Zhai, Z., Chen, Q., Haves, P. and Klems, J.H. 2002. "On approaches to couple energy simulation and computational fluid dynamics programs", *Building and Environment*, 37, 857-864.

On approaches to couple energy simulation and computational fluid dynamics programs

Zhiqiang Zhai[a], Qingyan Chen[a,], Philip Haves[b], Joseph H. Klems[b]*
[a]Building Technology Program, Massachusetts Institute of Technology, Cambridge, MA, USA
[b]Lawrence Berkeley National Laboratory, Berkeley, CA , USA

Abstract

Energy simulation (ES) and computational fluid dynamics (CFD) can play an important role in building design by providing complementary information of the building performance. However, separate applications of ES and CFD usually cannot give an accurate prediction of building thermal and flow behavior due to the assumptions used in the applications. An integration of ES and CFD can eliminate many of these assumptions, since the information provided by ES and CFD is complementary. This paper describes some efficient approaches to integrate ES and CFD, such as static and dynamic coupling strategies, in order to bridge the discontinuities of time-scale, spatial resolution and computing speed between ES and CFD programs. This investigation further demonstrates some of the strategies through two examples by using the EnergyPlus and MIT-CFD programs.

Keywords: Energy simulation; Computational fluid dynamics (CFD); Integration; Building design

1. Introduction

Energy simulation (ES) and computational fluid dynamics (CFD) programs provide complementary information about building performance. ES programs, such as EnergyPlus [1], provide energy analysis for a whole building and the heating, ventilating and air conditioning (HVAC) systems used. Space-averaged indoor environmental conditions, cooling/heating loads, coil loads, and energy consumption can be obtained on an hourly or sub-hourly basis for a period of time ranging from a design day to a reference year. CFD programs, on the other hand, make detailed predictions of thermal comfort and indoor air quality, such as the distributions of air velocity, temperature, relative humidity and contaminant concentrations. The distributions can be used further to determine thermal comfort and air quality indices such as the predicted mean vote (PMV), the percentage of people dissatisfied (PPD) due to discomfort, the percentage dissatisfied (PD) due to draft, ventilation effectiveness, and the mean age of air. With the information from both ES and CFD calculations, a designer can design an energy-efficient, thermally comfortable, and healthy building.

However, most ES programs assume that the air in an indoor space is well mixed. Those programs cannot accurately predict building energy consumption for buildings with non-uniform air temperature distributions in an indoor space, such as those with displacement ventilation systems. Moreover, the spatially averaged comfort information generated by the single node model of ES cannot satisfy advanced design requirements. The convective heat transfer coefficients used in ES programs are usually empirical and may not be accurate.

* Corresponding author. Room 5-418, 77 Massachusetts Avenue, Cambridge, MA 02139-4307, USA. Phone: (617) 253-7714, Fax: (617) 253-6152, Email: qchen@mit.edu

Furthermore, most ES programs cannot determine accurate airflow entering a building by natural ventilation, while room air temperature and heating/cooling load heavily depend on the airflow.

On the other hand, CFD can determine the temperature distribution and convective heat transfer coefficients. CFD can also accurately calculate natural ventilation rate driven by wind effect, stack effect, or both. However, CFD needs information from ES as inputs, such as heating/cooling load and wall surface temperatures.

Therefore, coupling ES with CFD is very attractive, and is the objective of the present investigation. After a brief introduction of the principles of ES and CFD, the paper describes possible approaches to couple ES and CFD. The current study emphasizes the explicit coupling of individual ES and CFD programs by exchanging information linking the two programs. Due to the different physical models and numerical methods employed by ES and CFD, this study suggests staged coupling strategies that consist of the static and dynamic coupling for different problems. The strategies effectively reduce the computing costs but preserve the accuracy and details of the computed results, due to the complementary information from ES and CFD. This paper finally uses an office and an indoor auto racing space to demonstrate the strategies.

2. Fundamentals of ES and CFD thermal coupling

2.1. The principles of ES

Energy balance equations for room air and surface heat transfer are two essential equations solved by many ES programs. The energy balance equation for room air is

$$\sum_{i=1}^{N} q_{i,c} A_i + Q_{other} - Q_{heat\_extraction} = \frac{\rho V_{room} C_p \Delta T}{\Delta t} \tag{1}$$

where $\sum_{i=1}^{N} q_{i,c} A_i$ is convective heat transfer from enclosure surfaces to room air, $q_{i,c}$ is convective flux from surface I, N is number of enclosure surfaces, A_i is area of surface i, Q_{other} is heat gains from lights, people, appliances, infiltration, etc., $Q_{heat\_extraction}$ is heat extraction rate of the room, $\frac{\rho V_{room} C_p \Delta T}{\Delta t}$ is the energy change in room air. The ρ is air density, V_{room} is room volume, C_p is specific heat of air, ΔT is temperature change of room air, and Δt is sampling time interval, normally one hour.

The heat extraction rate is the same as the cooling/heating load when the room air temperature is maintained as constant ($\Delta T = 0$). The convective heat flux from a wall is determined from the energy balance equation for the wall surface, as shown in Figure 1. A similar energy balance can be obtained for each window. The energy balance equation for a surface (wall/window) can be written as:

$$q_i + q_{ir} = \sum_{k=1}^{N} q_{ik} + q_{i,c} \tag{2}$$

where q_i is conductive heat flux on surface i, q_{ir} is radiative heat flux from internal heat sources and solar radiation, and q_{ik} is radiative heat flux from surface i to surface k.

The q_i can be determined by transfer functions, by weighting factors, or by solutions of the discretized heat conduction equation for the enclosure surface using the finite-difference method. The radiative heat flux is

$$q_{ik} = h_{ik,r}(T_i - T_k) \tag{3}$$

where $h_{ik,r}$ is linearized radiative heat transfer coefficient between surfaces i and k, T_i is temperature of interior surface i, and T_k is temperature of interior surface k. And

$$q_{i,c} = h_c\,(T_i - T_{room}) \tag{4}$$

where h_c is convective heat transfer coefficient and T_{room} is room air temperature.

The convective heat transfer coefficient, h_c, is unknown. Most energy programs estimate h_c by empirical equations or as a constant. If the room air temperature, T_{room}, is assumed to be uniform and known, the interior surface temperatures, T_i, can be determined by simultaneous solving Equations (2). Space cooling/heating load can then be determined from Equation (1). Thereafter, the coil load is determined from the heat extraction rate and the corresponding air handling processes and HVAC system selected. With a plant model and hour-by-hour calculation of the coil load, the energy consumption of the HVAC system for a building can be determined. It is obvious that the interior convective heat transfer from enclosures is the explicit linkage between room air and surface energy balance equations. Its accuracy will directly affect the energy calculated.

2.2. The principles of CFD

CFD applies numerical techniques to solve the Navier-Stokes (N-S) equations for fluid flow. CFD also solves the conservation equation of mass for the contaminant species and the conservative equation of energy for building thermal comfort and indoor air quality analysis. All the governing conservation equations can be written in the following general form:

$$\frac{\partial \Phi}{\partial t} + (V \bullet \nabla)\Phi - \Gamma_\phi\, \nabla^2\Phi = S_\phi \tag{5}$$

where Φ is V_j for the air velocity component in the j direction, 1 for mass continuity, T for temperature, C for different gas contaminants, t is time, V is velocity vector, Γ_ϕ is diffusion coefficient, and S_ϕ is source term. The Φ could also stand for turbulence parameters.

The C can stand for water vapor and various gaseous contaminants. For buoyancy-driven flows, the Buossinesq approximation, which ignores the effect of pressure changes on density, is usually employed. The buoyancy-driven force is treated as a source term in the momentum equations. Because most room airflows are turbulent, a turbulence model must be applied to make the flow solvable with present computer capacity and speed.

Since the governing equations are highly non-linear and self-coupled, it is impossible to obtain analytical solutions for room airflow. Therefore, CFD solves the equations by discretizing the equations with the finite volume method. The spatial continuum is divided into a finite number of discrete cells, and finite time-steps are used for dynamic problems. The discrete equations can be solved together with the corresponding boundary conditions. Iteration is necessary to achieve a converged solution [2].

The accuracy of CFD prediction is highly sensitive to the boundary conditions supplied (assumed) by the user. The boundary conditions for CFD simulation of indoor airflows relate to the inlet (supply), outlet (exhaust), enclosure surfaces, and internal objects. The temperature, velocity, and turbulence of the air entering from diffusers or windows determine the inlet conditions, while the interior surface convective heat transfers in terms of surface temperatures or heat fluxes are for the enclosures. These boundary conditions are crucial for the accuracy of the CFD results.

2.3. The coupling approaches

The previous two sections show that the convective heat transfer from interior surfaces of a space is equally important to both ES and CFD. On one hand, ES needs accurate convective heat transfer coefficient and room air temperature that can be calculated by CFD. On the other hand, CFD requires interior surface temperatures that can be determined by ES. Therefore, it is necessary to couple the two programs in order to improve their accuracy. This section focuses on how to treat the convective heat transfer in ES and CFD.

One may argue that a CFD program can be extended to solve heat transfer in solid materials, such as building walls, with an appropriate radiation model. This is the conjugate heat transfer method and many applications are available [3-6]. With energy model for the HVAC systems and plant, the CFD can include the function of ES. This method sounds powerful but it is very computationally expensive [4]. The reason for this is twofold. First, when the CFD calculates the heat transfer in solid materials, the calculation becomes stiffer and the computing time goes up dramatically [7]. Room air has a characteristic time of a few seconds while building envelope has a few hours. CFD simulation must be performed over a long period for the thermal performance of the building envelope, but it must use a small time step to account for the room air characteristics. Secondly, the computing time grows exponentially with building size. Hence, the conjugate heat transfer method is not practical for immediate use in a design context with current computer capabilities and speed.

Therefore, it is necessary to couple directly ES and CFD programs. This coupling involves the exchange the convective heat transfer information between the two programs. In principle, a fully iterated ES and CFD coupling program can provide a solution that is equivalent to the conjugate heat transfer method, provided that the ES program subdivides surfaces sufficiently small to model any significant temperature variations. In this coupling, the time step is considerably large in ES (a few minutes to an hour), the impact of the transient variation is small for CFD. CFD solution at a specific time step is actually quasi-steady, consistent with the given boundary conditions for that time step. Such a calculation, thus, has the advantage that it does not attempt to solve the flow field during the transition from one time step to the next, and therefore greatly saves computing time.

Some early work includes Chen and van der Kooi [8] who used the airflow pattern determined from CFD to calculate room air temperature and consider the impact of air temperature distribution on the cooling/heating loads. Srebric et al. [9] improved Chen and van der Kooi's study by directly coupling a CFD program with an ES program for design heating/cooling load calculation. The ESP-r program [10-13] has also used a CFD solver for whole-building simulation using three handshaking methods. These studies have indicated that the coupling can improve the solutions with acceptable computing efforts, and the convective heat transfer from enclosures is most important for the coupling.

The air temperature in the boundary layer of a surface and the convective heat transfer coefficient are two key factors determining the convective heat transfer. However, most ES programs assume a complete mixing in room air in solving the energy balance equation for room air. CFD can determine the air temperatures near the surfaces from the air temperature distribution, and the convective heat transfer coefficients as:

$$h_{i,c} = C_p \frac{\mu_{eff}}{Pr} \frac{1}{\Delta x} \tag{6}$$

where C_p is air specific heat, μ_{eff} is effective kinetic viscosity, Pr is Prandtl number, and Δx is normal distance from a point near a wall to the wall. A straightforward coupling method is to pass the air temperature, $T_{i,air}$, closed to a wall surface and the corresponding averaged convective heat transfer coefficient, $h_{i,c}$, to ES. The implementation is to Equation (4) as:

$$q_{i,c} = h_{i,c} (T_i - T_{i,air}) = h_{i,c}(T_i - T_{room}) - h_{i,c} \Delta T_{i,air} \tag{7}$$

where T_{room} is the desired air temperature of the room and $\Delta T_{i,air} = T_{i,air} - T_{room}$. ES use the updated $T_{i,air}$ and $h_{i,c}$ from each call of a CFD program and substitute them into Equation (7). Then, by solving the heat balance equations (1) and (2) together with Equation (7), the surface temperatures and heat extraction can be used to update the boundary conditions for the next CFD run.

In each of the CFD run, the use of the surface temperatures obtained from the ES is straightforward. The heat extraction rate from ES is used to determine the inlet boundary conditions in the CFD calculation. For a constant-air-volume HVAC system with a known air supply airflow rate V, the supply air temperature, T_{supply}, is

$$T_{supply} = Q_{heat\_extraction}/\rho C_p AV + T_{outlet} \tag{8}$$

where A is diffuser air supply area and T_{outlet} is return air temperature. For a variable-air-volume system, T_{supply} is constant, the V becomes

$$V = Q_{heat\_extraction}/ \rho C_p A \,(T_{supply}-T_{outlet}) \tag{9}$$

Since the heat flows and surface temperatures vary with time in buildings, it is necessary theoretically to run CFD for each time step. Even at each time step, iteration between ES and CFD may be needed to reach a convergence. The structure of the coupled simulation is illustrated in Figure 2.

3. Staged strategies for ES/CFD code coupling

Although coupling approach discussed above is straightforward, the coupling is not very practical due to the considerable disparities of the physical models and numerical schemes between ES and CFD programs. Three main discontinuities exist between ES and CFD programs. The first one is a time-scale discontinuity: ES has a characteristic time-scale of hours for heat transfer in building enclosure, but CFD has a few seconds for room air. The second one is a modeling discontinuity: the indoor environmental conditions predicted for each space in ES are spatially averaged, while CFD presents field distributions of the variables. The last one is a speed discontinuity: the computing time for ES is a few seconds per zone for an annual energy analysis and requires little computer memory, while a CFD calculation for a zone may take a few hours and require a large amount of memory [9].

To bridge these discontinuities between ES and CFD, this investigation develops special coupling strategies. For the time-scale discontinuity, the current study partitions the whole calculation into a long-time process for ES and a short-time scale (strictly speaking, a quasi-static process at a given time-step) process for CFD. As illustrated in Figure 3, ES handles a long-term simulation, such as a design day, while CFD runs only at some specific time steps, such as 8:00 am, with the boundary conditions provided by ES at that time step. ES then uses the updated information from CFD for the next two hours running till the next CFD call at 10:00 am. Space model discontinuity can also be bridged by appropriate numerical approximation. Although different numerical approximation algorithms may have different impacts on the coupling performance depending on the problems studied, sufficient subdivisions of enclosure surfaces in ES always can diminish this effect. However, the computational demands of CFD simulation make the coupling almost impractical. In addition to using more numerical approximations, such as simpler turbulence models, to reduce the computing time of CFD programs directly, it is necessary to develop simplified coupling strategies to minimize the number of CFD runs. The present study proposes *static coupling* and *dynamic coupling* as illustrated in Figure 4. The dynamic coupling process performs continuous (dynamic) information exchange between ES and CFD while the static coupling process has occasional (static) information exchange for a simulation.

The static coupling involves one-step or two-step exchange of information between ES and CFD programs, depending on the sensitivity of building thermal performance and user's accuracy requirement on solutions. With only a few coupling steps, the static coupling can be performed manually. Generally, the one-step static coupling is good in the cases where ES or CFD or both are not very sensitive to the exchanged variables. For example, ES is rather insensitive to $\Delta T_{i,air}$ and $h_{i,c,}$ in an air-conditioned room with low velocity mixing ventilation. To provide CFD inlet conditions and wall temperatures as inputs, one-step static coupling from ES to CFD is a good choice.

If the information from CFD, such as $h_{i,c}$, differs significantly from that used in the ES calculation, ES may use that from CFD as inputs for the next ES run. This is the ES-CFD-ES two-step static coupling. The coupling is good for buildings with little changes in the exchanged information, and the results of ES do not strongly depend on the exchanged data.

The dynamic coupling, which involves coupling between the two programs at every time step, is needed when both ES and CFD solutions are sensitive to the transient boundary conditions. This investigation proposes four kinds of dynamic coupling. The first one is *one-time-step dynamic coupling*, which focuses on the ES/CFD coupling at one specific time step interested. At that time step, the iteration between ES and CFD is performed to reach a converged solution. This coupling is for cases a designer is interested in only a few typical scenarios (design conditions) and both ES and CFD are very sensitive to the exchanged information.

Many building designs require the flow and energy information over a period of time, such as startup and shutdown periods. The ES/CFD coupling may be conducted at every time step over this period. When the time-step is small (for instance, a few minutes), it may not be necessary to couple the two programs at every time-step because the changes of the required information may not be significant. Further, the coupling requires no iteration between ES and CFD in order to reduce the computing time. This is *quasi-dynamic coupling*. Regular office building is a good example to have this coupling strategy applied.

If ES and CFD iterate for a couple of times at each time step to reach a converged solution, the coupling is *full dynamic coupling*. Full dynamic coupling is undoubtedly the most accurate, but also most intensive computationally. Fully dynamic coupling may be necessary for poorly insulated buildings with dynamical loads.

One way to reduce the computational costs is to use *virtual dynamic coupling,* as proposed by Chen and van der Kooi [8]. The room air temperatures and the convective heat transfer coefficients required by ES are generated by CFD as the functions of cooling/heating loads (for conditioned periods) or indoor-outdoor air temperature difference (for unconditioned periods). At each time step, ES determines $\Delta T_{i,air}$ and $h_{i,c}$ by interpolating the CFD results. Virtual dynamic coupling is suitable for buildings without dramatic changes of heat/cooling load and outdoor air temperature because the dramatic changes make the curve-fitted functions less accurate.

Note that the iteration of ES and CFD may result in convergence and stability problems due to the physical and numerical differences between ES and CFD programs. Different data-exchange methods in iteration may produce differences in convergence and stability behaviors. More analysis will be reported in the future.

In general, the building characteristics and the simulation purpose determine which coupling strategy is most suitable. One may use several coupling strategies to achieve the best solution for a specific case. For example, the virtual dynamic coupling may be best for a whole year energy analysis, while one-time-step dynamic coupling may be adequate for thermal comfort and indoor air quality analysis.

4. Case studies

The coupling strategies described above have been implemented by using the EnergyPlus and MIT-CFD programs. EnergyPlus, developed by the U.S. Department of Energy, is an energy simulation program based on DOE-2 [14] and BLAST [15]. The program uses the heat balance method. Developed at the Massachusetts Institute of Technology, MIT-CFD is a CFD program that solves steady and unsteady laminar and turbulent flow problems with arbitrary geometry. Standard numerical methods and turbulence models are employed in MIT-CFD. A prototype version of the coupled EnergyPlus/MIT-CFD codes has been used here to demonstrate different coupling strategies.

4.1. An office in Boston

This case uses an office to demonstrate a *quasi-dynamic coupling* for the winter design day in Boston. The office is assumed in a middle floor of a large building. It has only one south-facing exterior wall. The properties of the enclosure materials are listed in Table 1. There are no internal heat gains in the office so that the heating load is solely due to the heat loss through the south exterior wall. The room is conditioned 24 hours a day with a VAV system. The exhaust is located above the air-supply diffuser on the west wall. The supply air temperature is 30ºC, and the room air temperature is controlled at 16ºC.

In this case, the CFD calculation is called every hour by ES whose time-step length is 15 minutes for a period of four design days. With the quasi-dynamic coupling strategy, the ES part first produces a set of surface temperatures and a heating load at the first hour and passes them to the CFD part. Based on these boundary conditions, the CFD part calculates the flow and temperature distributions for the time step. Then the ES part uses the $\Delta T_{i,air}$ and $h_{i,c}$ from the CFD results for the next hour running, and so on.

The CFD uses a zero-equation turbulence model [16]. The convergence criterion for the CFD is that the normalized residuals are less than 1% for all the variables solved. The total computing time for the coupled ES and CFD simulation is only 83 seconds on a Pentium III 600 MHz PC because the CFD solution uses an extremely coarse grid (10x5x6).

The results show that the heating load variation during the design day is not significant because of the weak winter solar effect in Boston and the good insulation of the south wall. In the room, as seen in Figure 5, the low-velocity warm air from the diffuser goes up directly due to the strong buoyancy effect. It travels back along the center-line and forms a warm re-circulation region in the top part of the space. The temperature difference between the top and bottom air of the room is about 3-4 K. However, the average air temperature close to the south wall is almost as same as the controlled room air temperature (a small $\Delta T_{r,i}$ in Table 2). The convective heat transfer coefficient on the interior south wall calculated by MIT-CFD is almost twice as large as the one originally determined in EnergyPlus. With this improved convective heat transfer coefficient, EnergyPlus predicts a larger heat flow from the room air to the surface, which also increases the surface temperature, as shown in Table 2.

Figure 6 presents the conductive, convective, and radiative heat transfer of the south wall. The south wall gains heat from room air and other surfaces by convection and radiation, and transfers the heat to the outside through the wall by conduction. The increased convective heat transfer in the coupled ES/CFD calculation increases the total heating load requirement by about 10%. The heating load increase would be even greater if there are windows on the south wall [17].

4.2. An indoor auto racing complex in Pittsburgh

The second case uses an indoor auto racing complex to demonstrate a *two-step static coupling*. The auto racing complex is a single space building with a 250,000 m^2 floor area and a 46 m ceiling height. It has seats for up to 120,000 spectators and can house 45 racing cars running simultaneously on the track at an average speed of 250 km/h, as shown in Figure 7. Zhai et al [18] used CFD to assist the ventilation system design for this building under both summer and winter design conditions. The predictions of the airflow, temperature, and contaminant concentrations distributions help to evaluate thermal comfort and indoor air quality in the complex and to improve the ventilation system designs. The CFD simulations need wall temperatures as its boundary conditions, which can be obtained from an ES program. The case is not completely mixed and has a very unusual heat transfer coefficients on the wall surfaces due to the strong forced convection caused by the cars. The energy simulation needs the heat transfer coefficients and temperature gradients computed by CFD. Because one CFD run under steady-state conditions may take about 10 hours to obtain a reasonable result for this case with a grid resolution of 100×100×55, it is impractical to perform any dynamical coupling process. The current study, therefore, employs the two-step static coupling.

The investigation focuses on a typical summer design day in Pittsburgh with a three-hour racing event from 9:00 to 12:00 AM. The event is with maximum spectators, lights and racing cars. The building has R-4 walls and a R-13 roof. The heat gains come from the outdoor air, solar radiation, spectators, lights, and cars. About 1400 m^3/s fresh air is supplied by the overhead duct system, the underneath displacement ventilation system, and a partial air curtain system to maintain an acceptable indoor air quality and thermal comfort during the event. In the ES-CFD-ES two-step static coupling, ES first calculates the surface temperatures and cooling loads using the default convective heat transfer coefficients. Using these surface temperatures and cooling loads as boundary conditions, CFD then calculates the heat and airflow distribution in the space. The $\Delta T_{i,air}$ and $h_{i,c}$ from the CFD results are fed back to the ES to obtain more accurate cooling loads for sizing the ventilation systems.

Figure 8 shows a strong air momentum caused by the cars on the track in the space. The strong momentum brings the heat and contaminants from the racing zone to the occupied zone, while the ventilation systems attempt to reduce this adverse effect. The required peak cooling energy is 30.67 MW by the coupled ES and CFD simulation, as shown in Figure 9. With only ES, the required cooling energy is 27.05 MW. The difference of 3,620 KW is considerable. The main reason for this is due to the dramatic increase of the convective heat transfer coefficients. Table 3 compares the convective heat transfer coefficients computed by the CFD with the original ones used by ES. The coefficients from ES are undoubtedly too small for such a strong forced convection case, while those from CFD seem more reasonable. The convective heat transfer coefficient on the west wall is about the same as that from ES due to the low air velocity there. Figure 9 also shows that the surface temperature changes after the coupled calculation. Based on the new surface temperatures and cooling load determined from ES, another CFD calculation could be performed to update those $\Delta T_{i,air}$ and

$h_{i,c}$. For this case, it is estimated that those changes in boundary conditions may not be significant. The two-step static coupling simulation, therefore, is sufficient for the design purpose.

5. Conclusions

This paper outlines several strategies to couple an energy simulation (ES) program with a computational fluid dynamics (CFD) program. With the coupling, most assumptions used in the two programs for thermal and flow boundary conditions can be eliminated due to the complementary information provided by the two programs. However, there is a gap in computing speed between ES and CFD programs. This paper presents two staged coupling strategies, static and dynamic coupling strategies, to bridge the gap to reduce the computing costs while achieving accurate results. The use of the coupling strategies depends on building characteristics and results needed.

Using a coupled ES and CFD program (a combination of EnergyPlus and MIT-CFD programs), this paper demonstrates the coupling strategies for an office under winter design conditions in Boston and an indoor auto racing complex under summer design conditions in Pittsburgh. The office case uses the quasi-dynamic coupling strategy and the auto-racing complex a two-step static coupling strategy. The results show that the coupling can improve at least 10% the cooling/heating load prediction, due to the improvement in obtaining convective heat transfer coefficients. Moreover, the coupling can provide accurate enclosure surface temperature that is an important comfort parameter.

References

[1] Crawley D.B., Lawrie L.K., Pedersen C.O., Winkelmann, F.C. EnergyPlus: energy simulation program. ASHRAE Journal 2000; 42(4): 49-56.

[2] Patankar S.V. Numerical heat transfer and fluid flow. New York: Hemisphere/ McGraw-Hill, 1980.

[3] Holmes M.J., Lam J.K.-W., Ruddick K.G., Whittle G.E. Computation of conduction, convection, and radiation in the perimeter zone of an office space. Proc. ROOMVENT ’90, Oslo, Norway, 1990.

[4] Chen Q., Peng X., and van Paassen A.H.C. Prediction of room thermal response by CFD technique with conjugate heat transfer and radiation models. ASHRAE Transactions, 1995; 3884: 50-60.

[5] Moser A., Schalin A., Off F., Yuan X. Numerical modeling of heat transfer by radiation and convection in an atrium with thermal inertia. ASHRAE Transactions, 1995; SD-95-14-4.

[6] Schild P. Accurate prediction of indoor climate in glazed enclosures. PhD Thesis, Norwegian University of Science and Technology, Trondheim, Norway, 1997.

[7] Thompson C.P., Leaf G.K. Application of a multigrid method to a buoyancy-induced flow problem. Multigrid Methods--Theory, Applications and Supercomputing. S.F. McCormick. New York: arcel Dekker Inc., 1988.

[8] Chen Q., van der Kooi J. ACCURACY — A computer program for combined problems of energy analysis, indoor airflow, and air quality. ASHRAE Transactions, 1988; 94(2): 196-214.

[9] Srebric J., Chen Q., Glicksman L.G. A coupled airflow-and-energy simulation program for indoor thermal environmental studies. ASHRAE Transactions, 2000; 106(1): 465-476.

[10] Negrao C.O.R. Conflation of computational fluid dynamics and building thermal simulation. PhD Thesis, University of Strathclyde, Glasgow, UK, 1995.
[11] Clarke J.A., Dempster W.M., Negrao C.O.R. The implementation of a computational fluid dynamics algorithm within the ESP-r system. Proc. Building Simulation '95, Madison, USA, Int. Building Performance Simulation Association, 1995, 166-75.
[12] Clarke J.A., Hensen J.L.M., Negrao C.O.R. Predicting indoor air flow by combining network approach, CFD and thermal simulation. Proc. 16th AIVC Conference, 1995, 145-53.
[13] Beausoleil-Morrison I. The adaptive coupling of heat and air flow modeling within dynamic whole-building simulation. PhD Thesis, University of Strathclyde, Glasgow, UK, 2000.
[14] Birdsall B.E., etc. The DOE-2 computer program for thermal simulation of buildings. Energy Sources: Conservation and Renewables, American Institute of Physics, Ed. David Hafmeister, 1985.
[15] Hittle D.C. Building loads analysis and system thermodynamics (BLAST) users manual (Version 2.0). Technical Report E-153, Vol. 1 and 2, U.S. Army Construction Engineering Research Laboratory (USA-CERL), Champaign, IL, 1979.
[16] Chen Q., Xu W. A zero-equation turbulence model for indoor airflow simulation. Energy and Buildings, 1998; 28: 137-144.
[17] Kendrick J.F. An overview of combined modeling of heat transport and air movement. Technical Note 40, Air Infiltration and Ventilation Center, Coventry, UK, 1993.
[18] Zhai Z., Chen Q., Scanlon P.W. Design of ventilation system for an indoor auto racing complex. ASHRAE Transactions, 2002; 108(1).

Fig. 1. Energy balance on the interior surface of a wall, ceiling, floor, roof or slab
Fig. 2. Structure of coupling simulation
Fig. 3. Illustration of time coupling (ES handles a long-term simulation, such as a design day, while CFD runs only at some specific times, such as 8:00 am)
Fig. 4. Illustration of the staged coupling strategies (The arrow from CFD to ES indicates the transfer of $\Delta T_{i,air}$ and $h_{i,c}$ while the arrow from ES to CFD indicates the transfer of T_i and $Q_{heat\_extraction}$)
Fig. 5. Configuration of the office and flow pattern
Fig. 6. Heat transfer on the south wall of the office
Fig. 7. The CFD model of the auto racing complex
Fig. 8. A strong air momentum caused by the cars at 5 m above the track in the auto racing complex
Fig. 9. Comparison of the surface temperatures and cooling load computed with and without CFD results

Table 1 The properties of the building materials used for the office
Table 2 Comparison of the day-averaged convective heat transfer coefficients, temperature difference between the room air and wall surface, and the wall temperature for the south wall with and without CFD for the office
Table 3 The convective heat transfer coefficients and temperature differences between the room air and wall surfaces for the auto racing complex with and without CFD

Table 1
The properties of the building materials used for the office

Enclosure	Thickness (m)	Density (kg/m^3)	Specific heat (J/kgK)	Thermal cond (W/mK)
Ceiling/Floor	0.175	2300	840	1.9
Walls	0.140	700	840	0.23

Table 2
Comparison of the day-averaged convective heat transfer coefficients, temperature difference between the room air and wall surface, and the wall temperature for the south wall with and without CFD for the office

South Wall	$h_{i,conv}$ (W/m^2K)	$\Delta T_{i,r}$ (°C)	T_{wall} (°C)	Q (W)
Without CFD	2.41	0	9.62	583
With CFD	4.37	-0.1077	11.65	638

Table 3
The convective heat transfer coefficients and temperature differences between the room air and wall surfaces for the auto racing complex with and without CFD

	With CFD		Without CFD	
Enclosure surfaces	$\Delta T_{i,r}=T_{i,a}-T_{room}$ (°C)	h_c (W/m^2K)	$\Delta T_{i,r}=T_{i,a}-T_{room}$ (°C)	h_c (W/m^2K)
South	-0.72	77.19	0	2.55
East	0.36	91.53	0	2.51
North	0.62	52.94	0	2.33
West	-2.63	2.81	0	2.32
Ground	-0.95	111.41	0	1.42
Roof	0.18	12.80	0	1.45

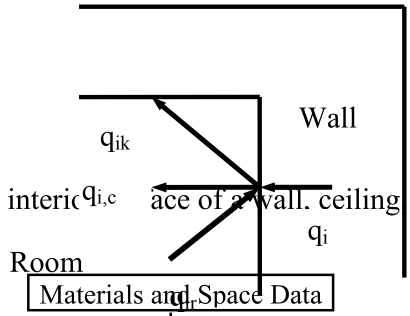


Fig. 1. Energy balance on the interior surface of a wall, ceiling, floor, roof or slab

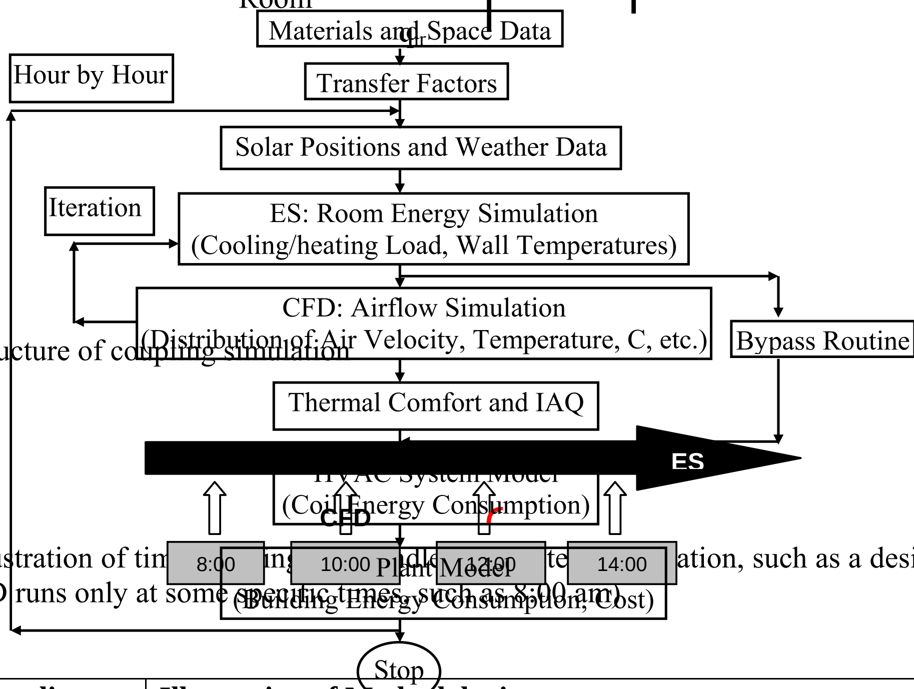
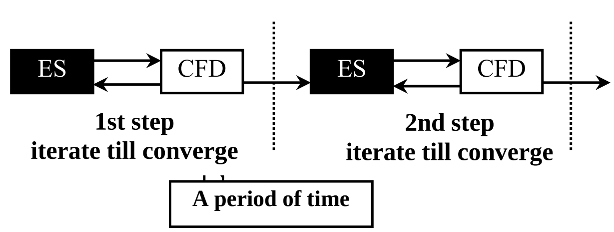


Fig. 2. Structure of coupling simulation

Fig. 3. Illustration of tim[illegible]ation, such as a design day, while CFD runs only at some specific times, such as 8:00 am)

Staged Coupling	**Illustration of Methodologies**	
Static Coupling	One Step ES → CFD CFD → ES	Two Step ES → CFD → ES CFD → ES → CFD
Dynamic Coupling	One-Time-Step Dynamic Coupling ES ⇄ CFD **Iterate till converge at one specific time step**	
	Quasi-Dynamic Coupling ES → CFD → ES → CFD → **1st step** **2nd step** **A period of time**	



	Full Dynamic Coupling
	Virtual Dynamic Coupling

Fig. 4. Illustration of the staged coupling strategies (The arrow from CFD to ES indicates the transfer of $\Delta T_{i,air}$ and $h_{i,c}$ while the arrow from ES to CFD indicates the transfer of T_i and $Q_{heat\_extraction}$)

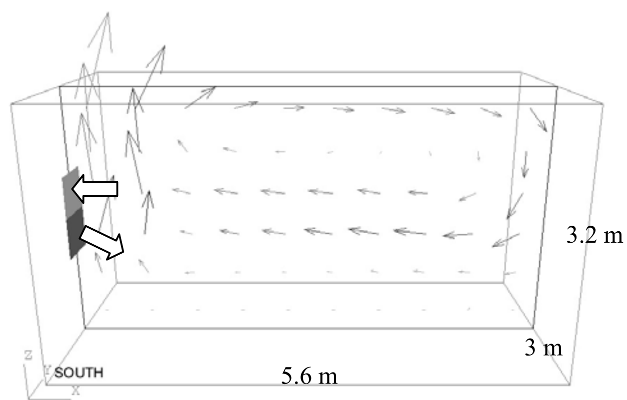


Fig. 5. Configuration of the office and flow pattern

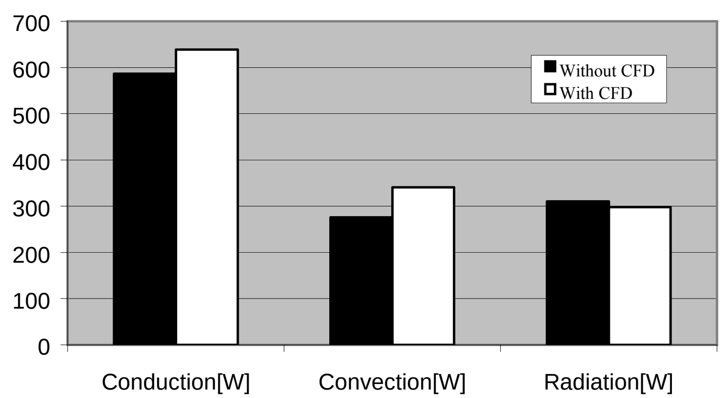


Fig. 6. Heat transfer on the south wall of the office

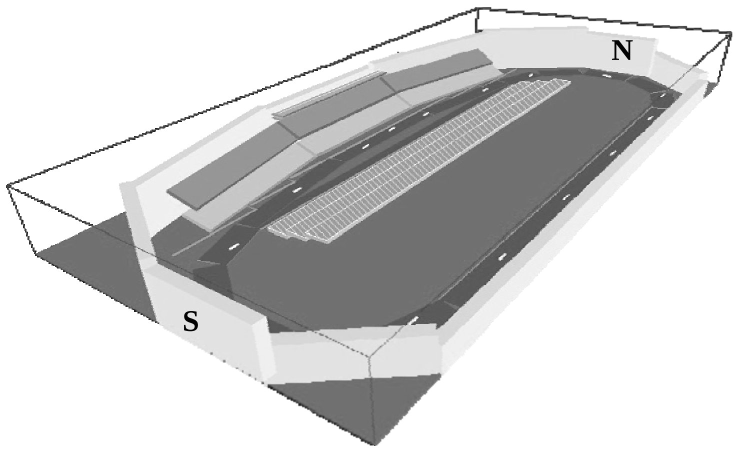
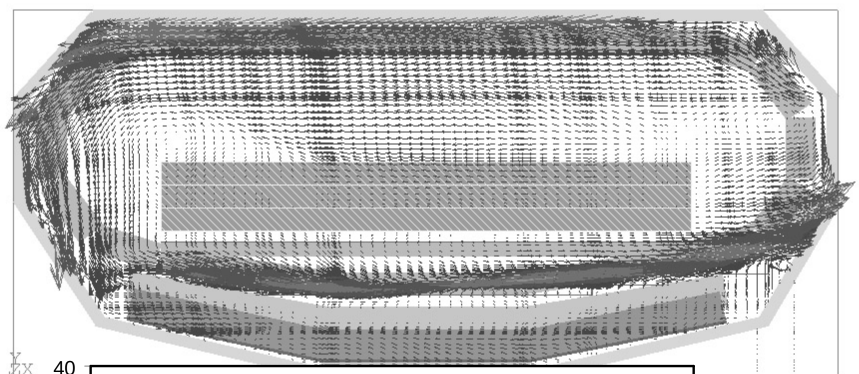


Fig. 7. The CFD model of the auto racing complex



cing

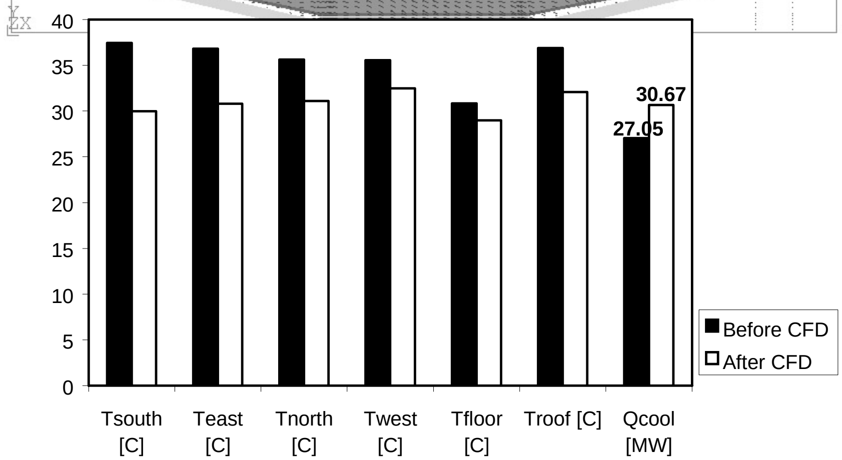


Fig. 9. Comparison of the surface temperatures and cooling load computed with and without CFD results