

## C717154 : MAJOR CHEMICAL ENGINEERING  
KEY WORD: TEE MIXER / T-JUNCTION / COMPUTATIONAL FLUID DYNAMICS

SUPACHAI TERMSINSUWAN : SIMULATION OF FLUID MIXING INDUCED AT THE TEE MIXER HAVING A SQUARE CROSS-SECTION. THESIS ADVISOR : SOMPRASONG SRICHAI, Ph. D. THESIS COADVISOR : JACK ASAVANANT, Ph. D. 204 pp. ISBN 974-636-696-3.

Computational Fluid Dynamics (CFD) , which is the numerical method for solving a set of the equations such as the continuity equation , the conservation of momentum equation and the conservation of energy equation including turbulent flow model in small control volume , was applied to simulate the mixing of fluid induced at tee mixer having a square cross-section with circular jet exit. The simulation results could provide more details of the fluid mixing phenomena. The velocity and temperature values of mixing fluid obtained from CFD technique were verified and found to be in close agreement to those from experimental data of Kamotani and Greber [1974]. In addition, CFD technique was also used to predict three features of jet exits, namely, single row of square exit, single row of transverse slot exit and single row of longitudinal slot exit, respectively. It was found that the side distribution of jet stream from single row of transverse slot exit is better than that from single row of square exit, and that from single row of square exit is better than that from single row of longitudinal slot exit. On the other hand, the penetration of jet stream from single row of longitudinal slot exit is better than that from single row of square exit, and that from single row of square exit is better than that from single row of transverse slot exit.

ภาควิชา.....วิศวกรรมเคมี  
สาขาวิชา.....วิศวกรรมเคมี  
ปีการศึกษา.....2539

ลายมือชื่อนิสิต.....  
ลายมือชื่ออาจารย์ที่ปรึกษา.....  
ลายมือชื่ออาจารย์ที่ปรึกษาร่วม.....