AN ADJOINT METHOD FOR OPTIMAL VENTILATION DESIGN

Wei Liu[2,1], Qingyan Chen[1,2]
[1] School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA
[2] School of Environmental Science and Engineering, Tianjin University, Tianjin 300072, China

ABSTRACT

This investigation studied an adjoint method to achieve the optimal design of ventilation in enclosed environment and validated it by two-dimensional cases. This study defined a part of the flow field and/or temperature field as the design objective and determined the thermo-fluid boundary conditions as the design variables. By using the adjoint method together with the optimality condition that was implemented in OpenFOAM, this investigation could find the optimal air supply parameters. With the air supply parameters, this study used CFD to calculate the flow and/or temperature fields, which are in a good agreement with the design objective. However, the adjoint method may lead to multiple optimal solutions.

INTRODUCTION

Ventilation is one of the most important factors for maintaining acceptable indoor environment in enclosed spaces, such as buildings and transportation vehicles. Ventilation could control the air temperature, relative humidity, air speed, and chemical species concentrations in the air of the enclosed spaces. There are many standards (ASHRAE, 2007) to formulate the requirements of indoor environment, and ventilation design should be optimal for creating and maintaining an environment to satisfy the requirements.

Traditionally, researchers and engineers applied a "trial-error" process in designing ventilation, which means predicting and evaluating the ventilation performance with different design variables to find a scenario that has the best agreement with the design objective. According to Chen (2009), researchers and engineers normally predicted or evaluated the ventilation performance typically by analytical and empirical models, experimental measurements, and computer simulations. With the development of computer technology, the Computational Fluid Dynamics (CFD) simulations in computer simulations are most popular for predicting ventilation performance recently. The CFD simulation could provide the field distributions of air velocity, air temperature, species, etc. With a validated turbulence model, the CFD simulation would be more accurate and informative than the analytical models, empirical models, multizone models, and zonal models and much faster than the experimental measurements. However, since this "trial-error" process requires CFD simulations for many scenarios, it would need days or months to obtain an optimal design for a ventilated space. At the same time, the "trial-error" process with the other methods may be inaccurate, non-informative, expensive, and/or time-consuming. Most importantly, because of the subtlety and complexity of fluid flow, it is unlikely that the repeated trials in an interactive analysis and design procedure can lead to a truly optimal design. Jumping out the notion of this "trial-error" process, the optimization approach could achieve this design procedure.

The optimization approach has been widely applied in identifying groundwater pollution source as linear optimization method (Gorelick et al., 1983), maximum likelihood method (Wagner, 1992), and nonlinear optimization method (Mahar and Datta, 2000). The approach involving the solution of an adjoint system of equations has recently attracted substantial interests from mathematicians and computational scientists. Systematic mathematical and numerical analysis of optimal control problems of different types are available such as Dirichlet, Neumann, and distributed controls as well as finite-dimensional controls for the steady state Navier-Stokes system (Gunzburger et al., 1989, 1991, 1992). For the time-dependent Navier-Stokes system, Cuverlier (1976) conducted the mathematical treatments of optimal boundary for heat flux control with free convection. The existence of optimal solutions was proven and necessary conditions were derived for characterizing optimal controls and states. Jameson (1988, 1995) developed an adjoint approach for potential flow. He used Euler equations and Navier–Stokes equations to find the geometry that minimizes some objective function subject to a set of constraints. Gunzburger (1999) built up adjoint equations and their solving method for a suppression of instabilities in boundary layer flows by injection/suction control and stress matching problem (Joslin et al. 1997). The optimization approach solved by the adjoint method shows a great potential (fast and most optimal). They could couple multiple parameters in the optimal design of ventilation in an enclosed environment.

Due to the successes in the previous studies, this investigation established an adjoint method for the optimal design of indoor airflow with causal aspect as thermo-fluid boundary conditions and flow and/or temperature fields as design objective. By implementing this method into the CFD solver OpenFOAM (Open Field Operation And Manipulation) (2007), this study determines the air supply parameters in two two-dimensional ventilated spaces.

METHOD

To apply the optimization approach, this study firstly transformed the design problem into a control problem. Then an adjoint system can be established and implemented in OpenFOAM.

Design problem as control problem

In ventilation design, the thermo-fluid boundary conditions are design variables and flow and/or temperature fields are the design objective. Then, the control problem consists of the following components:

- State variables: velocity V , pressure p, temperature T ;
- Design variables: inlet air velocity V_{inlet} and inlet air temperature T_{inlet};
- State equations: Navier-stokes equations denoted by $S=(S_1,S_2,S_3,S_4,S_5)$ and:

$$S_1 = -\nabla\cdot V = 0 \quad (1)$$

$$(S_2,S_3,S_4)^T = (V\cdot\nabla)V + \nabla p - \nabla\cdot(2\nu D(V)) - \gamma\vec{g}(T-T_{op}) = 0 \quad (2)$$

$$S_5 = \nabla\cdot(VT) - \nabla\cdot(\kappa\nabla T) = 0 \quad (3)$$

- Objective function:

$$J(V_{inlet},T_{inlet}) = \alpha\int_{\theta_1}(V-V_0)^2 d\theta_1 + \beta\int_{\theta_2}(T-T_0)^2 d\theta_2 \quad (4)$$

In the state equations, S_1, S_2, S_3, S_4, and S_5 are vector components of S, ν is the effective viscosity, $D(V)=(\nabla V+(\nabla V)^T)/2$ the rate of strain tensor, κ is the effective conductivity, T_{op} is the operating temperature, γ is the thermal expansion cofficient of the air, $\vec{g}$ is the gravity vector. In the objective function, velocity distribution V_0 on domain θ_1 and temperature distribution T_0 on domain θ_2 are the design objective and V and T are calculated from the state function. α and β are chosen to adjust the relative importance. The Boussinesq approximation is applied to simulate the thermal effect, while air density is assumed constant that has been a common approach for room airflow simulations. Then, the optimization approach is to minimize objective function $J(V_{inlet},T_{inlet})$ subject to the state equations.

Adjoint equations

By introducing a Lagrangian function L, the constrained control problem can be transformed into an unconstrained control problem. An augmented objective could be:

$$L = J + \int_{\Omega}(p_a,V_a,T_a)Sd\Omega \quad (5)$$

where Ω stands for the flow domain, V_a the adjoint velocity, p_a the adjoint pressure, and T_a the adjoint temperature. These parameters are Lagrangian multipliers. Then the total variation of L is:

$$\delta L = \delta_{V_{inlet}}L + \delta_{T_{inlet}}L + \delta_V L + \delta_P L + \delta_T L \quad (6)$$

that includes the contributions from changes in V_{inlet} and T_{inlet} and the corresponding changes in state variables V , p, and T . The state equations should be calculated once for each variable to satisfy $S=0$. To find the relationship between the variations of V_{inlet} and T_{inlet} and the δL, the adjoint velocity V_a, the adjoint pressure p_a and the adjoint temperature T_a are chosen to satisfy:

$$\delta_V L + \delta_P L + \delta_T L = 0 \quad (7)$$

Based on Equation (7), this study has developed the adjoint equations. Due to the limitation on the paper length, the detailed derivation procedure is not shown here and only the adjoint equations in final form are presented as follows:

$$-\nabla\cdot V_a = 0 \quad (8)$$

$$-\nabla V_a\cdot V - (V\cdot\nabla)V_a - \nabla\cdot(2\nu D(V_a)) + \nabla p_a + 2(V-V_0) = 0 \text{ for domain } \theta_1 \quad (9)$$

$$-\nabla V_a\cdot V - (V\cdot\nabla)V_a - \nabla\cdot(2\nu D(V_a)) + \nabla p_a = 0 \text{ for domain } \Omega\backslash\theta_1 \quad (10)$$

$$-V\cdot\nabla T_a - \nabla\cdot(\kappa\nabla T_a) + 2(T-T_0) = 0 \text{ for domain } \theta_2 \quad (11)$$

$$-V\cdot\nabla T_a - \nabla\cdot(\kappa\nabla T_a) = 0 \text{ for domain } \Omega\backslash\theta_2 \quad (12)$$

At the same time, this investigation has developed the corresponding adjoint boundary conditions. On the inlet and wall, the adjoint velocity boundary condition is $V_a\big|_{inlet\text{ and }wall} = 0$ and the adjoint pressure boundary condition is zero gradient. On the outlet, the adjoint velocity and pressure boundary conditions are:

$$V_{at}V_n + \nu(\vec{n}\cdot\nabla)V_{at} = 0 \quad (13)$$

$$(\vec{n}\cdot\nabla)V_{an} = -\nabla_t\cdot V_{at} \quad (14)$$

$$p_a = V_a\cdot V + V_{an}V_n + \nu(\vec{n}\cdot\nabla)V_{an} \quad (15)$$

The subscripts t and n mean tangential and normal components, respectively. On the wall, the adjoint temperature boundary condition of zero gradient is applied. On the inlet and outlet, this study calculated the temperature as:

$$T_a V_n + \kappa(\vec{n}\cdot\nabla)T_a = 0 \qquad (16)$$

According to Equations (6) and (7), the variation of L is calculated as

$$\delta L = \delta_{V_{inlet}} L + \delta_{T_{inlet}} L = \delta_{V_{inlet}} J + \delta_{V_{inlet}} J + \int_\Omega (p_a, V_a, T_a)\delta_{V_{inlet}} S d\Omega + \int_\Omega (p_a, V_a, T_a)\delta_{T_{inlet}} S d\Omega \qquad (17)$$

Thus, the sensitivity of the augmented objective becomes:

$$\frac{\delta L}{\delta V_{inlet}} = \int_\Omega (p_a, V_a, T_a)\frac{\partial S}{\delta V_{inlet}} d\Omega \qquad (18)$$

$$\frac{\delta L}{\delta T_{inlet}} = \int_\Omega (p_a, V_a, T_a)\frac{\partial S}{\partial T_{inlet}} d\Omega \qquad (19)$$

We set

$$\delta V_{inlet} = -\lambda_1 [\int_\Omega (p_a, V_a, T_a)\frac{\partial S}{\delta V_{inlet}} d\Omega]^T \qquad (20)$$

$$\delta T_{inlet} = -\lambda_2 [\int_\Omega (p_a, V_a, T_a)\frac{\partial S}{\delta T_{inlet}} d\Omega]^T \qquad (21)$$

where λ_1 and λ_2 are positive constants. The variation of L is always negative and the value of L would always decreases until an optimum condition is achieved. Therefore, by using the simple steepest descent algorithm, the variation of V_{inlet} and T_{inlet} at the boundary face cell can be approximately as:

$$\delta V^i_{inlet} = -\lambda_1 [V^i_{inlet} \cdot V^i_{a(inlet)} - (\nabla T_{inlet})^i T_{a(inlet)}] \qquad (22)$$

$$\delta T_{inlet} = -\lambda_2 (-\gamma \vec{g} \cdot V_{a(inlet)} + \vec{n} \cdot V_{inlet} T_{a(inlet)}) \qquad (23)$$

where the $V_{a(inlet)}$ and $T_{a(inlet)}$ are the calculated adjoint velocity and temperature at the cell adjacent to the corresponding boundary face cell. The superscript i denotes a vector component.

Numerical method

Figure 1 shows that the calculation begins with an initial inlet boundary conditions of air velocity and temperature. With the initial boundary conditions, our method firstly solves the state equations with N_1 iterations. Then the method initializes and calculates the adjoint equations with N_2 iterations. Based on the results of state equations and adjoint equations and by using the optimality conditions, one can obtain new inlet air velocity and temperature. This forms a design cycle. The new boundary conditions are used to calculate the state equations again until the convergence criterion is satisfied.

N_1 and N_2 are number of iteration for solving the state equations and adjoint equations, respectively, until converge in each design circle. If N_1 and N_2 is only one, the method is called continuous adjoint, which is all-at-once or one-shot method.

The solver uses a Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm to couple the velocity and pressure in state equations and adjoint velocity and pressure.

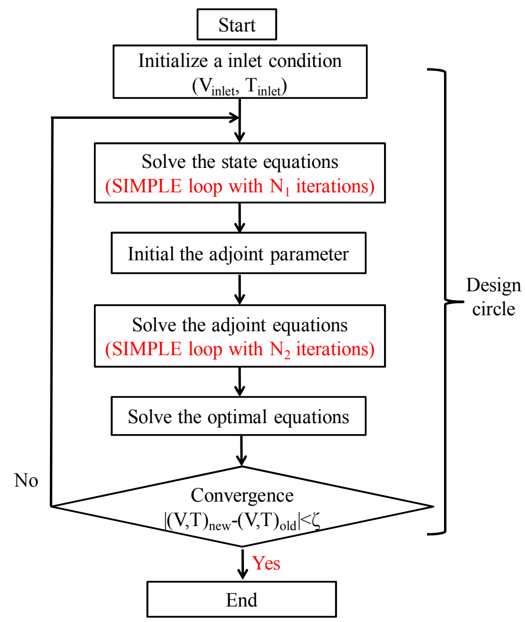


Figure 1 Solving flow chart of the adjoint method

This investigation applied the standard k-ε model (Launder and Sharma, 1974) to simulate turbulence in the state equations. To decrease the calculation load, this study assumed the turbulence to be "frozen" (Dwight et al., 2006; Othmer, 2008) and the turbulent viscosity in the state equations was re-used for the adjoint diffusion term. For the convection terms, this study adopted the standard finite volume discretisation of Gaussian integration with the first-order upwind-differencing-interpolation (Gauss Upwind) scheme. The diffusion terms adopted the standard finite volume discretisation of Gaussian integration with central-differencing-interpolation (Gauss Linear) scheme. The adjoint equations applied the Gauss Linear scheme for the gradient term. Neither Gauss upwind scheme nor Gauss liner scheme is the most accurate, but these two schemes could make the calculation stable. This investigation solved the state and adjoint continuity equation by the generalised Geometric-Algebraic Multi-Grid (GAMG) solver. GAMG is faster than standard methods when the increase in speed by solving first on coarser meshes outweighs the additional costs of mesh refinement and mapping of field data. The adjoint method was implemented in OpenFOAM, which is a CFD toolbox and can be used to simulate a broad range of physical problems.

RESULTS

This study has validated the adjoint method by applying it to two two-dimensional ventilation cases: one was with isothermal flow and the other non-isothermal flow.

Isothermal case

The isothermal case was from Nielsen (1978) who provided detailed experimental data. Figure 2 shows the geometry of the case where L/H=3.0, h/H=0.056, and t/H=0.16, where H=3.0 m. The inlet velocity was V_x=0.455 m/s, and V_y=0 with a turbulence intensity of 4%.

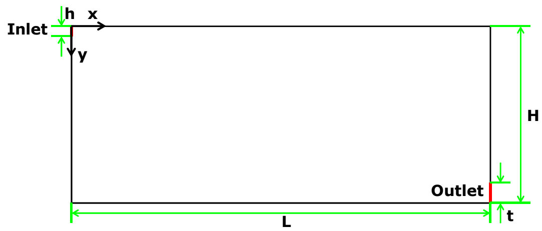


Figure 2 Sketch of the two-dimensional isothermal case.

This study firstly conducted a forward CFD simulation with the standard k-ε model by assigning the inlet air velocity as V_x=0.455 m/s, and V_y=0. The forward CFD simulation was to obtain a flow field in the room and a part of it can be used as the design objective. Figure 3 shows the computed air velocity profile at the lower part of line x=6 m that was selected as the design objective. Then, this study aimed to find the optimal inlet air velocity based on this design objective (V_0 in Equation (4)). This study chose the objective because the location could be some area around the occupant in an indoor environment. The optimized inlet air velocity may be the same as V_x = 0.455 m/s, and V_y = 0 since this value is obviously the most optimal. However, the solution may lead to some other values as long as the objective is reached.

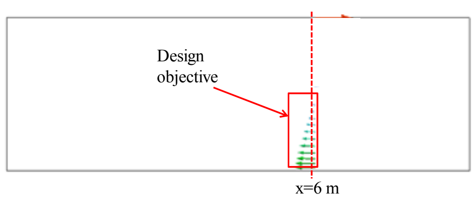


Figure 3 Design objective for the two-dimensional isothermal case

Since the isothermal case was simple, the adjoint method could lead to a converged solution with limited iterations. For simplicity, this study used N_1=N_2=1 (referred to Figure 1) in the calculation. This investigation used four different inlet air velocities as the initial inlet conditions, which were different from the true value of V_x = 0.455 m/s, and V_y = 0.0 m/s:

(1) V_x = 1.0 m/s and V_y = 0.0 m/s

(2) V_x = 1.0 m/s and V_y = 0.1 m/s

(3) V_x = 1.0 m/s and V_y = -0.1 m/s

(4) V_x = 1.0 m/s and V_y = 0.3 m/s

With the initial boundary conditions, the flow and adjoint equations were calculated for 2000 design circles with N_1=N_2=1. Figures 4 and 5 show that the first three initial conditions could gradually changed to the true result of V_x = 0.455 m/s and V_y = 0.0 m/s. The error for V_x was as small as 0.25~1.8%. The error for V_x was as small as 0.25~1.8% and the V_y calculated was exactly 0.0 m/s. Unexpectedly, the calculation with the initial velocity of V_x = 1.0 m/s and V_y = 0.3 m/s led to a final velocity of V_x = 0.132 m/s and V_y = 2.578 m/s. Figure 6 further shows that all the calculations could lead to a small value of objective function. Figures 4-6 illustrate that the calculations at the beginning of were unstable. This was caused by the partial converged flow field and adjoint equations as N_1=N_2=1. The calculation was also unstable when the cycle equals 500 and it is difficult to identify the exact cause.

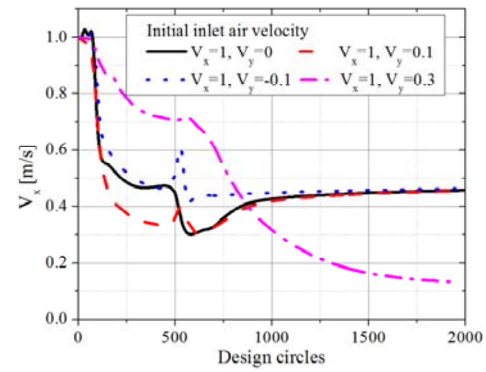


Figure 4 V_x change with the design circles

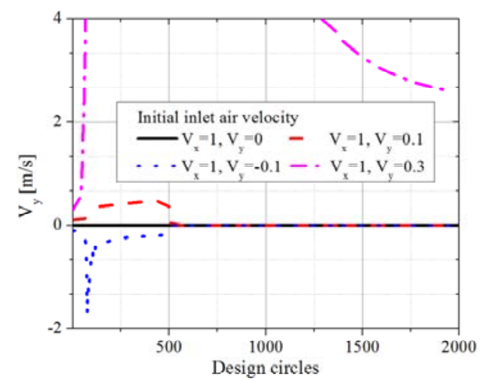


Figure 5 V_y change with the design circle

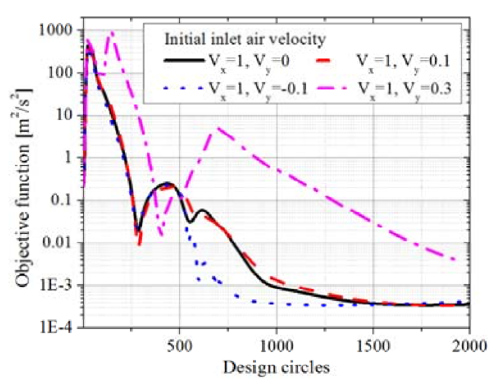
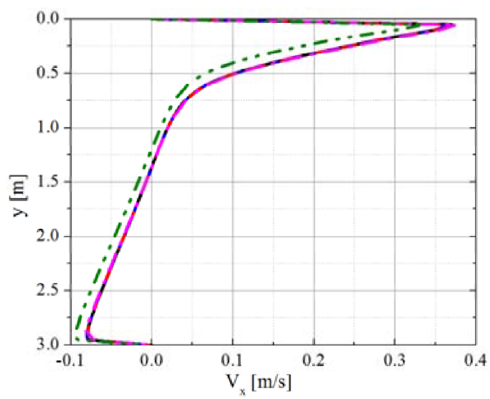
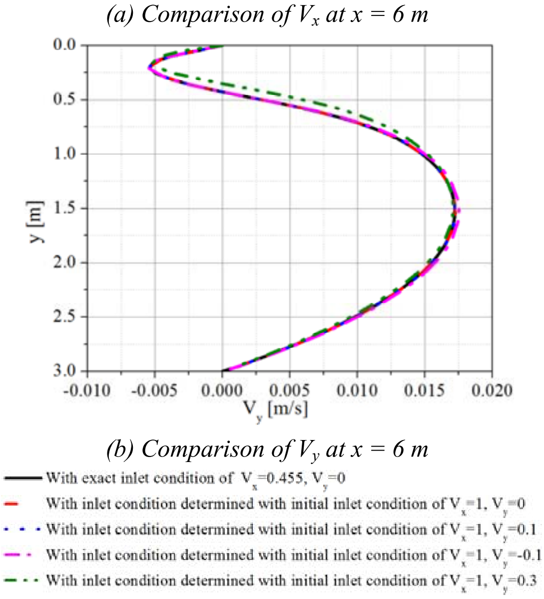


Figure 6 The objective function change with the design circle

This study further conducted forward CFD simulations with the inlet air velocities determined by the adjoint method. Figure 7 compares the computed velocity profiles at x = 6 m with that computed by the exact inlet air velocity of V_x=0.455 m/s and V_y=0 m/s. Only the calculation with the inlet velocity of V_x=0.132 m/s and V_y = 2.578 m/s had a minor difference. Therefore, this adjoint method could find the optimal inlet air velocity with a small value of objective function. In other words, this adjoint method could inversely find the desirable inlet air velocity by setting the velocity profile in the lower part of x = 6 m as the design objective. However, the results show that to achieve the design objective, the solution may not be unique.



(a) Comparison of V_x at x = 6 m



(b) Comparison of V_y at x = 6 m

Figure 7 Comparison of the design objective (velocity profiles) determined by adjoint method and that by the specified inlet condition of V_x = 0.455 m/s, V_y = 0 m/s at x = 6 m.

Non-isothermal case

This investigation also applied the adjoint method to a two-dimensional, non-isothermal case as shown in Figure 8. The dimension of the flow domain was 1.04 m × 1.04 m, the inlet height h = 18 mm, and outlet height t = 24 mm. The inlet air velocity was V_x = 0.57 m/s, V_y = 0.0 m/s and inlet air temperature was 15ºC. The temperature of the walls was 15ºC and that of the floor was 35ºC. Blay et al. (1992) conducted experimental measurements of the airflow and temperature distributions for the case.

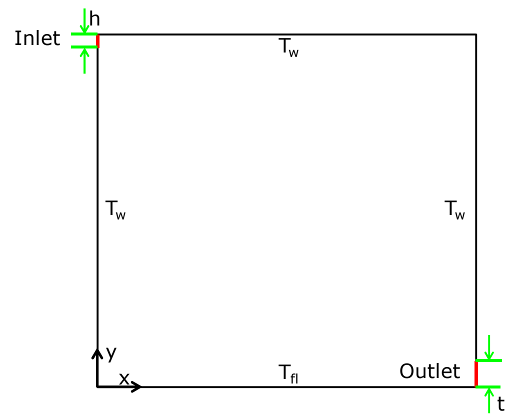


Figure 8 Sketch of the two-dimensional, non-isothermal case.

Again, this study conducted a CFD simulation with inlet air velocity V_x =0.57 m/s and V_y = 0.0 and inlet air temperature 15ºC to generate a design objective. The design objective selected for this case was the air velocity profile and/or air temperature profile at the mid-cavity as shown in the red line in Figure 9:

Scenario 1: Air velocity profile
Scenario 2: Air temperature profile
Scenario 3: Air velocity and temperature profiles

The design variables would be the inlet air temperature and velocity. For Scenario 3, α and β in Equation (4) were set to be 1.

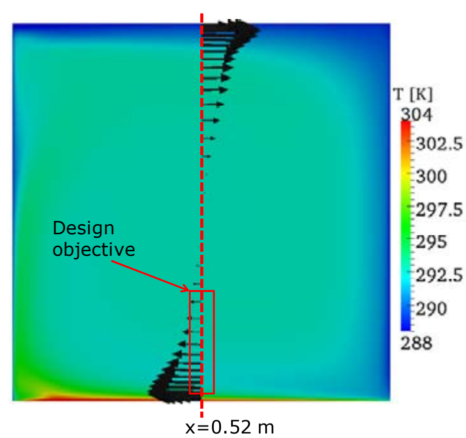
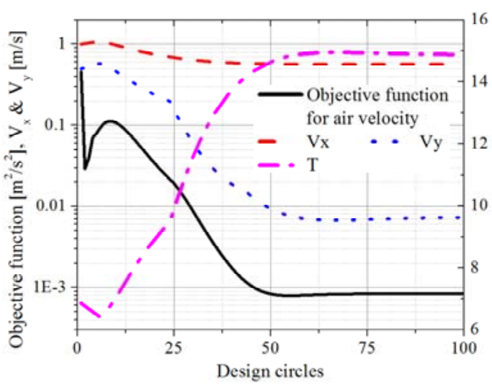


Figure 9 Design objective for the two-dimensional, non-isothermal case: velocity and/or temperature profiles in the red line.

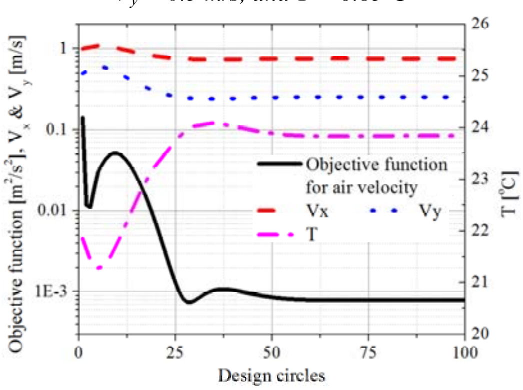
Since this is a non-isothermal case, the adjoint method solved also the energy equation. Without iterations (N_1=N_2=1), the solution would diverge due to very significant and sometimes unreasonable changes in the variables. So this study set $N_1=N_2>1$ (referred to the solving flow chart) in the calculations to achieve a stable and converged solution.

- Scenario 1: Using air velocity as objective

For this scenario, this study initialized the inlet velocity and temperature as V_x = 1 m/s, V_y = 0.5 m/s, and T = 6.85 ºC as Case S1a and V_x = 1 m/s, V_y = 0.5 m/s, and T = 21.85 ºC as Case S1b, respectively. Furthermore, this investigation used $N_1=N_2=100$ for each design circle and conducted 100 design circles. Figure 10 shows that the objective function was as small as 10^{-3} at the end of the calculation, indicating that an optimal design was reached. However, the two initial conditions led to two different optimal inlet conditions. Figure 10(a) shows a final inlet air velocity of V_x = 0.563 m/s and V_y = 0.007 m/s and a final inlet air temperature of T = 14.85ºC. The final condition was very close to the true inlet condition of V_x =0.57 m/s, V_y = 0 m/s, and T = 15ºC. However, Figure 10(b) shows the final inlet condition of V_x = 0.755 m/s, V_y = 0.254 m/s, and T = 21.85ºC that is quite different. Since the velocity field was used as the only design objective, multiple optimal solutions could exist.



(a) Case S1a with initial inlet condition of V_x = 1 m/s, V_y = 0.5 m/s, and T = 6.85ºC



(b) Case S1b with initial inlet conditions of V_x = 1 m/s, V_y = 0.5 m/s, and T = 21.85ºC

Figure 10 The changes of V_x, V_y, and T at the inlet and the objective function vs. design circles

Very similar to the isothermal case, this study also conducted forward CFD simulations with final inlet air velocity and temperature determined by the adjoint method. Figure 11 compares the computed velocity profiles at the mid-cavity (x = 0.52 m) with that using the exact inlet condition of V_x =0.57 m/s, V_y = 0 m/s, and T = 15 ºC. It is easy to notice that the two final inlet conditions had almost the same velocity profiles as the design objective.

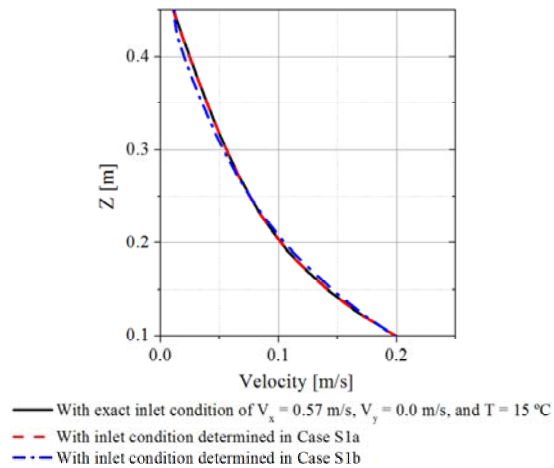


Figure 11 Comparison of the design objective (velocity profiles) determined by adjoint method and that by the specified inlet condition of V_x =0.57 m/s, V_y = 0 m/s, and T = 15ºC at x=0.52 m.

- Scenario 2: Using air temperature as objective

Since it is much harder to obtain a converged result after adding the energy equation into the adjoint method, the N_1 and N_2 were set to be 5000. Figure 12 shows that, with an initial inlet condition of V_x = 1 m/s, V_y = 0.1 m/s, and T = 6.85ºC, the objective function went down gradually to about 0.02 after 60 design cycles. Then the corresponding average temperature difference between the temperature design objective and the calculated temperature distribution in the lower part of the mid-cavity was about 0.01 K. However, the calculated inlet air velocity and temperature led to V_x = 0.816 m/s, V_y = 0.073 m/s, and T = 23.84 that is quite different from the true value of V_x =0.57 m/s, V_y = 0 m/s, and T = 15 ºC. This study also tried some other initial conditions and the calculation always approached to different optimal solutions from the true value, although the objective function could become as small as 10^{-2}. Since different inlet conditions could lead to similar temperature distribution in the lower part of the mid-cavity, multiple optimal solutions are not completely surprised to us.

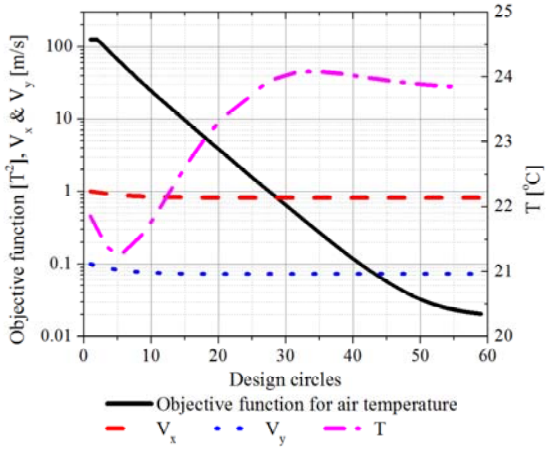
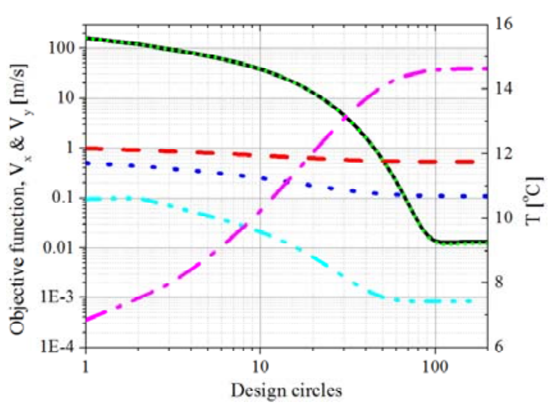


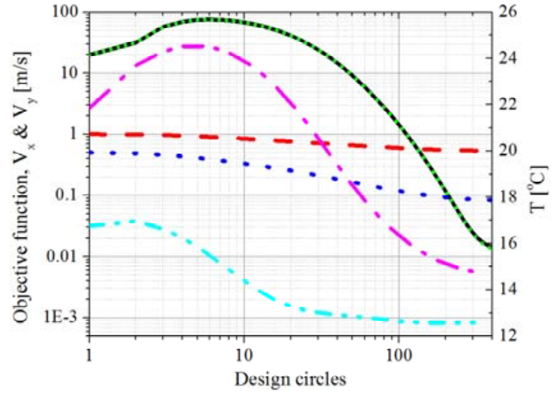
Figure 12 The changes of V_x, V_y, and T at the inlet and objective function vs. the design circle

- Scenario 3: Using air velocity and temperature as objective

The design objective in Scenario 3 is to satisfy the air velocity and temperature profile in the lower part of the mid-cavity. This study used initial inlet condition of V_x = 1 m/s, V_y = 0.5 m/s, and T = 6.85ºC as Case S3a and that of V_x = 1 m/s, V_y = 0.5 m/s, and T = 21.85ºC as Case S3b. The N_1 and N_2 were set to be 1000 that is a trade-off between the convergence of the calculations and the computing effort. For each condition, this study calculated no more than 400 design circles. Figure 13 shows that the final thermo-fluid boundary condition at the inlet for Case S3a was V_x = 0.528 m/s, V_y = 0.109 m/s, and T = 14.63 ºC and for Case S3b V_x = 0.537, V_y = 0.084 m/s, and T = 14.68ºC. The two final inlet conditions agree well with the true condition at the inlet as V_x =0.57 m/s, V_y = 0 m/s, and T = 15 ºC. It seems that the optimal solution was unique when both the air velocity and temperature profiles were specified as the design objective.



(a) Case S3a with initial inlet condition of V_x = 1 m/s, V_y = 0.5 m/s, and T = 6.85ºC



(b) Case S3b with initial inlet condition of V_x = 1 m/s, V_y = 0.5 m/s, and T = 21.85ºC

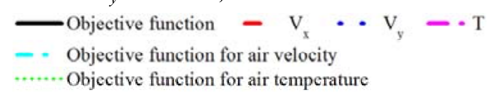


Figure 13 V_x, V_y, and T on the inlet and design objective change with design circles, partial flow and temperature fields as design objective.

Figure 13 also shows that the objective function for air velocity became 10^{-3} and that for air temperature 10^{-2}. The corresponding average velocity in the lower part of the mid-cavity differed only 0.003 m/s from the design objective and average air temperature differed only 0.01 K from the design objective.

DISCUSSION

In the isothermal case, within 1 iteration/cycle × 2000 cycles, the adoint method could find the optimal inlet air velocity. The calculation time is roughly twice of that needed by forward CFD simulation for 4000-iteration (no internal iteration) in solving the flow. However, in Scenario 3 of the non-isothermal case, the adjoint method could also find the optimal inlet conditions in 1,000 iterations/cycle × 400 design circles. The more complex the flow is, the more computing effort is required.

The more design objectives are, such as Scenario 3, the less possibility multiple solution would occur. It is possible that no solution could be obtained if too many design objectives are specified. That might be the limitation of this method in the application of optimal ventilation design.

This method could be further applied to design the ventilation by using thermal comfort index such as PMV, PPD, etc. as the design objective.

CONCLUSION

This investigation developed an adjoint method and implemented it in OpenFOAM to determine the optimal thermo-fluid boundary conditions for designing best indoor environment. By applying the method to solve the inlet conditions by using air velocity and/or air temperature in rooms as design objective, the inlet conditions could be inversely identified.

For the isothermal case, this study used a velocity profile in the lower part of the room computed by a forward CFD simulation as the design objective. The results showed that this adjoint method could accurately find the optimal inlet air velocity by using different initial inlet conditions. However, the adjoint method can lead to different optimal inlet conditions, which implies the existence of multiple solutions.

For the non-isothermal case, if only the air velocity profile or only the air temperature profile in the lower part of the mid-cavity was used as the design objective, the calculation can lead to multiple solutions. When both the air velocity and temperature profiles were used as design objective, this method found a unique solution of the inlet condition.

ACKNOWLEDGEMENT

The research presented in this paper was partially supported by the National Basic Research Program of China (The 973 Program) through grant No. 2012CB720100.

REFERENCES

ASHRAE. 2007. Ventilation for acceptable indoor air quality. Atlanta, GA.

Blay, D., Mergui, S., Niculae, C. 1992. Confined turbulent mixed convection in the presence of a horizontal buoyant wall jet. Fundamentals of Mixed Convection, ASME HTD, 213, 65-72.

Chen, Q. 2009. Ventilation performance prediction for buildings: A method overview and recent applications. Building and Environment, 44(4), 848-858.

Cuverlier, P. 1976. Optimal control of a system governed by the Navier-Stokes equations coupled with the heat equation, in New Developments in Differential Equations. W. Eckhaus, ed., North-Holland, Amsterdam, 81-98.

Dwight, R.P., Brezillon, J. 2006. Effect of various approximations of the discrete adjoint on gradient-based optimization. AIAA-2006-0690.

Gorelick, S.M., Evans, B.E., Remson, I. 1983. Identifying sources of groundwater pollution: an optimization approach. Water Resources. Res., 19(3), 779-790.

Gunzburger, M., Hou, L., Svobodny, T. 1989. Numerical approximation of an optimal control problem associated with the Navier-Stokes equations. Applied Mathematics, 2, 29-31.

Gunzburger, M., Hou, L., Svobodny, T. 1991. Analysis and finite element approximation of optimal control problems for the stationary Navier-Stokes equations with distributed and Neumann controls. Mathematics of Computation, 57, 123-151.

Gunzburger, M., Hou, L., Svobodny, T. 1991. Boundary velocity control of incompressible flow with an application to viscous drag reduction. SIAM Journal on Control and Optimization, 30, 167-181.

Gunzburger, M. 1999. Sensitivities, adjoints and flow optimization. International Journal for Numerical Methods in Fluids, 31, 53-78.

Jameson, A. 1988. Aerodynamic design via control theory. Journal of Scientific Computing. 3, 233-260.

Jameson, A. 1995. Optimum aerodynamic design using CFD and control theory. AIAA paper, 1995-1729.

Joslin, R., Gunzburger, M., Nicolaides, R., Erlebacher, G., Hussaini, M. An automated methodology for optimal flow control with an application to transition delay. AIAA Journal, 35, 816-824.

Launder, B., Sharma, B. 1974. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. Letters in Heat Mass Transfer, 1, 131-138.

Mahar, P.S., Datta, B. 2000. Identi cation of pollution sources in transient groundwater system. Water Resource Management, 14(6), 209-227.

Nielsen, P.V., Restivo, A., Whitelaw, J.H. 1978. The velocity characteristics of ventilated rooms. Journal of Fluids Engineering, 100, 291-298.

OpenFOAM: The Open Source CFD Toolbox. 2007. http://www.opencfd.co.uk/openfoam/index.html.

Othmer, C. 2008. A continuous adjoint formulation for the computation of topological and surface sensitivities of ducted flows. International Journal for Numerical Methods in Fluids, 58, 861-877.

Wagner, B.J. 1992. Simultaneously parameter estimation and contaminant source characterization for coupled groundwater flow and contaminant transport modelling. Journal of Hydrology, 135, 275-303.