Natural and Forced Conjugate Heat Transfer in Complex Geometries on Cartesian Adapted Grids

The Cartesian incompressible RANS solver with immersed boundaries, IRRANS, recently developed at Stanford, has been extended to include conjugate heat transfer modeling and used for the simulation of the electrical motor of an automotive engine cooling fan system. Such applications are particularly challenging, as they involve very complex geometries with tight tolerances and rotating parts. The new conjugate heat transfer capability of IRRANS has been verified on natural and forced convection flows. The former involves flows in enclosures around a sphere and electronic chips. The latter focuses on heated cylinders for Reynolds numbers covering flow regimes ranging for a steady laminar flow to unsteady turbulent flows. Excellent agreement is achieved with similar simulations with a conventional body-fitted solver (FLUENT 6.1) using equivalent turbulent models. First three-dimensional simulations of the flow and heat transfer within the complete electrical motor are presented. The numerical predictions of the pressure drop through the motor as a function of flow rate agree very well with the measured data over the complete operating range. [DOI: 10.1115/1.2201625]

1 Introduction

In the design process of new automotive engine cooling fan systems, one increasingly important problem is the interaction of the fan system with a tighter and hotter underhood environment. This creates aerodynamic and acoustic constraints on the fan system but also large thermal loads on the electrical motor driving the fan system. The aerothermal management within the fan system requires detailed conjugate heat transfer (CHT) analysis within the electrical motor.

The conventional computational fluid dynamic (CFD) approach based on body-fitted grids provides such capabilities but several outstanding issues emerge. First, the computer aided design (CAD) to CFD transfer of such large geometrical models requires user intervention even when direct interfaces between the CAD and the grid generation software are available. Surfaces representation has very different meanings in CAD and CFD environments. In the former, they are the end product and they serve as basis for manufacturing. The surfaces are typically converted to a set of points to drive a Computer Aided Manufacturing (CAM) machine or to a set of triangles for rapid-prototyping manufacturing. The distribution of the output points (or the size of the triangles) is typically based on surface curvature and it is directly controlled by the user in the CAD system. In a CFD environment, the surface representation is only a starting point and it is used as a support for the surface mesh generation procedure. Once this surface mesh is obtained, the volume mesh is built to allow for a CFD solution. There are several constraints on the surface mesh that make the direct use of CAD surfaces nearly impossible; the most important is the quality of the mesh elements. Usually, the CAD model is broken into several smaller components and surface meshes are generated in patches enforcing quality constraints. The volume meshes are also typically built in subdomains. Therefore, the grid generation for a complex geometry, such as an electric motor, is extremely time consuming and requires very skilled users. As quoted by Péniguel in [1]: “This step is representing sometimes more than 80% of the time devoted to a study.” Moreover, in a typical design cycle several parametric studies must be carried out to evaluate the effect of various design solutions. This can hardly be done within typical time constraints due to the grid reconstruction step required for each new simulation. Current industrial practice is to use simplified model problems and to extrapolate the results to build several initial prototypes that are consequently tested. However, such procedures can hardly yield an optimized system and can be still very costly.

In the present study, an innovative approach based on the immersed boundary technique is proposed to overcome most of the above simulation obstacles. The method is based on the use of Cartesian grids nonconforming with the physical boundaries. Body forces are then added at the interface to enforce the boundary constraints. This recently developed approach will be described after stating the present industrial context and simulation background in the next section. Particular emphasis will be put on the grid generation and the extension of the flow solver to conjugate heat transfer problems. Verification examples focused on natural and forced convection at various flow regimes are then shown. The extension to the complete electrical motor is then tackled and the first milestone results are shown.

2 Design Background

The increasing need of flat fan systems in automotive engine cooling modules often leads to the use of compact electric engine cooling (EEC) motors as shown in Fig. 1. These compact motors should have the same efficiency as earlier designs with reduced volume and then increased heat dissipation density. Using high magnetic characteristic materials and high thermal insulation class materials would achieve this goal. Unfortunately, EEC motors are very cost sensitive products, and compromise need to be found between cost and performance. Improving heat transfer inside an EEC motor is an important way to meet these challenges, which should lead to reduced temperatures of critical parts of the motor. Reliability is then increased and high power density in the electrical motor is no longer compromised by motor life consideration. In [2], Hong et al. focused on the interaction between motor
temperature and airflow inside the EEC motor. Theoretically, the thermal analysis of these electric motors includes:

- heat source analysis
- airflow and conjugate heat transfer (CHT) analysis

Both numerical analysis and experimental validation were carried out in [2] to establish a robust and practical methodology for thermal analysis of EEC motors. The turnaround of the three-dimensional simulations involved in the second step (the CHT analysis) limited the possibility of varying the geometrical parameters of these EEC motors and did not allow mapping its complete flow performances quickly. To achieve such simulations several hurdles had to be overcome. First, to simulate the inside of an electrical motor, one has to deal with an extreme geometrical complexity similar to the underhood environment of a vehicle. An electrical motor consists of about a hundred parts with thirty moving components and a wide range of clearances between these parts. The generation of the grid (Fig. 6 in Ref. [2]) required further simplification of the original CAD models. For instance, the crimping teeth, the screws and nuts were removed. In addition, the conjugate heat transfer analysis requires the meshing of both the fluid and solid regions. Overall, the complete CAD preparation and mesh generation required several months. The resulting numerical model had several millions nodes, which required several CPU days on a SGI Octane workstation. This makes a quick design or a quick adaptation to a particular thermal issue currently impossible.

3 Immersed Boundary Method

The present computational approach, namely IBRANS, is based on a simple, widely available description for geometrical components, the stereo-lithography (STL) format. In this context any three-dimensional surface is represented by a collection of triangles. These models are the same as those used for rapid prototyping and no constraints are imposed on the surface triangulation (other than being a sufficiently accurate representation of the geometry). In particular, highly irregular and skewed triangles can be successfully handled. The relation between the underlying Cartesian grid and the STL surface is constructed using a ray-tracing algorithm; this allows to tag the computational cells that are cut by the immersed surface interface and identify the fluid and solid cells as volumes that are completely outside and inside the STL surface, respectively. Our immersed boundary (IB) method uses interpolants to reconstruct the behavior of the solution in the neighborhood of the embedded surfaces so that the boundary conditions are enforced at the physical location, as opposed to the closest Cartesian location as in the stairstep approach used, for instance, in the commercial Cartesian RANS solver UH3D [3-4]. Note that the current approach is also different from some of the earlier versions of the IB methods where the governing equations are modified by adding forcing terms designed to enforce the boundary conditions. A detailed discussion of the differences between several IB approaches can be found in Refs. [5-7]; here we only describe the most important aspects focusing on the application of the technique for heat transfer calculation (see next section).

The approach is based on the solution of the RANS equations for an incompressible fluid using a simple approach [8,9]. The fluid cells are treated without any modifications while in the solid and the interface cells a special treatment is introduced. In the solid cells the velocity is set to zero, whereas in the interface cells a reconstruction is used to bridge the solution in the fluid cells and the desired no-slip boundary condition on the STL surface. The interpolant used is in the form

$$\phi = a_1 n^2 + a_2 n + a_3 t_1 + a_4 t_1 t_2 + a_5 t_1 + a_6 t_2 + a_7$$  \hspace{1cm} (1)

where $\phi$ represents any independent variable (i.e., the velocity components), $n$, $t_1$, and $t_2$ are the normal and tangential directions to the surface and the $a_i$ are the unknown interpolation coefficients. These coefficients are determined by imposing the boundary condition and the values of $\phi$ in the fluid cells surrounding the interface. Equation (1) can be rewritten in a form that explicitly shows the fluid cells contributions

$$\phi_p = \sum_{nb} b_{nb} \phi_{nb} + b_{bc} \phi_{bc}$$  \hspace{1cm} (2)

The reconstruction Eq. (2) allows for an implicit treatment of the interface value $\phi_p$ thus it does not introduce any stability constraint.

It is worth mentioning that Eq. (1) requires the determination of eight coefficients, therefore seven fluid values and one boundary condition. In general, only nearest neighbors of the interface cell—cells that share at least one node—are used. If the specific conformation of the STL surface does not allow the use of enough neighbors, the interpolant is simplified by eliminating the quadratic terms.

The current approach uses the Boussinesq approximation to close the RANS equations. The eddy viscosity is computed using simple one- or two-equation turbulence models; the models are described in detail in [10]. The one-equation model is the Spalart-Allmaras model [11], whereas the two-equation model is a transformation of the $k$-$\omega$ model developed by Wilcox [12]; in the present approach, the variable $g$ defined as $g = 1/\sqrt{\beta_0}$ is used instead of $\omega$. The complete set of model damping functions, closure coefficients and auxiliary relationships for both turbulence models can be found in Ref. [10]. Both models are characterized by simple homogeneous wall boundary conditions for all the turbulent scalars and therefore the same reconstruction operators used for the velocity components are used.

A note of caution is related to the treatment of the pressure. In the current approach, the pressure equation is solved everywhere in the domain, including inside solid regions. This is done to simplify the solution of the elliptic equations without requiring a reconstruction of the pressure field. The pressure field is clearly
used only in the fluid cells. Note that for no-slip boundary conditions this treatment is completely equivalent to imposing a Neumann condition on the pressure.

To increase the grid resolution and, therefore, the accuracy, the Cartesian cells are successively refined locally thus introducing hanging nodes. The procedure is anisotropic such that each cell can be subdivided in two, four or eight subcells; the grid refinement approach is described in detail in [13]. In summary, once the interface cells are identified, their size is compared to the desired normal and tangential resolution at the STL surface. If necessary, the cells are split in one or more directions and the tagging procedure applied again. An additional refinement procedure is applied to avoid jumps in cell size larger than one-to-eight; this smoothing also limits the number of neighbors for each cell at 24 (four per cell side). The advantages of the anisotropic grid refinements can be clearly seen in Fig. 2 where the necessary number of cells to achieve a given resolution around an elliptic profile is plotted. It is clear that the more alignment between the STL surfaces and the Cartesian direction, the more the anisotropic grid refinement is effective. The latter capability has first been validated on an actual turbine blade with its internal cooling passage [14]. The application of the present procedure to the complex geometry of an electric motor is presented in Fig. 3 with two cutting planes through the mesh. The left plot cuts through the motor stack at midspan of the motor case; the right plot goes through the motor front plate behind the drive plate.

4 Conjugate Heat Transfer

In a previous work [15], Iaccarino et al. showed that the heat transfer in heated ribbed passages is significantly modified when the heat conduction through the passage walls is accounted for. This motivates the need for solid/fluid CHT analysis.

As mentioned before, in the present IBRANS solver, a Cartesian grid nonconforming to the boundaries is used. The availability of a grid throughout the computational domain, including the inside of the solid embedded region, allows for a CHT calculation to be carried out naturally. The energy equation is solved in the entire domain consisting of the fluid plus the solid, in the form

\[
\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x_i} = \frac{1}{\kappa} \left( k_\text{eff} \frac{\partial T}{\partial x_j} \right) + S_f
\]

where \( k \) is the thermal conductivity, \( k_\text{eff} \) the turbulent diffusivity modeled here using a simple gradient diffusion hypothesis, \( k_\text{eff} \)}
\[ S_b = \alpha (\rho - \rho_{ref}) g_i \]

where \( g_i \) represents the gravity vector, \( \rho \) and \( \rho_{ref} \) the local density and a reference density, and, \( \alpha \) is the expansion ratio of the fluid considered.

Appropriate material properties are considered for each solid and the fluid in the system, and (eventually) source terms are added locally in the solid regions. The energy equation is solved, similarly to the turbulence equations, segregated from the mean flow. The only other aspect to consider is the treatment of the interface cells, which correspond to the solid/fluid interfaces. From an IB point of view the reconstruction procedure (Eq. (1)) must be applied on both sides of the interfaces. In physical terms, the interface temperature is not known but the continuity of both temperature, \( T_w \), and heat flux, \( \Phi_{ws} \), must be enforced

\[
\begin{align*}
T_{ws} &= T_{wf}, \\
\Phi_{ws} &= \kappa \left( \frac{\partial T}{\partial n} \right)_{ws} = \kappa \left( \frac{\partial T}{\partial n} \right)_{wf} = \Phi_{wf}
\end{align*}
\]

where the subscripts \( s \) and \( f \) refer to the solid and fluid side of the interface. In the present approach, a linear interpolation procedure is used on both sides of the interface; this together with the conditions (5) allows to obtain the temperature in the interface cells.

\[
\begin{align*}
T_f &= a_1 T_b + a_2 T_s + a_3 T_2 + a_4, \\
T_c &= b_1 T_f + b_2 T_s + b_3 T_2 + b_4, \\
\frac{\partial T}{\partial n_f} &= a_1, \\
\frac{\partial T}{\partial n_s} &= b_1.
\end{align*}
\]

The present reconstruction requires three cells on each side of the interface.

5 Verification of the IBRANS Solver

The IBRANS solver has already been applied to several test cases to evaluate its accuracy. The flow in a wavy-wall channel at a Reynolds number \( Re_f = 11,000 \) was reported in [5], together with the challenging piston/cylinder assembly. The flow around road vehicles has then been recently tackled to study the unsteady dynamics of the wake and the modiﬁcation induced by drag reduction devices [5,6]. The veriﬁcation of the conjugate heat transfer capability in IBRANS is close to a typical electronic process. Two hot components within a mold are placed in an isothermal cavity. This last test case can also be seen as representative of the heat transfer within the sealed volume containing the power electronics of the electronized motor shown in Fig. 1. The resulting CHT analysis would then help properly designing its heat sink with fins. The cavity has a \( 0.25 \times 0.25 \times 0.25 \) m dimension. The mold has a \( 0.1 \times 0.05 \times 0.025 \) m dimension and is placed in the bottom corner of the cavity as shown in Fig. 7. The two hot spots have much smaller thicknesses. No slip at a fixed temperature of 300 K is assumed on the cavity walls. No slip with coupled thermal conditions are imposed on the mold with equal temperature and heat flux in the fluid and solid. The heat load is specified uniformly in the sphere to 100 W/m². The flow is assumed laminar and a steady state simulation is searched. Second order discretization is applied to both momentum and energy equations. The grid resolution is set to 0.05 m in the cavity and to 0.025 m on the sphere surface. Fig. 5 compares the body-fitted unstructured grid with the corresponding Cartesian mesh in a plane cutting through the center of the sphere. The adapted Cartesian grid has about two times less cells. The convergence is also faster on this grid. The resulting temperature ﬁelds for both methods are compared in Fig. 6 in the same plane as the grid. Very similar diffusion of temperature to the top of the cavity is found with both codes.

The second veriﬁcation of the natural convection conjugate heat transfer capability in IBRANS is close to a typical electronic process. Two hot components within a mold are placed in an isothermal cavity. This last test case can also be seen as representative of the heat transfer within the sealed volume containing the power electronics of the electronized motor shown in Fig. 1. The resulting CHT analysis would then help properly designing its heat sink with fins. The cavity has a \( 0.25 \times 0.25 \times 0.25 \) m dimension. The mold has a \( 0.1 \times 0.05 \times 0.025 \) m dimension and is placed in the bottom corner of the cavity as shown in Fig. 7. The two hot spots have much smaller thicknesses. No slip at a fixed temperature of 300 K is assumed on the cavity walls. No slip with coupled thermal conditions are imposed on the mold with equal temperature and heat flux in the fluid and solid. The heat load is specified uniformly in the sphere to 100 W/m². The flow is assumed laminar and a steady state simulation is searched. Second order discretization is applied to both momentum and energy equations. The grid resolution is set to 0.05 m in the cavity and to 0.025 m on the sphere surface. Fig. 5 compares the body-fitted unstructured grid with the corresponding Cartesian mesh in a plane cutting through the center of the sphere. The adapted Cartesian grid has about two times less cells. The convergence is also faster on this grid. The resulting temperature fields for both methods are compared in Fig. 6 in the same plane as the grid. Very similar diffusion of temperature to the top of the cavity is found with both codes.

5.1 Natural Convection Within Enclosures. The first verification of the natural convection conjugate heat transfer capability in IBRANS has been performed by studying the flow around a heated sphere in a simple cubic enclosure (Fig. 4). The cavity has a 2 m side and the sphere has a 1 m diameter and is centered within the cavity. No slip at a fixed temperature of 300 K is assumed on the cavity walls. No slip with coupled thermal conditions are imposed on the sphere with equal temperature and heat flux in the fluid and solid. The heat load is specified uniformly in the sphere to 100 W/m². The flow is assumed laminar and a steady state simulation is searched. Second order discretization is applied to both momentum and energy equations. The grid resolution is set to 0.05 m in the cavity and to 0.025 m on the sphere surface. Fig. 5 compares the body-fitted unstructured grid with the corresponding Cartesian mesh in a plane cutting through the center of the sphere. The adapted Cartesian grid has about two times less cells. The convergence is also faster on this grid. The resulting temperature fields for both methods are compared in Fig. 6 in the same plane as the grid. Very similar diffusion of temperature to the top of the cavity is found with both codes.
and $5.12 \times 10^7$ W/m$^3$ for the smaller one. The flow is assumed laminar and a steady state simulation is searched. Second order discretization is applied to both momentum and energy equations. The grid resolution is set to 0.06 m in the cavity and to $10^{-4}$ m on the much smaller components. Figure 8 compares the body-fitted unstructured grid with the corresponding Cartesian mesh in a plane cutting through the center of the mold. The adapted Cartesian grid has about 20% less cells. The convergence is again faster on this grid. The resulting temperature fields for both methods are compared in Fig. 9 in the same plane as the grid. Figure 10 shows the temperature contours on the mold surface looking at the bottom of the mold where the hot chips are. In both cases, the IB temperature field matches the FLUENT one closely.

5.2 Heated Cylinders in Cross Flow. The verification of the forced convection conjugate heat transfer capability in IBRANS has been performed by studying the flow around a heated cylinder at various Reynolds numbers based on the cylinder diameter, $Re_D$. The cases selected correspond to three main flow regimes [16–18]: at $Re_D=23$, the flow is two dimensional, laminar and steady; at $Re_D=120$, the flow is two-dimensional, laminar and unsteady (counterrotating two-dimensional vortices are shed behind the cylinder); at $Re_D=3900$, the flow is three dimensional, turbulent and unsteady.

Figures 11–14 present the results of the heated cylinder at $Re_D=23$. Figure 11 compares the body-fitted unstructured grid with the corresponding Cartesian mesh. Figure 12 then compares temperature fields of the FLUENT simulation with that of the corresponding IBRANS simulation. A constant point heat source is applied in the cylinder near its center, for all simulations. However, two different thermal conductivities of the cylinder are considered to represent extreme conditions of solid conductivity. In the first case (Figs. 11(a) and 11(b)), the cylinder conductivity $k_s$ nearly matches the fluid conductivity $k_f$, $k_s = k_f$. The diffusion of temperature within the solid material is clearly evidenced. Figure
13 shows the velocity and temperature profiles along the centerline of the computational domain. Within the cylinder the temperature gradient is clearly seen and a maximum temperature of about 700 K is reached. In the second case \( k_f/k_s \ll 1 \), the solid conductivity is much higher than the fluid one: \( k_s \gg k_f \). The hot spot produced by the point heat source has now been fully conducted to the cylinder surface (complete temperature diffusion within the solid material). Figure 14 again shows the velocity and temperature profiles along the centerline of the computational domain. Compared to the above case, the velocity profiles are similar, as the effect of temperature on the fluid properties is not accounted for here. On the contrary, the temperature is now constant within the cylinder, equal to a larger maximum temperature of about 1000 K. In both cases similar heat convection is observed in the cylinder wake as the Reynolds number is not changed and the fluid properties are not affected by the local temperature. The comparison between the two cases clearly shows the different role of conduction within the cylinder. Overall, the IB temperature contours compare very well with the corresponding body-fitted ones. Similarly, the velocity and temperature profiles obtained with IBRANS do not show any significant difference with the FLUENT body-fitted results.

Similar results are obtained at the other Reynolds numbers. For both \( \text{Re}_D = 120 \) and \( \text{Re}_D = 3900 \), the temperature contours follow the vortex shedding pattern leaving the cylinder in a very similar way between IBRANS and FLUENT. Figure 15 stresses that for the turbulent case \( \text{Re}_D = 3900 \), the periodic variation of the lift coefficient is almost identical between the body-fitted simulation with FLUENT and the Cartesian grid one with IBRANS, yielding similar Strouhal number and mean lift coefficient. Similar results are found on the drag coefficient. They also compare favorably with the existing experimental data for this well documented turbulent case [16–18].

6 Present Realistic 3D Model

The next step has been to tackle the complete electronized EEC motor shown in Fig. 1. The CAD transfer of the latest version of this motor, termed V6+, was successful without any modification and simplification of the industrial model meant for prototypes. To simulate the flow through the motor V6+, a box has been drawn around the motor and a wall has been put around the motor to force the flow through the motor. Figure 16 shows the resulting computational domain and topology. A uniform air stream is imposed on the left of the box to create airflow through the back plate of the motor. Three different velocities have been considered 1, 2 and 4 m/s, which cover the motor operating range on the fan system. This range can be found by separate CFD simulations, which couple the main flow through the fan and the internal flow under the fan hub through the motor modeled as a porous medium [19]. To remove the numerical difficulty associated with the wiring rotating closely to the magnets with a very narrow rotor-stator interface, only the 0 rpm case is presently considered (cold run).

A first qualitative verification of the flow field has been achieved by comparing the air paths at the design condition (2 m/s), with the previous body-fitted STAR-CD simulations in a similar EEC motor described in [2]. Figure 17 shows the results of this cold run through the motor in two different planes: in the middle of the wiring and collector on the left and close to the brush card on the right. As expected, the larger flow rate is achieved in the motor stack or around the collector where the larger air gaps are present to cool the copper wiring. Similarly the large gaps between the magnets allow a greater flow rate than the tiny gap between these magnetic poles and the rotor, which is kept

![Fig. 12 Heated cylinder Re=23. Temperature contours. Top: body fitted (FLUENT). Bottom: IB Cartesian.](image)

![Fig. 13 Heated cylinder \((k_f=k_s)\) Re=23. Mid-line velocity and temperature.](image)
as small as possible to create the maximum electromagnetic flux and optimize the motor efficiency. The holes made in the brush card also bring maximum airflow around the hottest components, the brushes and the selfs.

To quantitatively verify the IBRANS predictions the motor flow characteristics have been compared to available measured data. The experimental apparatus is shown in Fig. 18. The setup consists of a flowmeter which forces a known flow rate through the motor. Two pressure taps close to venting holes in the front and back plates of the motor provide the pressure drop through the motor. The influence of the selected venting holes in the back and front plates has been shown to be negligible. More details can be found in Ref. [19]. Figure 19 shows the pressure drop through the electrical motor as a function of the flow rate. Two sets of data, termed V5 and V6+ in Fig. 19, correspond to two different prototypes of the same motor with slight differences on the brush card (minor design modifications) and heat sink. None of them should affect the flow paths significantly. Yet some scatter in the tolerance of assembly is expected at this prototype stage. Therefore, these two curves provide a good estimate of the combined experimental uncertainty and process scattering on the motor flow characteristics. The IBRANS results (solid square symbols) lay in between these two curves and therefore provide sufficient practical accuracy over the whole operating range.

7 Conclusions

The use of a Cartesian RANS solver for fluid and heat transfer problem in realistic, industrial configuration is presented. The approach is based on the use of the immersed boundary technique to represent the geometry on a non-body conformal grid. The major advantage is the ability to handle complex geometries defined in a CAD environment without the need for surface mesh generation.

A solid/fluid heat transfer module has been implemented and verified by several simple test cases for both natural and forced convection. The former deals with flows in enclosures around a sphere and electronic components. The latter copes with “classi-
cal” flows around heated cylinders. Both heat transfer problems are relevant to the industrial issues encountered in electrical motors. In fact, the natural convection heat transfer is found in the electronic box that might drive the speed of these machines and in sealed motors. The forced convection is also always present around the motor by the airflow created by the driven fan and inside the motor when it is open. All results are compared to body-fitted calculations over a range of Reynolds numbers and for different solid/fluid thermal conductivity. The consistency between the two methods is remarkable showing the feasibility of solving conjugate heat transfer problem with a non-body-fitted mesh.

Preliminary simulations of a complete electric motor consisting of about 100 different CAD parts have also been carried out. An adaptively refined mesh was automatically generated and flow calculations performed over a range of flow rates. Comparison to experimental integral measurements (pressure drop versus flow rate) also demonstrated the sufficient accuracy of the present approach and the capability of handling extremely complicated geometry in a fully automatic fashion. The whole simulation process with the IB method took less than one day compared to the six months needed for the corresponding body-fitted simulation.

Acknowledgment

The authors wish to thank G. Kalitzin for providing significant support in the development of the IBRANS code and the implementation of the conjugate heat transfer model.

References

Briefs, Stanford University.


