2 ปริญญา รวมภักดีกุล นิยม ศรีศิริสิทธิกุล และ พงษ์เจต พรหมวงศ์ "การศึกษาเชิงตัวเลขของการไหลแบบปั่นป่วนในเตา" วารสารพระจอมเกล้าลาดกระบังปีที่ 8 ฉบับที่ 3, ธันวาคม 2543, หน้า 13-18 (หน้า ก81)
- 3.3.3 ศุภชัย เลื่อนลอย นิวัติ พิริยะรุ่งโรจน์ และ พงษ์เจต พรหมวงศ์ "อิทธิพลความชื้นของแกลบนต่อคุณลักษณะการเผาไหม้" วารสารพระจอมเกล้าลาดกระบังปีที่ 9 ฉบับที่ 1, เมษายน 2544 (หน้า ก88)
- 3.3.4 สมศักดิ์ โพธิ์ถวิลเกียรติ์ นิวัติ พิริยะรุ่งโรจน์ และ พงษ์เจต พรหมวงศ์ "อิทธิพลของขนาดของแกลบนต่อคุณลักษณะการเผาไหม้" วารสารพระจอมเกล้าลาดกระบังปีที่ 9 ฉบับที่ 1, เมษายน 2544 (หน้า ก94)

เอกสารอ้างอิง

- Ahmed, S.A. 1997a. "An Isothermal Experimental investigation of Turbulence Transport Through an Abrupt Axisymmetric Expansion." **Proc. Instn. Mech. Engrs.** Vol.212 Part G : 45-55.
- Ahmed, S.A. 1997b. "Three Component Velocity Measurements of an Isothermal Confined Swirling Flow." **Proc. Instn. Mech. Engrs.** Vol.211 Part G : 113-122.
- Ahmed, S.A. and Nejad, A.S. 1992. "Premixed, Turbulent Combustion of Axisymmetric Sudden Expansion Flows." **Int. J. Heat and Fluid Flow.** Vol.13 No.1 : 15-21.
- Boysan, F. *et. al.* 1982. "A Fundamental Mathematical Modelling Approach to Cyclone Design." **Trans. Inst. Chem. Eng.** Vol. 60 : 222-230.
- Chen, C.J. and Jaw, S.Y. 1997. **Fundamentals of Turbulence Modeling.** London. : Taylor&Francis.
- Ehalil, E.E. *et. al.* 1975. "The Calculation of Local Flow Properties in Two-Dimensional Furnaces." **Heat Mass Transfer.** Vol.18 : 775-791.
- El-Mahallawy, F.M. and Habib, M.A. 1978. "Study of Mixing of Two Coaxial Swirling Jets in a Cold Model Furnace." 35-47. in Khalil, K.H. *et. al.* **Flow, Mixing and Heat Transfer in Furnace.** London : Pergamon Press
- Fraser, S.M. *et. al.* 1997. "Computation and Experimental Investigations in a Cyclone Dust Separator." **Proc. Instn. Mech. Engrs.** Vol.211 Part E : 247-257.
- Gaski, T.B. *et. al.* 1996. **Simulation and Modelling of Turbulent Flows.** New York. : Oxford University Press.
- Gaski, T.B. and Peyret, R. Editor. 1986. **Handbook of Computational Fluid Mechanics.** London. : Academic Press Ltd.
- Halthore, R.N. and Gouldin, F.C. July 1986. "Laser Scatter Measurement for Gas Densities in a Swirling Flow Combustor." **AIAA J.** Vol.24 No.7 : 1129-1136.
- Hogg, S. and LeschZiner, M.A. January 1989. "Computation of Highly Swirling Confined Flow with a Reynold Stress Turbulence Model." **AIAA J.** Vol.27 No.1 : 57-63.
- Holzapfel, F. *et. al.* September 1999. "Quintuple Hot-wire Measurements of the Turbulence Structure in Confined Swirling Flows." **Trans. ASME J. of Fluid Engineering.** Vol. 121 : 517-525.

- Hwang, C.C. *et. al.* March 1993. "A Comparison of the Linear and Nonlinear $k-l$ Turbulence Models in Combustors." **Trans. ASME J. of Fluid Engineering**. Vol.115 : 93-102.
- Jones, W.P. and Pascau, A. September 1989. "Calculation of Confined Swirling Flows with a Second Moment Closure." **Trans. ASME J. of Fluid Engineering**. Vol.111 : 248-255.
- Kim, K.Y. and Chung, M.K. March 1988. "Calculation of a Strongly Swirling Turbulent Round Jet with Recirculation by an Algebraic Stress Model." **Heat and Fluid Flow J.** Vol.9 No.1 : 62-68.
- Kitoh, O. 1990. "Experimental study of Turbulent Swirling Flow in a Straight Pipe." **Fluid Mechanic J.** Vol.25 : 445-479.
- Launder, B.E. *et. al.* 1975. "Progress in the Development of a Reynold Stress Turbulence Closure." **J. Fluid Mechanics**. Vol.68 : 537-566.
- Lin, C.A. and Lu, C.M. July 1994. "Modeling Three-Dimensional Gas Turbine Combustor Model Flow Using Second-Moment Closure." **AIAA J.** Vol.32 No.7 : 1416-1422.
- Lin, C.H. *et. al.* 1997. "Evaluation of the Combustion Efficiency and Emission of Pollutants by Coal Particles in a Vortexing Fluidized Bed." **Combustion and Flame**. Vol. 110 ; 163-172.
- Lin, C.H. *et. al.* 1998. "A Study on the Swirling Flow Flied in the Freeboard of a Vortexing Fluidized Bed Combustor." **JSME international Journal, Series B**. Vol. 41 : 538-545.
- Lixing, Z. 1993. **Theory and Numerical Modeling of Turbulent Gas-Particle Flows and Combustion**. China. : Science Press and CRC Press.
- Lockwood, F.C. *et. al.* 1974. "An Experimental and Theoretical Investigation of Turbulent Mixing in a Cylindrical Furnace." **Combustion and Flame**. Vol.23 : 283-293.
- Nieh, S. and Yang, G. 1986. "Modeling of Solid Flows in a Fluidized Bed with Secondary Tangential Air Injection in the Freeboard." **AIChE Annual Meeting**, Paper No. 41K.
- Patankar, S.V. 1980. **Numerical Heat Transfer and Fluid Flow**. Washington D.C. : Hemisphere.
- Promvonge, P. and Vongsarnpigoon, L. March-June 1998. "A Simplified Algebraic Reynolds Stress Model for Calculating Confined Swirling Flows." **Eng. Trans. A**

Research Publication of Mahanakorn University of Technology. Vol.1 No.2 :150-161.

Rhode, D.L. *et. al.* April 1983. "Mean Flowfields in Axisymmetric Combustor Geometries with Swirl." **AIAA J.** Vol.21 No.4 : 593-600.

Rodi, W.A. 1976. "New Algebraic Relations for Calculating the Reynolds Stresses." **Z. Angew. Math. Mech. (ZAMM).** Vol.56 : T219-T221.

Ronald, M.C.S. *et. al.* 1984. **An Experimental Investigation of Gas Jets in Confined Swirling Air Flow.** NASA CR 3832.

Serag-Eldin, M.A. and Spalding, D.B. July 1979. "Computations of Three-Dimensional Gas-turbine Combustion Chamber Flows." **Journal of Engineering for Power.** Vol.101 : 327-336.

Sloan, D.G. *et. al.* 1986. "Modeling of Swirl in Turbulent Flow System." **Progr. Energy Combust. Sci.** Vol.12 : 163-250.

Speziale, C.G. 1987. "On Non-Linear $k-l$ and $k-\varepsilon$ models of Turbulence." **J. of Fluid Mechanics.** Vol.178 : 459-475.

Speziale, C.G. 1991. "Annual Review of Fluid Mechanics." **Annual Review Inc.** Vol.23 : 107-157.

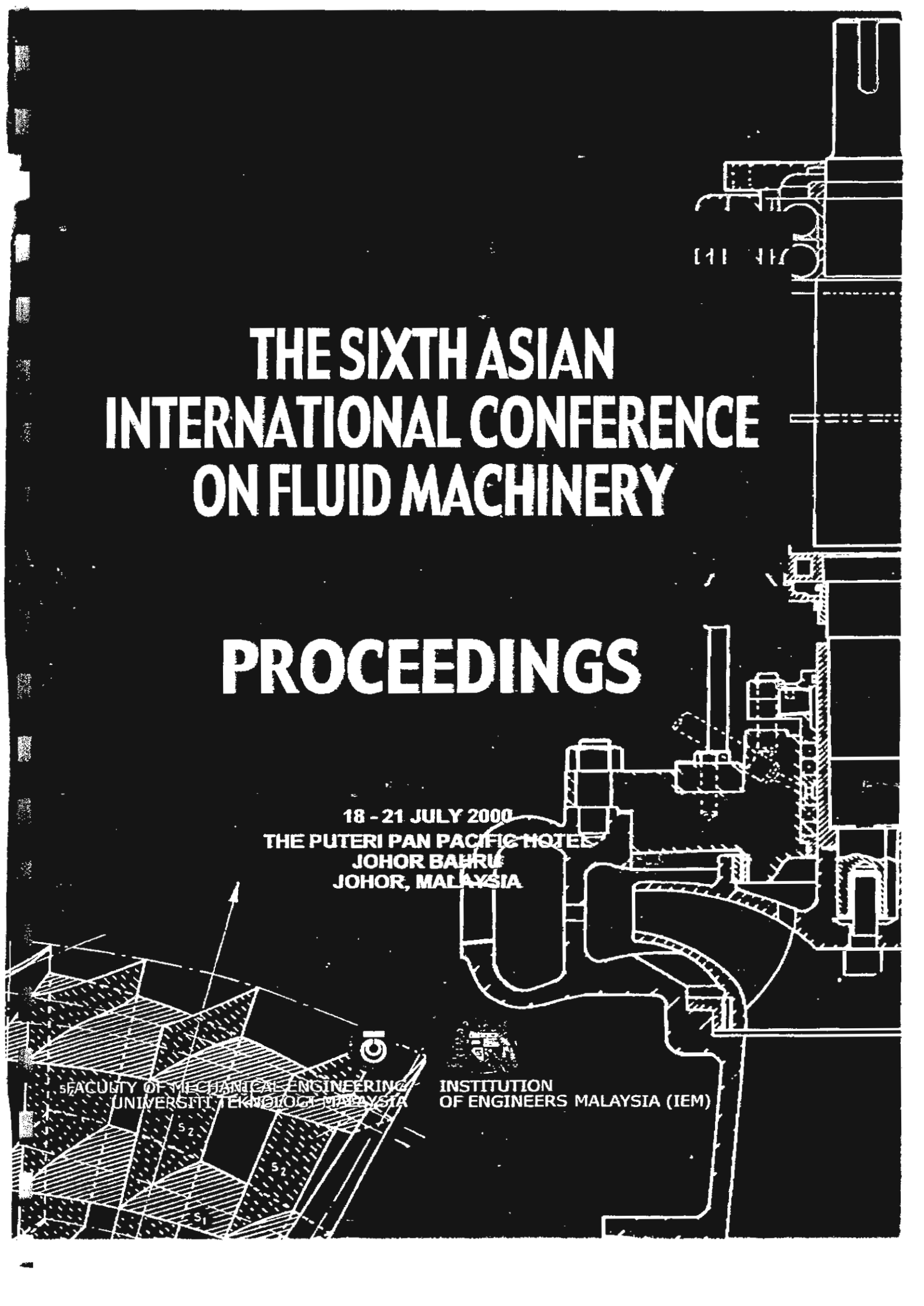
Versteeg, H.K. and Malalasekera, W. 1995. **An Introduction to Computational Fluid Dynamics: The Finite Volume Method.** English. : Longman Scientific & Technical, Longman Group Limited.

Willcox, C.D. 1993. **Turbulence Modelling for CFD.** California. : DCW Industries Inc.

Xia, J.L. *et. al.* July 1999. "Study of Swirling Flow in a Combustion Chamber." **Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference.**

Zhang, J. *et. al.* 1992. "A New Version of Algebraic Stress Model for Simulating Strongly Swirling Turbulent Flows." **J. Numerical Heat Transfer.** Vol.22 Part B : 49-62.

ภาคผนวก ก
บทความตีพิมพ์



THE SIXTH ASIAN INTERNATIONAL CONFERENCE ON FLUID MACHINERY

PROCEEDINGS

18 - 21 JULY 2000
THE PUTERI PAN PACIFIC HOTEL
JOHOR BAHRU
JOHOR, MALAYSIA

FACULTY OF MECHANICAL ENGINEERING
UNIVERSITI TEKNOLOGI MALAYSIA

INSTITUTION
OF ENGINEERS MALAYSIA (IEM)

SIMULATION OF A TURBULENT CHANNEL FLOW WITH INJECTION THROUGH A SLIT

Pongjet Promvonge

Department of Mechanical Engineering, Faculty of Engineering,
King Mongkut's Institute of Technology Ladkrabang, Bangkok 10520, THAILAND.
Email: kppongje@kmitl.ac.th

ABSTRACT

The paper presents the simulation of a turbulent channel flow with injection through a slit. A Finite Volume approach with a non-uniform and staggered grid system is employed in the present simulation. To account for the turbulence nature of the flow, an algebraic Reynolds stress model (ASM) and the standard $k-\epsilon$ turbulence model are incorporated in the time-averaged governing equations. Effect of numerical diffusion on the calculated results is also investigated by comparing between a second-order-differencing scheme for the convection transport and a first-order one. The influences of injection through the slit on mean flow properties in a fully developed turbulent channel flow are investigated and compared with available measurements. The computation reveals that use of ASM leads to slightly accurate results than that of the standard $k-\epsilon$ turbulence model.

1. INTRODUCTION

The injection of a side-jet into a cross-flow is encountered in several engineering applications, such as transpiration cooling of turbine blades, the mixing process for the combustor cooling, the transition from hover to forward flight of a V/STOL aircraft, the entrainment of jet pump and drag reduction. Injection of fluid from a wall to the flow is a way to control wall turbulence. When the jet is injected into a cross-flowing stream, there is a complicated interaction between the jet and the cross-flow. Its nature of this kind of flow is still not well understood and need to clarify how mean flow and turbulence characteristics are affected by the fluid injection. There have been many reports published on the jet in a turbulent cross-flow. Andreopoulos [1] presented an investigation of velocity fluctuation statistics in the jet-pipe flow normal to a cross-stream for different ratios of jet-to-crossflow velocities. Sumitani and Kasagi [2] studied turbulent channel flow with uniform injection and suction on the two opposite channel walls by a direct numerical simulation (DNS) and obtained various statistic characteristics on turbulence. Kato et al. [3] presented a method to reduce turbulent frictional drag by injection of high viscosity fluid into the boundary layer. The flow characteristics in a two-dimensional channel with nonuniform injection using both laser-Doppler velocimetry measurements and DNS for comparison can be found in Liou et al. [4]. Sano [5] investigated experimentally the turbulent channel flow with various side-jet injection flow rates. The brief survey presented above reveals that there has been no report of numerical investigation on the mean flow properties in a channel flow with side injection with an algebraic Reynolds stress model.

This article deals with the numerical simulation of a fully developed turbulent channel flow with a side-jet injection of Sano [5] by utilizing the ASM and various numerical schemes. The process of recovery to equilibrium state from non-equilibrium state is investigated numerically by predicted wall static pressure and velocity profiles.

2. MATHEMATICAL FORMULATION

2.1 Governing Equations

The phenomenon under consideration is governed by the steady two-dimensional axisymmetric form of the continuity and the time-averaged incompressible Navier-Stokes equations. In the Cartesian tensor system these equations can be written in the following form:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \tau_{ij}) \quad (2)$$

where ρ , u_i , p and x_i are the density, mean velocity tensor, mean pressure and coordinate tensor respectively. The mean viscous stress tensor is approximated as:

$$\bar{\tau}_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

where μ is laminar viscosity. The time-averaged Reynolds stress tensor, $\tau_{ij} = -\overline{\rho u_i' u_j'}$, in the above equation is not known and thus, models are needed to express it in terms of the solution variables. In the present study, two turbulence models are used, namely the k - ϵ model and an algebraic stress model (ASM). The k - ϵ model has been reviewed in references [6] and it will be described briefly. The standard k - ϵ model version relates the turbulent eddy viscosity to the turbulence kinetic energy k and the dissipation rate ϵ through Boussinesq's approximation as:

$$\tau_{ij} = -\frac{2}{3} \delta_{ij}(\rho k) + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

where $\mu_t = \rho C_\mu k^2/\epsilon$ is the turbulent eddy viscosity and ϵ is the dissipation rate of turbulence kinetic energy (TKE). The modelled equation of the TKE, k is given by:

$$\frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G - \rho \epsilon \quad (5)$$

in which $\mu_e = \mu_t + \mu$ is effective viscosity. Similarly the dissipation rate of TKE is given by the following equation:

$$\frac{\partial}{\partial x_j}(\rho u_j \epsilon) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + \frac{\epsilon}{k} (C_{\epsilon 1} G - C_{\epsilon 2} \rho \epsilon) \quad (6)$$

where G is the rate of generation of the TKE while $\rho \epsilon$ is its destruction rate. G is given by:

$$G = \mu_e \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \right] \quad (7)$$

The boundary values for the turbulent quantities near the wall are specified with the wall function method [7]. $C_\mu = 0.09$, $C_{\epsilon 1} = 1.44$, $C_{\epsilon 2} = 1.92$, $\sigma_k = 1.0$, and $\sigma_\epsilon = 1.3$ are empirical constants [6] in the turbulence transport equations. Reynolds-averaged transport equations can be solved for τ_{ij} , [6] the modelled equations for which are:

$$\frac{\partial \tau_{ij}}{\partial t} + \frac{\partial (u_k \tau_{ij})}{\partial x_k} = -G_{ij} - \Phi_{ij} + D_{ij} + \epsilon_{ij} \quad (8)$$

where $G_{ij} = \text{local production} = \rho P_{ij} = - \left(\overline{\rho u_i' u_k'} \frac{\partial u_j}{\partial x_k} + \overline{\rho u_j' u_k'} \frac{\partial u_i}{\partial x_k} \right)$

$\Phi_{ij} = \text{local pressure strain} = -C_1 \frac{\rho \epsilon}{k} \left(\overline{u_i' u_j'} - \frac{2}{3} k \delta_{ij} \right) - C_2 \left(G_{ij} - \frac{2}{3} G \delta_{ij} \right)$

$D_{ij} = \text{net diffusive transport} = - \frac{\partial}{\partial x_k} \left(\left(\frac{\mu_e}{\sigma_T} \right) \frac{\partial \overline{u_i' u_j'}}{\partial x_k} \right)$

$\epsilon_{ij} = \text{local dissipation tensor} = \frac{2}{3} \rho \epsilon \delta_{ij}$

in which $C_1 = 2.5$, and $C_2 = 0.55$ are model constants.

2.2 Algebraic Reynolds Stress Model (ASM)

For simplicity in solving the six Reynolds stresses, Rodi's approximation [8] is used in this study and the Reynolds stress transport can be expressed in algebraic form as follows:

$$\frac{D\tau_{ij}}{Dt} - D_{ij} = \frac{\tau_{ij}}{\rho k} \left(\frac{Dk}{Dt} - D_k \right) \quad (9)$$

Substitution of eqs. (5) and (8) into eq. (9) gives the desired algebraic expression for τ_{ij} :

$$-G_{ij} - \Phi_{ij} + \frac{2}{3}\delta_{ij}\rho\epsilon = \frac{\tau_{ij}}{\rho k} (G - \rho\epsilon) \quad (10)$$

The ASM expressions can thus be rewritten as:

$$\overline{\rho u_i u_j} = \frac{2}{3}\delta_{ij}\rho k + \frac{\lambda k}{\epsilon} \left(G_{ij} - \frac{2}{3}\delta_{ij}G \right) \quad (11)$$

where the empirical constant λ , was found to be 0.135, [9] is defined as:

$$\lambda = \frac{1 - C_2}{C_1 - 1 + \frac{G}{\rho\epsilon}} \quad (12)$$

2.3 Common Form for the Equations

All the governing equations can be re-organised and expressed in a standard form that includes the convection, diffusion, and source terms for 2-D axisymmetric flows as follows:

$$\frac{\partial}{\partial x}(\rho u \phi) + \frac{\partial}{\partial y}(\rho v \phi) - \frac{\partial}{\partial x} \left(\Gamma_{\phi x} \frac{\partial \phi}{\partial x} \right) - \frac{\partial}{\partial y} \left(\Gamma_{\phi y} \frac{\partial \phi}{\partial y} \right) = S_{\phi} \quad (13)$$

where ϕ may stand for any variable including the velocity components, $\Gamma_{\phi x}$ and $\Gamma_{\phi y}$ are the exchange coefficients for ϕ , and S_{ϕ} is the source term.

2.4 Solution Procedure

In the present computation, the time-averaged Navier-Stokes equations, the TKE equation, and the TKE dissipation rate equation are solved numerically by a control-volume finite-difference method [10] together with the turbulence model equations, equation (4) for the k - ϵ model or equation (11) for the ASM. All equations are in a generalised form of equation (13). The SIMPLE algorithm is utilised for pressure-velocity de-coupling and iteration [10, 11]. The first-order upwind (FOU) and the second-order upwind (SOU) [12] schemes were used for discretising convection and diffusion transports on a staggered grid cell. The under-relaxation iterative TDMA line-by-line sweeping technique is used for solving the resultant finite-difference equations. The computation was carried out using a personal computer (Pentium III-450 MHz). About 7,000 iterations were needed to achieve satisfactory convergence for each calculation case, which requires about 1.0 hr of computer time.

3. CHANNEL FLOW WITH SIDE-INJECTION OF SANO [5]

A flow in channel with a side jet of Sano [5] was employed in the present simulation. A schematic configuration of the channel is shown in Fig. 1 below. The channel with a length of about 3,050-mm has a width (h) of 30-mm and the slit width (L) for injection is 10 mm. Profile measurements of axial velocity, wall-static pressure, Reynolds stresses, and wall shear stresses were provided and measured by using a constant-temperature-type hot-wire anemometer. The mean velocity u_{in} at the channel inlet was set at 10.0 m/s while side-injection flow rate, $Q = vL/hu_{in}$ where v = injection velocity, was set at three values: 0.067, 0.133 and 0.2. The measurements were made at eight stations, namely, $x = 70, 110, 150, 250, 350, 500, 700$, and 950 mm from the slit.

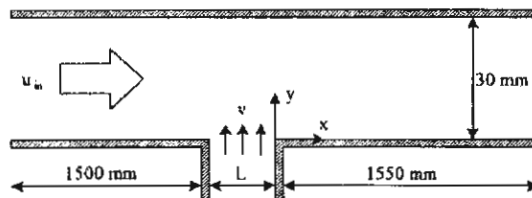


Fig. 1 Geometry of channel with side-jet injection.

4. RESULTS AND DISCUSSION

The comparisons between the predicted results by the $k-\epsilon$ turbulence model and the ASM with different numerical schemes and the measured data of the flow are presented in Figs. 2 through 7. The predicted axial velocities are compared with the measurements, where solid or dash curves are represented for the calculated results while open circles for the measured data. The computational results are based on an 80×40 non-uniform grid with refinement in the vicinity of the slit. Grid independence of the numerical results was verified with a 120×60 finer grid. It is found that the differences for both the base grid and the finer grid in local flow properties are marginal. This suggests that grid independent solutions can be obtained with an 80×40 grid, which is used throughout the computations.

4.1 Effects of Numerical Schemes

The axial mean velocity profiles predicted with the $k-\epsilon$ model and the FOU and SOU schemes are compared with the measured data in Figs. 2 and 3 for side-injection flow rates of 0.067 and 0.2 respectively. A closer examination reveals that predictions with both the schemes generally are in good agreement with the measurements. However, in prediction with the SOU scheme in Fig. 3, under-predicted results are seen in the vicinity of the slit while excellent agreement is found in the upper wall region in comparison with experimental data. Overall, use of the SOU leads to slight improvement so the FOU is sufficient for computation of this flow.

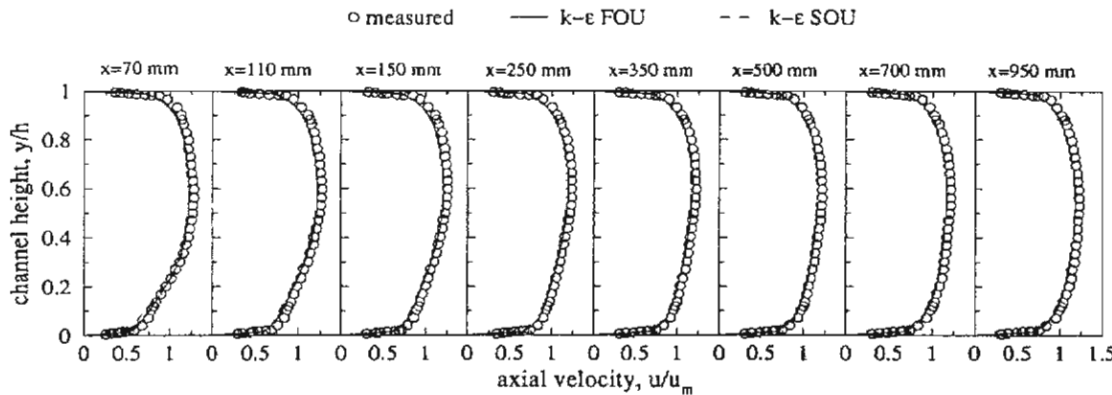


Fig. 2 Effects of numerical schemes on streamwise velocity profiles with measurements for $Q = 0.067$

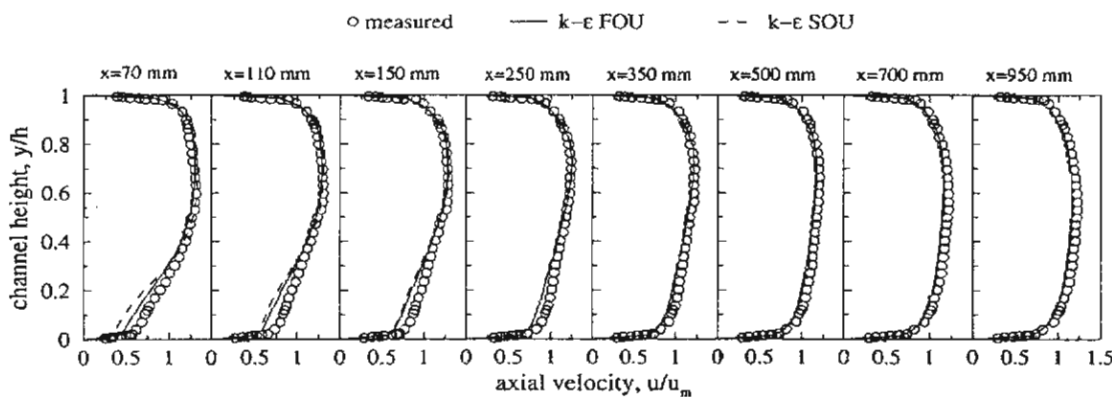


Fig. 3 Effects of numerical schemes on streamwise velocity profiles with measurements for $Q = 0.2$

4.2 Flow Field

Velocity Field. Predictions using the $k-\epsilon$ model and the ASM are compared with experimental data in Figs. 4, 5 and 6 for the injection flow rates of 0.067, 0.133 and 0.2 respectively. It is found that for lower injection flow rate, $Q = 0.067$, the predicted profiles of streamwise mean velocity are in excellent agreement with measurements for both turbulence models as depicted in Fig. 4. However, increase in side-injection flow rate leads to significant deviations between predicted and measured results in the vicinity of side-jet port but both predicted and measured profiles agree well after station $x=350$ -mm as can be seen in Figs. 5 and 6. Close examination

reveals that as the flow proceeds toward the injection slit, the mean velocity profiles gradually becomes nonuniform and asymmetrical near the wall and returns to uniform and symmetric after advancing in the downstream direction. The ASM gives slight under-prediction close to the injection port while the $k-\epsilon$ model predicts slightly better. But for further downstream from the slit, results predicted by both turbulence models are similar and agree very well with measurements.

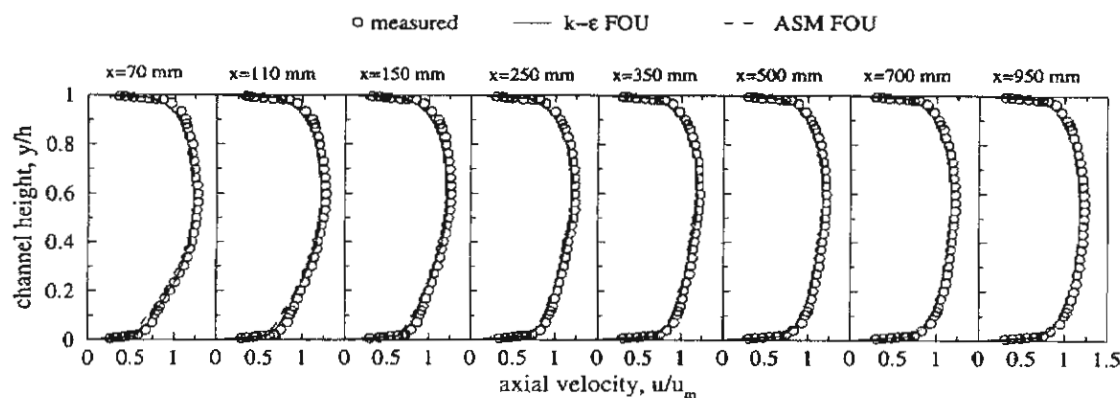


Fig. 4 Comparison of predicted streamwise velocity profiles with measurements for $Q = 0.067$

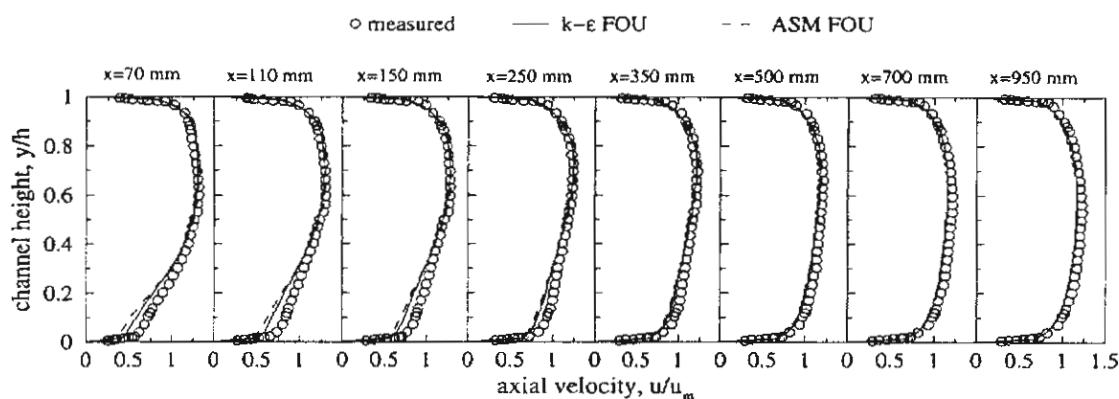


Fig. 5 Comparison of predicted streamwise velocity profiles with measurements for $Q = 0.133$

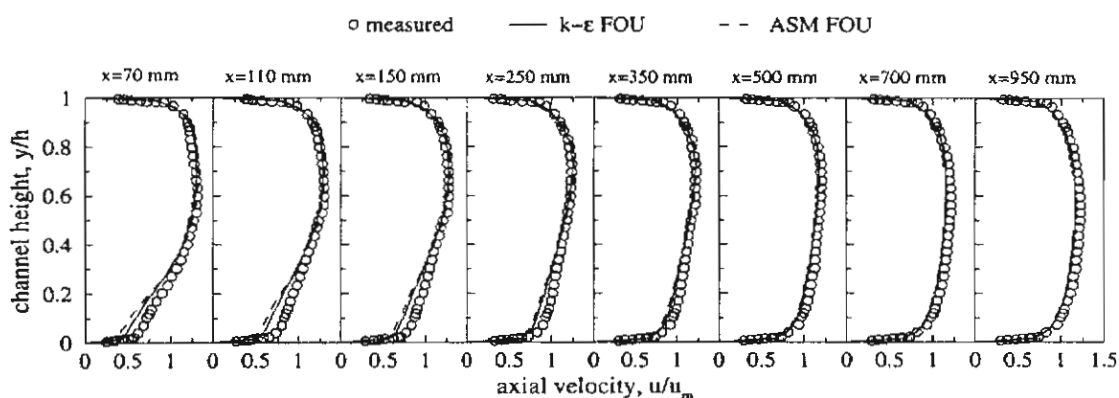


Fig. 6 Comparison of predicted streamwise velocity profiles with measurements for $Q = 0.2$

Pressure Field. Figure 7 compares the non-dimensional profiles of static pressure along the upper and lower walls using the FOU scheme with the measurements. It is found that the pressure profiles for the lower wall are nearly similar to those for the upper wall except for the

vicinity of the slit. A close look reveals that there is a high-pressure drop across the side-jet port for both walls. This suggests that injection to the flow suffers substantially from a pressure loss. Predictions with both turbulence models show favourable agreement only in the downstream region of the slit. However, for upstream of the slit, the calculated wall pressures for both walls rise faster than the experiment shows. At upstream regions from the slit, the ASM results mimic experimental data better while the $k-\epsilon$ model ones are over-predicted especially at high flow rates of injection. Again, use of the ASM results in slight improvement for this flow.

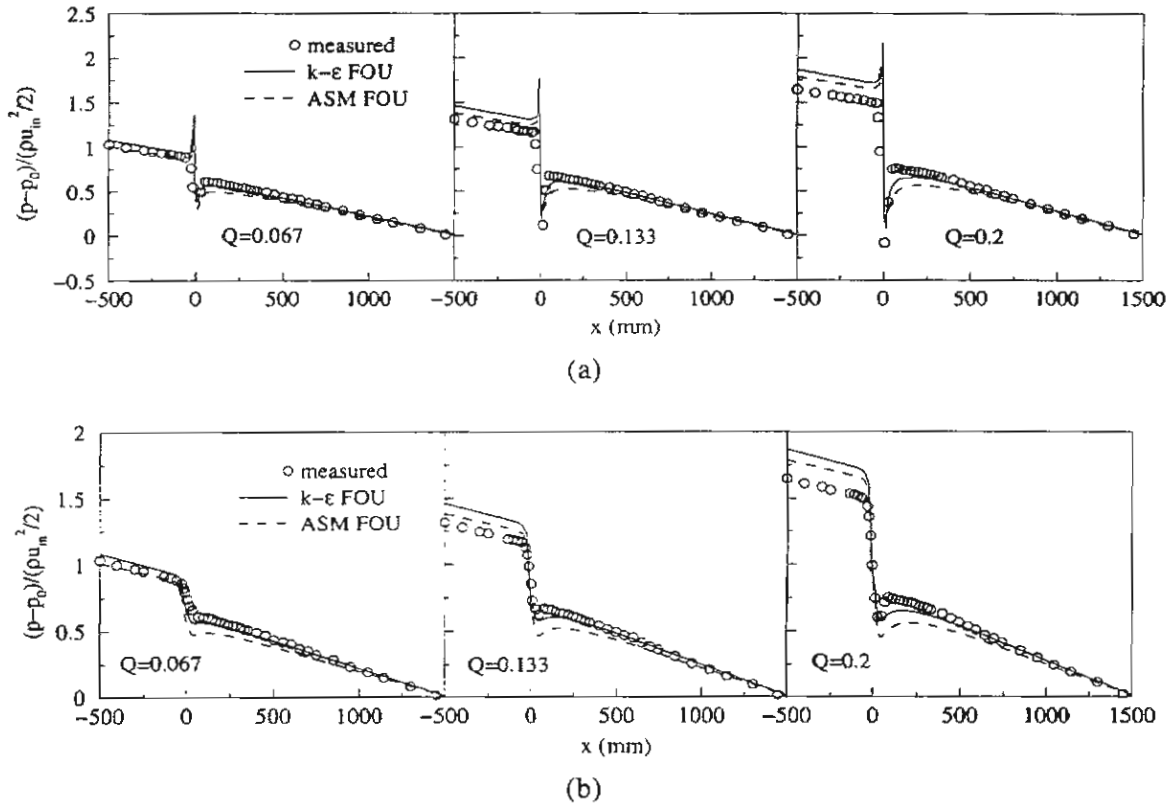


Fig. 7 Comparison of predicted wall static pressure distributions with measurements under different side-injection flow rates, Q , for (a) lower wall and (b) upper wall (p_0 = exit pressure).

5. CONCLUDING REMARKS

Simulations of turbulent channel flow with side-injection have been carried out by utilizing the $k-\epsilon$ model and ASM with two numerical schemes. The prediction of mean flow properties was compared with measurements. The conclusions from the investigation can be drawn as follows:

1. The predicted axial velocity and wall-static pressure profiles by the FOU and SOU schemes for both turbulence models are generally in good agreement with measurements.
2. ASM with the FOU scheme shows small improvement over the $k-\epsilon$ model.
3. The FOU and SOU schemes give slightly under-predicted in the vicinity of the slit for the lower wall but slightly over-predicted results for the upper wall in comparison with measurements.
4. The higher side-injection flow rate leads to substantial under-prediction in the vicinity of the side-jet slit.

6. ACKNOWLEDGEMENTS

The authors would like to gratefully acknowledge the Thailand Research Fund for financial support of this research.

7. REFERENCES

1. Andreopoulos, J., Measurements in a Jet-Pipe Flow Issuing Perpendicularly into a Cross Stream, ASME J. of Fluids Engineering, Vol. 104, pp. 493-499, 1982.
2. Sumitani, Y. and Kasagi, N., Direct Numerical Simulation of Turbulent Transport with Uniform Wall Injection and Suction, AIAA J., Vol. 33, No. 7, pp. 1220-1228, 1995.
3. Kato, H., Fujii, Y., Yamaguchi, H., and Miyanaga, M., Frictional Drag Reduction by Injecting High-Viscosity Fluid into Turbulent Boundary Layer, Trans. ASME, J. Fluids Engineering, Vol. 115, 1993.
4. Liou, T.M., Lien, W.Y. and Tsai, G.S., Experimental and Computation Studies of Fluid Flow in a Two-dimensional Porous-walled Channel, 9th Symposium on Turbulent Shear Flows, 5-4, 1993.
5. Sano, M., Mean Flow and Turbulence Characteristics in a Turbulent Channel Flow with Injection through a Slit, Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference, Paper No. FEDSM99-7283, 18-23 July, San Francisco, California, 1999.
6. Wilcox, C.D., Turbulent Modelling for CFD, DCW Industries, Inc., California, 1993.
7. Patankar, S.V. and Spalding, D.B., Heat and Mass Transfer in Boundary Layers, Intertext Books, London, 1970.
8. Rodi, W.A., New Algebraic Relations for Calculating the Reynolds Stresses, Z. Angew. Math. Mech. (ZAMM), vol. 56, pp. T219-T221, 1976.
9. Promvonge, P., Numerical Simulation of Turbulent Compressible Vortex-Tube Flow, Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference, Paper No. FEDSM99-6972, 18-23 July, San Francisco, California, 1999.
10. Patankar, S.V., Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington, D.C., 1980.
11. Versteeg, H.K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Longman Scientific & Technical, Longman Group Limited, Essex, England, 1995.
12. Shyy, W., Thakur, S., and Wright, J., Second Order Upwind and Central Difference Schemes for Recirculating Flow Computation, AIAA J., Vol.30, No.4, pp.931-932, 1992.



The First Regional Conference on Energy Technology Towards a Clean Environment

1st-2nd December 2000

**The Empress Hotel,
Chiang Mai,
Thailand**

**Efficient Energy and Clean Environment
A Review of The Past and a Look at
the Next Decade**



The Joint Graduate School of Energy and Environment

A Low Emission Annular Vortex Combustor Firing Rice Husk Fuel: Part I – Numerical Simulation

P. Promvonge¹, L. Vongsarnpigoon², S. Sripattanapipat¹ and K. Silapabanleng³

1. Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok 10520, Thailand. Email: kppongje@kmitl.ac.th
2. Department of Mechanical Engineering, Faculty of Engineering, Mahanakorn University of Technology, Bangkok 10530, Thailand.
3. Department of Mechanical Engineering, Faculty of Engineering, Chulalongkorn University, Bangkok 10330, Thailand.

Abstract

The paper presents the simulation of the strongly swirling turbulent flow and combustion characteristic in an annular vortex combustor (VC) for burning rice husk. Computations, based on a staggered finite volume method, were carried out by utilising the standard k- ϵ turbulence model for the closure of the second-order correlation moments in the governing equations. Cold flow and mixture fraction combustion models for gas fuel were used to investigate the effects of various operating parameters on the mean flow properties in the VC. Some of calculated results are compared with available measurements.

Introduction

To help increase rice husk utilisation in an efficient, clean manner, a novel non-slugging vortex combustor (VC) was recently developed and investigated for drying/heating application purposes. The VC technique first developed for coal fuel [1] is evolved from the basic understanding of swirling gas-particle flows and combustion in vortex chambers [2]. It integrates many advantages from cyclone combustor, fluidized bed combustor, multistage combustion and pulverized coal-fired combustor; while eliminates most of their inherent disadvantages. This combustor is featured with two vertical, concentric tubes with multiple air injection. Due to the strong swirl and low temperature combustion environments, the VC has demonstrated to pose many inherent advantages: vigorous gas-particle turbulence, long particle residence time, high combustion intensity, and broad fuel flexibility.

As schematically shown in Fig. 1, primary air is provided into the lower chamber (or the bottom), together with solid fuel. Secondary air is tangentially injected into the annular chamber of the VC through the nozzles arranged at various strategic levels to form a strong swirl. Combustion occurs primarily in the annular space. Then, hot flue gas spirally ascends to the top before leaving through the exhaust center tube. The combined effects of centrifugal, gravitational, and fluid drag forces give rise to fuel particles to be trapped along the height of the combustor. As particles are burned they continually reduce in mass and size until completely burned out. The majority of ash particles become light or small enough to be entrained by the flue gas and exit the combustor as flyash. Some may remain in the combustor bottom as dry ash. Hence, it is necessary to know the detailed aerodynamic structure and the distinct flow characteristics of the VC. However, little study pertaining to its flow properties has been made and numerical study of combusting rice husk flow in the VC has never come across due to its difficulty in modelling.

This article deals with the numerical results of the isothermal and combusting gas flows in a VC of Nieh and Zhang [1], along with only the measured cold flow results for comparison. This investigation is to help expand the technical database, better understand the involved physical processes, and guide the design and operation of the vortex combustor.

Mathematical Modelling and Solution Procedure

Governing Equations for Flow Model: The phenomenon under consideration is governed by the steady two-dimensional axisymmetric form of the continuity and the time-averaged incompressible

Navier-Stokes equations. In the Cartesian tensor system these equations can be written in the following form:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \tau_{ij}) \quad (2)$$

where ρ , u_i , p and x_i are the density, mean velocity tensor, mean pressure and coordinate tensor respectively. The mean viscous stress tensor is approximated as:

$$\bar{\tau}_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

where μ is laminar viscosity and $\tau_{ij} = -\overline{\rho u_i u_j}$ is the time-averaged Reynolds stress tensor. The standard k - ε model relates the turbulent eddy viscosity to the turbulence kinetic energy k and the dissipation rate ε through Boussinesq's approximation as:

$$\tau_{ij} = -\frac{2}{3} \delta_{ij}(\rho k) + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

where $\mu_t = \rho C_\mu k^2 / \varepsilon$ is the turbulent eddy viscosity and ε is the dissipation rate of turbulence kinetic energy (TKE). The modelled equation of the TKE, k is given by:

$$\frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G - \rho \varepsilon \quad (5)$$

in which $\mu_e = \mu_t + \mu$ is effective viscosity. Similarly the dissipation rate of TKE is given by the following equation:

$$\frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} (C_{\varepsilon 1} G - C_{\varepsilon 2} \rho \varepsilon) \quad (6)$$

where G is the rate of generation of the TKE while $\rho \varepsilon$ is its destruction rate. G is given by:

$$G = \mu_e \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \right] \quad (7)$$

The boundary values for the turbulent quantities near the wall are specified with the wall function method [3]. $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1.0$, and $\sigma_\varepsilon = 1.3$ are empirical constants [4] in the turbulence transport equations.

Gaseous Phase Combustion: The combustion model is based upon the assumption of single chemical reaction with fast chemistry with respect to the turbulent time scale. The mass fractions of fuel, oxidant and products are only functions of the mixing rate. In a combustion system, a fuel stream flows into the chamber at a rate of f kg/s and an oxidant stream at a rate of $1-f$ kg/s. They mix in the combustion chamber to form a mixture which flows out at a rate of 1.0 kg/s. For adiabatic, steady flow, the fuel and oxidant enthalpies can be related to f as follows [5]:

$$f = \frac{h - h_{ox}}{h_{fu} - h_{ox}} \quad \text{or} \quad f = \frac{(C_p T + m_{fu}) - (C_p T)_{ox}}{(C_p T + H)_{fu} - (C_p T)_{ox}} \quad (8)$$

where f called the mixture fraction, is between 0.0 and 1.0, H and m_j are the heat of combustion and mass fraction of species j . From the partial differential equation of f , the flame temperature (T) can be

obtained. Thus, the density ρ , is calculated from ideal gas equation, $\rho = MP/\bar{R}T$ in which $1/M = \sum(m_j/M_j)$, $\bar{R}$ is the universal gas constant and M is the mean mixture molar mass.

Common Form for the Equations: All the governing equations can be re-organised and expressed in a standard form that includes the convection, diffusion, and source terms for 2-D axisymmetric flows as follows:

$$\frac{\partial}{\partial x}(\rho u \phi) + \frac{\partial}{\partial y}(\rho v \phi) - \frac{\partial}{\partial x}\left(\Gamma_{\phi} \frac{\partial \phi}{\partial x}\right) - \frac{\partial}{\partial y}\left(\Gamma_{\phi} \frac{\partial \phi}{\partial y}\right) = S_{\phi} \quad (9)$$

where ϕ may stand for any variable including the velocity components, Γ_{ϕ_x} and Γ_{ϕ_y} are the exchange coefficients for ϕ , and S_{ϕ} is the source term.

Solution Procedure: In the present computation, the time-averaged Navier-Stokes equations, the TKE equation, the TKE dissipation rate equation and the mixture fraction equation are solved numerically by a control-volume finite-difference method [6] together with the turbulence model equations. All equations are in a generalised form of equation (9). The SIMPLE algorithm is utilized for pressure-velocity de-coupling and iteration [6, 7]. The first-order upwind, hybrid and the QUICK [8] schemes were used for discretising convection and diffusion transports on a staggered grid cell. The under-relaxation iterative TDMA line-by-line sweeping technique is used for solving the resultant finite-difference equations. The computation was carried out using a personal computer (Pentium III-550 MHz). About 20,000 iterations were needed to achieve satisfactory convergence for each calculation case, which requires about 2.0 hr of computer time.

Vortex Combustor Flow

A strongly swirling flow in a VC [1] was selected and its schematic diagram of the VC is shown in Fig. 1. It comprised a 0.11 m diameter central tube, concentrically placed inside a cylindrical chamber of 0.66 m height and 0.25 m diameter. The center tube was set 0.51 m into the chamber. Air injected into the chamber tangentially through 3 sets of nozzles spirally ascended through the annular space and exhausted through the central tube. The measured swirl number (S) was 13.9. Experimental data of mean velocities are provided at ten axial locations, namely, $x = 0.051, 0.112, 0.173, 0.234, 0.295, 0.358, 0.42, 0.483, 0.544$, and 0.605 m. The specifications of the combustor are summarised in Table 1 below.

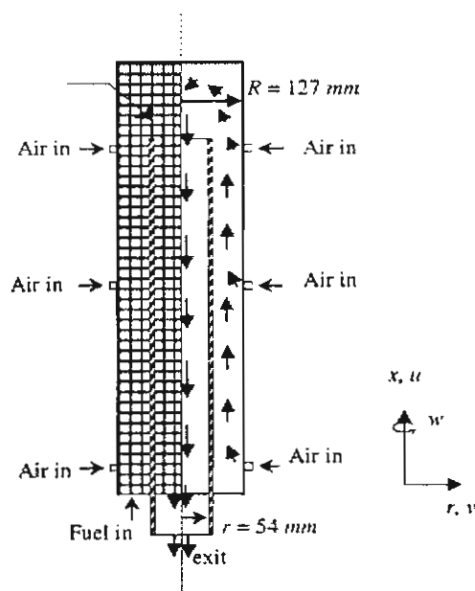


Table 1 Geometry and flow conditions for the vortex chamber [1].

Parameter	Magnitude		
Vortex chamber geometry			
chamber inner dia., m		0.25	
chamber height, m		0.66	
centre tube outer dia., m		0.13	
centre tube inner dia., m		0.11	
centre tube height, m		0.51	
Multiple air injection			
nozzle height distribution, m	0.051	0.295	0.483
flow rate distribution, m^3/h	226	226	226
tangential velocity, m/s	24.6	24.6	24.6
air temperature, K	300	300	300

Fig. 1 Flow arrangement, computational domain and grid for the VC [1].

For modelling of the combustor VC flow, methane (CH_4) gas fuel was employed instead of rice husk because of simplicity. The gas fuel was injected at the bottom into the lower chamber. To study the effect of inlet fuel injection on combustor flow properties for keeping air injection constant, two types of the injection were made. One was the axial injection and the other was the oblique fuel jet (30-degree from a tangential line).

Results and Discussion

Prediction of Cold VC Flow: The influence of grid mesh distribution on the predicted velocity profiles is investigated by comparison with measurements for different grid densities. It is found that a grid size of 40×20 or finer is sufficient to ensure grid-independence of results. The axial and tangential velocity variations in Figs. 2a and 2b, respectively, for various numerical schemes along with the measurements show that, except for minor differences, all three schemes for the convective terms yield almost the same results. This indicates that any of the numerical schemes can be used to calculate the flow with adequate accuracy and thus, only upwind scheme was carried out throughout. These figures indicate that the predictions with the $k-\varepsilon$ model resulted in poorer agreement with the experimental data especially in the top cylindrical region. For example, the $k-\varepsilon$ model fails to predict the central toroidal recirculation zone and the combined forced and free vortex motion in the upper region of the combustor (at $x = 0.544$ and 0.605 m). However, in the present study, the $k-\varepsilon$ model was still used for giving a trend of the combustor flow characteristics in the VC since it is robust and easy for use.

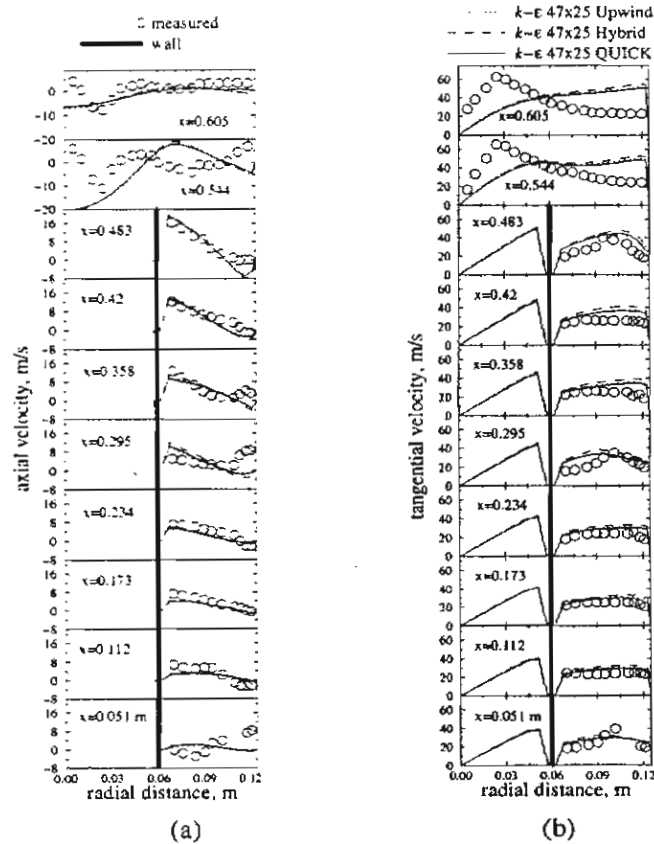


Fig. 2 Comparison of measurements with predictions using different schemes, (a) axial and (b) tangential velocities.

Prediction of Reacting VC Flow: The influence of inlet fuel injection on the flow and temperature fields was studied by using two different inlet fuel injection types. The first type was made by setting the fuel inlet nozzle to obtain a fuel jet parallel to the axial axis, called the axial fuel jet. The second type was set to obtain an oblique fuel jet with an inclination angle of 30° along the tangential line at

the fuel inlet. The predicted axial velocity, tangential velocity, static pressure and temperature profiles for both fuel injections are depicted in Fig. 3a, 3b, 3c and 3d, respectively. It is worth noting that both fuel injections have no effect on the mean flow properties such as velocities and pressure as can be seen in Fig. 3a, 3b and 3c. When compared to the cold flow model, the axial velocity at the outlet for this case is higher (about 3 times). For the temperature profiles, It is found that the oblique fuel injection leads to a higher temperature in the upper chamber while gives the same temperature in the annular region as shown in Fig. 3d. This indicates that inlet fuel should be tangentially injected into the chamber when uniform temperature inside the VC is needed.

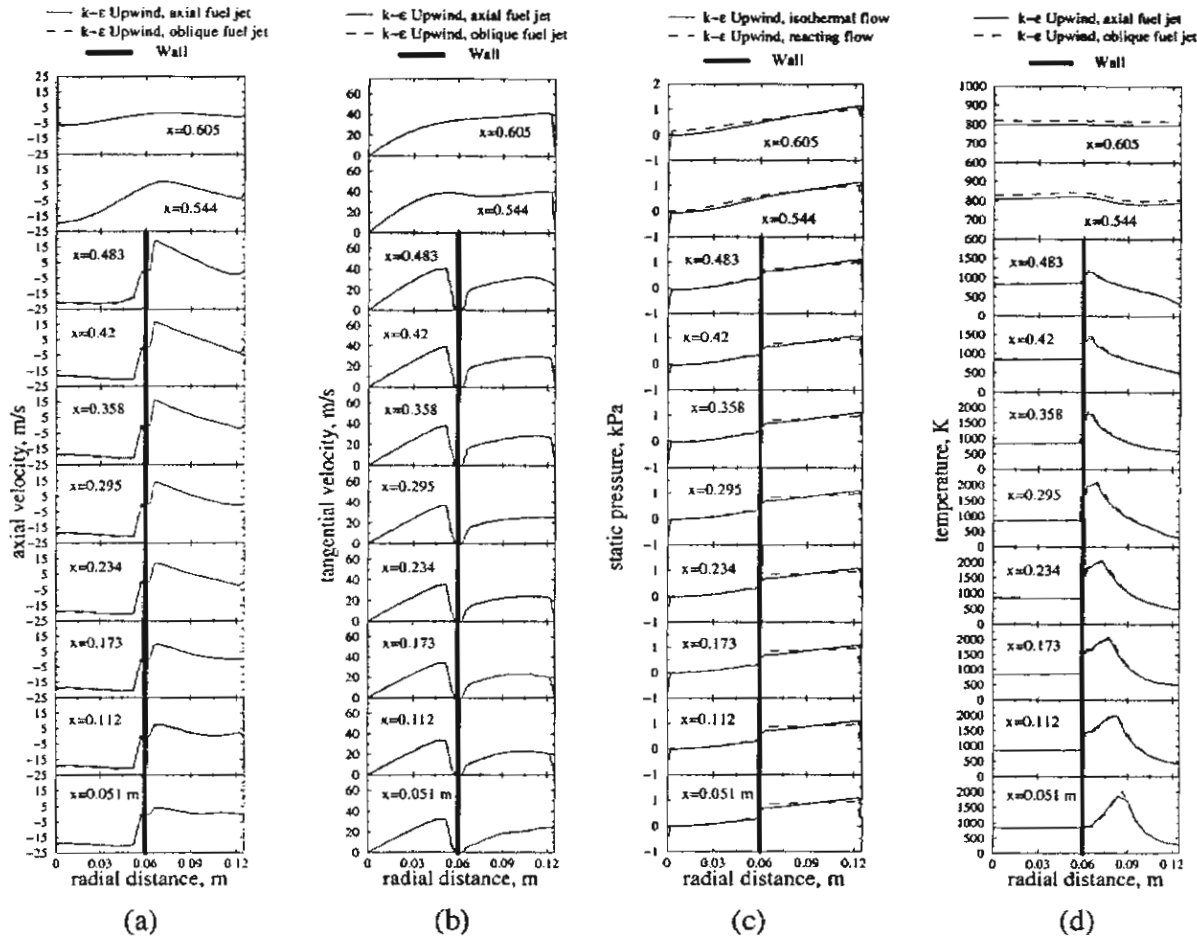


Fig. 3 Radial profiles of (a) axial and (b) tangential velocities (c) static pressure, and (d) temperature calculated with different fuel injections.

Streamlines and velocity vectors are presented in Fig. 4a while Fig. 4b is for the cold flow for comparison. In Figs. 4a and 4b, each has two parts: the left half contains the velocity vectors whereas the right half contains the streamlines. For the vector plots of velocity, it should be noted that the flow patterns are similar for both flow models. The velocity profile in the exhaust center pipe region for the reacting flow is relatively uniform. The velocity is more concentrated in the vicinity of the center pipe walls for both models as can be seen in Fig. 4a and 4b. For streamlines, the stronger and wider recirculation zone is observed for the reacting flow.

The contour plots of temperature and pressure are exhibited in Figs. 4c and 4d respectively. Also, in the figures the left half is for the axial fuel injection while the right half for oblique fuel injection. For the temperature contours, it is found that both inlet fuel injections yield almost similar results but the oblique fuel injection performs a nearly uniform temperature along the core region as can be seen in Fig. 4c. From the contours of static pressure in Fig. 4d, it is clear that both fuel injections give identical results.

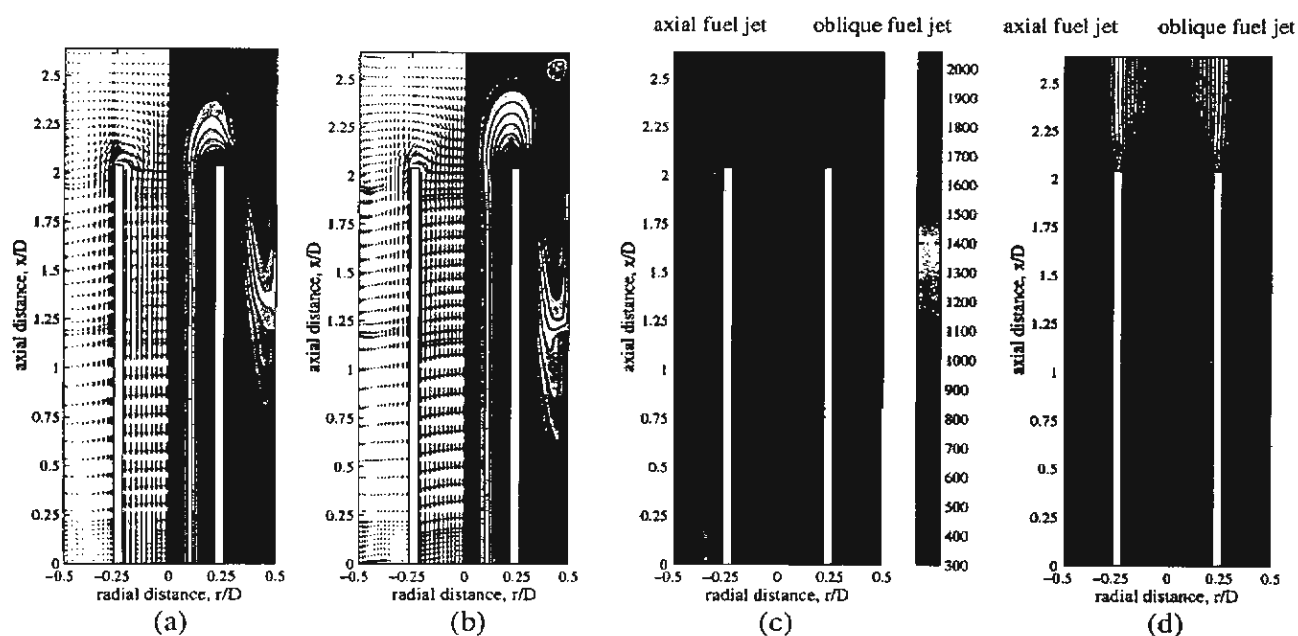


Fig. 4 Velocity vectors and streamlines of (a) reacting flow and (b) isothermal flow; contour plots of (c) predicted temperature and (d) pressure.

Conclusions

Numerical investigations have been carried out to predict a strongly swirling turbulent flow and combustion characteristic in a vortex combustor using the $k-\epsilon$ model. The present numerical results for the cold flow are compared with available experimental data. Major findings can be summarised as follows: For the cold flow, the gas flow in the combustor is characterised as a strongly swirl, recirculating, and nonisotropic turbulent flow with a dominant tangential velocity component. Local recirculating flows occur near the injecting nozzles, at the chamber bottom, at the top of the center tube, and in the vortex core. For the reacting flow model, the computation shows that the flow pattern is similar to the cold flow model but the magnitude of velocity, recirculation zone are greater.

Acknowledgements

The authors would like to acknowledge with appreciation the Thailand Research Fund (TRF) for financial support of this research.

References

- [1]. Nieh, S. and Zhang, J. (1992), Simulation of the Strongly Swirling Aerodynamic Field in a Vortex Combustor, *Trans. ASME J. of Fluids Engineering*, Vol.114, Sept.
- [2]. Gupta, A.K., Lilley, D.G., and Syred, N. (1984), *Swirl Flows*, Abacus Press, Turnbridge Wells, England.
- [3]. Patankar, S.V. and Spalding, D.B. (1970), *Heat and Mass Transfer in Boundary Layers*, Intertext Books, London.
- [4]. Sloan, D.G., Smith, P.J., and Smoot, L.D. (1986), Modeling of Swirl in Turbulent Flow System, *Progr. Energy Combust. Sci.*, vol. 12, pp. 163-250.
- [5]. Turns, S. R., *An Introduction to Combustion*, McGraw-Hill, 1996.
- [6]. Patankar, S.V. (1980), *Numerical Heat Transfer and Fluid Flow*, Hemisphere, Washington, D.C.
- [7]. Wilcox, C.D. (1993), *Turbulent Modelling for CFD*, DCW Industries, Inc., California.
- [8]. Leonard, B.P. (1979), A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation, *Comp. Meth. Appl. Mech. Eng.*, 19, pp.59-98.

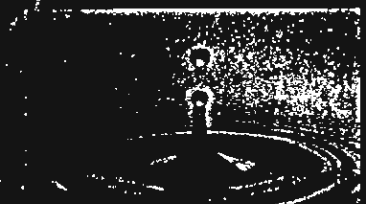


1st. Third Regional Conference on Energy Policy and Environment

1st-2nd December 2000

The Empress Hotel,
Chiang Mai,
Thailand

Efficient Energy and Clean Environment
A Review of The Past and a Look at
the Next Decade



The Joint Graduate School of Energy and Environment

A Low Emission Annular Vortex Combustor Firing Rice Husk Fuel: Part II – Experimental Investigation

P. Promvonge¹, L. Vongsarnpigoon² and N. Piriyaungroj¹

1. Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok 10520, Thailand. Email: kppongje@kmitl.ac.th
2. Department of Mechanical Engineering, Faculty of Engineering, Mahanakorn University of Technology, Bangkok 10530, Thailand.

Abstract

This paper presents the study of flow and combustion characteristics of rice husk fuel in an annular vortex combustor (VC). The temperature distributions for selected locations inside the combustor and the fly ash and smoke from its flue gas were measured and observed respectively. Measurements were made by setting constant mass flow rate of rice husk to be 0.3 kg/min and by varying the equivalence ratio, Φ , to be 0.8, 1.0 and 1.2. The ratio of volumetric flow rates of the primary air to the secondary air, (λ), was set to be 4 and 1.5. The experiment shows the maximum temperature of about 1300 K in the annular chamber with less smoke of flue gas. Besides, the sizes of flyash particles from the exhaust stack can be controlled by the flow rate ratios of the primary air to the secondary air, λ . The VC shows an excellent performance, high stabilization and ease of operation in firing the fine rice husk and its important feature is that some small ashes remain in the bottom of the VC.

Keywords: annular vortex, vortex combustor, rice husk, biomass.

Introduction

The conventional sources of energy have been depleting at an alarming rate and the price of conventional energy is going up. Thus, the focus on alternative renewable sources of energy has been increasing and biomass is one of them that has been getting continued and increased attention. In general, paddy rice is one of the mostly produced crops throughout the world. Since it is not possible to take the whole of it as food, about 22% by weight of this paddy rice is generated as a waste, known as rice husk. It is estimated that over 60 million tons of rice husk are generated each year worldwide. Thailand generates 4.4-4.6 million tons of rice husk annually, which their thermal potentials are equivalent to 1.46-1.53 million tons of crude oil. About 10% of rice husk are utilized as a source of heat energy in Thailand. It is creating waste management problem, especially in the rice milling sites. The characteristic of rice husk is presented in Table 1. Therefore, an attempt to energy recovery from this rice husk waste by combustion technique may be worthwhile. With this the burning of rice husk by an annular vortex combustor (VC) can be considered.

There have been many reports published on various combustors using biomass materials or coals as a fuel. Tuatawin and Watcharaapapaibul [1] studied the performance of a concentric-vortex furnace firing rice husk. The relation between air-fuel ratio and efficiency of the furnace was reported. Jittrejit [2] and Nanumchok [3] presented an experimental investigation of combustion characteristics in a cyclone combustor for burning sawdust. The combustion characteristics in a fluidized bed furnace using rice husk fuel for fluidized bed paddy dryer can be found in Chujinda [4]. Srisawas [5] also presented an improvement of Chujinda's furnace. An extensive study of a vortex combustor for burning dry ultrafine coal and coal water fuel was reported by Nieh and Fu [6]. For above combustors burning rice husk fuel, effects of ash inside the combustors on combustion efficiency including its problem in elimination were reported.

This article deals with a preliminary study of combustion characteristics and temperature distributions in a vortex combustor burning rice husk fuel. Effect of feeding secondary air on flame

stability, temperature control in the combustor, ashes and smokes from the exhaust stack are studied and observed for a future reference in improving the performance of the combustor.

Table 1 Composition of rice husk [4]

Carbon	36.8 %
Hydrogen	5.8 %
Oxygen	37.4 %
Nitrogen	0.14 %
Sulfur	0.1 %
Moisture	7.8 %
Ash	18.9 %
Density, kg/m ³	100.
Gross heat of combustion, kcal/kg	3,308.
Stoichiometric air	3.2731 kg/kg fuel

Experimental Setup

The rice husk particles were milled and grinded, sieved up to about 1-mm size (between 1.19 and 1.41 mm), and stored in the laboratory under dry condition (9.2% of moisture content). The arrangement of experimental system of the combustor is shown in Fig. 1 below. The combustor is a concentric cylindrical pipe made of steel while of stainless steel for the exhaust center pipe. It is 0.755-m high and 0.195-m inside diameter with multiple injection nozzles of 0.005-m dia. each for the secondary air (Q_s) as depicted in Fig. 2. The exhaust center pipe is 0.078 m in diameter.

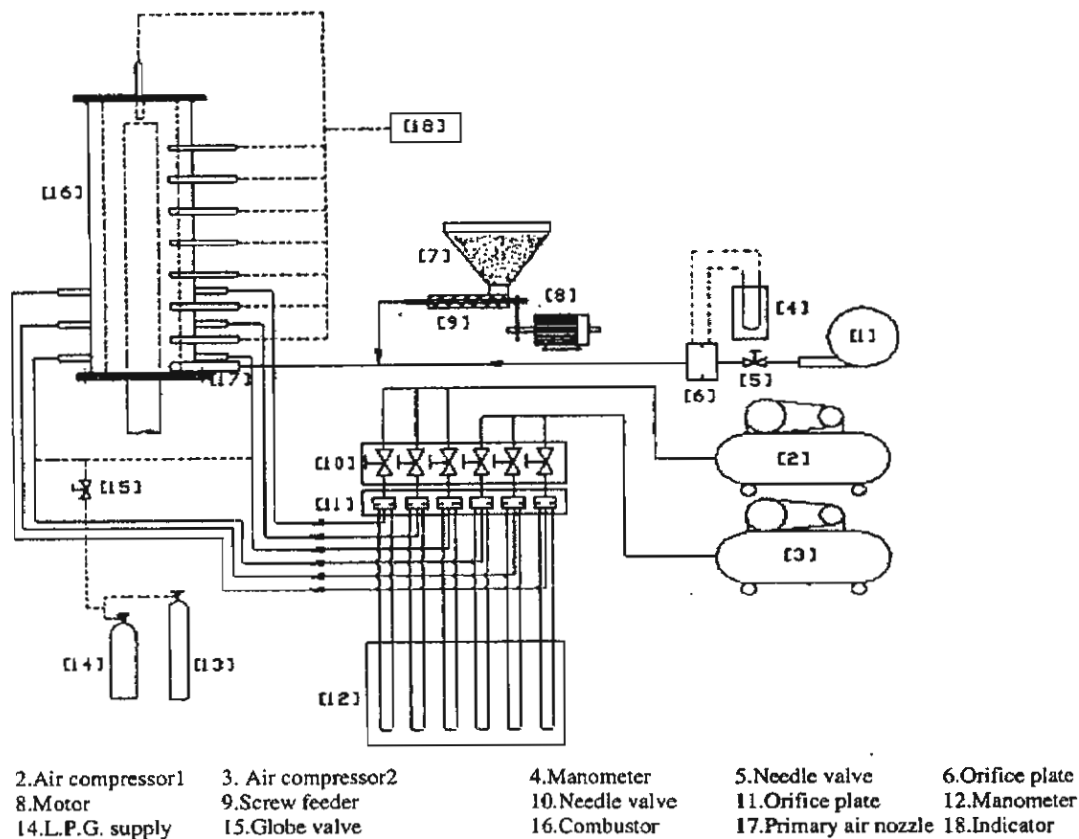


Fig. 1 Schematic diagram of experimental setup of combustion system.

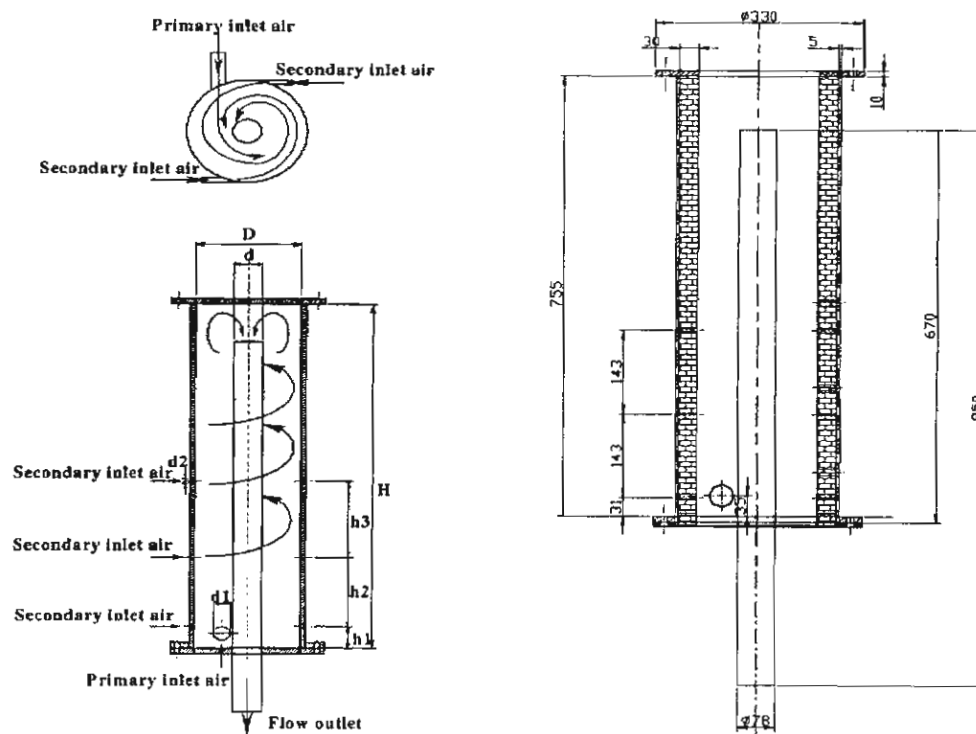


Fig. 2 Configuration of vortex combustor (unit in mm.)

The combustor was operated over a temperature range from 800 – 1000 K. A blower and two compressors were used for providing both primary and secondary combustion airs. Rice husk fuel was fed through a screw feeder and injected to the bottom chamber by pneumatic conveying via the primary air (Q_p). Start up process was commenced by heating up the VC with LPG torch inserted at the lower air nozzle slot. The preheating took about 20 minutes for the chamber to raise its temperature to be about 680 K. Then feeding commenced through the hopper, slowly with the rice husk until 0.3 kg/min and kept constant. When the temperature in the chamber reached 950 K, stop of preheating with LPG was made. A thick, black smoke was seen at the beginning and slowly thinned out by adjusting the air/fuel (A/F) ratio. Excess air at various levels for equivalence ratio of 0.8, 1.0 and 1.2 was tried for each of λ . The temperatures were monitored at various selected locations with chromel-alumel (type K) thermocouples while volumetric flow rates of primary and secondary airs were measured by using orifice meters. All data collection was taken at steady state condition.

Results and Discussion

The VC was originally designed to accommodate various types of coal fuels as sources of heat. The VC responded significantly to the various fuels as expected. For rice husks, the level of equivalence ratio (or excess air) was varied to optimize the temperature inside the VC. The measurements of temperature were made at seven axial locations namely, $x = 0.1, 0.2, 0.3, 0.4, 0.5, 0.6$ and 0.7 m.

Effects of Equivalence Ratios

The radial profiles of temperature inside the VC for $\Phi = 0.8$ (rich), $\Phi = 1.0$ (stoich.) and $\Phi = 1.2$ (lean) are presented in Figs. 3a, 3b and 3c respectively. The feed-rate of fine rice husk with 9.2% moisture content was 0.3 kg/min and kept constant throughout.

For $\Phi = 0.8$, Fig. 3a, it is found that the temperature profiles for all locations generally are not uniform. The temperature in the vicinity of the center pipe is higher than that near the chamber walls. Maximum temperature of about 1400 K takes place in the top of chamber while lower temperature can be seen near the chamber wall. Unburned fuel and black smoke emission from the exhaust stack were

observed for this case. This indicates the incomplete combustion (high CO concentration) or combustion reaction takes place very slowly.

Fig. 3b shows the temperature distributions for $\Phi = 1.0$. It can be seen that except for few early stations, the temperature profiles are nearly uniform for all stations especially at the top chamber. The maximum temperature around 1300 K is found in the top annular space whereas the minimum temperature in the vicinity of the chamber walls at early stations. This shows insufficient turbulence at the bottom chamber, which was due to too many rice husks, resulting in inadequate mixing and consequently the presence of fuel rich zones within the bottom space. This case yielded a less black smoke emission from the exit pipe.

The temperature distribution for $\Phi = 1.2$ is depicted in Fig. 3c. It is worth noting that the temperature profiles similar to the case for $\Phi = 1.0$, are nearly uniform for all stations. The peak temperature of some 1200 K is also seen in the top annular space while lowest temperature at the bottom one. The peak one for every station is found in the middle between the chamber and center pipe walls. The maximum temperature for this case has the lowest value among the three cases, as expected, since an increase in the excess air leads to decreasing temperature in the combustion system. White and thin smoke emission and small size of flyash particles are visible. This points out that complete combustion takes place for this case.

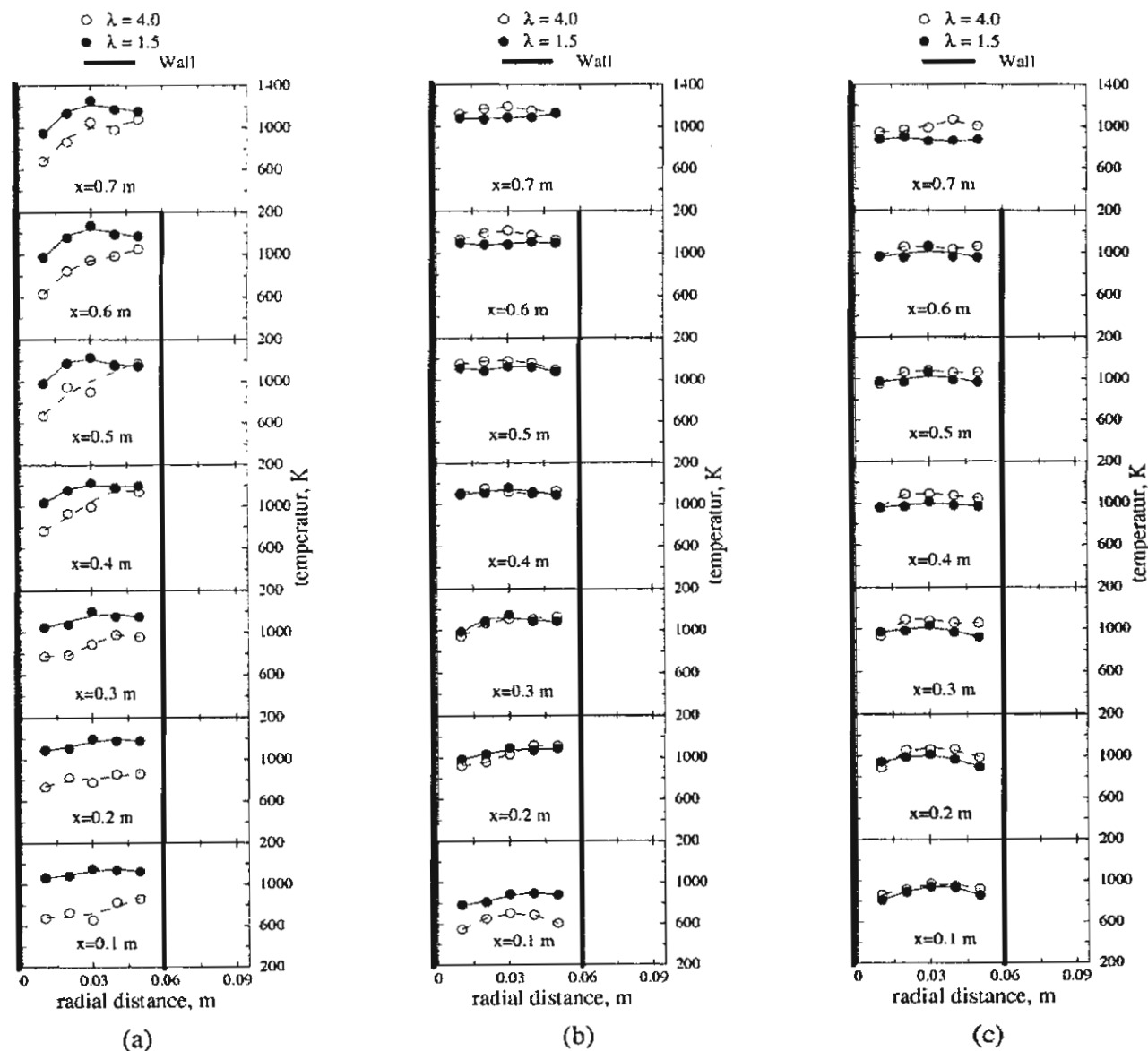


Fig. 3 Temperature distributions inside the combustor, (a) $\Phi = 0.8$, (b) $\Phi = 1.0$, and (c) $\Phi = 1.2$.

Effects of Volumetric flow rate ratios of Primary to Secondary Airs (λ)

The ratio of volumetric flow rate of the primary air to the secondary air, defined as $\lambda = Q_p/Q_s$, is an indication of the strength of the vortex of the flow. The decrease in value of λ results in the higher tangential velocity component (strong vortex). This means that the impact of swirling phenomenon of the secondary air on the flow and temperature fields becomes more pronounced as the value of λ is reduced. Comparison of the temperature distributions between $\lambda = 4$ and $\lambda = 1.5$ for various equivalence ratios is presented in Fig. 3.

For $\Phi = 0.8$, it is obvious that adjusting of $\lambda = 1.5$ leads to a flatter and higher temperature distribution curve than that of $\lambda = 4$. This indicates that the use of a small value of λ can improve the temperature distributions in the VC for the rich mixture as can be seen in Fig. 3a. When $\Phi = 1.0$, it is interested to note that the influence of λ on the temperature profiles shows only the first two stations at the bottom while no effect of λ on the temperature profiles is found at other stations as illustrated in Fig. 3b. As $\Phi = 1.2$, it is worth noting that no effect of λ on the temperature distributions is seen at all stations as exhibited in Fig. 3c. It is concluded that for the value of Φ above 1.0, use of a small value of λ results in no significant effect on temperature profiles in the VC but enhances swirling intensity instead. However, the increased swirling intensity leads to an abatement of particle elutriation, and to enlargement of the region of recirculating zone.

Conclusions

The measurements of temperature distributions and the observation of flue gas emission have been conducted during the combustion of fine rice husk in the VC. The conclusions derived from the results of these experiments are as follows: combustion of rice husk with equivalence ratio between 1.0 and 1.2 using the VC yields good combustion efficiency, flame stabilization, low emission and reliable furnace condition. For equivalence ratio ≥ 1.0 , the increase in secondary air has no significant effect on the temperature distributions inside but helps reduce the size of flyash particles.

Acknowledgements

The authors would like to acknowledge with appreciation, the Thailand Research Fund (TRF) for the financial support in part and the Mechanical Engineering Department, Mahanakorn University of Technology (MUT) for their support in this effort and the MUT undergraduate students for data collection.

References

- [1]. Tuatawin, T and Watcharaapapaibul, S., Bachelor Thesis, Mechanical Engineering Department, King Mongkut's University of Technology Thonburi, Bangkok, 1991.
- [2]. Jittrejit, P., Master Thesis, Mechanical Engineering Department, King Mongkut's Institute of Technology North Bangkok, Bangkok, 1992.
- [3]. Nanumchok, S., Master Thesis, Mechanical Engineering Department, King Mongkut's Institute of Technology North Bangkok, Bangkok, 1996.
- [4]. Chujinda, A., Master Thesis, Department of Energy Technology, King Mongkut's University of Technology Thonburi, Bangkok, 1997.
- [5]. Srisawas, B., Master Thesis, Department of Energy Technology, King Mongkut's University of Technology Thonburi, Bangkok, 1998.
- [6]. Nieh, S. and Fu, T.T., Proceeding of The 5th International Coal Conference, 1988, Pittsburgh, pp. 761-768.

**4th SEA Regional Conference On Higher Engineering Education Network
(RECHEEN 2000)
15-17 November 2000
Kuala Lumpur, Malaysia.**

THEME: ADVANCED RESEARCH IN SCIENCE, ENGINEERING AND TECHNOLOGY

15 September 2000

Pongjet Promvonge, Lerkiat Vongsarnpigoon & Kulthorn Silapabanleng
Dept. of Mech. Eng, King Mongkut's Inst of Tech. Ladkrabang, Bangkok,, Thailand

Title of Paper : Computation of isothermal confined swirling flow in a
combustor

→ Dear Pongjet Promvonge,

I am pleased to inform you that your paper entitled as above has been accepted
for the conference. Please prepare your paper based on the author's kit enclosed
and return to me by 5th October 2000.

Thank you and see you at the conference.

Best regards.



Prof. M. Mahmoud Moghavvemi
Chairman Technical Committee

COMPUTATION OF ISOTHERMAL CONFINED SWIRLING FLOW IN A COMBUSTOR

Pongjet Promvonge

Dept. of Mechanical Engineering, Faculty of Engineering,
King Mongkut's Institute of Technology Ladkrabang,
Bangkok 10520, Thailand. Email: kppongje@kmitl.ac.th

Lerkiat Vongsarnpigoon

Dept. of Mechanical Engineering, Faculty of Engineering,
Mahanakorn Univeristy of Techonology, Bangkok, Thailand.

Kulthorn Silapabanleng

Dept of Mechanical Engineering, Faculty of Engineering,
Chulalongkorn University, Bangkok 10330, Thailand.

Abstract: The paper presents the simulation of strongly swirling confined flow in a combustor with an algebraic Reynolds stress model (ASM). The k - ϵ turbulence model was also employed in the present simulation for comparison. For the flows compared, the predicted results of gas tangential and axial velocities based on this model are in good agreement with measurements. The computation shows that the ASM is superior to the k - ϵ model in capturing the stabilising effect of the swirl.

Keywords: swirl, turbulent flow, algebraic stress model.

1. INTRODUCTION

Swirl flows have been of considerable interest over the past decades because of their occurrence in industrial applications, such as furnaces, utility boilers, internal combustion engines, gas turbine combustors and dust collectors [2, 5]. Swirl has been used in combustion systems to enhance the flame stability, the mixing and heat transfer besides prolonging the fuel residence time and abating the pollutants. This is because under appropriate conditions, swirl can be employed to induce a central recirculation zone. The recirculating flow generates additional turbulence in the shear layer between the reverse flow and the surrounding forward flow and helps stabilize the flame in combustors. Swirling turbulent flows are physically complex in nature due to the effect of a swirl-turbulence interaction. The turbulence structure in swirling turbulent flows is generally highly non-isotropic and nonhomogeneous.

Computation of swirling flows is a difficult and challenging task. Large velocity gradients appear in these flows, so numerical problems and turbulence modelling play a significant role in their analysis. The commonly used, the k - ϵ model may not be suitable for simulating swirling turbulent flows [3, 6, 7]. It is also found that the use of modified k - ϵ models or even the non-linear k - ϵ model [13] leads to no significant improvement of the predictions in swirling flows. The second-order moment closure models, i.e., the Reynolds stress model (RSM) and the algebraic Reynolds stress model provide better methods for the simulation of swirling turbulent flows [6, 7]. The

RSM is regarded as a most logical approach to the turbulence closure problem, which does not need any ad hoc modification for extra strain rates. However, in the prediction of swirling flows with the RSM, it is necessary to solve a total of 11 governing differential equations of elliptic type: a continuity equation, three momentum equations, an ϵ -equation, and six equations for the Reynolds stresses. This leads to much extra computational effort to solve six Reynolds stress transport equations simultaneously [6, 7] and much attention needs to be paid to numerical stability and inlet boundary conditions. It is for this reason that a simplified algebraic Reynolds stress turbulence model [16] in axisymmetric cylindrical coordinates is employed for simulating strongly swirling flows.

In this paper, the formulation of the present model, the procedure of numerical solution based on the present ASM and the validation results of the proposed model are described in detail. The validation was conducted through successive comparisons of the calculated gas tangential, radial and axial velocities with the test data measured in a cold flow combustor.

II. MATHEMATICAL FORMULATION

2.1 Governing Equations

The phenomenon under consideration is governed by the steady two-dimensional axisymmetric form of the continuity and the time-averaged incompressible Navier-Stokes equations. In the Cartesian tensor system these equations can be written in the following form:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \tau_{ij}) \quad (2)$$

where ρ , u_i , p and x_i are the density, mean velocity tensor, mean pressure and coordinate tensor respectively. The mean viscous stress tensor, $\bar{\tau}_{ij}$, is approximated as:

$$\bar{\tau}_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

where μ is laminar viscosity. The time-averaged Reynolds stress tensor, $\tau_{ij} = -\rho \overline{u_i u_j}$, in the above equation is not known and thus, models are needed to express it in terms of the solution variables. In the present study, two turbulence models are used, namely the k - ϵ model and an algebraic stress model (ASM). The k - ϵ model has been reviewed in references [4, 15] and it will be described briefly. The standard k - ϵ model version relates the turbulent eddy viscosity to the turbulence kinetic energy k and the dissipation rate ϵ through Boussinesq's approximation as:

$$\tau_{ij} = -\frac{2}{3}\delta_{ij}(\rho k) + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

where $\mu_t = \rho C_\mu k^2/\epsilon$ is the turbulent eddy viscosity and ϵ is the dissipation rate of turbulence kinetic energy (TKE). The modelled equation of the TKE, k is given by:

$$\frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G - \rho \epsilon \quad (5)$$

in which $\mu_e = \mu + \mu_t$ is effective viscosity. Similarly the dissipation rate of TKE is given by the following equation:

$$\frac{\partial}{\partial x_j}(\rho u_j \epsilon) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + \frac{\epsilon}{k} (C_{\epsilon 1} G - C_{\epsilon 2} \rho \epsilon) \quad (6)$$

where G is the rate of generation of the TKE while $\rho \epsilon$ is its destruction rate. G is given by:

$$G = \mu_e \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \right] \quad (7)$$

The boundary values for the turbulent quantities near the wall are specified with the wall function method [10]. $C_\mu = 0.09$, $C_{\epsilon 1} = 1.44$, $C_{\epsilon 2} = 1.92$, $\sigma_k = 1.0$, and $\sigma_\epsilon = 1.3$ are empirical constants [4, 15] in the turbulence transport equations. Reynolds-averaged transport equations can be solved for τ_{ij} , [4, 15] the modelled equations for which are:

$$\frac{\partial \tau_{ij}}{\partial t} + \frac{\partial (u_k \tau_{ij})}{\partial x_k} = -G_{ij} - \Phi_{ij} + D_{ij} + \epsilon_{ij} \quad (8)$$

where

$$G_{ij} = \text{local production} = \rho P_{ij} = - \left(\overline{\rho u_i u_k} \frac{\partial u_j}{\partial x_k} + \overline{\rho u_j u_k} \frac{\partial u_i}{\partial x_k} \right)$$

$$\Phi_{ij} = \text{local pressure strain} = -C_1 \frac{\rho \epsilon}{k} \left(\overline{u_i u_j} - \frac{2}{3} k \delta_{ij} \right) - C_2 \left(G_{ij} - \frac{2}{3} G \delta_{ij} \right)$$

$$D_{ij} = \text{net diffusive transport} = - \frac{\partial}{\partial x_k} \left(\frac{\mu_e}{\sigma_\tau} \frac{\partial \tau_{ij}}{\partial x_k} \right)$$

$$\epsilon_{ij} = \text{local dissipation tensor} = \frac{2}{3} \rho \epsilon \delta_{ij}$$

in which $C_1 = 2.5$, and $C_2 = 0.55$ are model constants.

2.2 Algebraic Reynolds Stress Model (ASM)

For simplicity in solving the six Reynolds stresses, Rodi's approximation [11] is used in this study and the Reynolds stress transport can be expressed in algebraic form as follows:

$$\frac{D \tau_{ij}}{Dt} - D_{ij} = \frac{\tau_{ij}}{\rho k} \left(\frac{Dk}{Dt} - D_k \right) \quad (9)$$

Substitution of eqs. (5) and (8) into eq. (9) gives the desired algebraic expression for τ_{ij} :

$$-G_{ij} - \Phi_{ij} + \frac{2}{3} \delta_{ij} \rho \epsilon = \frac{\tau_{ij}}{\rho k} (G - \rho \epsilon) \quad (10)$$

The ASM expressions can thus be rewritten as:

$$\overline{\rho u_i u_j} = \frac{2}{3} \delta_{ij} \rho k + \frac{\lambda k}{\epsilon} \left(G_{ij} - \frac{2}{3} \delta_{ij} G \right) \quad (11)$$

where the empirical constant λ , was found to be 0.135, [16] is defined as:

$$\lambda = \frac{1 - C_2}{C_1 - 1 + \frac{G}{\rho \epsilon}} \quad (12)$$

2.3 Common Form for the Equations

All the governing equations can be re-organised and expressed in a standard form that includes the convection, diffusion, and source terms for 2-D axisymmetric flows as follows:

$$\frac{\partial}{\partial x}(\rho u \phi) + \frac{\partial}{\partial y}(\rho v \phi) - \frac{\partial}{\partial x} \left(\Gamma_{\phi x} \frac{\partial \phi}{\partial x} \right) - \frac{\partial}{\partial y} \left(\Gamma_{\phi y} \frac{\partial \phi}{\partial y} \right) = S_\phi \quad (13)$$

where ϕ may stand for any variable including the velocity components, $\Gamma_{\phi x}$ and $\Gamma_{\phi y}$ are the exchange coefficients for ϕ , and S_ϕ is the source term.

2.4 Solution Procedure

In the present computation, the time-averaged Navier-Stokes equations, the TKE equation, and the TKE dissipation rate equation are solved numerically by a control-volume finite-difference method [9, 14] together with the turbulence model equations, equation (4) for the k - ϵ model or equation (11) for the ASM. All equations are in a generalised form of equation (13). The SIMPLE algorithm is utilised for pressure-velocity de-coupling and iteration [9, 14]. The hybrid, the upwind, the quadratic upstream interpolation for convective kinematics (QUICK) [8] and the second-order upwind (SOU) [12] schemes were used for discretising convection and diffusion transports on a staggered grid cell. The under-relaxation iterative TDMA line-by-line sweeping technique is used for solving the resultant finite-difference equations. The computation was carried out using a personal computer (Pentium III-550 MHz). About 20,000 iterations were needed to achieve satisfactory convergence for each calculation case, which requires about 3.0 hr of computer time.

III. SWIRLING FLOW OF AHMED (1997)

The performances of the k - ϵ model and the ASM are compared for a turbulent swirling flow in a combustor of Ahmed (1997) [1], shown in Fig. 1. The details of the geometry and fluid properties are given in Table 1 below. A single air stream entered the test section through a secondary annulus and passed through an adjustable vane swirler. Experimental mean velocity profiles for u , v and w , turbulence intensities and the Reynolds stresses were

measured at downstream locations of the expansion corner. All data in radial profiles were obtained with a two-components LDV system.

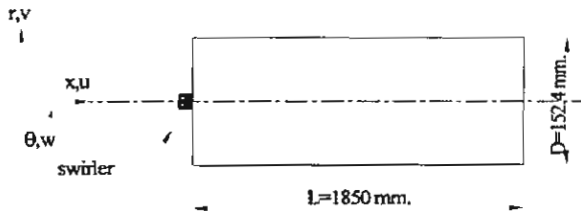


Fig. 1 Flow geometry and computational domain.

Table 1 Data for flow of Ahmed (1997)

Test section characteristics	
swirl vane angle (degree)	45
chamber length (m)	1.85
chamber dia. (m)	0.1524
Inlet flow conditions	
Reynolds number	200,000
average velocity, U_0 (m/s)	30.4

Since axial, radial and tangential velocities and turbulence intensities including shear stresses were provided at $x/D = 0.0633$ downstream of the inlet, computations are started at that location to reduce uncertain inlet conditions. In the present computation, performances of both the standard $k-\epsilon$ model and the ASM are examined by comparing the predicted velocity profiles at ten locations, namely, $x/D = 0.0633, 0.167, 0.25, 0.333, 0.417, 0.5, 0.583, 0.669, 0.833$ and 1.0 .

IV. RESULTS AND DISCUSSION

4.1 Effects of Grid Densities and Numerical Schemes

The comparisons between the predicted results by the $k-\epsilon$ turbulence model and the ASM and the measured data of the flow are presented in Figs. 2 through 7. The predicted axial, radial and tangential velocities are compared with the measurements, where solid or dash curves are represented for the calculated results while open circles for the measured data. The computational results are based on a 95×40 non-uniform grid with refinement in the vicinity of the slit. Grid independence of the numerical results was verified with a 120×60 finer grid. It is found that the differences for both the base grid and the finer grid in local flow properties are marginal. This suggests that grid independent solutions can be obtained with a 95×40 grid, which is used throughout the computations.

The axial, radial and tangential mean velocity profiles predicted with the $k-\epsilon$ model and four different numerical schemes are compared with the measured data in Fig. 2. It is found that predictions with the four schemes generally are in fairly good agreement with the measurements. However, in prediction with hybrid, QUICK and SOU, under-predicted tangential velocity profiles are seen in all locations while good agreement is found for axial and radial velocity profiles in comparison with experimental data. For the four numerical schemes, the FOU scheme performs the best agreement in prediction of tangential velocity but shows poorer agreement for axial and radial velocity profiles. Thus, in terms of the strength of the tangential velocity, only the FOU scheme is considered for computation of this flow.

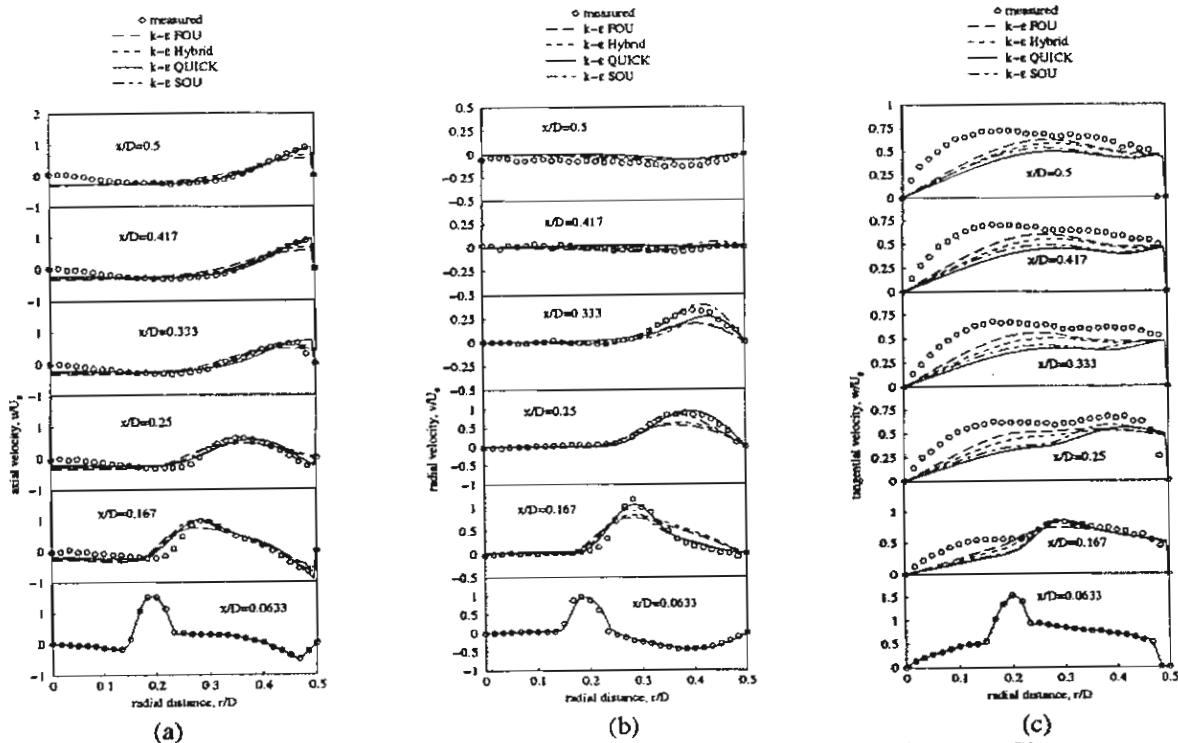


Fig. 2 Effect of numerical schemes on the axial, radial and tangential velocity profiles.

4.2 Flow Field

Velocity Field. Predictions of velocity profiles using the $k-\epsilon$ model and the ASM are compared with experimental data in Figs. 2a, 2b and 2c for the axial, radial and tangential velocity profiles respectively. For the radial profile of axial velocity in Fig. 2a, it is seen that the predicted profiles show no difference for both turbulence models and generally are in favorable agreement with measurements especially for $x/D \geq 0.333$. The ASM. However, both turbulence models give under-predicted results in the core region. Predictions with the ASM show closer agreement with measurements than those with the $k-\epsilon$ model in some locations as can be seen at $x/D = 0.167$ and 0.25 between $r/D = 0.35$ and $r/D = 5$. The results of Fig. 2a indicate that the predicted axial velocity recovers and progresses to uniformity at a faster rate than in the measurements.

The tangential velocity profiles predicted by the $k-\epsilon$ model and the ASM along with the measurements are shown in Fig. 2c. It is worth noting that the tangential velocity predictions with the $k-\epsilon$ model display a rapid decay to a forced vortex profile (solid-body rotation) while the experimental data correspond to a combined free and forced vortex profile. It is clear that the tangential velocity predictions do not agree with the measured data even for trends. Since the calculations were fairly free of numerical diffusion, discrepancies between the data and the predictions can be attributed to two sources: - improper boundary conditions at the inlet plane and deficiencies of the turbulence model.

All inlet values except the ϵ and the k , were obtained indirectly from the experiment. The values of ϵ were derived from a constant length-scale assumption and those of k from an approximation of total kinetic energy. To study the sensitivity of the inlet ϵ and k profiles to the flow field, calculations were made using different ϵ and k distributions, by increasing or decreasing their estimated values by a factor of 10. It was found that the calculated velocity was not affected significantly. The poor agreement between the predictions and measurements is, therefore, more likely to be due to the deficiencies of the turbulence model.

Fig. 2b compares the radial profile of radial velocities predicted by the $k-\epsilon$ model and the ASM with measurements. It is of interest to note that prediction with the ASM shows closer agreement with experimental data than that with the $k-\epsilon$ model. The ASM results mimic very well the measurements while the $k-\epsilon$ model gives under-predicted results, especially in the vicinity of the inlet. For the downstream region far away from the inlet ($x/D > 0.417$), both turbulence models yield similar results and agree well with the measurements.

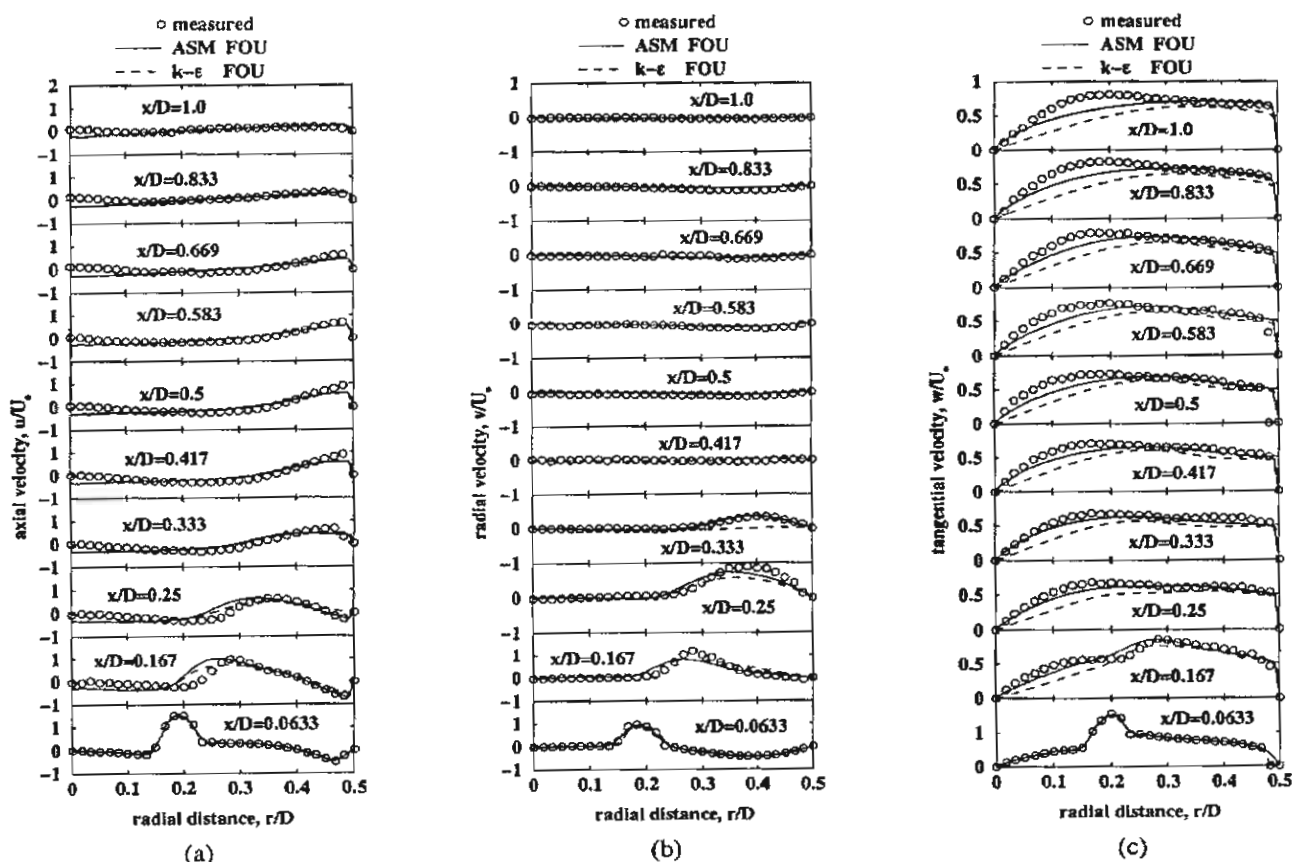


Fig. 3 Comparison of predicted velocity profiles with measurements; (a) axial velocity, (b) radial velocity and (c) tangential velocity profiles.

The use of ASM results in better overall agreement with the measurements than possible with the $k-\epsilon$ model. The major difference between calculations with the two turbulence models is most clear in the recirculation region and in the tangential velocity profiles. The agreement between the calculation with the ASM and the experimental data is fairly good. The peak values and a combined forced and free (Rankine) vortex motion for the tangential velocity profiles are well predicted. The recirculation zone (IRZ) is, however, longer and wider than that with the $k-\epsilon$ model.

Streamlines predicted by the ASM and the $k-\epsilon$ model are shown in Figs. 4 and 5 respectively. The contour plot of streamlines is in a range of $x/D = 0$ to 3 and $r/D = 0$ to 0.5, which has a high change of velocities. From the figures, there is no change of velocities at about $x/D = 1.5$ for the ASM and at some $x/D = 3$ for the $k-\epsilon$ model. Two

recirculation zones are identified; one is at the corner and the other, an internal recirculation zone (IRZ) or central toroidal recirculation, near the inlet. It is observed that the size of the recirculation zone calculated by the ASM is larger than that by the $k-\epsilon$ model.

Vector plots of velocity predicted with the ASM and the $k-\epsilon$ model are illustrated in Fig. 6 and Fig. 7 respectively. The plots for both turbulence model are also in a range of $x/D = 0$ to 3 and $r/D = 0$ to 0.5. It is clear to note that for the ASM, the centers of the corner recirculation and the central toroidal recirculation are, respectively, at about $x/D = 0.1$ and $r/D = 0.42$ and at $x/D = 0.35$ and $r/D = 0.35$. For the $k-\epsilon$ model, they are at some $x/D = 0.05$ and $r/D = 0.42$ and at $x/D = 0.3$ and $r/D = 0.3$. In the inlet region, the reverse flow occurs for both turbulence models but the flow predicted by the ASM is longer than that by the $k-\epsilon$ model as can be seen from the figures.

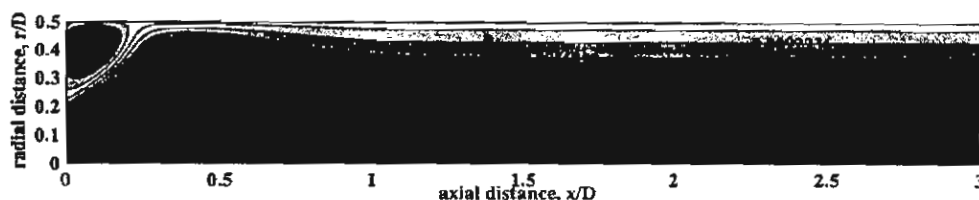


Fig. 4 Streamline predicted by ASM

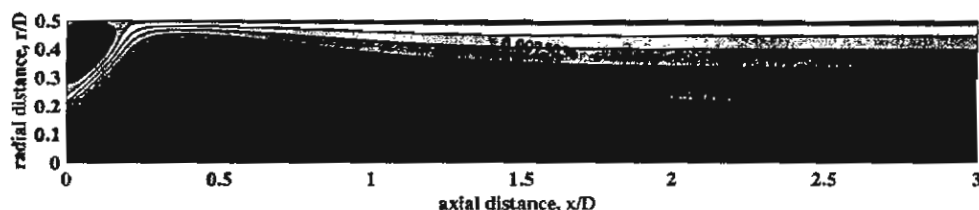


Fig. 5 Streamline predicted by the $k-\epsilon$ model

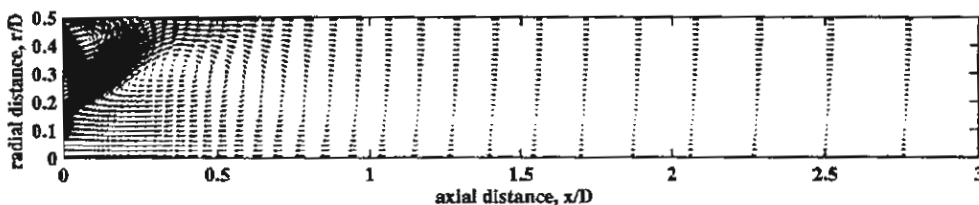


Fig. 6 Velocity vectors predicted by ASM

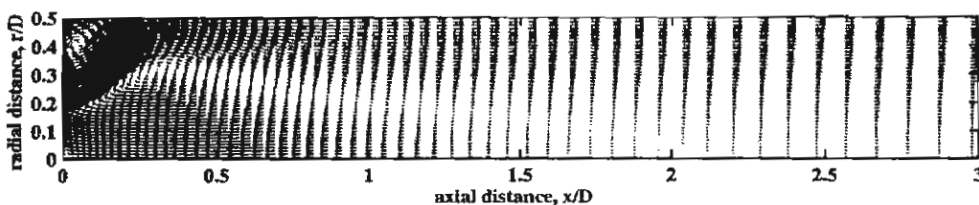


Fig. 7 Velocity vectors predicted by $k-\epsilon$ model

V. CONCLUDING REMARKS

The assessments of the $k-\epsilon$ model and the ASM have been carried out in a swirling incompressible flow. The conclusions from the validation study are as follows: The numerical procedure incorporating the ASM was tested for a swirling turbulent flow. Since the ASM represents better the anisotropy of the turbulence, the ASM results with the FOU scheme is in closer agreement with measurements than those with the $k-\epsilon$ model which predicts a fast decay of tangential velocity profiles to a solid-body rotation. The computations of the flow showed that the ASM performs better than the $k-\epsilon$ model in capturing the mean flow behaviour, superiority rooted in the response of the ASM to turbulence interaction.

VI. ACKNOWLEDGEMENTS

The authors would like to gratefully acknowledge the Thailand Research Fund (TRF) for financial support of this research.

VII. REFERENCES

- [1] Ahmed S.A. (1997), "Three Component Velocity Measurements of an Isothermal Confined Swirling Flow", Proc. Instn Mech. Engrs, Part G, J. of Aerospace Engineering, 211(G2), pp. 113-122.
- [2] Boysan, F, Ayers, W.H. and Swithenbank, J. (1982), A Fundamental Mathematical Modelling Approach to Cyclone Design, Trans. Inst. Chem. Eng., Vol.60, pp.222-230.
- [3] Chen, C.P. (1986), Calculation of Confined Swirling Jets, Commu. Appl. Numer. Methods, Vol.2, pp.333-338.
- [4] Gatski, T.B. (1996), Turbulent Flows: Model Equations and Solution Methodology, in Handbook of Computational Fluid Mechanics, Edited by Roger Peyret, Academic Press Ltd, London.
- [5] Gupta, A.K., Lilley, D.G., and Syred, N. (1984), Swirl Flows, Abacus Press, Turnbridge Wells, England.
- [6] Hogg, S. and Leschziner, M.A. (1989), Computation of Highly Swirling Confined Flow with a Reynolds Stress Turbulence Model, AIAA J., Vol.27, pp.57-63.
- [7] Jones, W.P. and Pascau, A. (1989), Calculation of Confined Swirling Flows with a Second Moment Closure, Trans. ASME J. of Fluid Engineering, Vol.111, Sept, pp.248-255.
- [8] Leonard, B.P. (1979), A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation, Comp. Meth. Appl. Mech. Eng., 19, pp.59-98.
- [9] Patankar, S.V. (1980), Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington, D.C.
- [10] Patankar, S.V. and Spalding, D.B. (1970), Heat and Mass Transfer in Boundary Layers, Intertext Books, London.
- [11] Rodi, W.A. (1976), New Algebraic Relations for Calculating the Reynolds Stresses, Z. Angew. Math. Mech. (ZAMM), vol. 56, pp. T219-T221.
- [12] Shyy, W., Thakur, S., and Wright, J. (1992), Second Order Upwind and Central Difference Schemes for Recirculating Flow Computation, AIAA J., Vol.30, No.4, pp.931-932.
- [13] Sloan, D.G., Smith, P.J., and Smoot, L.D. (1986), Modeling of Swirl in Turbulent Flow System, Progr. Energy Combust. Sci., vol. 12, pp. 163-250.
- [14] Versteeg, H.K. and Malalasekera, W. (1995), An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Longman Scientific & Technical, Longman Group Limited, Essex, England.
- [15] Wilcox, C.D. (1993), Turbulent Modelling for CFD, DCW Industries, Inc., California.
- [16] Zhang, J., Nieh, S. and Zhou, L. (1992), A New Version of Algebraic Stress Model for Simulating Strongly Swirling Turbulent Flows, J. Numerical Heat Transfer, Part B, Vol. 22, pp. 49-62.



July 29 – August 2, 2001
at the Westin Hotel, Savannah, Georgia



co-sponsored by



AT-21: WIRELESS ENERGY TRANSFER/SPACE SOLAR POWER SYSTEMS

Tuesday, July 31, 2001
8:30 - 11:30 AM

Organizers and Chairs: Dr. Frank Little, Texas A&M University
Dr. Henry Brandhorst, Auburn University

2001-AT-05 *Comparison of Space Solar Power Systems and Terrestrial Photovoltaic Arrays* — Finley R. Shapiro

2001-AT-44 *Air-Launch Lunar Tourism with Beamed-Power Electrical Propulsion Costing* — Simon Rowland

2001-AT-56 *Effects of Hypervelocity Impact on Solar Cell Modules at High Voltage* — Henry Brandhorst

CT-2: ADVANCED CYCLES II

Tuesday, July 31, 2001
8:30 - 11:30 AM

2001-CT-02 *I/V Characteristic of ZnSe Schottky Diodes* — Dirk M. Chiu

2001-CT-08 *A Novel Integrated Gas/ Steam Cycle with Ericsson Bottoming Gas Turbine* — Onkar Singh

2001-CT-11 *Prediction of Performance of Simple Combined Gas/ Steam Cycle and Cogeneration Plants with Different Means of Cooling* — Sanjay Yadav, Mechanical Engg., Regional Institute of Technology, Jamshedpur, Jharkand, India.

2001-CT-13 *Test and Demonstration Results of a New Magnetically Coupled Adjustable Speed Drive* — Kenneth Anderson

2001-CT-25 *Fast Rise of Seebeck Coefficient in CdTe with Decreasing temperature: Theoretical Explanation* — Svetla Vackova

ET-3: BATTERIES FOR TERRESTRIAL APPLICATIONS

Tuesday, July 31, 2001
8:30 - 11:30 AM

2001-ET-12 *Optimization of Nanostructured Hydrous RuO₂/ Carbon Supercapacitor* — Kim Hansung, Bala Haran, Slavkov Dragan, Ralph E. White and Branko N. Popov

2001-ET-13 *Optimization of Energy Density and Power Density of Li-ion Based Hybrid Devices* — Bala S. Haran, Anand Durairajan, Ralph E. White and Branko N. Popov

2001-ET-16 *Nickel-Hydrogen Battery Design for Terrestrial Applications* — Jeff Dermott

2001-ET-17 *45-Ah Lithium Ion cells for the Team Northern Sun Solar Powered Car* — Gregg Bruce

2001-ET-19 *OSCR program for Primary Pouch Batteries* — Christo Brand

PE-2: POLICY AND GLOBAL ISSUES

Tuesday, July 31, 2001
8:30 - 11:30 AM

Organizer and Chair: William D. Jackson, HMJ Corp.

2001-PI-03 *Structural Problems with the Danish CO₂ Tax System* — Jens Anderson

2001-EI-14 *Eco-Energy City Project - the Concept and the results of the research* — Yoshihiro Ogisu

2001-PI-05 *An Efficient Time-Space Numerical Method For Global Warming Studies* — Takeo S. Saitoh and Shinichiro Wakashima

2001-EI-08 *Electric and Magnetic Fields Associated with the Use of Electrical Power in the Workplaces in Underdeveloped Countries* — Ahmed Farag

2001-EI-09 *Electromagnetic Fields Policy Implication For Risk Assessment* — Ahmed Farag

2001-EI-10 *Power Frequency Electromagnetic Fields Management Technologies* — Ahmed Farag

ES-3: THERMAL AND MECHANICAL ENERGY STORAGE

Tuesday, July 31, 2001
8:30 - 11:30 AM

2001-ES-20 *Evaluation and Improvement of Eddy Current Position Sensors for Use in Magnetically Suspended Flywheel Systems* — Timothy Dever

2001-ES-24 *Storage and Transport of "Cold" Using Clathrate Slurries: Fundamental Thermophysical Characteristics of Aqueous Solution of TBAB* — I.Tanasawa, S. Takao

2001-ES-44 *Heat Balance of Long-Term Supercooled Thermal Energy Storage* — Satoshi Hirano

RE-3: HYDROGEN AND BIOMASS ENERGY

Tuesday, July 31, 2001
8:30 - 11:30 AM

2001-RE-06 *Hydrogen from Solar Energy: An Overview of Theory and Current Technological Status* Paul A. Erickson and D. Yogi Goswami, Department of Mechanical Engineering, University of Florida

2001-RE-03 *On-Board Hydrogen Storage for Fuel Cell Vehicles* — Thomas Vernersson, Kristina Johansson, and Per Alvfors, Royal Institute of Technology, Department of Chemical Engineering and Technology, Division of Energy Processes

2001-RE-07 *Premixed Combustion of Hydrogen/Methane Mixtures in a Porous Medium Burner* Chung-Jen Tseng Department of Mechanical Engineering, National Central University

2001-RE-04 *Biomass Briquetting Technology with Preheating* — Sk Ahad Ali

2001-RE-17 *Experimental Study of Combustion Characteristics in a Rice Husk Fired Vortex Combustor* — Pongjet Promvonge, Department of Mechanical Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok

TM-2: SPACECRAFT AND AIRCRAFT THERMAL MANAGEMENT

Tuesday, July 31, 2001
8:30 - 11:30 AM

Organizer and Chair: Dr. Michael K. Choi
NASA Goddard Space Flight Center

2001-TM-16 *Off-design Performance of a Heat Exchanger for Regenerative Aircraft Engines* — S. Pasini, U. Ghezzi, and R. Andriani

2001-RE-17

EXPERIMENTAL STUDY OF COMBUSTION CHARACTERISTICS IN A RICE HUSK FIRED VORTEX COMBUSTOR

Pongjet Promvonge

Mechanical Engineering Department, Faculty of Engineering,
King Mongkut's Institute of Technology Ladkrabang,
Bangkok 10520, Thailand
Email: kppongje@kmitl.ac.th

Kulthorn Silapabanleng

Former Associate Professor
Mechanical Engineering Department, Faculty of Engineering,
Chulalongkorn University,
Bangkok 10330, Thailand

ABSTRACT

This paper presents the study of combustion characteristics of rice husk fuel in an annular vortex combustor (VC). The temperature distributions for selected locations inside the combustor and the fly ash and smoke from its flue gas were measured and observed respectively. Measurements were made by setting a constant mass flow rate of rice husk to be 0.2 kg/min and by varying the equivalence ratio, Φ , to be 0.8, 1.0 and 1.2. To study the effect of feeding secondary air on flame stability, three values of the ratio of volumetric flow rates of the secondary air to the total air, (λ), were used and set to be 0.0, 0.2 and 0.4. The experiment shows the maximum temperature of about 1000° C in the upper chamber with less smoke of flue gas. Besides, emissions and the sizes of flyash particles from the exhaust stack can be controlled by the flow rate ratios of the secondary air to the total air, λ . The VC shows an excellent performance, low emissions, high stabilization and ease of operation in firing the fine rice husk.

Keywords: annular vortex combustor, rice husk, biomass.

INTRODUCTION

The conventional sources of energy have been depleting at an alarming rate and the price of conventional energy is going up. Thus, the focus on alternative renewable sources of energy has been increasing and biomass is one of them that has been getting continued and increased attention. In general, paddy rice is one of the mostly produced crops throughout the world. Since it is not possible to take the whole of it as food, about 22% by weight of this paddy rice is generated as a waste, known as rice husk. It is estimated that over 60 million tons of rice husks are generated each year worldwide. Thailand generates 4.4-4.6 million tons of rice husks annually, which their thermal potentials are equivalent to 1.46-1.53 million tons of crude oil.

About 10% of rice husk are utilized as a source of heat energy in Thailand. It is creating waste management problem, especially in the rice milling sites. Therefore, an attempt to energy recovery from this rice husk waste by combustion technique may be worthwhile. With this the burning of rice husk by an annular vortex combustor (VC) can be considered.

There have been many reports published on various combustors using biomass materials or coals as a fuel. Singh et al. [1] designed a cyclonic rice husk furnace for drying a ton of paddy and its moisture was reduced from 35% to 14%d.b. Different furnace efficiencies were found for various rice husk feed and air flow rates. Tumambing [2] also investigated a rice husk furnace for drying paddy. Xuan et al. [3] studied two types of husk furnaces. One was a furnace with inclined grate and cylindrical combustion chamber with heat exchanging in the furnace upper part. Inlet air entered at the lower part of inclined grate and burnt the rice husk on grate. The other was a pneumatic-fed furnace (vortex type) including combustion chamber and rice husk feed system. Rice husk was fed into the combustion chamber with primary air and burnt and fell to the lower part of the furnace. The secondary air entered tangentially the chamber to create the vortex flow with a view to eliminating dust from flue gas. Soponronnarit et al [4] applied a rice husk fired furnace to a fluidized bed paddy dryer. An extensive study of a vortex combustor for burning dry ultrafine coal and coal water fuel was studied by Nieh and Fu [5]. A vortex combustor similar to [5] but using rice husk fuel instead was experimentally investigated by Promvonge et al. [6]. For above combustors burning rice husk fuel, effects of ash inside the combustors on combustion efficiency including its problem in elimination were reported.

This article deals with a preliminary study of combustion characteristics and temperature distributions in a vortex combustor burning rice husk fuel. Effect of feeding secondary

air on flame stability, temperature control in the combustor, emissions, ashes and smokes from the exhaust stack are studied and observed for a guideline in design and improving the performance of the combustor.

Table 1 Composition of fine rice husk

Carbon	38.0 %
Hydrogen	5.70 %
Oxygen	41.6 %
Nitrogen	0.69 %
Sulfur	0.06 %
Moisture	10.3 %
Ash	14.0 %
Density, kg/m ³	100.00
Gross heat of combustion, kcal/kg	3,580.00
Stoichiometric air	4.850 kg/kg fuel

EXPERIMENTAL SETUP

The rice husk particles were milled and grinded, sieved up to about 1.3-mm size (between 1.19 and 1.41 mm), and stored in the laboratory under dry condition (10.3% of moisture content). The arrangement of experimental system of the combustor is shown in Fig. 1 below. The combustor was a concentric cylindrical pipe made of steel with castable refractory cement lining as insulation while the exhaust center pipe was made of stainless steel. It is 0.755-m high and 0.20-m

inside diameter (D) with multiple injection nozzles of 0.005-m diameter each for the secondary air (Q_2) as depicted in Fig. 2. The exhaust center pipe is 0.078 m in diameter.

The combustor was operated over a temperature range from 600 – 1000° C. A blower and two compressors were used for providing both primary and secondary combustion airs. Rice husk fuel was fed through a screw feeder and injected to the bottom chamber by pneumatic conveying via the primary air (Q_1). Start up process was commenced by heating up the VC with LPG torch inserted at the lower air nozzle slot. The preheating took about 20 minutes for the chamber to raise its temperature to be about 400°C. Then feeding commenced through the hopper, slowly with the rice husk until 0.2 kg/min and kept constant. When the temperature in the chamber reached 700° C, stop of preheating with LPG was made. A thick, black smoke was seen at the beginning and slowly thinned out by adjusting the air/fuel (A/F) ratio. Excess air at various levels for equivalence ratio, defined as $\Phi = (A/F)_{\text{stoi}} / (A/F)_{\text{actual}}$, of 0.8, 1.0 and 1.2 was tried for each of λ . The temperatures were monitored at various selected locations with chromel-alumel (type K) thermocouples while volumetric flow rates of primary and secondary airs were measured by using orifice meters. Flue gas emissions were measured by a gas analyzer. All data collection was taken at steady state condition.

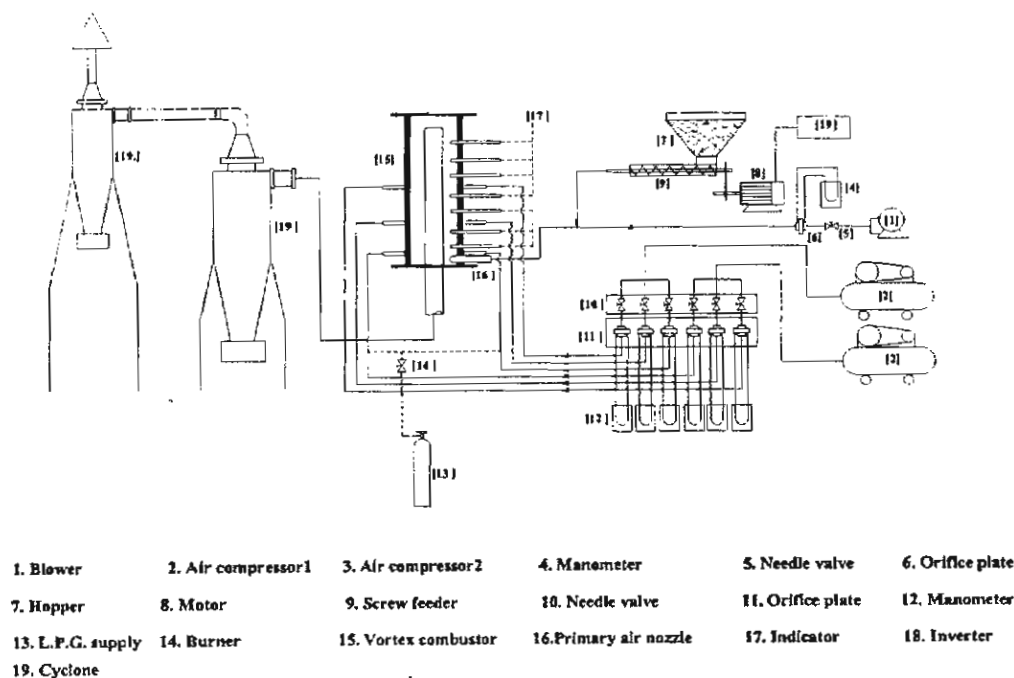


Fig. 1 Schematic diagram of experimental setup of combustion system.

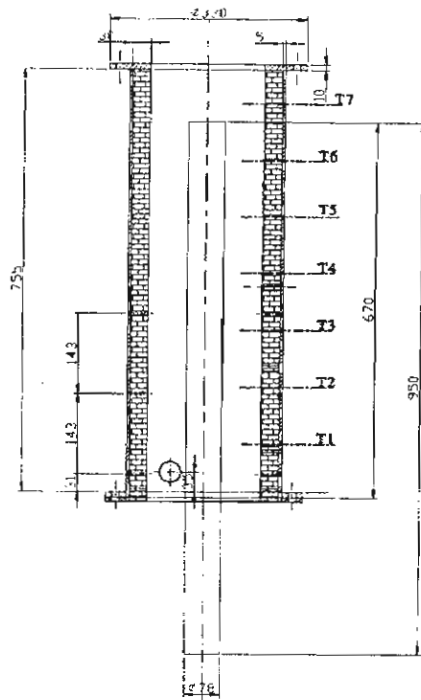


Fig. 2 Configuration of vortex combustor (unit in mm.)

RESULTS AND DISCUSSION

The VC was originally designed to accommodate various types of coal fuels as sources of heat. The VC responded significantly to the various fuels as expected. For rice husks, the level of equivalence ratio (or excess air) was varied to optimize the temperature inside the VC. The measurements of temperature were made at seven axial locations namely, $x/D = 0.5, 1.0, 1.5, 2.0, 2.5, 3.0$ and 3.5 .

Effects of Equivalence Ratios

The radial profiles of temperature inside the VC for $\Phi = 1.2$ (rich), $\Phi = 1.0$ (stoich.) and $\Phi = 0.8$ (lean) are presented in Figs. 3a, 3b and 3c respectively. The feed-rate of fine rice husk with 10.3% moisture content was 0.2 kg/min and kept constant throughout.

For $\Phi = 1.2$ and $\lambda = 0$, Fig. 3a, it is found that the temperature profiles for all locations generally are not uniform. The temperature in the vicinity of the center pipe is higher than that near the chamber walls. Maximum temperature of about 800°C takes place in the top of chamber while lower temperature can be seen near the chamber wall. Unburned fuel and black smoke emission from the exhaust stack were observed for this case. This indicates the incomplete combustion (high CO concentration) or combustion reaction takes place very slowly.

Fig. 3b shows the temperature distributions for $\Phi = 1.0$. It can be seen that (for $\lambda = 0$) the temperature profiles are nearly

uniform for all stations except at the top chamber. The maximum temperature above 800°C is found in the top annular space whereas the minimum temperature in the vicinity of the chamber walls at early station. This shows insufficient turbulence at the bottom chamber, which was due to too many rice husks, resulting in inadequate mixing and consequently the presence of fuel rich zones within the bottom space. This case yielded a less black smoke emission from the exit pipe.

The temperature distribution for $\Phi = 0.8$ is depicted in Fig. 3c. It is worth noting that the temperature profiles similar to the case for $\Phi = 1.0$, are nearly uniform for all stations. The peak temperature of some 800°C is also seen in the top annular space while lowest temperature at the bottom one. The peak one for every station is found in the middle between the chamber and center pipe walls. The maximum temperature for this case has the lower value than the case for $\Phi = 1.0$, as expected, since an increase in the excess air leads to decreasing temperature in the combustion system. White and thin smoke emission and small size of flyash particles are visible. This points out that complete combustion takes place for this case.

Effects of Volumetric flow rate ratios of Secondary to total Airs (λ)

The ratio of volumetric flow rate of the primary air to the secondary air, defined as $\lambda = Q_2/(Q_1+Q_2)$, is an indication of the strength of the vortex of the flow. The increase in value of λ results in the higher tangential velocity component (strong vortex). This means that the impact of swirling phenomenon of the secondary air on the flow and temperature fields becomes more pronounced as the value of λ is increased. Comparison of the temperature distributions for $\lambda = 0.0, 0.2$ and 0.4 for various equivalence ratios is also presented in Fig. 3.

For $\Phi = 1.2$, it is obvious that adjusting of $\lambda = 0.4$ leads to a flatter and higher temperature distribution curve than that of $\lambda = 0.0$ and 0.2 . The use of a value of λ can improve substantially the temperature distributions in the VC as can be seen in Fig. 3a. This indicates that for the rich mixture, λ should be used to help increase combustion efficiency.

When $\Phi = 1.0$, it is interesting to note that the influence of λ on the temperature profiles shows slightly improvement on the first three stations at the bottom while significant improvement due to λ on the temperature profiles is found at other stations as illustrated in Fig. 3b. Again, use of λ leads to higher combustion temperature in the VC at about 20%.

As $\Phi = 1.2$, it is worth noting that effect of λ on the temperature distributions is seen at early stations while slightly influence on the top part of the VC as exhibited in Fig. 3c. Also, use of λ shows better combustion temperature obtained.

It is concluded that for all values of Φ , use of λ results in significant improvement of temperature profiles and yields higher temperature of combustion in the VC. Besides, the increased swirling intensity leads to an abatement of particle elutriation, and to enlargement of recirculating region.

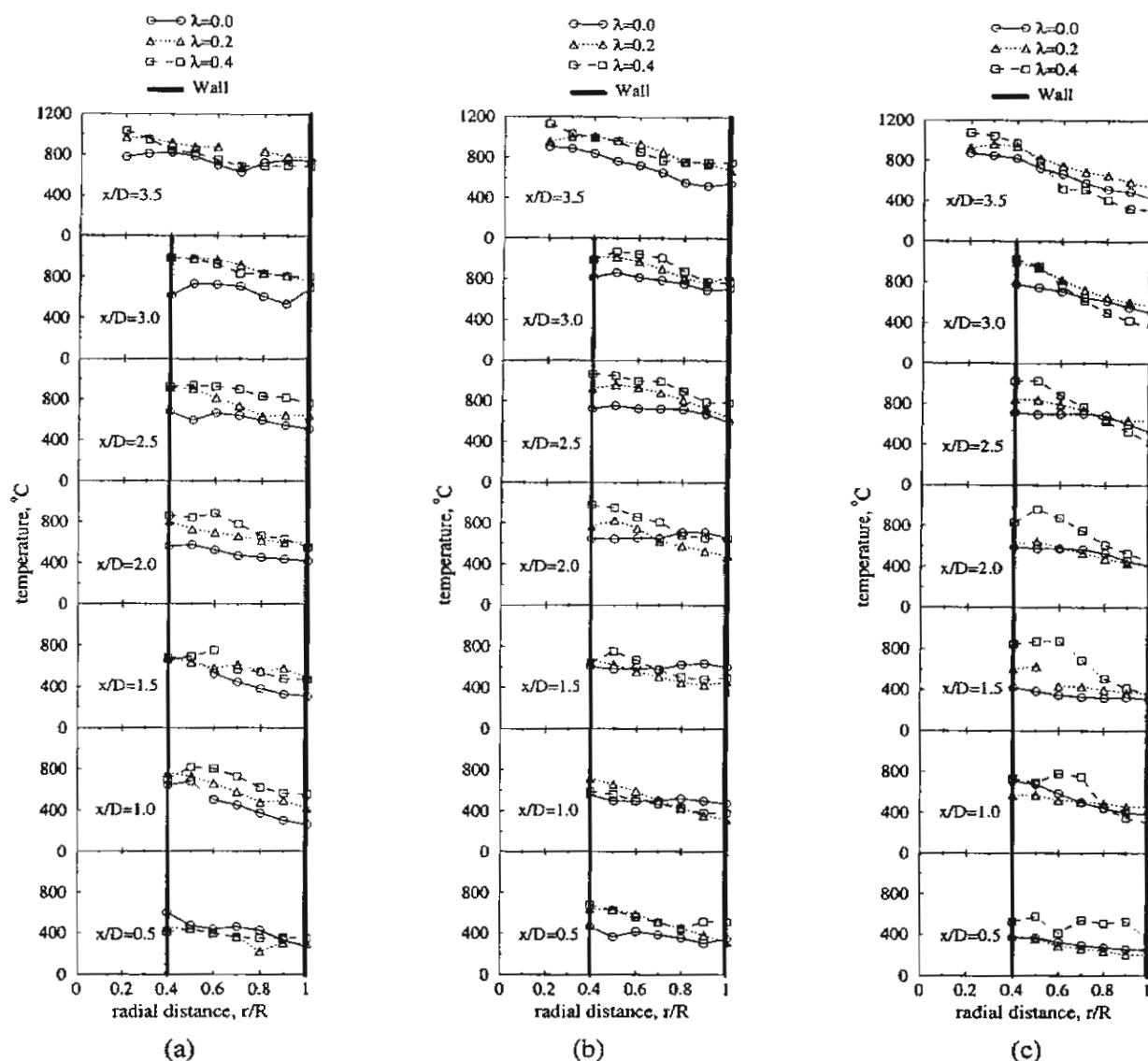


Fig. 3 Temperature distributions inside the combustor, (a) $\Phi = 1.2$, (b) $\Phi = 1.0$, and (c) $\Phi = 0.8$.

Emissions of exhaust gas

The flue gas emissions of CO , CO_2 , NO_x and O_2 for various equivalence ratios and different λ are studied and presented in Figs. 4a, 4b, 4c and 4d, respectively.

For CO emissions in Fig. 4a, it is interesting to note that the emissions of CO are reduced substantially when Φ is decreased (or lean mixture). The formation of CO is very low once λ is introduced for the three equivalence ratio cases. The emissions of CO_2 can be also reduced by increasing the excess air (or decreasing Φ). Nevertheless, the application of λ leads to the reduction of CO_2 as can be seen in Fig. 4b.

For NO_x emissions, Fig. 4c shows that lower equivalence ratio favours the formation of NO_x . Also, use of λ results in higher NO_x emissions. This can be attributed to higher temperature of combustion in the VC. The level of O_2 emissions is presented in Fig. 4d. It is found that the O_2 level increases for lower equivalence ratio and using λ .

CONCLUSIONS

The measurements of temperature distributions and flue gas emissions have been conducted during the combustion of fine rice husk in the VC. The conclusions derived from the results of these experiments are as follows: combustion of rice husk in the

VC with equivalence ratio between 0.8 and 1.0 yields better combustion efficiency, flame stabilization, low emission and reliable furnace condition. For all equivalence ratios, use of

secondary air has a significant effect on the temperature distributions inside and helps reduce emissions and the size of flyash particle.

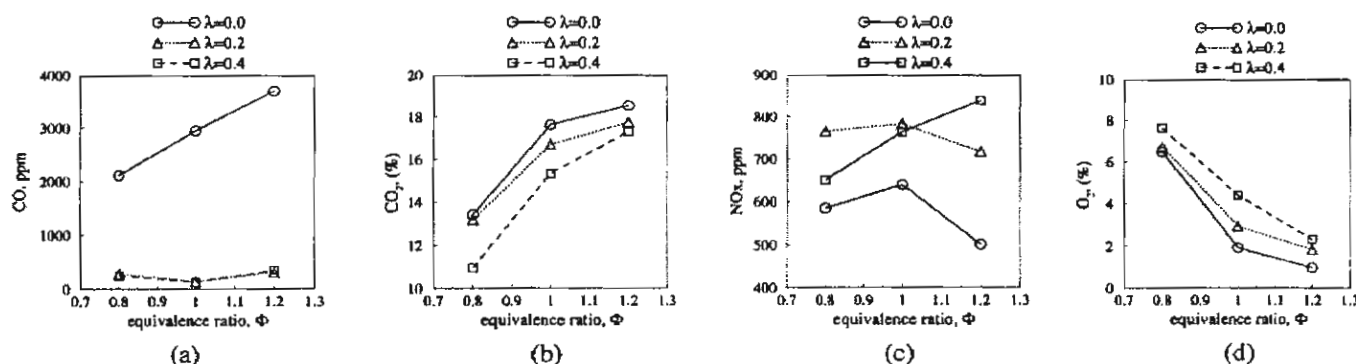


Fig. 4 Emissions of flue gas, (a) CO, (b) CO₂, (c) NO_x, and (d) O₂.

ACKNOWLEDGEMENTS

The authors would like to acknowledge with appreciation, the Thailand Research Fund (TRF) for the financial support and the Mechanical Engineering Department, Mahanakorn University of Technology (MUT) for their support in this effort and the MUT undergraduate students for data collection.

REFERENCES

- [1]. Singh, R., Maherhweri, R.C., and Ojha, T.P., 1980, "Development of Husk Fired Furnace", The British Society for Research in Agricultural Engineering, India, pp. 109-120
- [2]. Tumaming, J.A., 1984, "Testing and Evaluation of Rice Hull-Fed Furnaces for Grain Drying", National Post Harvest Institute for Research and Extension, pp. 197-214.
- [3]. Xuan, N.V., Vinh, T., Anh, P.T. and Hien, P., 1995, "Development of Rice Husk Furnaces for Grain Drying",

Proceeding of the International Conference on Grain Drying in Asia, 17-20 October, Bangkok, Thailand, pp.109-120.

- [4]. Soponronnarit, S., Prachayawarakorn, S. and Wangi, M., 1996, "Comercial Fluidized Bed Paddy Dryer", Proceedings of 10th International Drying Symposium (IDS'96), Krakow, Poland, 30 July-2 August, Vol. A, pp. 638-644.
- [5]. Nieh, S. and Fu, T.T., 1988, "Development of a Non-slugging Vortex Combustor (VC) for Space/Water Heating Applications", Proceeding of The 5th International Coal Conference, Pittsburgh, pp. 761-768.
- [6]. Promvong, P., Vongsampigoon, L. and Piriyaungroj, N., 2000, "A Low Emission Annular Vortex Combustor Firing Rice Husk Fuel: Part II - Experimental Investigation", Proceeding of The First Regional Conference on Energy Technology Towards a Clean Environment, 1st - 2nd December, Chiang Mai, Thailand, pp. 279-284.



สัมมนาวิชาการวิศวกรรมเครื่องกล แห่งประเทศไทยครั้งที่ ๑๓



บทความทางวิชาการ เล่มที่ ๒/๒

จัดโดย

เครือข่ายวิศวกรรมเครื่องกล

ดำเนินงานโดย

ภาควิชาวิศวกรรมเครื่องกล

คณะวิศวกรรมศาสตร์

สถาบันเทคโนโลยีพระจอมเกล้า

เจ้าคุณทหารลาดกระบัง

วันที่ ๒-๓ ธันวาคม ๒๕๕๒

Simulation of Turbulent Flow through a Circular Orifice

Pongjet Promvongse

Department of Mechanical Engineering, Faculty of Engineering,
King Mongkut's Institute of Technology Ladkrabang, Bangkok 10520, Thailand.
Email: kppongje@kmitl.ac.th

and

Kulthorn Silapabanleng

Department of Mechanical Engineering, Faculty of Engineering,
Chulalongkorn University, Bangkok 10330, Thailand.

Abstract

The paper presents a numerical simulation of a steady turbulent airflow in a circular duct containing an orifice plate. A Finite Volume approach with a non-uniform and staggered grid system is employed in the present simulation. To account for the turbulence nature of the flow, the standard k- ϵ turbulence model is incorporated in the time-averaged governing equations. Effects of numerical diffusion on the calculated results are also investigated by comparing between a second-order-differencing scheme for the convection transport and the first-order hybrid scheme. The calculated solutions are in close agreement with 3D LDA measurements. The computations of the flow reveal that the use of a second-order scheme leads to more accurate results than that of a first-order scheme.

1. Introduction

The orifice meter is a device commonly used for measuring fluid flow in industrial processes such as metering flow in the natural gas industry. Although more accurate metering methods are available, the orifice plate continues to be preferred. The popularity of the orifice meter can be attributed primarily to its simplicity, relatively low cost and little maintenance requirements in comparison with other fluid meters. The orifice plate becomes the essential part of a fluid flow meter when installed in a pipe such that the fluid stream must negotiate the constriction.

By far the most common orifice plate installation is that of the concentric round orifice plate. In this type of arrangement the orifice is round and the plate is mounted between pipe flanges. The plate is positioned perpendicular to a fully developed pipe flow while at the same time the circular orifice is concentric with respect to the (circular) pipe interior. Other types of orifice plate exist, such as square orifices, series and non-concentric. Also, the inner edge of the orifice is machined in one of several different ways. Some orifice plates are square edged while others are rounded and beveled. This study is

concerned with the concentric, round, beveled orifice plate.

Most of the work thus far on orifice meters has focused almost entirely on the determination of discharge coefficients. There have been a handful of attempts to study in detail the flow field in the vicinity of the orifice plate. It is believed that knowledge concerning details of the orifice flow field will lead the way to improvements in metering accuracy. These improvements could come via improved determination and prediction of discharge coefficients.

This paper deals with the simulation of turbulent flow through an orifice plate with a view to increasing the knowledge of orifice meter flow. The mathematical model including the k- ϵ turbulence model, numerical solution and other computational details is described. Comparisons of the calculated gas axial velocity with 3D LDV measured data [4] are made to evaluate the turbulence models and the numerical schemes used.

2. Mathematical Modelling

2.1 Governing Equations and Closures

For constant density, isothermal turbulent flows, the time-averaged incompressible Navier-Stokes equations in the Cartesian tensor notation can be written in the following form:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \tau_{ij}) \quad (2)$$

The mean viscous stress tensor is approximated as:

$$\bar{\tau}_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

where μ is laminar viscosity. Due to the nonlinearity of equation (2), the averaging process employed introduces the unknown correlation the time-averaged Reynolds stress tensor, τ_{ij} ($= -\rho \overline{u_i' u_j'}$) that are obtained from

turbulence models [1, 2, 3, 10]. In the present study, the standard k - ϵ turbulence model [1, 10] is adopted and the Reynolds stress is linearly related to the mean rate of strain by a scalar eddy viscosity. The standard version relates the turbulent eddy viscosity to the turbulence kinetic energy k and the dissipation rate ϵ through Boussinesq's approximation [8] as:

$$\tau_{ij} = -\frac{2}{3}\delta_{ij}(\rho k) + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

where μ_t is the turbulent eddy viscosity. In the k - ϵ model the turbulent viscosity is related to k and ϵ by

$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon} \quad (5)$$

The two turbulence quantities in equation (5), k and ϵ , are obtained from the following transport equations which are solved simultaneously with governing equations (1) and (2):

$$\frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G - \rho \epsilon \quad (6)$$

$$\frac{\partial}{\partial x_j} (\rho u_j \epsilon) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + \frac{\epsilon}{k} (C_{\epsilon 1} G - C_{\epsilon 2} \rho \epsilon) \quad (7)$$

in which G represents the rate of generation of turbulent kinetic energy while $\rho \epsilon$ is its destruction rate. G is given by:

$$G = \mu_t \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \right] \quad (8)$$

The boundary values for the turbulent quantities near the wall are specified with the wall function method [6]. The empirical constants C_μ , $C_{\epsilon 1}$, $C_{\epsilon 2}$, σ_k , and σ_ϵ in the turbulent transport equations are assumed to have the values of 0.09, 1.44, 1.92, 1.0 and 1.3 [8, 10] respectively.

2.2 Common Form for the Equations

All the governing partial differential equations can be re-organised and expressed in a standard form that includes the convection, diffusion, and source terms [5, 9] for 2-D axisymmetric flows as follows:

$$\frac{\partial}{\partial x} (\bar{\rho} u \phi) + \frac{1}{r} \frac{\partial}{\partial r} (r \bar{\rho} v \phi) - \frac{\partial}{\partial x} \left(\Gamma_{\phi x} \frac{\partial \phi}{\partial x} \right) - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma_{\phi r} \frac{\partial \phi}{\partial r} \right) = S_\phi \quad (8)$$

where ϕ may stand for any variable including the velocity components, $\Gamma_{\phi x}$ and $\Gamma_{\phi r}$ are the exchange coefficients for ϕ , and S_ϕ is the source term.

Detailed expressions of $\Gamma_{\phi x}$, $\Gamma_{\phi r}$ and S_ϕ for different ϕ 's are summarised in Table 1 below.

Nomenclature

$C_{\epsilon 1}$, $C_{\epsilon 2}$	constants in the dissipation rate equation
C	convection term
C_μ	constant in the k - ϵ turbulence model
D	diffusion term; dimension
D	pipe or duct diameter
d	orifice diameter
G	stress generation
k	turbulence kinetic energy
l	turbulence characteristic length scale
p	mean pressure
p_{in}	inlet wall pressure
p_{out}	outlet wall pressure
r	radial co-ordinate; radius
R	pipe radius
Re_D	Reynolds number based on pipe diameter
S	general source term; swirl number
τ_{ij}	viscous stress tensor
u_i	fluctuating velocities in direction x_i
$\overline{\rho u_i u_j}$	Reynolds stresses
u	time-averaged velocity in x -direction
v	time-averaged velocity in r -direction
x	axial co-ordinate

Greek Symbols

δ_{ij}	Kronecker delta tensor
ϵ	dissipation
ϕ	generalised dependent variable
Γ_ϕ	exchange coefficient
β	ratio of pipe to orifice diameter (d/D)
μ , μ_t	dynamic viscosity, eddy-viscosity
μ_e	effective viscosity, ($= \mu_t + \mu$)
ρ	density
σ_ϕ	Schmidt or Prandtl numbers for the scalar ϕ
τ_{ij}	Reynolds stress tensor

Subscripts

e	effective
t	turbulence
i, j, k	Cartesian indices

Superscripts and Overbars

$'$	fluctuating quantity in time-averaging
$-$	mean quantity

Table 1 Summary of the governing equations.

Conservation of	ϕ	$\Gamma_{\phi x}$	$\Gamma_{\phi r}$	S_{ϕ}
Mass	1	0	0	0
x-momentum	u	μ_e	μ_e	$\frac{\partial}{\partial x} \left(\mu_e \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial r} \left(r \mu_e \frac{\partial v}{\partial x} \right) - \frac{\partial}{\partial x} \left(\bar{p} + \frac{2}{3} \bar{\rho} k \right)$
r-momentum	v	μ_e	μ_e	$\frac{\partial}{\partial x} \left(\mu_e \frac{\partial u}{\partial r} \right) + \frac{\partial}{\partial r} \left(r \mu_e \frac{\partial v}{\partial r} \right) - \frac{\partial}{\partial r} \left(\bar{p} + \frac{2}{3} \bar{\rho} k \right) - \frac{2 \mu_e \bar{v}}{r^2}$
Turbulent kinetic energy	k	μ_e / σ_k	μ_e / σ_k	$G - \rho \epsilon$
TKE dissipation rate	ϵ	$\mu_e / \sigma_{\epsilon}$	$\mu_e / \sigma_{\epsilon}$	$(C_{\epsilon 1} G - C_{\epsilon 2} \rho \epsilon) \frac{\epsilon}{k}$

where $\sigma_k = 1.0$, $\sigma_{\epsilon} = 1.3$, $C_{\epsilon 1} = 1.44$, $C_{\epsilon 2} = 1.92$, $\mu_e = \mu_t + \mu$, $G_k = \mu_e \left\{ 2 \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial r} \right)^2 + \left(\frac{v}{r} \right)^2 \right] + \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial r} \right)^2 \right\}$.

2.3 Solution Procedure

In the present computation, the time-averaged Navier-Stokes equations, the TKE equation, and the TKE dissipation rate equation are solved numerically by a control-volume finite-difference method [5, 9] together with the the k - ϵ model, equation (4). All equations are in a generalised form of equation (8). The SIMPLE algorithm is utilised for pressure-velocity de-coupling and iteration [5, 8]. The hybrid [9] and the second order upwind (SOU) [7] schemes were used for discretising convection and diffusion transports on a staggered grid cell. The underrelaxation iterative TDMA line-by-line sweeping technique is used for solving the resultant finite-difference equations. The computation was carried out using a PC (Pentium III - 450 MHz) computer. About 5,000 iterations were needed to achieve satisfactory convergence for each calculation case, which requires about 15 minutes of computer time.

3. Flow through an Orifice of Nail [4]

A flow in pipe with a circular orifice of Nail [4] was employed in the present simulation. A schematic configuration of the duct orifice is shown in Fig. 1 below. The pipe with a length of $9D$ has a diameter of 25.4 mm and the 3.2-mm thick orifice diameter is 12.7 mm. Profile measurements of centerline axial velocities, wall-static pressure, Reynolds stresses, and wall shear stresses were measured by using Laser Doppler anemometer (LDA). The flow had the Reynolds number (Re) of 1.84×10^4 and rate of mass flow of 1.356×10^{-2} kg/s with temperature at 300K (see Table 2 below for specifications).



Fig. 1 Geometry of pipe with the orifice.

Table 2 Data for flow through an orifice plate [4].

Parameter	Magnitude
Test section characteristics	
Pipe diameter (D), m	0.0254
Orifice diameter (d), m	0.0127
pipe length (L), m	1.8
Inlet fluid properties (air)	
Mean axial velocity, m/s	5.6
Temperature, K	300
Reynolds number	1.84×10^4

4. Results and Discussion

The comparisons between the predicted results by the k - ϵ turbulence model with different numerical schemes and the measured data of the flow are presented in Figs. 2 through 7. The predicted gas pressure and centerline axial velocities are compared with the measurements, where solid or dash curves are represented for the calculated results while open circles for the measured data.

The computational results are based on a 70×30 non-uniform grid with refinement in the vicinity of the orifice. Grid independence of the numerical results was verified with a 90×50 finer grid. It is found that the differences for both the base grid and the finer grid in local flow properties are marginal. This suggests that grid independent solutions can be obtained with a 70×30 grid, which is used throughout the computations. In order to reduce uncertainties in the inlet profiles of the mean flow field, the inlet boundary conditions were specified at $x/D = -2.0$ for which measured data was available, apart from the radial velocity v which is set to zero.

The distributions of centerline axial velocity predicted with the hybrid and the SOU schemes are compared with the measured data in Fig. 2. A closer examination reveals that predictions with both the

schemes are in generally good agreement with the measurements. However, for prediction with the hybrid scheme, under-predicted results are seen in the orifice region in comparison with experimental data. The use of the SOU leads to substantial improvement for this flow.

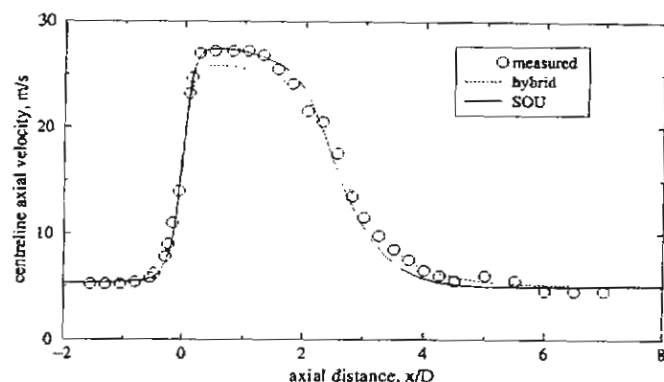


Fig. 2 Effects of numerical schemes on centerline axial velocity profiles with measurements.

Figure 3 compares the profiles of static pressure along the wall using the hybrid and the SOU schemes with the measurements. It is found that there is a high-pressure drop across the orifice. Predictions with both numerical schemes show favourable agreement in upstream region of the orifice. However, immediately after the orifice plate, the calculated wall pressure rises faster than the experiment shows. At downstream regions from the orifice, the SOU results mimic experimental data very well while the hybrid scheme ones are over-predicted. Again, the use of the SOU results in significant improvement for this flow

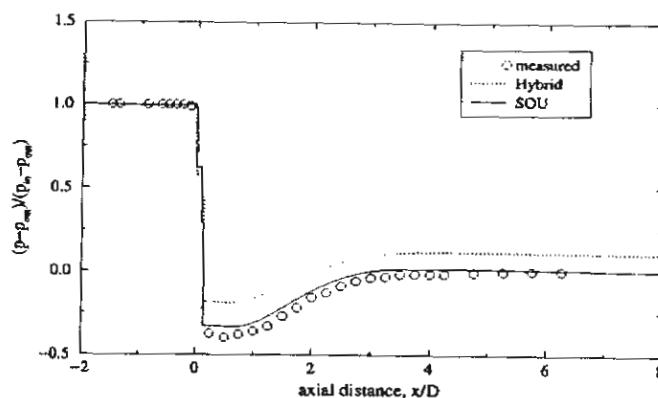


Fig. 3 Effects of numerical schemes on wall-static pressure profiles with measurements.

Streamlines predicted with the hybrid and SOU schemes are shown in Fig. 4 and Fig. 5 respectively. Two recirculation zones are found; one is at the corner upstream of the orifice and the other, a large recirculation zone, a downstream region one. The size of the recirculation zone calculated by the hybrid is slightly larger than that by the SOU. The center of recirculation predicted by the SOU is at about $x/D = 1$ and $r/R = 0.7$ while at about $x/D = 0.85$ and $r/R = 0.7$ is seen for the hybrid. The reattachment length, an important measure of the quality of numerical results, is well predicted ($x_{r,exp} = 2.25D$, $x_{r,calc} = 2.23D$ for SOU, and $x_{r,calc} = 2.1D$ for hybrid).

Velocity vectors predicted by the hybrid and SOU schemes are depicted in Figs. 6 and 7 respectively. High velocities are observed in the core region in a range of $x/D = 0$ to 3 from downstream of the orifice.

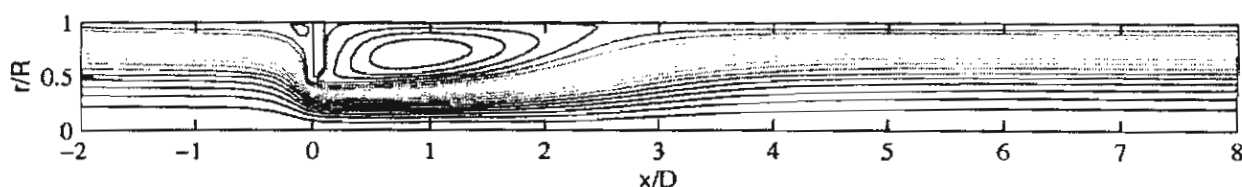


Fig. 4 Streamlines predicted by the hybrid scheme

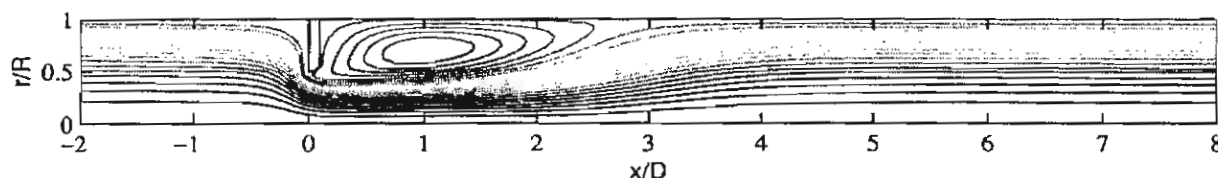


Fig. 5 Streamlines predicted by the SOU scheme

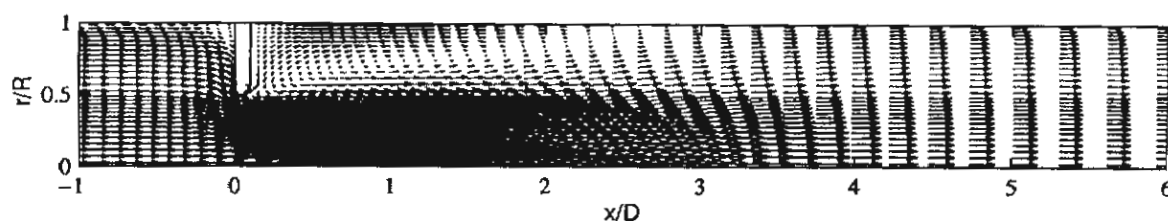


Fig. 6 Velocity vectors predicted by the hybrid scheme

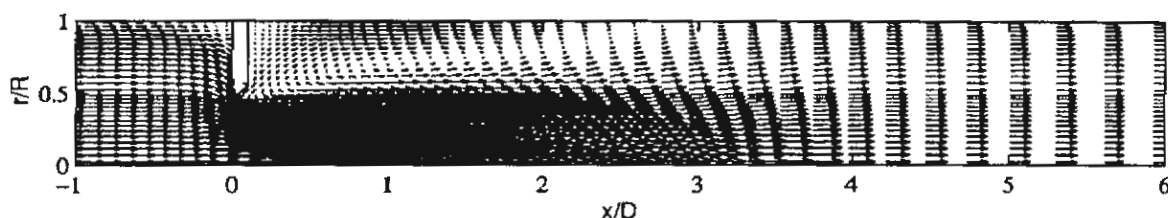


Fig. 7 Velocity vectors predicted by the SOU scheme

5. Concluding Remarks

Simulations of turbulent flow through a circular orifice plate have been carried out by utilizing the $k-\epsilon$ model and two different numerical schemes. The predicted results of mean flow properties were compared with measurements. The conclusions from the investigation can be drawn as follows:

1. The predicted centerline axial velocity and wall-static pressure profiles by the hybrid and SOU schemes are in generally good agreement with measurements.
2. The SOU shows a significant improvement over the hybrid scheme and both schemes give slightly over-predicted results in the core region and slightly under-predicted near the walls in comparison with measurements.

6. Acknowledgements

The authors would like to gratefully acknowledge the Thailand Research Fund for financial support of this research.

7. References

1. Gatski, T.B. (1996), *Turbulent Flows: Model Equations and Solution Methodology*, in *Handbook of Computational Fluid*

Mechanics, Edited by Roger Peyret, Academic Press Ltd, London.

2. Hogg, S. and Leschziner, M.A. (1989a), Computation of Highly Swirling Confined Flow with a Reynolds Stress Turbulence Model, *AIAA J.*, Vol.27, pp.57-63.
3. Jones, W.P. and Pascau, A. (1989), Calculation of Confined Swirling Flows with a Second Moment Closure, *Trans. ASME J. of Fluid Engineering*, Vol.111, Sept, pp.248-255.
4. Nail, G.H. (1991), A Study of 3-dimensional flow through orifice meters, Ph.D. Dissertation, Texas A&M University.
5. Patankar, S.V. (1980), *Numerical Heat Transfer and Fluid Flow*, Hemisphere, Washington, D.C.
6. Patankar, S.V. and Spalding, D.B. (1970), *Heat and Mass Transfer in Boundary Layers*, Intertext Books, London.
7. Shyy, W., Thakur, S., and Wright, J. (1992), Second Order Upwind and Central Difference Schemes for Recirculating Flow Computation, *AIAA J.*, Vol.30, No.4, pp.931-932.
8. Sloan, D.G., Smith, P.J., and Smoot, L.D. (1986), Modeling of Swirl in Turbulent Flow System, *Progr. Energy Combust. Sci.*, vol. 12, pp. 163-250.
9. Versteeg, H.K. and Malalasekera, W. (1995), *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Longman Scientific & Technical, Longman Group Limited, Essex, England.
10. Wilcox, C.D. (1993), *Turbulent Modelling for CFD*, DCW Industries, Inc., California.

The Fifth Annual National Symposium
on Computational Science
and Engineering



ANSCSE 2001

Organizers :

National Electronics and Computer Technology Center

June 19 - 20, 2001

Bangkok Convention Center (BCC), Central Plaza

Numerical Simulation of Isothermal Swirling Turbulent Flow in a Combustor

P. Promvonge¹, L. Vongsarnpigoon² and K. Silapabanleng³

¹Dept. of Mechanical Engineering, Faculty of Engineering,
King Mongkut's Institute of Technology Ladkrabang,
Bangkok 10520, Thailand. Email: kppongje@kmitl.ac.th

²Dept. of Mechanical Engineering, Faculty of Engineering,
Mahanakorn Univeristy of Techonology, Bangkok 10530, Thailand.

³Dept of Mechanical Engineering, Faculty of Engineering,
Chulalongkorn University, Bangkok 10330, Thailand.

Abstract

The paper presents the simulation of strongly swirling confined flow in a combustor with an algebraic Reynolds stress model (ASM). The k - ϵ turbulence model was also employed in the present simulation for comparison. For the flows compared, the predicted results of gas tangential and axial velocities based on this model are in good agreement with measurements. The computation shows that the ASM is superior to the k - ϵ model in capturing the stabilising effect of the swirl.

Keywords: swirl, turbulent flow, algebraic stress model.

1. Introduction

Swirl flows have been of considerable interest over the past decades because of their occurrence in industrial applications, such as furnaces, utility boilers, internal combustion engines, gas turbine combustors and dust collectors [2, 5]. Swirl has been used in combustion systems to enhance the flame stability, the mixing and heat transfer besides prolonging the fuel residence time and abating the pollutants. This is because under appropriate conditions, swirl can be employed to induce a central recirculation zone. The recirculating flow generates additional turbulence in the shear layer between the reverse flow and the surrounding forward flow and helps stabilize the flame in combustors. Swirling turbulent flows are physically complex in nature due to the effect of a swirl-turbulence interaction. The turbulence structure in swirling turbulent flows is generally highly non-isotropic and nonhomogeneous.

Computation of swirling flows is a difficult and challenging task. Large velocity gradients appear in these flows, so numerical problems and turbulence modelling play a significant role in their analysis. The commonly used, the k - ϵ model may not be suitable for simulating swirling turbulent flows [3, 6, 7]. It is also found that the use of modified k - ϵ models or even the non-linear k - ϵ model [13] leads to no significant improvement of the predictions in swirling flows. The second-order moment closure models, i.e., the Reynolds stress model (RSM) and the algebraic Reynolds stress model provide better methods for the simulation of swirling turbulent flows [6, 7]. The RSM is regarded as a most logical approach to the turbulence closure problem, which does not need any ad hoc modification for extra strain rates. However, in the prediction of swirling flows with the RSM, it is necessary to solve a total of 11 governing differential equations of elliptic type: a continuity equation, three momentum equations, an ϵ -equation, and six equations for the Reynolds stresses. This leads

to much extra computational effort to solve six Reynolds stress transport equations simultaneously [6, 7] and much attention needs to be paid to numerical stability and inlet boundary conditions. It is for this reason that a simplified algebraic Reynolds stress turbulence model [16] in axisymmetric cylindrical co-ordinates is employed for simulating strongly swirling flows.

In this paper, the formulation of the present model, the procedure of numerical solution based on the present ASM and the validation results of the proposed model are described in detail. The validation was conducted through successive comparisons of the calculated gas tangential, radial and axial velocities with the test data measured in a cold flow combustor.

2. Mathematical Formulation

2.1 Governing Equations

The phenomenon under consideration is governed by the steady two-dimensional axisymmetric form of the continuity and the time-averaged incompressible Navier-Stokes equations. In the Cartesian tensor system these equations can be written in the following form:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \tau_{ij}) \quad (2)$$

where ρ , u_i , p and x_i are the density, mean velocity tensor, mean pressure and coordinate tensor respectively. The mean viscous stress tensor, $\bar{\tau}_{ij}$, is approximated as:

$$\bar{\tau}_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

where μ is laminar viscosity. The time-averaged Reynolds stress tensor, $\tau_{ij} = -\overline{\rho u_i u_j}$, in the above equation is not known and thus, models are needed to express it in terms of the solution variables. In the present study, two turbulence models are used, namely the k - ϵ model and an algebraic stress model (ASM). The k - ϵ model has been reviewed in references [4, 15] and it will be described briefly. The standard k - ϵ model version relates the turbulent eddy viscosity to the turbulence kinetic energy k and the dissipation rate ϵ through Boussinesq's approximation as:

$$\tau_{ij} = -\frac{2}{3} \delta_{ij}(\rho k) + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

where $\mu_t = \rho C_\mu k^2 / \epsilon$ is the turbulent eddy viscosity and ϵ is the dissipation rate of turbulence kinetic energy (TKE). The modelled equation of the TKE, k is given by:

$$\frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G - \rho \epsilon \quad (5)$$

in which $\mu_e = \mu_t + \mu$ is effective viscosity. Similarly the dissipation rate of TKE is given by the following equation:

$$\frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} \left(\frac{\mu_e}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} (C_{\varepsilon 1} G - C_{\varepsilon 2} \rho \varepsilon) \quad (6)$$

where G is the rate of generation of the TKE while $\rho \varepsilon$ is its destruction rate. G is given by:

$$G = \mu_e \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \right] \quad (7)$$

The boundary values for the turbulent quantities near the wall are specified with the wall function method [10]. $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1.0$, and $\sigma_\varepsilon = 1.3$ are empirical constants [4, 15] in the turbulence transport equations. Reynolds-averaged transport equations can be solved for τ_{ij} , [4, 15] the modelled equations for which are:

$$\frac{\partial \tau_{ij}}{\partial t} + \frac{\partial (u_k \tau_{ij})}{\partial x_k} = -G_{ij} - \Phi_{ij} + D_{ij} + \varepsilon_{ij} \quad (8)$$

where

$$\begin{aligned} G_{ij} &= \text{local production} = \rho P_{ij} = - \left(\overline{\rho u_i' u_k'} \frac{\partial u_j}{\partial x_k} + \overline{\rho u_j' u_k'} \frac{\partial u_i}{\partial x_k} \right) \\ \Phi_{ij} &= \text{local pressure strain} = -C_1 \frac{\rho \varepsilon}{k} \left(\overline{u_i' u_j'} - \frac{2}{3} k \delta_{ij} \right) - C_2 \left(G_{ij} - \frac{2}{3} G \delta_{ij} \right) \\ D_{ij} &= \text{net diffusive transport} = - \frac{\partial}{\partial x_k} \left(\frac{\mu_e}{\sigma_\tau} \frac{\partial \overline{u_i' u_j'}}{\partial x_k} \right) \\ \varepsilon_{ij} &= \text{local dissipation tensor} = \frac{2}{3} \rho \varepsilon \delta_{ij} \end{aligned}$$

in which $C_1 = 2.5$, and $C_2 = 0.55$ are model constants.

2.2 Algebraic Reynolds Stress Model (ASM)

For simplicity in solving the six Reynolds stresses, Rodi's approximation [11] is used in this study and the Reynolds stress transport can be expressed in algebraic form as follows:

$$\frac{D \tau_{ij}}{Dt} - D_{ij} = \frac{\tau_{ij}}{\rho k} \left(\frac{Dk}{Dt} - D_k \right) \quad (9)$$

Substitution of eqs. (5) and (8) into eq. (9) gives the desired algebraic expression for τ_{ij} :

$$-G_{ij} - \Phi_{ij} + \frac{2}{3} \delta_{ij} \rho \varepsilon = \frac{\tau_{ij}}{\rho k} (G - \rho \varepsilon) \quad (10)$$

The ASM expressions can thus be rewritten as:

$$\overline{\rho u_i' u_j'} = \frac{2}{3} \delta_{ij} \rho k + \frac{\lambda k}{\varepsilon} \left(G_{ij} - \frac{2}{3} \delta_{ij} G \right) \quad (11)$$

where the empirical constant λ , was found to be 0.135, [16] is defined as:

$$\lambda = \frac{1 - C_2}{C_1 - 1 + \frac{G}{\rho \epsilon}} \quad (12)$$

2.3 Common Form for the Equations

All the governing equations can be re-organised and expressed in a standard form that includes the convection, diffusion, and source terms for 2-D axisymmetric flows as follows:

$$\frac{\partial}{\partial x}(\rho u \phi) + \frac{1}{r} \frac{\partial}{\partial r}(\rho v r \phi) - \frac{\partial}{\partial x} \left(\Gamma_{\phi} \frac{\partial \phi}{\partial x} \right) - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma_{\phi} \frac{\partial \phi}{\partial r} \right) = S_{\phi} \quad (13)$$

where ϕ may stand for any variable including the velocity components, Γ_{ϕ_x} and Γ_{ϕ_r} are the exchange coefficients for ϕ , and S_{ϕ} is the source term.

2.4 Solution Procedure

In the present computation, the time-averaged Navier-Stokes equations, the TKE equation, and the TKE dissipation rate equation are solved numerically by a control-volume finite-difference method [9, 14] together with the turbulence model equations, equation (4) for the $k-\epsilon$ model or equation (11) for the ASM. All equations are in a generalised form of equation (13). The SIMPLE algorithm is utilised for pressure-velocity de-coupling and iteration [9, 14]. The hybrid, the upwind, the quadratic upstream interpolation for convective kinematics (QUICK) [8] and the second-order upwind (SOU) [12] schemes were used for discretising convection and diffusion transports on a staggered grid cell. The under-relaxation iterative TDMA line-by-line sweeping technique is used for solving the resultant finite-difference equations. The computation was carried out using a personal computer (Pentium III-550 MHz). About 20,000 iterations were needed to achieve satisfactory convergence for each calculation case, which requires about 3.0 hr of computer time.

3. Swirling Flow of Ahmed (1997)

The performances of the $k-\epsilon$ model and the ASM are compared for a turbulent swirling flow in a combustor of Ahmed (1997) [1], shown in Fig. 1. The details of the geometry and fluid properties are given in Table 1 below. A single air stream entered the test section through a secondary annulus and passed through an adjustable vane swirler. Experimental mean velocity profiles for u , v and w , turbulence intensities and the Reynolds stresses were measured at downstream locations of the expansion corner. All data in radial profiles were obtained with a two-components LDV system.

Since axial, radial and tangential velocities and turbulence intensities including shear stresses were provided at $x/D = 0.0633$ downstream of the inlet, computations are started at that location to reduce uncertain inlet conditions. In the present computation, performances of both the standard $k-\epsilon$ model and the ASM are examined by comparing the predicted velocity profiles at ten locations, namely, $x/D = 0.0633, 0.167, 0.25, 0.333, 0.417, 0.5, 0.583, 0.669, 0.833$ and 1.0 .

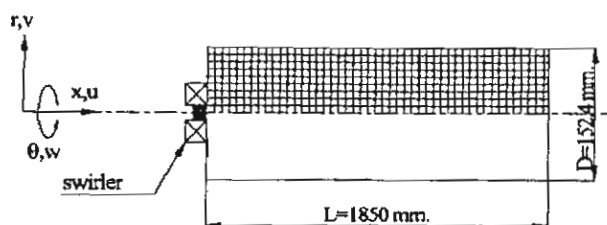


Fig. 1 Flow geometry and computational domain.

Table 1 Data for flow of Ahmed (1997)

Test section characteristics	
swirl vane angle (degree)	45
chamber length (m)	1.85
chamber dia. (m)	0.1524
Inlet flow conditions	
Reynolds number	200,000.
average velocity, U_o (m/s)	30.4

4. Results and Discussion

4.1 Effects of Grid Densities and Numerical Schemes

The comparisons between the predicted results by the $k-\epsilon$ turbulence model and the ASM and the measured data of the flow are presented in Figs. 2 through 7. The predicted axial, radial and tangential velocities are compared with the measurements, where solid or dash curves are represented for the calculated results while open circles for the measured data. The computational results are based on a 95×40 non-uniform grid with refinement in the vicinity of the slit. Grid independence of the numerical results was verified with a 120×60 finer grid. It is found that the differences for both the base grid and the finer grid in local flow properties are marginal. This suggests that grid independent solutions can be obtained with a 95×40 grid, which is used throughout the computations.

The axial, radial and tangential mean velocity profiles predicted with the $k-\epsilon$ model and four different numerical schemes are compared with the measured data in Fig. 2. It is found that predictions with the four schemes generally are in fairly good agreement with the measurements. However, in prediction with hybrid, QUICK and SOU, under-predicted tangential velocity profiles are seen in all locations while good agreement is found for axial and radial velocity profiles in comparison with experimental data. For the four numerical schemes, the FOU scheme performs the best agreement in prediction of tangential velocity but shows poorer agreement for axial and radial velocity profiles. Thus, in terms of the strength of the tangential velocity, only the FOU scheme is considered for computation of this flow.

4.2 Flow Field

Predictions of velocity profiles using the $k-\epsilon$ model and the ASM are compared with experimental data in Figs. 2a, 2b and 2c for the axial, radial and tangential velocity profiles respectively. For the radial profile of axial velocity in Fig. 2a, it is seen that the predicted profiles show no difference for both turbulence models and generally are in favorable agreement with measurements especially for $x/D \geq 0.333$. The ASM However, both

turbulence models give under-predicted results in the core region. Predictions with the ASM show closer agreement with measurements than those with the $k-\epsilon$ model in some locations as can be seen at $x/D = 0.167$ and 0.25 between $r/D = 0.35$ and $r/D = 5$. The results of Fig. 2a indicate that the predicted axial velocity recovers and progresses to uniformity at a faster rate than in the measurements.

Fig. 2b compares the radial profile of radial velocities predicted by the $k-\epsilon$ model and the ASM with measurements. It is of interest to note that prediction with the ASM shows closer agreement with experimental data than that with the $k-\epsilon$ model. The ASM results mimic very well the measurements while the $k-\epsilon$ model gives under-predicted results, especially in the vicinity of the inlet. For the downstream region far away from the inlet ($x/D > 0.417$), both turbulence models yield similar results and agree well with the measurements.

The tangential velocity profiles predicted by the $k-\epsilon$ model and the ASM along with the measurements are shown in Fig. 2c. It is worth noting that the tangential velocity predictions with the $k-\epsilon$ model display a rapid decay to a forced vortex profile (solid-body rotation) while the experimental data correspond to a combined free and forced vortex profile. It is clear that the tangential velocity predictions do not agree with the measured data even for trends. Since the calculations were fairly free of numerical diffusion, discrepancies between the data and the predictions can be attributed to two sources: - improper boundary conditions at the inlet plane and deficiencies of the turbulence model.

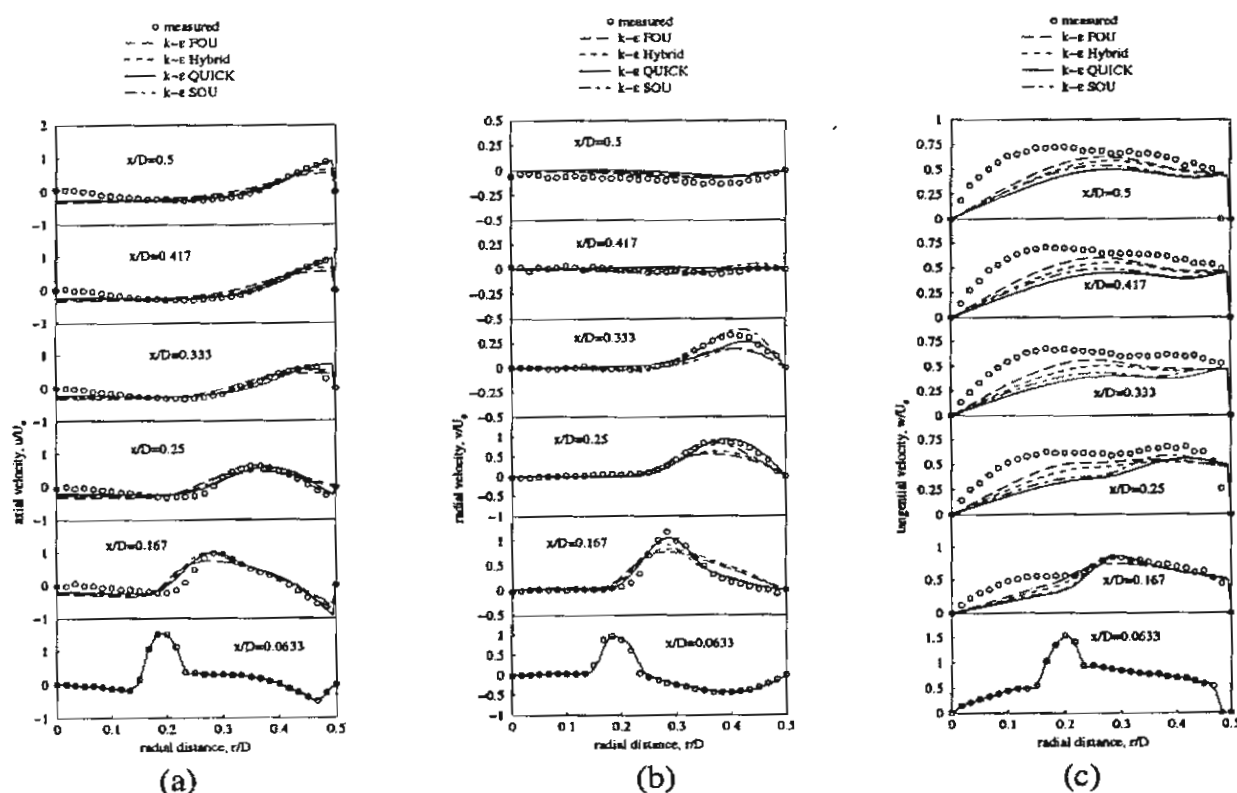


Fig. 2 Effect of numerical schemes on the axial, radial and tangential velocity profiles.

All inlet values except the ϵ and the k , were obtained indirectly from the experiment. The values of ϵ were derived from a constant length-scale assumption and those of k from an

approximation of total kinetic energy. To study the sensitivity of the inlet ε and k profiles to the flow field, calculations were made using different ε and k distributions, by increasing or decreasing their estimated values by a factor of 10. It was found that the calculated velocity was not affected significantly. The poor agreement between the predictions and measurements is, therefore, more likely to be due to the deficiencies of the turbulence model. The use of ASM results in better overall agreement with the measurements than possible with the k - ε model. The major difference between calculations with the two turbulence models is most clear in the recirculation region and in the tangential velocity profiles. The agreement between the calculation with the ASM and the experimental data is fairly good. The peak values and a combined forced and free (Rankine) vortex motion for the tangential velocity profiles are well predicted. The recirculation zone (IRZ) is, however, longer and wider than that with the k - ε model.

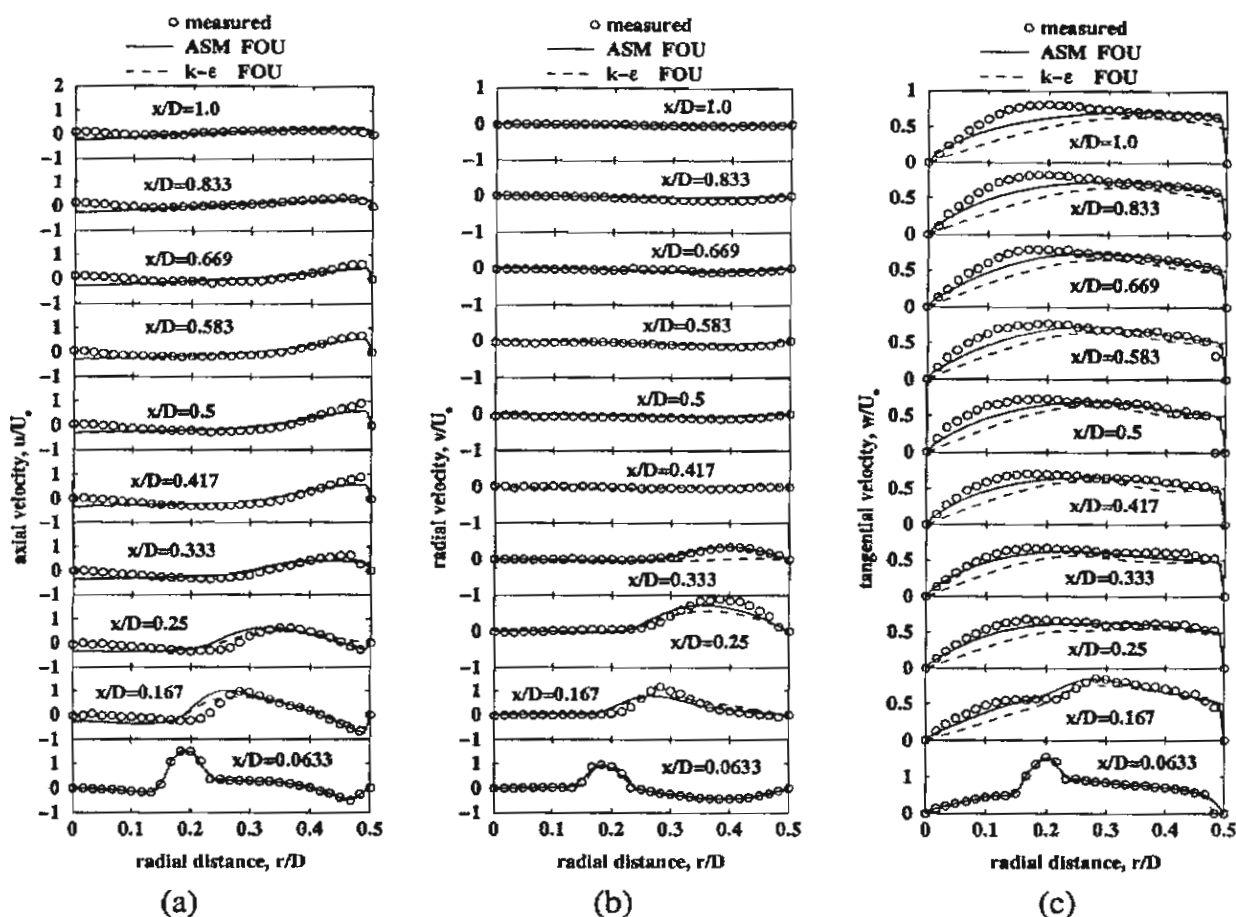


Fig. 3 Comparison of predicted velocity profiles with measurements; (a) axial velocity, (b) radial velocity and (c) tangential velocity profiles.

Streamlines predicted by the ASM and the k - ε model are shown in Figs. 4 and 5 respectively. The contour plot of streamlines is in a range of $x/D = 0$ to 3 and $r/D = 0$ to 0.5, which has a high change of velocities. From the figures, there is no change of velocities at about $x/D = 1.5$ for the ASM and at some $x/D = 3$ for the k - ε model. Two recirculation zones are identified; one is at the corner and the other, an internal recirculation zone (IRZ) or central