International Journal of Ignited Minds (IJIMIINDS)

Validation of the Fluid116 element usage for flow related problems in ANSYS thermal analysis

Aravind[a]

[a] *Team Lead, CAE simulation team, Control Component Inc, Bangalore, India*

ABSTRACT

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions). ANSYS has the capability of both Steady state (SS) & transient thermal analysis capabilities. In any thermal analysis, to deal the convection problems, an input of bulk temperature is required. Not always, bulk temperature approach is going to provide correct results. In the physical problem where flow effects are predominant, bulk temperature fails e.g. turbine cooling flows, secondary flow system of HPC or HPT, etc. Fluid116 element can address this efficiently. Current paper discusses the usage & validation of fluid116 element capabilities.

Keywords *–Fluid116, Convection, Thermal analysis, ANSYS, Validation, Fluid heat up*

1. INTRODUCTION

Steady state or transient thermal analysis provides temperature scenario of the component in a detailed fashion. The accuracy of thermal simulation depends upon the modeling of three basic modes of heat transfer i.e. conduction, convection & radiation. The general approach in convection problems is to use the heat transfer coefficient (HTC) & bulk temperature. HTC can be calculated based on geometry, flow behavior & any standard correlation. This approach doesn't account for any increase/decrease of flow temperature. It assumes as constant. In many convection applications, it is necessary to understand the rise in temperature of fluid. To account this effect, a CFD analysis can be performed. But, not always, CFD analysis can be performed because of issues with computational effort, proficiency & license issues. ANSYS provides fluid116 element to account these effects. In the current technical work, usage of FLUID116 element & its validation in calculation is discussed in detail. Validation calculations are performed based on 1D heat transfer equation [1].

2. WHAT IS FLUID116 ELEMENT?

Fig 1 gives element topology of FLUID116. It is a 3-D element with the ability to conduct heat and transmit fluid between its two primary nodes. Heat flow is due to the conduction within the fluid and the mass transport of the fluid. Convection may be accounted for either with additional nodes and convection areas or with surface elements SURF151 and SURF152 [2, 3]. In both cases, the film coefficient may be related to the fluid flow rate. The element may have two different types of degrees of freedom, temperature and/or pressure. But, for the current usage, temperature degree of freedom should be used. The thermal-flow element may be used in a steady-state or transient thermal analysis. There is no equivalent structural element for this. So, this element should be replaced by a null structural element. This element has a capability to calculate the heat transfer coefficients by inputting the relevant parameters in the form of real constants. As the heat transfer coefficients are calculated externally and are input to ANSYS, these features are not used. A hydraulic diameter of 1mm ($1e^{-3}$m) will be used as real constant to avoid any run time errors.

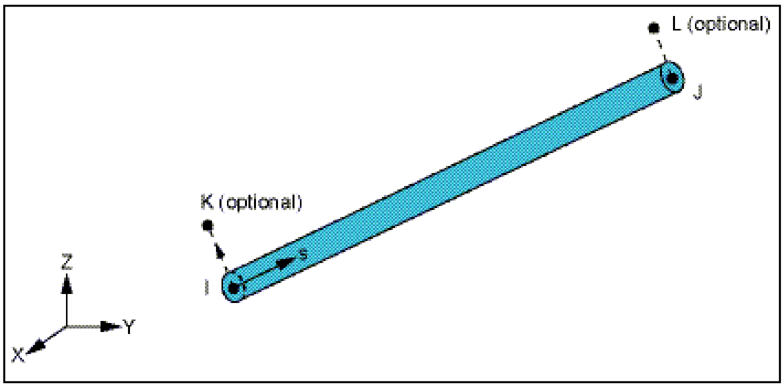

Fig 1: Element topology of Fluid116

3. USAGE OF FLUID116 AND ITS VALIDATION TO FIELD EXAMPLE

An example is taken to explain the usage of FLUID116 elements.
Problem Description: find out the temperature of the body at the steady state condition (i.e. after infinite amount of time). A block of 1mX0.25mX0.25m is experiencing a flow of air at 2.5kg/sec at 20degC on the top face. Initial block/body temperature is 200degC. Assume a HTC of 50W/m^2-degC on the top face because of air flow. Assume the block material properties as stainless steel. Assume all other faces are adiabatic.

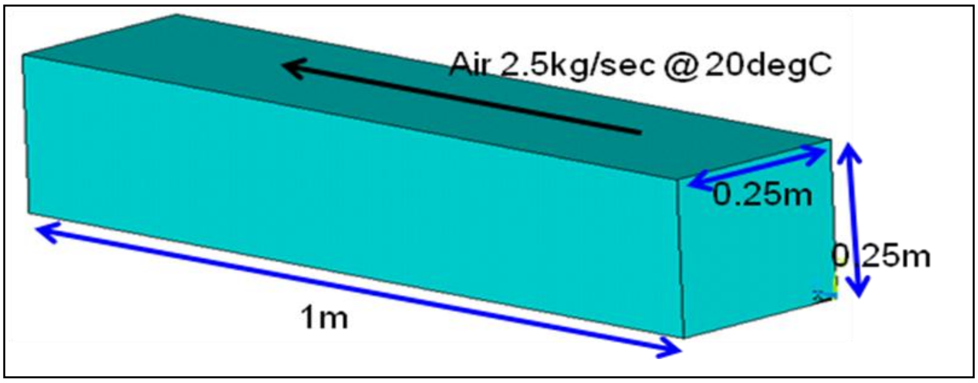

Fig 2: A block of 1mX0.25mX0.25m

Solution @ Filed: Initially, this body will get cooled by the 20degC of air. Transiently, heat transfer between body and air is observed. But, for an infinite amount of time, body should reach to 20degC only.
Solution @ ANSYS: Here, air is passing over the body for a length of 1m. Bulk temperature of 20degC can be used for the top face to run the analysis. But, this simulates that temperature of air is constant and is 20degC at both leading edge & trailing edge of the body. But in reality, the temperature of the air will increase over the length of the body because of the interaction with the hot body. This creates a necessity to estimate the temperature of fluid at several locations. This is a transient phenomenon. But in case of a steady state analysis (analysis for infinite amount of time with the given boundary conditions), body will reach to the temperature of air (20degC). So, based on the zeroth law of thermodynamics, irrespective of the initial component temperature of the body, both air and body will come to equivalent state of 20degC. Refer fig-3 for finite element model and fig-4 for temperature contour.

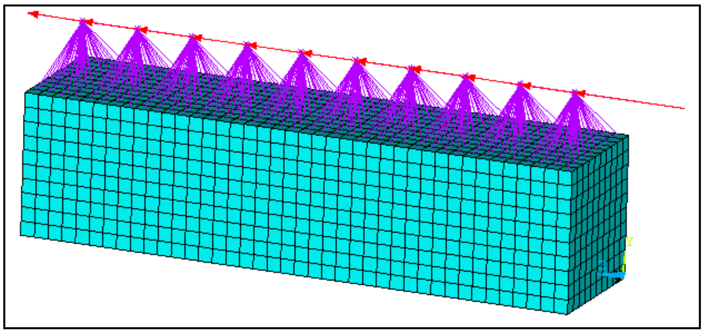

Fig 3: Snap shot of finite element model highlighting the direction of flow (red colored arrows)

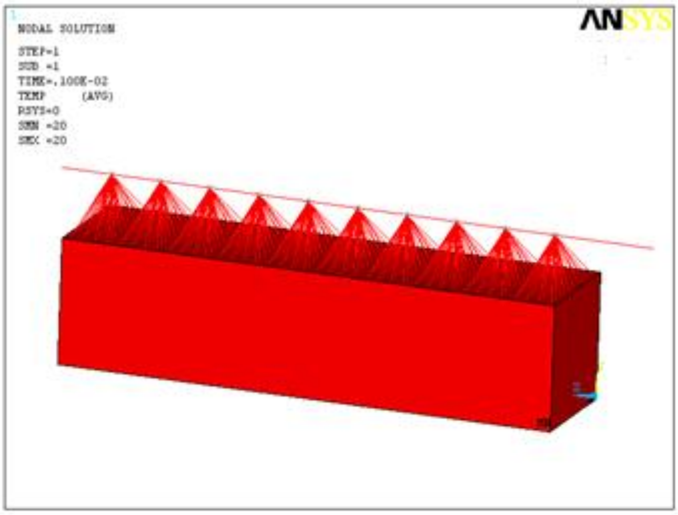

Fig 4: Steady state temperature contour-Both (body & fluid) are at 20degC only.

Example-2: For the problem given in example-1, a heat source is included in the bottom face with a HTC of 2000W/m^2-degC & temperature of 300degC. Find the temperature of the fluid at several locations b/w leading& trailing edges.

Solution @ Analytical: This is similar to the problem of slabs having two end boundary conditions. This can be solved in below procedure.

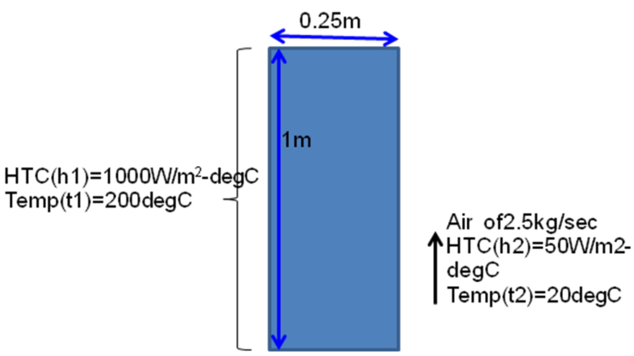

Fig 5: Boundary conditions of the block

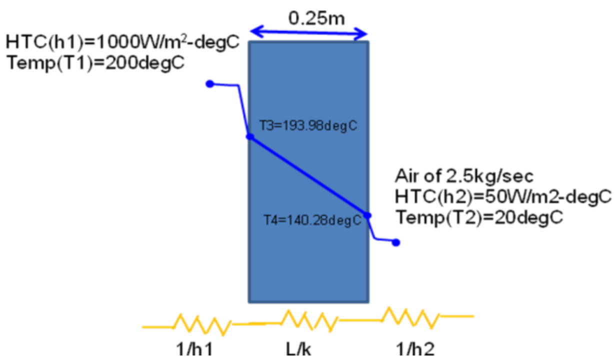

Fig 6: Analytical model highlighting heat resistance

Total resistance = (1/h1+L/K+1/h2)
= (1/1000+0.25/28+1/50)
= 0.0299

Net heat flux=q'' = (T1-T2)/ (total resistance)
= (200-20)/0.0299
= 6014.32W/m^2

From this, T3 and T4 can be estimated as T3= 193.98degC & T4= 140.28degC

Disadvantages in this formulation:

1. Temperature of flow is assumed as constant. But in reality, it increases from leading edge to trailing edge because of the interaction of hot surface. This change in fluid temperature cannot be captured.
2. Temperature of the end walls (T3 & T4) is constant for the entire length of the body. In reality, T4 will not be of constant in nature because of temperature rise of air from leading edge to trailing edge.

Solution @ ANSYS: To capture the above mentioned effects, a fluid network is modeled on the face of 20degC. As the flow temperature is 20degC, at the beginning of the fluid network, a source temperature of 20degC is applied. As the flow value is 2.5kg/sec, this will be boundary condition for fluid network. Refer fig-7 for ANSYS results.

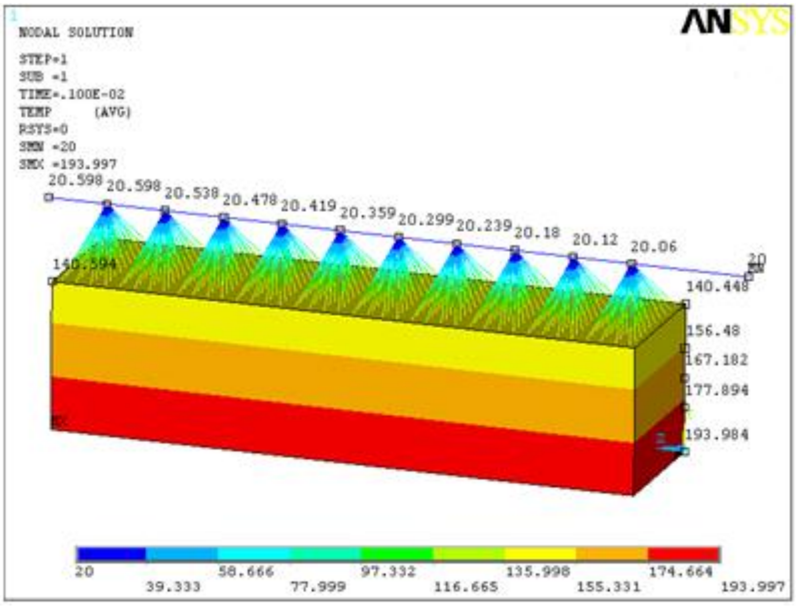

Fig 7: Steady state temperature contour with heated source at bottom face (in degC)

Interpretation:

1. Bottom face temperature is observed as 193.98degC which is equal to analytically derived temperature. Where as, top face temperature is observed to be of 140.59degC which is greater than analytical temperature of 140.28degC.
2. A clear temperature increase (0.6degC) in the fluid is observed which is realistic in nature. The increase is of very small quantity which is because of higher air flow rate.
3. An increase in the metal temperature is observed from leading edge to trailing edge which is inevitable. This increase is also of very small quantity because of minor changes of air temperature.

Example-3: For the problem given in example-2, flow rate of air has been decreased to 0.025kg/sec.
Find the temperature of the fluid at several locations b/w leading& trailing edges.
Solution @ ANSYS: Run the above analysis with a modified boundary condition of flow with 0.025kg/sec and below are the temperature contours along with fluid network temperatures.

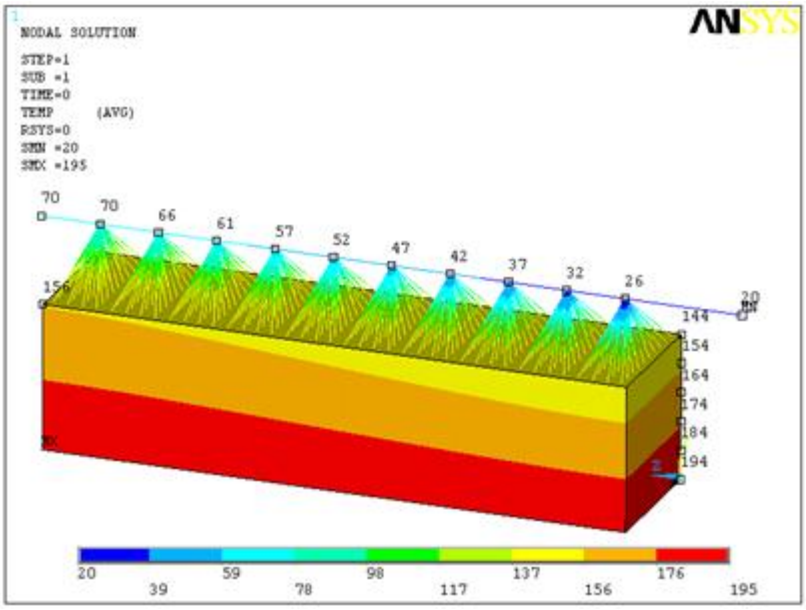

Fig 8: Steady state temperature contour with reduced air flowrate (in degC)

Interpretation:

1. Comparing with example-2, the fluid temperatures looks higher. Fluid temperature at trailing edge is 70degC which is higher than the leading edge temperature. This shows that fluid is getting heated up because of the hot body temperature.

An increase in metal temperatures from leading edge to trailing edge is also observed very distinctively from 144degC to 156degC which cannot be observed & captured in the analytical calculation given above. This rise in metal temperature is because of the rise in temperature of the fluid.

4. CONCLUSION

This technical work discusses the importance of fluid116 element in a thermal analysis. Usage of this element is discussed in detail. Multiple simulation results from ANSYS are compared with 1D heat transfer calculations and it can be observed that both results are matching very well. Hence, it can be to concluded that this element can increases the accuracy of analysis e.g. turbine blade cooling, analysis of secondary flow system etc. & the phenomenon of rise in flow temperature can be captured effectively.

REFERENCES

1. Frank P. Incropera & Dewitt et al, "Fundamentals of Heat & Mass transfer", by Wiley publisher, 6th edition, 2007, pp:2-5, pp: 96-101.
2. Collaborative solutions Inc., “ANSYS Tips & Tricks: Thermal surface effect elements”, Memo No: STI38:000409A, pp: 1-5.
3. ANSYS help 13.0