Modelling of Accurate and Fast Heat Transfer Analysis using FILM Subroutine

Abhilash P.M., Anup V.P.

Robert Bosch Engineering and Business Solution Pvt. Ltd, Bangalore, India

(Formerly Robert Bosch, India)

***Abstract**: In typical engineering applications, mixed convection is simulated using an assumed convection Heat Transfer Coefficient (HTC); but in reality, the HTC depends on various parameters like the orientation of geometry, the flow regimes, the temperature of the body, and the properties of the fluid. Hence the simulation using assumed HTC is less accurate. The thermal analysis can be made more realistic by taking into the account of variation of HTC due to several factors. A CFD analysis or a conjugate heat transfer analysis computes the HTC accurately but a major limitation is higher run time. On the other hand one can predict the HTC by using various correlations and dimensionless numbers. Keeping this in mind, a user subroutine (FILM) computing temperature state dependent HTC's has been developed and implemented in Abaqus. This results in accurate and fast uncoupled heat transfer simulation without making assumptions.*

For natural convection the Nusselt's number is a function of Grashof's number and Prandtl number, and for forced convection the Nusselt's number is function of Reynold's number, Grashof's number and Prandtl's number. Where Grashof's number and Prandtl number are functions of surface temperature and properties of fluid around the body. Once Nusselt's number is known, HTC can be estimated by the characteristic length of the geometry and the conductivity of the fluid. These temperature depended functions are fed into the FILM subroutine and recalculated the HTC at every increment of the transient heat transfer analysis. The advantage here is faster analysis compared to the conjugate heat-transfer analysis. Thus, the complex convection phenomena is easily included in the simulation and enhanced the functionality of Abaqus.

This paper describes in detail how to calculate the temperature dependent HTC using FILM subroutine and the simulation results comparison with conjugate heat transfer analysis carried out in Abaqus.

***Keywords**: Abaqus, Uncoupled Heat transfer, Natural convection, Heat transfer coefficient, Temperature, user subroutine, temperature dependent Heat transfer coefficient, FILM, Nusselt's number, Grashof's number, Prandtl number, CFD, conjugate heat transfer analysis*

1. Introduction

Convection is the heat transfer between a solid body and a moving fluid. In forced convection, the movement of fluid happens with the help of an external agency. In natural convection, the flow of the fluid takes place by the buoyant force. The buoyant force occurs because of temperature-induced density variations. Modelling of natural convection is important in many engineering applications such as heat exchangers, e-Motors, cooling of electronic equipment and solidification processes [1].

Generally for an uncoupled heat transfer analysis, the simulation is done using an assumed value of HTC [2]. The HTC is an important parameter in a thermal analysis [6]. Accurate HTC can give real time temperature profiles accurately, i.e. HTC value is one of the decisive accuracy factors for natural or mixed convection. Typical value of HTC varies from 2-25 W/m^2K for gases, and 50-1000 W/m^2K for liquids [4]. Due to this diverse range of HTC assumptions (values), there may be a high chance that the result deviate from reality, also it is less methodic. One can conduct a conjugate analysis to find out the heat distribution without this assumption of HTC, where the complete surroundings of air are modelled and simulated together. Nevertheless, this Conjugate Heat Transfer simulation is highly challenging and computationally expensive due to combined solving of Navier-Stoke's equation over heat equation [2]. Here, buoyancy/density depends on temperature, temperature depends on natural convection coefficient, and natural convection coefficient depends on density again. Due to this convoluted nature, the simulation is challenging and computationally expensive [9].

On other hand, using various dimensionless correlations, one can estimate the heat transfer coefficient of natural convection phenomena and incorporate it into an uncoupled heat transfer analysis, where heat equation is solved independently. The major hurdle here is its dependency on various factors like geometries, configuration, variation of temperature on surfaces and thermo-physical properties of fluid medium Using Abaqus subroutine environment, one can implement this temperature, geometry depended HTC into uncoupled heat-transfer analysis. The advantage here is less run time as the heat equation is solved independently here.

1.1 Objectives

The main objective of this work is to develop a subroutine for uncoupled heat transfer analysis by using various numerical correlations, thereby predict the HTC and avoid computationally challenging conjugate heat transfer analysis in a system undergoing natural convection. This paper compares the accuracy of uncoupled heat transfer analysis done using the developed subroutine over a conjugate simulation on a simple plate geometry. Comparison of simulation time is also described in the paper.

2. Estimation of Heat Transfer Coefficient

2.1 Newton's Law of Cooling

The rate of convection heat transfer is observed to be proportional to the temperature difference and is conveniently expressed by Newton's law of cooling as [10]

$$\dot{Q} = hA(T_S - T_\infty) \qquad \text{Equation (1)}$$

Where,

$\dot{Q}$ = the rate of heat transfer between the fluid and the surface, [W].

h = heat transfer coefficient [HTC], [W/m^2-K]

A = Area of the surface in contact with the fluid, [m^2]

T_S = Temperature of the solid surface, [°C]

T_∞ = Temperature of fluid far away from solid surface, [°C]

2.2 Convection coefficient and Non-dimensional numbers

Natural convection heat transfer on a surface depends on the geometry of the surface as well as its orientation. It also depends on the variation of temperature on the surface and the temperature dependent thermo-physical properties of the fluid involved.

The typical approach [10] for h in terms of *Nusselt's number*, Nu_L is written as

$$h = \frac{Nu_L k}{L_C} \qquad \text{Equation (2)}$$

Where, k = Thermal conductivity [W/^{2}K]

L_C = Characteristic length, [L]

The *Nusselt's number* Nu_L is the ratio of Convective heat transfer to conductive heat transfer, can also be written the form of

$$Nu_L = CRa_L{}^n \qquad \text{Equation (3)}$$

The values of C and n depend on the geometry of the surface and flow regime. The Nusselt's number for a variety of configurations are determined experimentally, can be found in literatures [5, 10]. For vertical plates, C=0.59, n=1/4 ($10^4 < Ra_L < 10^9$), and C=0.1, n=1/3 ($10^9 < Ra_L < 10^{13}$)

Here Ra_L is Rayleigh's number, describes the relationship between buoyancy and viscosity within a fluid, which can be written as the product of the Grashof's and Prandtl numbers:

$$Ra_L = Gr_L \Pr \qquad \text{Equation (4)}$$

Where, $\Pr$: Prandtl number is defined as the ratio of momentum diffusivity to thermal diffusivity. And written as,

$$\Pr = \frac{C_p \ v}{k} \qquad \text{Equation (5)}$$

Where,

C_p = specific heat of fluid [J-Kg-K]

v = kinematic viscosity of fluid [Ns/m^2]

k = Thermal conductivity [W/^{2}K]

Gr_L = Grashof's number, is a measure of the relative magnitudes of the buoyancy force and the opposing viscous force acting on the fluid, and is expressed as follows**,**

$$Gr_L = \frac{g\beta L_c{}^3 \left(T_S - T_\infty\right)}{\nu^2} \qquad \text{Equation (6)}$$

Where,

g = gravitational acceleration [m/s^2]

β = coefficient of volume expansion [1/°C]

T_S = Temperature of the surface [°C]

T_∞ =Temperature of the fluid sufficiently far from the surface [°C]

ν = kinematic viscosity of the fluid [m^2/s]

Here all the properties of fluid properties are evaluated at the film temperature T_f

$$T_f = \frac{\left(T_S + T_\infty\right)}{2} \qquad \text{Equation (7)}$$

By knowing type of geometry, surface temperature, and temperature depended fluid properties, one can calculate the heat transfer coefficient using the correlations expressed from the Equation 2 to Equation 7.

3. Working Algorithm of FILM subroutine.

The temperature depended HTC calculation in Abaqus can be implemented through FILM subroutine environment [3]. FILM subroutine environment is used to describe heat transfer coefficient behavior as a complex function of time, position, and surface temperature.

Inside subroutine, different correlations are embedded for different types of geometries based on the algorithm shown in the figure (1). During the pre-processing stage in the model, the surfaces in the geometries should be categorized and named accordingly to distinguish the type and orientation of the geometry. All the properties and dimensionless correlations are then calculated in terms of mean temperature as shown in the Equation 7. Thus, HTC depended on multiple parameters are accurately calculated here.

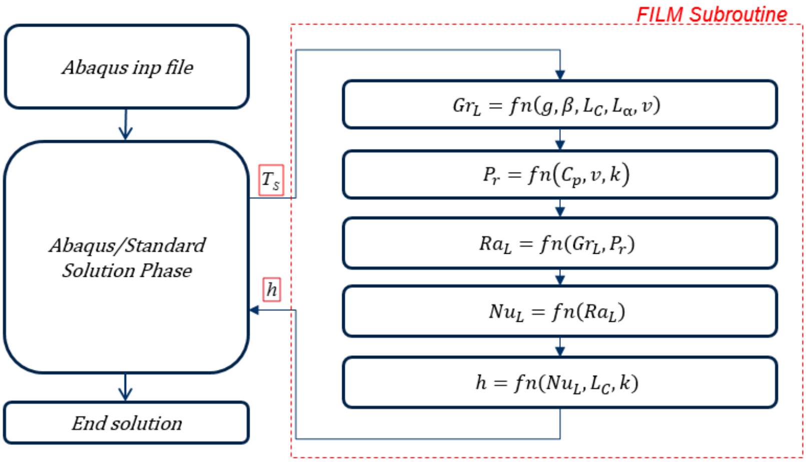

Figure 1. Working algorithm of FILM subroutine

As shown in the working algorithm [Figure 1], at every increment of the Abaqus/Standard solution phase, FILM subroutine is called and nodal HTCs are re-calculated as shown in the algorithm. FILM subroutine fetches the nodal surface temperature and surface name during each iterations. The nodal surface temperature and information is then used to calculate various dimensionless numbers as shown in the figure 1. At each increment, the corresponding nodal HTCs are calculated and fed back to the solver. Characteristic length and temperature dependent fluid properties are added in the subroutine which is used to calculate various dimensionless numbers.

4. Simulation Comparison between Conjugate Heat Transfer analysis and a FILM subroutine.

4.1 Problem statement:

Since the emphasis of this simulation is to see the convection behavior of the system, the cool down behavior of a hot Aluminum plate exposed to ambient air is simulated via convection, and radiation effect is neglected. It is assumed that the plate is at 100^{o}C at the beginning of the simulation as shown in the figure 2. When Plate is exposed to atmosphere, the hot surface of the plate induces a temperature-dependent density differential in the surrounding air, thereby setting up a buoyancy-driven natural convection cooling process to the external surface. Heat dissipation from the surface of the Plate to the atmosphere results into a heat transfer within the Plate due to conduction. Thus, the heat is from Plate body to the surfaces via conduction then to ambient air (sink) through the convection process. The simulation is carried out for 500 seconds.

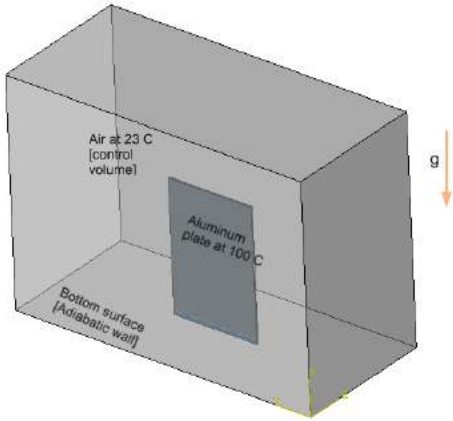

Figure 2. A hot Aluminum plate placed vertically in a fluid domain.

The simulation is done in three approaches:
(1) Conjugate Heat transfer analysis,
(2) Uncoupled heat transfer simulation with developed FILM subroutine,
(3) Conventional uncoupled heat transfer simulation with an assumed HTC of 10 W/m^2K.

The results of all the simulations are compared and the temperature deviation from the conjugate simulation is accounted. Here the Conjugate simulation approach is set as a baseline for the comparison with other approaches.

4.2 Simulation methodology for conjugate heat transfer

Conjugate heat transfer corresponds with the combination of heat transfer in solids and heat transfer in fluids. The governing equations are solved for each object in two subdomains (solids and fluids). Matching conditions for these solutions at the interface provide the distributions of temperature and heat flux along the body/flow interface.

4.2.1 Geometry, Mesh, and Materials

The Plate dimensions considered are 0.078×0.116×0.0016m. The size of the computational domain that encloses the plate is taken to be 0.278×0.020×0.01256 m for the Abaqus/CFD flow computations as shown in the Figure 2. The Aluminum plate is modelled with appropriate number of DC3D8 elements. Abaqus/CFD model is meshed with linear hexahedral (FC3D8) elements as shown in the figure 3.

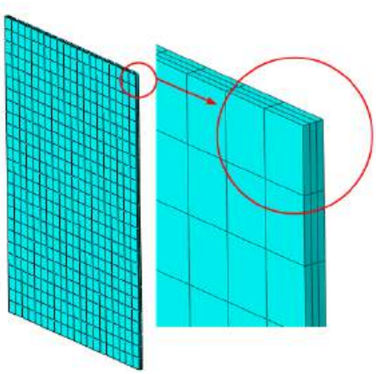

Figure-3a Aluminum Plate

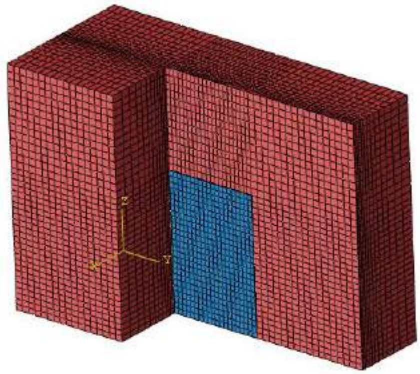

Figure-3b Fluid Volume

Figure 3. Finite element mesh for Aluminum Plate and Fluid volume

The Aluminum material has a thermal conductivity of 140 W/m-K, a density of 2960 kg/m^3 and a specific heat of 960 J/kg-K. For the external fluid domain, the temperature dependent properties of air [7] are taken from the table 2.

Temperature [^{o}C]	Density [kg/m^3]	Specific heat [J/kg-K]	Conductivity [W/m-K]	Viscosity x10^{-6} [m^2/s]	Expansion Coefficient x10^{-3}
0	1.293	1005	0.0243	13.3	3.67
20	1.205	1005	0.0257	15.11	3.43
40	1.127	1005	0.0271	16.97	3.2
60	1.067	1009	0.0285	18.9	3
80	1	1009	0.0299	20.94	2.83
100	0.946	1009	0.0314	23.06	2.68
120	0.898	1013	0.0328	25.23	2.55
140	0.854	1013	0.0343	27.55	2.43
160	0.815	1017	0.0358	29.85	2.32
180	0.779	1022	0.0372	32.29	2.21
200	0.746	1026	0.0386	34.63	2.11
250	0.675	1034	0.0421	41.17	1.91
300	0.616	1047	0.0454	47.85	1.75

Table 2. Temperature depended material properties of Air

4.2.2 Initial and Boundary Conditions

The initial temperature of the Aluminum plate is assumed to be 100^{o}C and the external fluid (ambient air) is at 23^{o}C. In addition, the fluid is initially assumed to be quiescent and hence, its velocity is set to zero everywhere.

Following assumptions are made for the simulation:

(1) No heat addition to the plate during the cooling process,
(2) The bottom surface of the external computational domain is a rigid floor,
(3) Bottom surface is an adiabatic wall.

The Plate is firmly attached to the bottom of the computational domain, as shown in Figure 2. The computational domain boundaries for the external flow and heat transfer calculations are positioned at sufficient distances from plate such that the effects of the boundaries on the results are negligible. A no-slip, no-penetration flow wall boundary condition is enforced at the bottom surface. At the top, an outlet boundary condition is specified with the fluid pressure set to zero.

For all other boundaries, free-stream conditions are applied for both the velocity and the temperature. Gravity loading is applied along the negative Z-axis as shown in the figure 2. FSI boundary condition is enabled between heat-transfer model and CFD Fluid domain. The conjugate heat transfer analysis is carried out for 500 Seconds.

4.3 Model setup for uncoupled heat transfer with FILM subroutine.

The same cooling behavior of heated aluminum plate explained in the session 4.2 is simulated using purely coupled-heat transfer analysis in conjunction with developed FILM subroutine. Unlike the conjugate heat transfer analysis, fluid domain is not simulated here. Instead the convection boundary condition is applied via developed FILM subroutine.

4.3.1 Geometry and Materials

Material, dimension, and mesh density are also kept same like above for aluminum plate.

4.3.2 Initial and Boundary conditions

The initial condition of the aluminum plate is assumed to be 100^{o}C, just like session 4.2.2. No separate fluid volume is simulated here; but the temperature dependent material property of air shown in the table-2 is included in the FILM subroutine.

4.3.3 Including subroutine in the model

The convection boundary condition is applied to the vertical surfaces of Plate where the surfaces are named as “VERTICAL_FACES” and activated *SFILM for the convection contact [3], during the solution phase, the subroutine is called and nodal heat transfer coefficient is calculated and fed back to the solver. Character length is taken as 0.116m.

4.4 Model setup for conventional uncoupled heat transfer with assumed HTC: 10W/^{2}K

Simulation approach is kept same like session 4.3; except the convection HTC. Here, instead of a subroutine, a constant value of 10 W/m^{2}K is directly applied on the Aluminum plate. No Fluid volume separately simulated here.

5. Results and Discussion

The heated air near the plate surface becomes less dense and rises upwards due to buoyancy. Because of incompressibility, the upward rising air entrains cold air from the surroundings, thereby setting up a natural convection flow which cools down the surface of the plate via convection. The heat inside the aluminum plate gets distributed to its surface by conduction thereby transferring heat to the atmosphere (sink). The transient contours comparison of temperature distribution (NT11) of plate for all 3 approaches are shown in Figure-4. The qualitative measurement of temperature at the middle of the plate is also shown in the Figure-5. Based on the quantitative measurement, the deviation of simulation result compared with conjugate heat transfer analysis is accounted as shown in Figure-6.

Uncoupled heat transfer analysis using subroutine (approach 2) resembles to the conjugate heat transfer simulation (approach 1) very closely with a maximum percentage deviation of result by 3.6%. Whereas the simulation approach with an assumed value of convection coefficient (approach-3) is observed to be deviating from the conjugate heat transfer simulation by 27%.

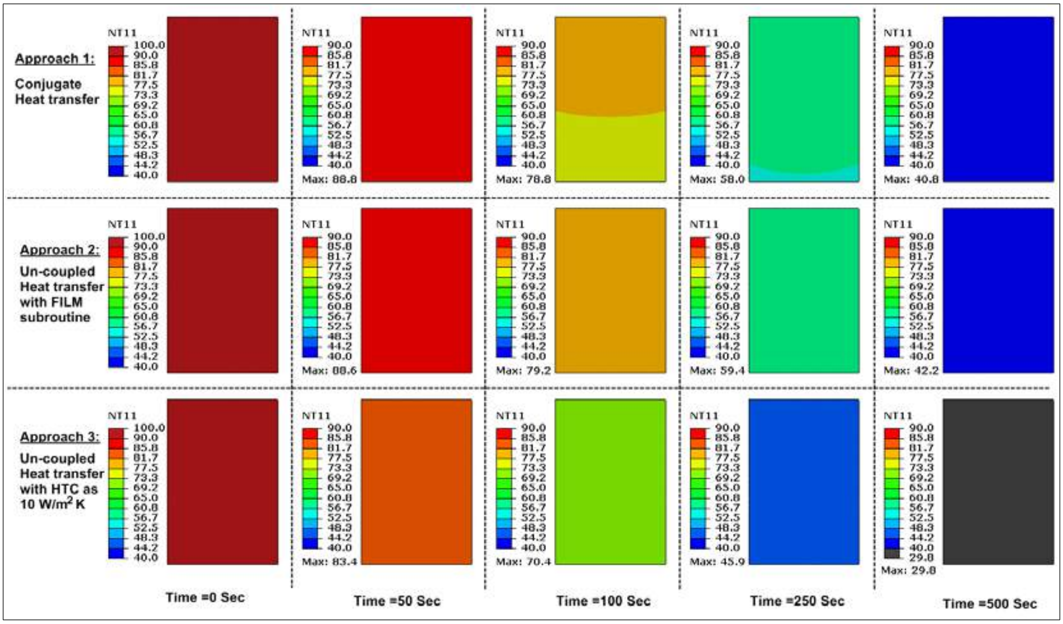

Figure 4. Transient temperature contour of Plate with different analysis approach.

The hot air sweeps from the bottom of the plate and thickens the boundary layer at the top of plate and due to this, temperature of the top portion of the plate is hotter than the bottom [9] as shown in the Figure-4 at 100 seconds. This is captured in the conjugate heat transfer analysis; whereas FILM subroutine fails to predict such kind of behaviors as there is no fluid volume separately taken into account in the simulation. This is a limitation of the subroutine. And fails in enclosed natural convection.

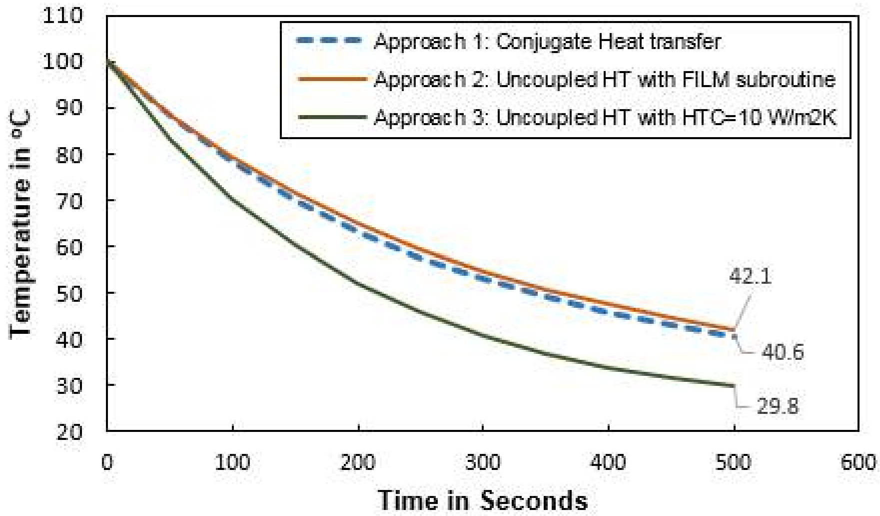

Figure-5. Temperature probe at mid-point of Plate in different analysis approach.

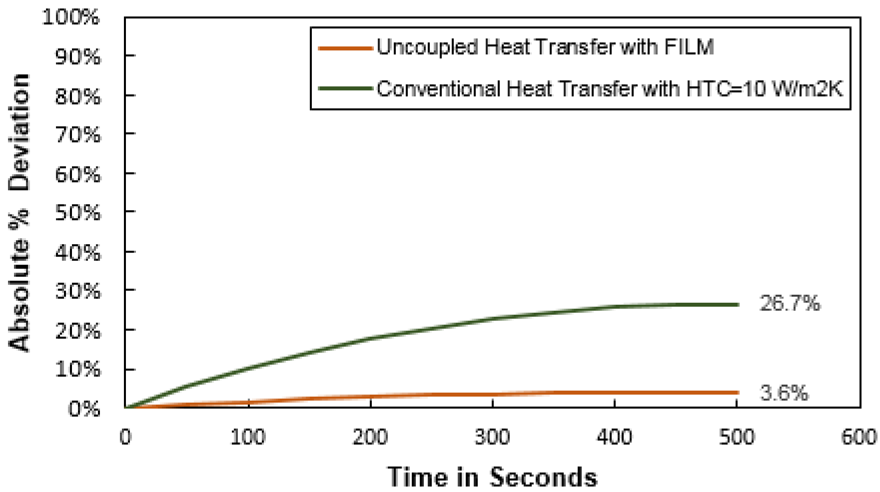

Figure-6. Percentage deviation of result compared with Conjugate Heat Transfer analysis

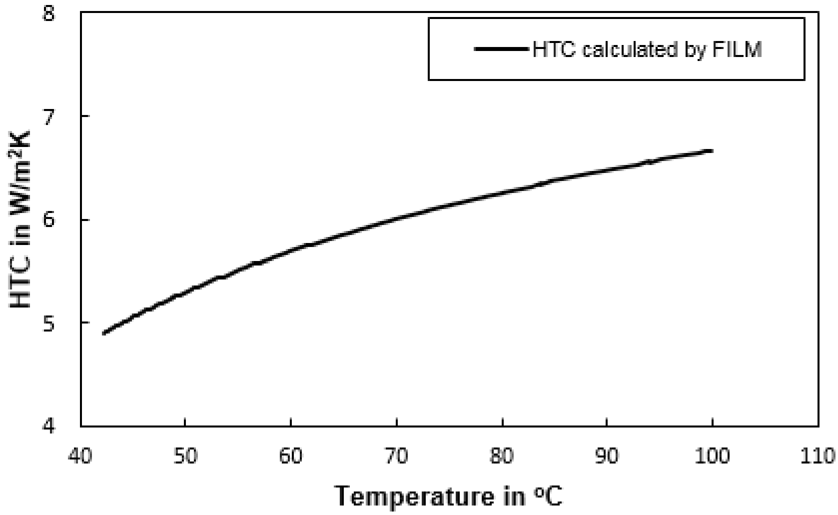

Figure-7. Temperature depended HTC; calculated inside the FILM subroutine during analysis.

HTC calculated by subroutine at every iteration of the simulation is written out and plotted in the figure-7. It is evident that the HTC is varying based on the temperature of the surface, and it is transient in nature; whereas in the conventional uncoupled heat transfer analysis the HTC is assumed to be constant through-out the analysis, thus approach 3 deviates from the reality.

Approach	Methodology	Solving time (in HP, Z800 HP machine with 8 cpus)
1	Conjugate Heat Transfer Analysis	~4 Hours(14400 Seconds)
2	Uncoupled HT with FILM subroutine	40 Seconds
3	Conventional uncoupled HT with HTC=10 W/m2K	40 Seconds

Table 3. Simulation Solving Time

The simulation time comparison is listed in the table 3, which shows that the uncoupled heat transfer analysis with subroutine solves in seconds whereas the conjugate heat transfer takes hours to solve the simulation. Also analysis with subroutine is as fast as a simulation with an assumed HTC.

6. Conclusion

Development and Implementation of FILM subroutine for predicting the heat transfer coefficient for a complex natural convection phenomena on a plate is presented in this paper. Based on the results, it is evident that this methodology reduces the computational cost over the conjugate heat transfer analysis with a very high accuracy of results. These results and the simulation time comparison shows that the developed subroutine has a high simulation efficiency with very small loss in the accuracy, and have great potential for replacing conjugate heat transfer analysis for open Natural convection simulations. However the present study is limited to simple shapes and its combinations. The study can be further expanded to complex geometries.

7. Acknowledgement

We would like to thank the management of Robert Bosch Engineering and Business Solutions Private Limited for the support in executing this work and for the encouragement in publishing the results. We express our gratitude is towards Mr. Thomas Heid and Mr. Kilian Kriener (Robert Bosch, GmbH), for their guidance and encouragement for the work. We would also like to thank Mr. Basvaraj Pyati (RBEI/ENF) for his invaluable inputs on conjugate heat transfer analysis.

8. References

1. A. Bairia, E. Zarco-Perniaa, J.-M. Garcia de Mariab, A review on natural convection in enclosures for engineering applications. The particular case of the parallelogrammic diode cavity,
2. Abaqus Users' Manual, Version 6.14-3, Dassault Systemes Simulia Corp., Providence, RI.
3. Abaqus Users Subroutines Reference Guide, Version 6.14-3, Dassault Systemes Simulia Corp., Providence, RI.
4. Incropera, F.P., DeWitt, D.P, Bergman, T.L., & Lavine, A.S., Fundamentals of Heat and Mass Transfer, 6th Ed., Hoboken, NJ, John Wiley & Sons, (2007).
5. Lienhard, J.H, IV and Lienhard, J.H. V, A Heat Transfer Textbook: A Free Electronic Textbook
6. S. Jahangeer, M.K. Ramis, G. Jilani, Conjugate heat transfer analysis of a heat generating vertical plate, International Journal of Heat and Mass Transfer, Volume 50, Issues 1–2, January 2007, Pages 85–93.
7. The Engineering Tool Box, Air Properties, < http://www.engineeringtoolbox.com/>
8. Warren M. Rohsenow, James P. Hartnett, Young I. Cho, Handbook of Heat Transfer, McGraw-Hill Education, 1998
9. Wei Shyy, Elements of Pressure-Based Computational Algorithms for Complex Fluid Flow and Heat Transfer, Advances in Heat Transfer, Volume 24, 1994, Pages 191–275.
10. Yunus A. Cengel, Afshin J. Ghajar, Heat and Mass Transfer- Fundamentals and applications, McGraw-Hill Education , 2013