

Calculation of the Heat Transfer Coefficient in the Outer Body for a Rotational Detonation Engine

John VanOsdol

**The 46th International Technical Conference on Clean Energy
August 1 to 4, 2022
Clearwater, Florida, USA**

Unsteady heat transfer characterization on the combustion surfaces of Rotational Detonation Engines (RDE) is not well understood. It is generally thought that the complex nature of the unsteady, reacting, compressible fluid flow inside the combustion annulus of the RDE causes the convective heat transfer coefficient to be significantly higher than it is in other applications. Empirical models that have been used to analyze this strictly apply to steady flows where dimensionless groups can be employed. The reacting flow fields that are characteristic of RDEs are inherently compressible, three dimensional, unsteady, and turbulent, having properties that change by orders of magnitude throughout the flow field. They will therefore contain multiple length scales and time scales operating everywhere in the flow during all times. Because of this it is not likely that the RDE flow fields will lend themselves to explanation using simple dimensionless parameters. The dimensionless groups have meaning only in situations where length scales and time scales are singular and well defined. In spite of this it may be possible to get a relatively good idea of what the convection heat transfer coefficient is. In this work a numerical study is performed where the inside wall surface temperature distribution in the RDE outer body is systematically changed over a given range that would be characteristic of the start-up flows inside an RDE. For each case, temperature distributions inside the outer containment wall of the RDE was calculated and compared with experimental data. The closest match can then be used to directly calculate the convection heat transfer coefficient on the inside surface of the RDE.

1.0 Introduction:

1.1 RDE Geometry:

A schematic of the general configuration of a typical experimental RDE is shown in Figure 1. The overall axial length of the outer and inner containment walls is assumed to be approximately 100 (mm). The outer wall is made of steel and has a thickness of 10 (mm). A hydrogen air mixture is made to flow through a combustion annulus located between the outer containment wall and the inner containment wall where reaction occurs. The energy release from this reaction is partially dissipated by the formation of a traveling shock wave which propagates around the combustion annulus in an angular direction. Experimental studies have shown that depending on operating conditions there can be either a single wave or multiple waves existing in the annulus at any given time but with proper adjustments of fuel and air flows a stable operating condition can be achieved, Johnson [1]. Thermocouple gauges placed at various axial locations and various depths inside the outer wall were used by Randall et al. [2] to measure the unsteady temperature distribution inside the outer wall during startup. Gauges were placed in rows 1, 2, 3 and 4 of Figure 1.1 at the depths

shown on Table 1.1. In this experimental work the results of these measurements were accompanied by conduction heat transfer calculations which were loosely coupled to Computational Fluid Dynamic (CFD) calculations which assumed that the average convection heat transfer coefficient for the inside surface of the outer body was constant in time and also along the axial dimension. This implies that the convection heat transfer coefficient in these calculations was not local. Calculation results were made and compared to gauge measurements using coefficients of 500 and 1000 (W/m²-K). Similar calculations were repeated by Roy et al. [3] and [4], where the convection heat transfer coefficient on the outer wall of the RDE was again non-local and loosely coupled to the CFD calculation. In this work an averaging procedure was used over the unsteady CFD results that allowed for a calculation of an averaged bulk gas temperature. Calculation results were presented for convection heat transfer coefficients of 500, 1000, and 2000 (W/m²-K) with the best results obtained for a convection heat transfer coefficient of $H=1000$ (W/m²-K).

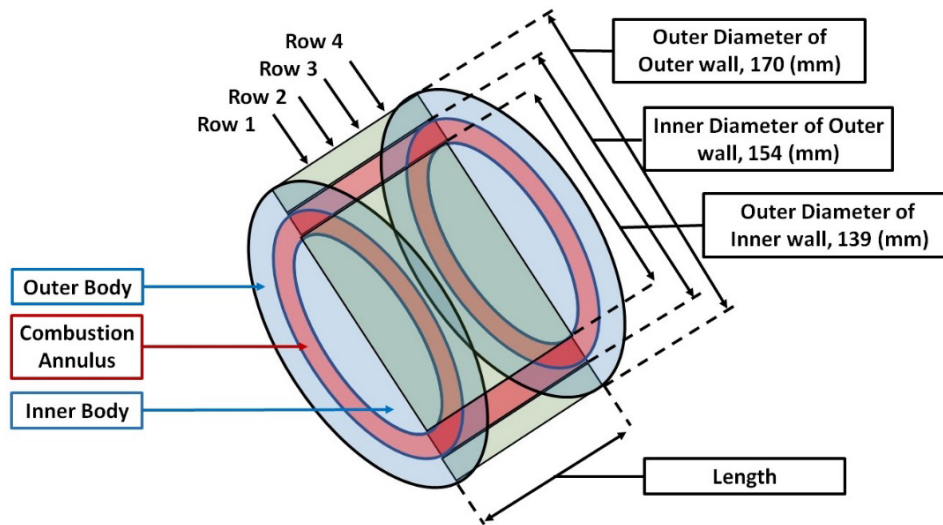


Figure 1.1: Geometry of a Rotational Detonation Engine (RDE) showing the combustion annulus situated between the Outer and Inner containment walls

TC	Row	Depth (mm)
1	2	1.6
2	4	2.4
3	2	4.9
4	4	6.1

Table 1.1 Thermocouple gauge axial location and depth

The results of these studies showed that it is not clear how the local convection heat transfer coefficient varies along the axial direction of the RDE, but that the average value is probably below at or below 1000 (W/m²-K). It should be possible however to use Randall's experimental data to directly calculate what the local convection heat transfer coefficient is without the need of any coupling to fluid dynamic calculations providing that the hot fluid temperature inside the combustion annulus can be estimated. Without either experimental measurement, or CFD

calculations this is generally unknown, but choosing a minimum and maximum range for the average temperatures inside the combustions annulus at any given axial position would allow for the convection heat transfer coefficient to be estimated between upper and lower bounds.

1.2 Heat Transfer Analysis:

Fully coupled calculation:

The heat transfer to the containment walls of a typical RDE is strongly coupled to the fluid flow inside the reaction zone. For a proper coupling it would be necessary that both the temperature, and the heat transfer rate at the surface interface between the solid wall and the fluid adjacent to the wall be equal at all locations and all time. This is completely specified using Fourier's heat conduction law [5] applied to both the solid wall just inside the surface, and the adjacent fluid immediately outside the surface. This is given by equations (1.1) and (1.2) where both the temperature and the heat transfer rate at the surface interface are equal for both the solid and the fluid.

$$T_s = T_w \quad (1.1)$$

$$\left(q_s = -k_s \frac{\partial T_s}{\partial y} \right) = \left(q_w = -k_w \frac{\partial T_w}{\partial y} \right) \quad (1.2)$$

One way to numerically achieve this coupling would be to first use equation (1.1) to specify a temperature distribution at the surface interface which would be used as a common boundary condition for both the fluid dynamic calculation and the heat transfer calculation. These two sets of calculations would then independently produce heat transfer rates at the interface for both the fluid and the solid which would not match. Thus, equation (1.1) would be satisfied but not equation (1.2). A second specification for the temperature boundary condition would then produce a second set of heat transfer rates. The relative difference in the heat transfer rates resulting from the two initial temperature specifications could then be used to specify a third temperature boundary condition establishing an iterative procedure where the boundary temperature is systematically changed until the heat transfer rates match. For unsteady three-dimensional fluid dynamic calculations this would require a lot of computational time.

Fully uncoupled calculation:

Because of these considerations most investigators have analyzed the problem in a decoupled or loosely coupled fashion. This is done using a convection heat transfer coefficient H to model the heat transfer rate at the wall where the heat flux at the surface of the wall is given using a combination of Fourier's conduction law for the solid material of the wall, and Newtons law of convection for the fluid just outside the wall. This condition is given by Incropera and Dewitt [6] as:

$$q_s = -k_s \frac{\partial T_s}{\partial y} = -k_w \frac{\partial T_w}{\partial y} = H(T_\infty - T_s) \quad (1.3)$$

This equation is unique in that it allows the heat transfer rate to be determined using a distributed temperature inside a solid medium without knowing the distributed temperature of the flow field

around it. Equation (1.3) is ideally used when the free stream fluid temperature T_∞ , the convective heat transfer coefficient H , and the wall temperature T_s are constant.

One Dimensional Steady Uncoupled Calculation:

Equation (1.3) has been extended to be used for steady compressible flows that are common in aerospace applications where there can be substantial variations in the flow properties along the free stream which is just outside of the viscous layer. Good examples of this are given by Bird, Stewart and Lightfoot [7], and Anderson [8]. In these applications we can write the free stream fluid temperature, pressure, and velocity as: $T_e = T_e(x)$, $p_e = p_e(x)$, and $u_e = u_e(x)$ where x is the distance along the streamline. These in turn are used to define a local Reynolds number, Prandtl number and Mach number given by:

$$Re = \frac{\rho_e u_e L}{\mu_e} \quad (1.4)$$

$$Pr = \frac{\mu_e C_p}{k_e} \quad (1.5)$$

$$M = \frac{u_e}{C_e} \quad (1.6)$$

Where the speed of sound in equation (1.6) for a thermally perfect gas is given as:

$$C_e = \sqrt{\gamma R T_e} \quad (1.7)$$

For these applications the length scale L in equation (1.4) is sometimes taken to be the convective distance along a streamline but should be taken as the distance across the viscous part of the flow where high shear stress and cross stream diffusion would occur. In steady flow applications where we have the growth of an unconfined boundary layer the diffusion length scale is normally the boundary layer thickness which is directly coupled to the convective length scale so the choice of L could be either, but in the RDE application this is not the case. For the RDE we could use either the hydraulic diameter, which is twice the gap distance between the inner and outer containment wall, or we would use the gap distance itself because it is the smaller of the two. The dimension chosen should be the dimension at which the largest turbulent eddies would exist because this provides the length and time scales where cross-stream diffusion would occur. The distinction between these length scales is important.

Using the free stream conditions stated above, an adiabatic wall temperature is calculated which is approximately equal to the temperature that the fluid would have in the viscous region near an insulated wall where the kinetic energy of the gas flow has been converted to thermal energy by viscous forces. This is written as:

$$T_{aw} = T_e + \gamma \frac{u_e^2}{2C_p} = T_e \left(1 + \frac{\gamma-1}{2} M_e^2 \right) \quad (1.8)$$

With the adiabatic wall temperature, the heat transfer at the wall for a steady compressible flow with variable free stream properties can now be calculated using the dimensionless Stanton number which can also be defined in terms of a local convection heat transfer coefficient $H(x)$ as:

$$C_H = \frac{q_w}{\rho_e u_e c_p (T_{aw} - T_w)} = \frac{H(x)}{\rho_e u_e c_p} \quad (1.9)$$

The Stanton number contains everything that is needed to calculate the local heat transfer rate for a 2-D steady compressible flow. It is considered to be a local parameter because it depends not only on variable free stream properties, but also on the local convection heat transfer coefficient $H(x)$. Equation (1.9) is easily transformed using equations (1.4) and (1.5) to form a local Nusselt number which is given by:

$$Nu = RePrC_H = \frac{q_w L}{k_e (T_{aw} - T_s)} = \frac{H(x)L}{k_e} \quad (1.10)$$

Assuming that the free stream conditions are known along with the local Stanton number, the heat transfer to the wall can be directly calculated using either equation (1.9) or (1.10). For an incompressible flow the Mach number is small making the adiabatic wall temperature to be approximately equal to the free stream temperature. This leaves equations (1.9) and (1.10) to be essentially in the same form as equation (1.3). Notice that the cross-stream diffusion length scale L in equation (1.10) cancels out because the effects of cross-stream diffusion as they naturally change along the streamline are intrinsically contained in the local heat transfer coefficient $H(x)$, and this must be determined experimentally. The derivation of equation (1.10) implies that we are not free to arbitrarily assign exponents to the various dimensionless groups that are used. If the proper local free stream conditions are known along with the local convection heat transfer coefficient the problem would be completely specified.

As previously mentioned, it is important that the various dimensionless parameters used in the above equations have significant meaning for steady flows of singular length scale. For example, the Reynolds number in equation (1.4) can be thought of as a ratio of time scales. It is the time required for the largest turbulent eddies generated by the steady state mean flow to dissipate their energy into heat at the molecular level by viscous action. In an unsteady flow both the length scale and the time scale of the mean flow changes; so, in this case the Reynolds number effectively has no meaning.

1.3 RDE application:

Fortunately, that although the equations developed above strictly apply to a 1-D steady compressible boundary layer, it may be possible to extend these to be used for the RDE. This is because the most significant unsteadiness of the flow in the combustion anulus is due to the periodic motion of the shock wave. This motion occurs with a time scale that is very much shorter than the rate of temperature change in the containment walls. For a typical RDE having a mean diameter of 143 (mm) and operating with a frequency of 8000 Hz, the wave speed would be 1,144 (m/sec). This gives a time period for the shock wave to move around the anulus of 0.000125 (sec). The rate of temperature change in the outer containment wall during start up is approximately 100 (deg K/sec) which gives a time scale of 0.01 (sec/K). This is 80 times slower than the motion of the shock wave in the reaction anulus and because of this, the unsteady heat transfer calculation can be simplified by assuming quasi-unsteady values of temperature, pressure and velocity which are averaged around the angular direction. It is likely that these averaged values would then change with the same time scale as the conduction heat transfer in the containment wall. Consequently,

several seconds after startup these averaged values will then reach a steady state value along with the outer containment wall, and at this time equations (1.4-1.10) might be useful. During start up, however, it is likely that only equation (1.9) could be useful, and this would implicitly assume that at each axial location $H(x)$ would be constant in time.

Averaging:

In order to obtain values for a quasi-unsteady flow in the combustion annulus we would first model the flow as a 2-D unsteady flow where the dependent variables in the equations are written as functions of the axial distance along the annulus x , the angular distance around the annulus y , and time t . Because the equations are only two-dimensional it would be necessary to include empirical models as source/sink terms for both the momentum and energy loss that occurs because of friction and heat transfer through the walls in the radial direction. Assuming that these empirical models were formulated, the dependent variables resulting from the calculation results would then be averaged over an appropriate conduction time scale, and around the angular dimension producing values of the form:

$$\bar{\xi}_e(x, \bar{t}) = \frac{1}{\bar{t}} \int_{t=0}^{t=\bar{t}} \left[\frac{1}{\pi D} \int_{y=0}^{y=\pi D} \xi_e(x, y, t) dy \right] dt \quad (1.11)$$

Where D is the mean diameter of the annulus and $\bar{t}$ = an appropriate conduction time scale. The averaged dependent variables $\bar{\xi}_e$ would then be the set of values for the mean flow in the annulus, (i.e., $T_e(x, \bar{t})$, $p_e(x, \bar{t})$, and $u_e(x, \bar{t})$). Thus, the unsteadiness due to the periodic motion of the shock wave would be averaged over resulting in quasi-unsteady variables written as functions of only the axial dimension and the averaged time $\bar{t}$. These variables then would reach a steady state value along with the surface temperature of the containment walls according to the equation:

$$\bar{\xi}_e(x) = \lim_{\bar{t} \rightarrow \infty} \bar{\xi}_e(x, \bar{t}) \quad (1.12)$$

When this occurs the dimensionless groups in equations (1.4-1.10) might possibly have meaning. In order to obtain these averaged values, the 2-D unsteady CFD equations must be solved in conjunction with the conduction heat transfer in the containment walls. The boundary conditions would therefore be coupled in a similar way as was discussed for the fully coupled calculations with the difference being that instead of using equations (1.1) and (1.2) in an iterative scheme with a full 3-D unsteady CFD calculation, we would use equations (1.1) and (1.3) with the averaged results of the 2-D quasi-unsteady CFD calculation. This would substantially reduce the computational time required for the CFD calculation and it would establish a direct coupling using local values for the convection heat transfer coefficient. The first step in doing this would be to estimate what the local convection heat transfer coefficient would be.

2.0 Analytic and Numerical Models:

In order to estimate what the local convection heat transfer coefficient would be on the RDE outer body during start-up a one-dimensional finite difference conduction model was developed and validated by comparing its calculation results with two different analytic solutions each incorporating a different set of boundary conditions. The basic equation to solve is the one-dimensional unsteady heat conduction equation given as:

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \quad (2.1)$$

The finite difference approximation to equation (2.1) is given by:

$$T_i^{n+1} - T_i^n = \frac{\alpha(\Delta t)}{(\Delta x)^2} (T_{i+1}^n - 2T_i^n + T_{i-1}^n) \quad (2.2)$$

The solution is a function of both time, (indexed by n), and space, (indexed by i), with the time domain starting at (t = 0.0 sec) and extending to (t = 3.0 sec) which was the time used to collect the experimental data. The space domain is chosen to start on the external outer surface of the RDE outer body at (x = 0 mm) and extends to the inner surface of the RDE outer body at (x=L=10 mm). In the finite difference model and both analytic models a zero temperature gradient boundary condition on the external outer surface of the RDE outer body was observed so that:

$$\frac{\partial T(0,t)}{\partial x} = 0 \quad (2.3)$$

2.1 Solutions with constant temperature at (x=L):

Analytic solutions to equation (2.1) depend greatly on the boundary conditions which generally act as constraints on the solution so for the first analytic solution we impose a specified temperature at x = L where the boundary condition is given by:

$$T(L, t) = T_s = \text{constant} \quad (2.4)$$

The solution to equation (2.1) subject to the boundary conditions (2.3) and (2.4) is given by Arpaci [9] as:

$$\frac{T(x,t) - T_s}{T_i - T_s} = 2 \sum_{n=0}^{\infty} \left(\frac{-1^n}{\lambda_n L} \right) (e^{-\alpha \lambda_n^2 t}) \cos(\lambda_n x) \quad (2.5)$$

where the characteristic values are given by:

$$\lambda_n L = \pi(2n + 1)/2, \quad n = 0, 1, 2, 3, \dots, \infty \quad (2.6)$$

Calculation parameters that were used to validate the numerical solution of equation (2.2) using equation (2.5) are listed on Table 2.1 where the parameters chosen are what might be representative of the typical RDE operation. The calculation results for the unsteady temperature distribution at the locations of thermocouple gauges 2 and 4 are shown in Figure 2.1. There is essentially no observable difference between the results of the finite difference calculation and the analytic solution thus showing the accuracy of the finite difference approximation that was used.

	Eqn (2.2)	Eqn (2.5)
T_i	300 K	300 K
T_s	378 K	378 K
T_∞	Not needed	Not needed
K	16.3 (W/m-K)	16.3 (W/m-K)
α	4.2e-6 (m²/sec)	4.2e-6 (m²/sec)
L	0.01 (m)	0.01 (m)

nx	501	501
Δx	2.0E-5 (m)	2.0E-5 (m)
t_{max}	3.0 (sec)	3.0 (sec)
nt	69,300	451
Δt	4.3e-5	6.66e-3
H	Not needed	Not needed
Bi	Not needed	Not needed

Table 2.1: Parameters used in solving equations (2.2) and (2.5) with a constant wall temperature of T_s on the inner surface of the outer body

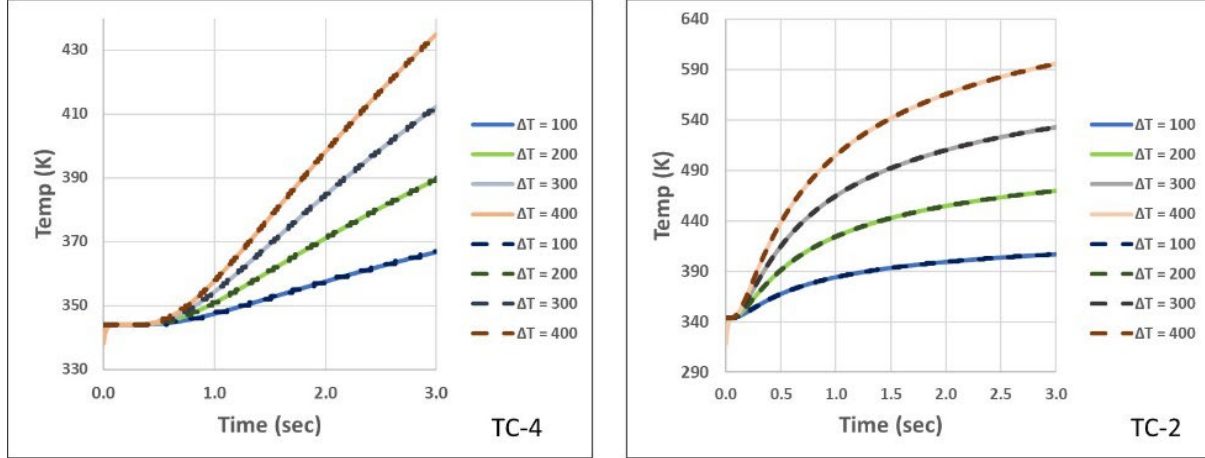


Figure 2.1: Comparison of equation (2.2), and equation (2.5), for a constant boundary temperature T_s on the inside surface of the RDE outer body

2.2 Solutions with constant convection coefficient at ($x=L$):

For the second analytic solution we specify the heat transfer rate on the inside surface of the RDE outer body using a constant convection heat transfer coefficient H and a constant bulk fluid temperature T_∞ by applying Fourier's law of heat conduction along with Newtons law of convection for the hot fluid. The boundary condition at ($x = L$) is thus written:

$$-k \frac{\partial T(L,t)}{\partial x} = H(T_\infty - T_s) \quad (2.7)$$

The analytic solution to equation (2.1) subject to the boundary conditions (2.3) and (2.7) is given by Arpaci, [9] as:

$$\frac{T(x,t)-T_\infty}{T_i-T_\infty} = 2 \sum_{n=0}^{\infty} \left(\frac{\sin(\lambda_n L)}{\lambda_n L + \sin(\lambda_n L) \cos(\lambda_n L)} \right) (e^{-\alpha \lambda_n^2 t}) \cos(\lambda_n x) \quad (2.8)$$

where the characteristic values are given as:

$$(\lambda_n L) \sin(\lambda_n L) = Bi * \cos(\lambda_n x) \quad (2.9)$$

The Biot number Bi used in equation (2.9) is given by:

$$Bi = \frac{HL}{k} \quad (2.10)$$

In equations (2.7) and (2.10) the thermal conductivity k is taken as that of the material of the RDE outer body and is assumed to be constant. For the purpose of code verification we choose a constant hot gas temperature of $T_\infty=1000$ K and with this, equation (2.8) will then generate a solution which intrinsically contains the inside wall surface temperature $T_s(L,t)$ as a function of time, and this is the essential difference between the analytic solutions given by equations (2.5) and (2.8).

	Eqn (2.2)	Eqn (2.8)
T_i	300 K	300 K
T_s	Specified Polynomial	Not needed
T_∞	Not needed	1000 (K)
K	16.3 (W/m-K)	16.3 (W/m-K)
α	4.2e-6 (m2/sec)	4.2e-6 (m2/secs)
L	0.01 (m)	0.01 (m)
nx	501	501
Δx	2.0E-5 (m)	2.0E-5 (m)
t_{max}	3.0 (sec)	3.0 (sec)
nt	69,300	451
Δt	4.3e-5	6.66e-3
H	Not needed	500 (W/m2-K)
Bi	Not needed	0.31

Table 2.2: Parameters used in solving equation (2.2) with a specified unsteady temperature for the inner surface of the RDE outer body and equation (2.8) for a constant heat transfer coefficient.

The solution of equation (2.8) thus generates the unsteady temperature profile for the inner wall of the RDE outer body shown as the red curve 3L/3 in Figure 2.2 and this is fit with a 6th order polynomial which is given as:

$$T_s(L, t) = 306.37 + 94.544 t - 119.59t^2 + 105.51t^3 - 51.553t^4 + 12.784t^5 - 1.2554t^6 \quad (2.11)$$

The polynomial fit can then be used as an unsteady boundary condition for the finite difference calculation given by equation (2.2) and from these results the convection heat transfer coefficient can be directly calculated and compared with the value that was used as an input parameter for the analytic solution of equation (2.8). Calculation results from equations (2.2) and (2.8) are compared at the ($x = L/3$), and ($x = 2L/3$) time traces, as well as on the right-hand side boundary where ($x = L$). Again, for points that are interior to the region there is essentially no observable difference between the finite difference calculation and the analytic solution thus showing the accuracy of the finite difference approximation that was used. The only observable difference is seen on the inner wall boundary where the polynomial oscillates slightly above and below the surface

temperature that was generated by the analytic solution. This important artifact will affect the convective heat transfer calculations as will be shown.

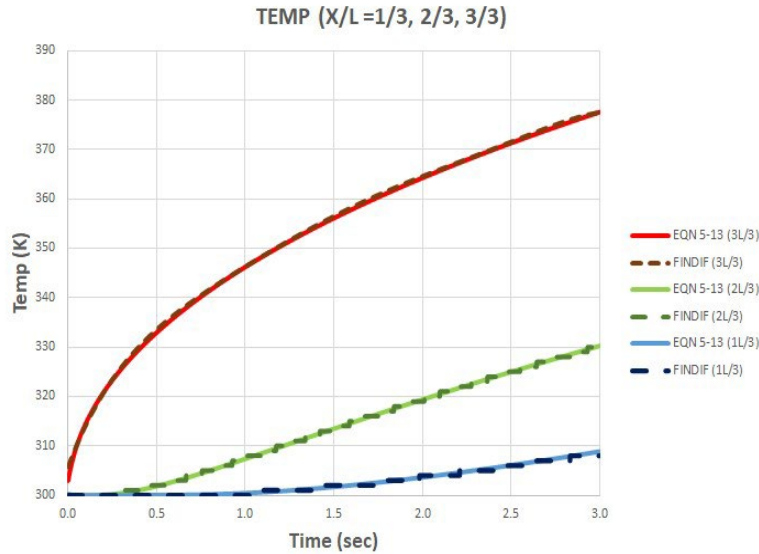


Figure 2.2: Comparison between equation (2.2) and equation (2.8) for the temperature on the inside of the RDE outer body

The numerical results of the solution of equations (2.2) and (2.8) were then used to directly calculate the convective heat transfer coefficient on the inner surface of the RDE outer body and compared. This is shown in Figure 2.4. The red solid curve is the numerical approximation to the convective heat transfer coefficient resulting from taking a finite difference of the temperature gradient from the analytic solution. As expected, this value is very close to the value that was used as an input parameter for this solution. The same numerical approximation to the convective heat transfer coefficient was calculated for the finite difference solution which results in the black dotted line. As can be seen the surface temperature gradient both overshoots and undershoots the same calculation from the analytic solution. This would be expected because the polynomial fit for the temperature boundary condition does the same, and the numerical derivative tends to amplify this behavior. However, the magnitude and average temporal behavior of the calculated value of the convection heat transfer coefficient produced by the finite difference scheme is in very good agreement with the input value to the analytic solution.

Figure 2.3 shows that the convective heat transfer coefficient may be directly calculated from the finite difference approximation providing that the unsteady temperature boundary condition used on the inner wall of the RDE outer body is reasonably well behaved and accurate. This provides the rational for our approach in determining the convective heat transfer coefficient for the inner surface of the RDE outer wall. We will use the finite difference calculation with a range of temperature boundary conditions and simply pick the solutions which match the thermocouple gage data the best. Using these solutions, the convective heat transfer coefficient can be directly calculated using a suitable bulk temperature for the hot fluid.

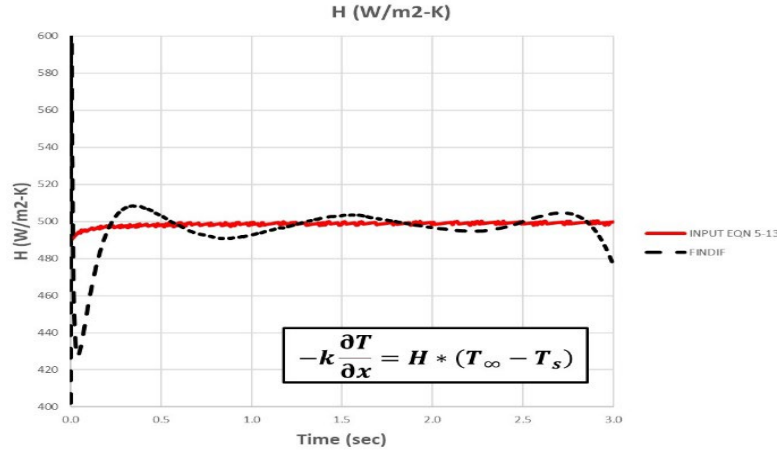


Figure 2.3: Comparison of the numerical calculations of the convective heat transfer coefficient resulting from equations (2.2) and (2.8)

3.0 Calculations:

Constant Temperature BC:

The first set of temperature boundary conditions that were used are shown in Figure 3.1. Calculations were done with four constant elevated temperatures on the inner wall of the RDE outer body. These temperatures were elevated above the initial uniform temperature of the RDE outer body which was taken to be 344.0 (K). For each additional case the inner wall temperature was increased by 100 (K) resulting in values for the ΔT 's of 100 (K), 200 (K), 300 (K), and 400 (K) above the initial starting temperature.

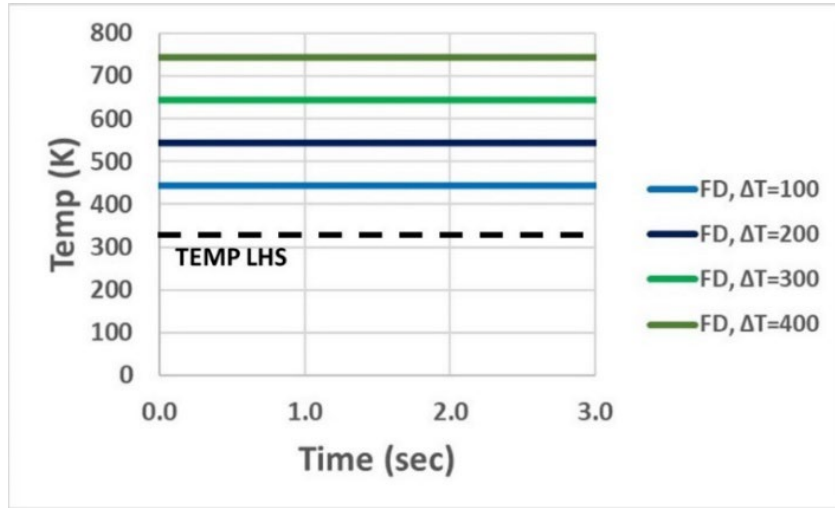


Figure 3.1: Constant temperature boundary conditions for the inner wall of the RDE outer body

The resulting unsteady temperature distributions obtained from the finite difference equation (2.2) at the locations of thermocouples 2 and 4 using these boundary conditions are shown in Figure 3.2 where they are compared with the actual temperature measured by the thermocouple gauges. The

results show that the best fit to the experimental data for these boundary conditions occurs for a ΔT of approximately 100.0 (K). The corresponding convection heat transfer coefficient H was calculated for each case and is shown in Figure 3.3. The calculations of H have assumed a bulk fluid temperature of $T_{\infty} = 1000$ (K).

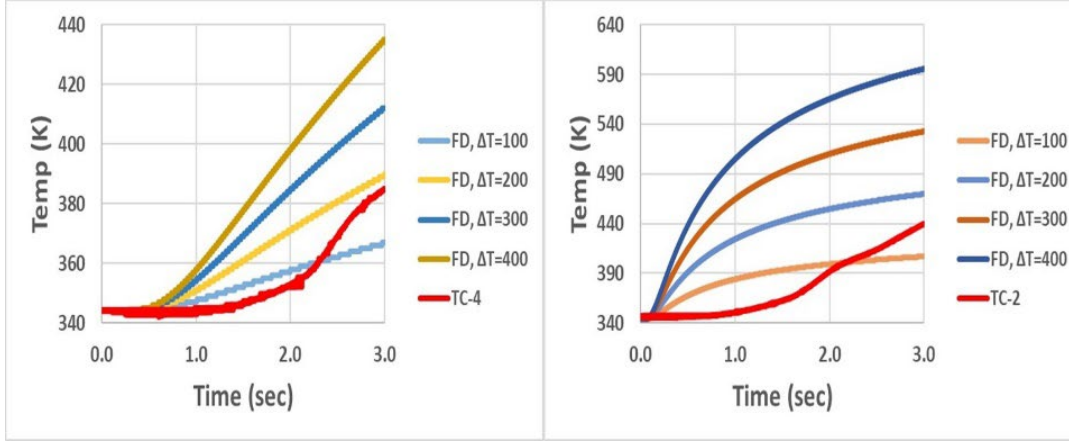


Figure 3.2: Comparison of unsteady temperature calculations using constant temperature boundary condition with thermocouple gauge data

It is noteworthy that in each case H is very high at the start but as time increases the value of H diminishes toward an asymptotically constant value. Physically, if the convection heat transfer coefficient is truly constant in time at any given local axial position, then the results indicate that in each case heat is being transferred to the wall initially much too fast, but as time increases the heat transfer rate diminishes showing the heat transfer coefficient to become more constant. For the best temperature fit to the gauges, (i.e. $\Delta T = 100$), the value of H is calculated to be just below 500 (W/m²-K)

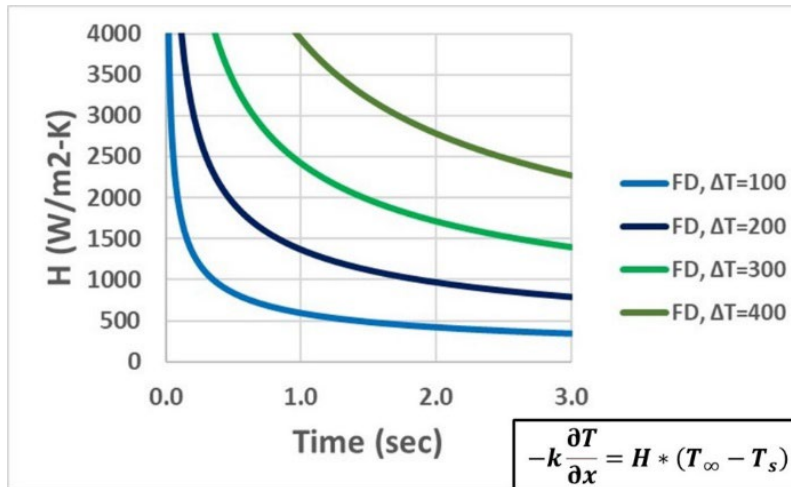


Figure 3.3: Convection heat transfer coefficient for the inner wall of the RDE outer body resulting from four constant temperature boundary conditions

Ramp Temperature BC:

In order to obtain a heat transfer coefficient that mitigates the initial effects shown in Figure 3.3, a more realistic approximation to the unsteady temperature boundary condition on the inner wall is needed. A better guess might be to start with the inner boundary at the initial temperature of 344.0 (K) and increase this linearly with time to a final temperature at the end of the 3 second test time. Each case would therefore have a different value for the total temperature increase. Calculations were thus done with four unsteady linear temperature distributions where the temperature increase at the end of the test time for each case is again given by ΔT 's of 100 (K), 200 (K), 300 (K), and 400 (K). These inner wall boundary conditions are shown in Figure 3.4.

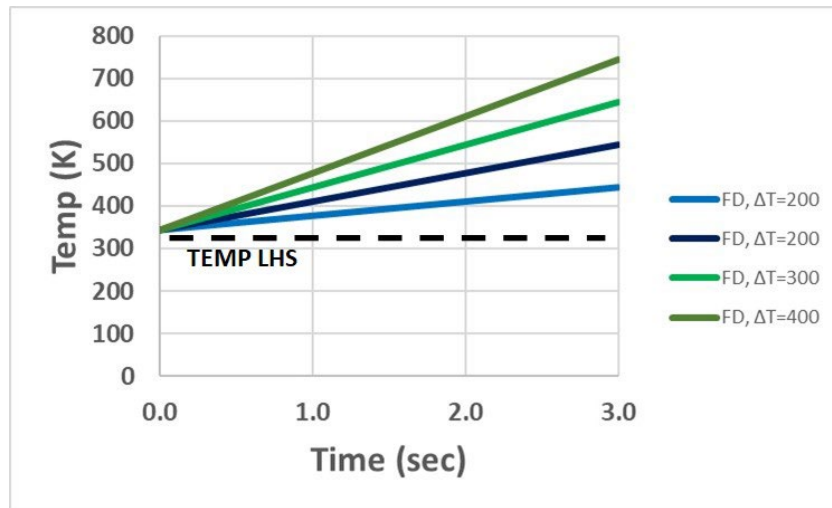


Figure 3.4: Ramp temperature boundary conditions for the inner wall of the RDE outer body

The resulting unsteady temperature distributions obtained from the finite difference equation (2.2) at the locations of thermocouples 2 and 4 using the ramped boundary conditions are shown in Figure 3.5 where they are compared with the actual temperature measured by the thermocouple gauges.

These results show that the best fit to the experimental data for the ramped boundary conditions shown in Figure 3.4 occurs for a ΔT of 400 (K) for thermocouple 4, and a ΔT of 200 (K) for thermocouple 2. The corresponding convection heat transfer coefficient was calculated for each case and is shown in Figure 3.6. It is noteworthy that for all the cases having ramped boundary condition H starts at zero and increases with time. The only calculation to show a slight leveling off from this increase is the one having a ΔT of 100 (K). In contrast to the four constant temperature boundary condition cases above, these cases show the opposite effect. These results indicate that at the initial time heat is not being transferred to the wall fast enough because H starts off with a very low value.

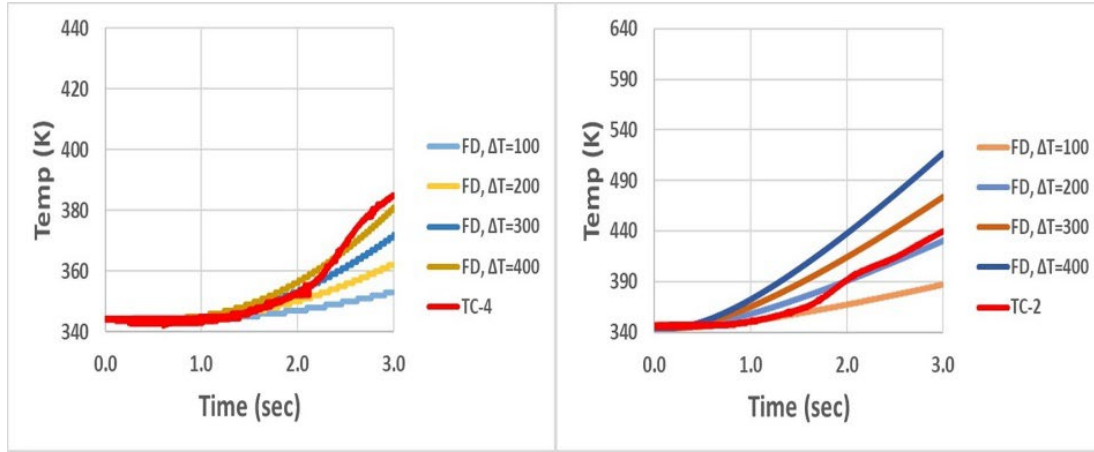


Figure 3.5: Comparison of unsteady temperature calculations with ramp temperature boundary condition with thermocouple gauge data

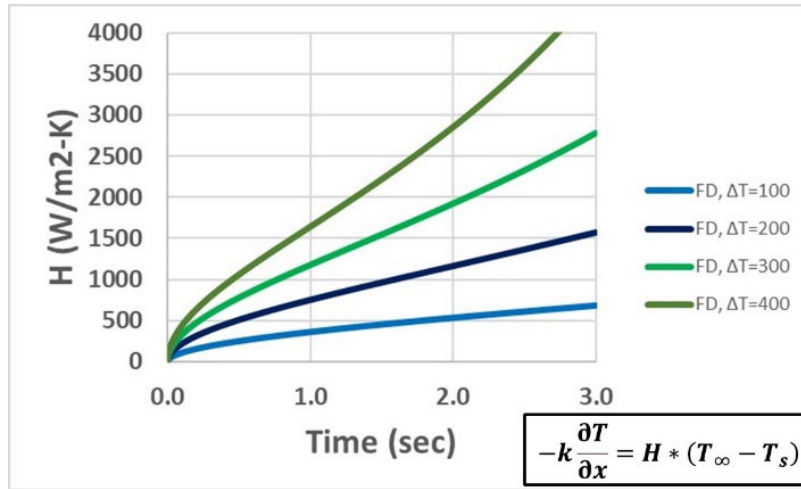


Figure 3.6: Convection heat transfer coefficient for the inner wall of the RDE outer body resulting from ramp temperature boundary conditions

A comparison between the calculation results of the constant temperature BC cases and the ramped temperature BC cases at the locations of thermocouple 4 and 2 is shown in Figures 3.7 and 3.8 respectively. At the location of thermocouple 4 the constant temperature BC cases mostly overshoot the experimental data and the ramp temperature BC cases completely undershoot. This again signifies that initially for the constant temperature BC cases heat is being imparted to the RDE outer body too fast, and for the ramped temperature BC cases heat is being imparted too slowly.

As seen in Figure 3.8 at the location of thermocouple 2 the constant temperature BC cases mostly overshoot the experimental data with the calculation for a ΔT of 100 (K) being the best fit to the data. The shape of the calculated curves here is notably quite different than the shape of the experimental data. The ramped temperature BC calculations produce curves having a shape that is more like the experimental data for thermocouple 4. The best fit to all the data is given for a ΔT between 100 and 200 (K).

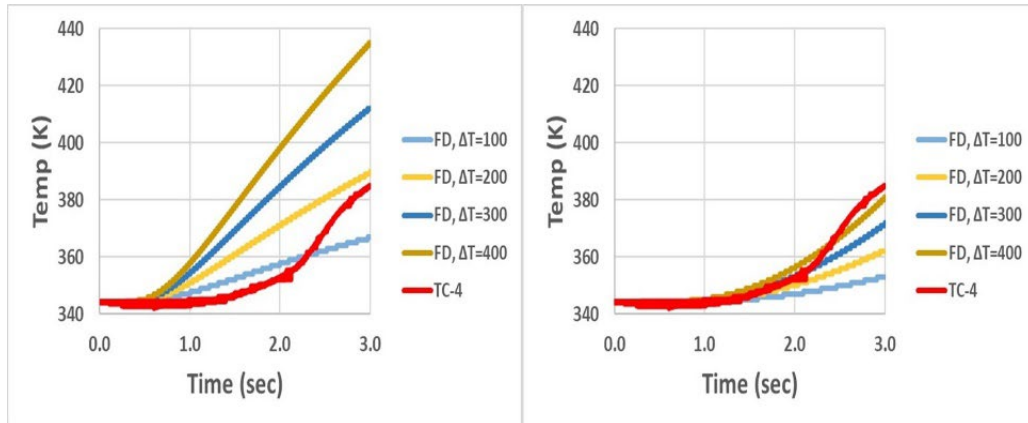


Figure 3.7: Comparison of unsteady temperature calculations with constant temperature BC's (left) and ramped temperature BCs (right) at thermocouple 4

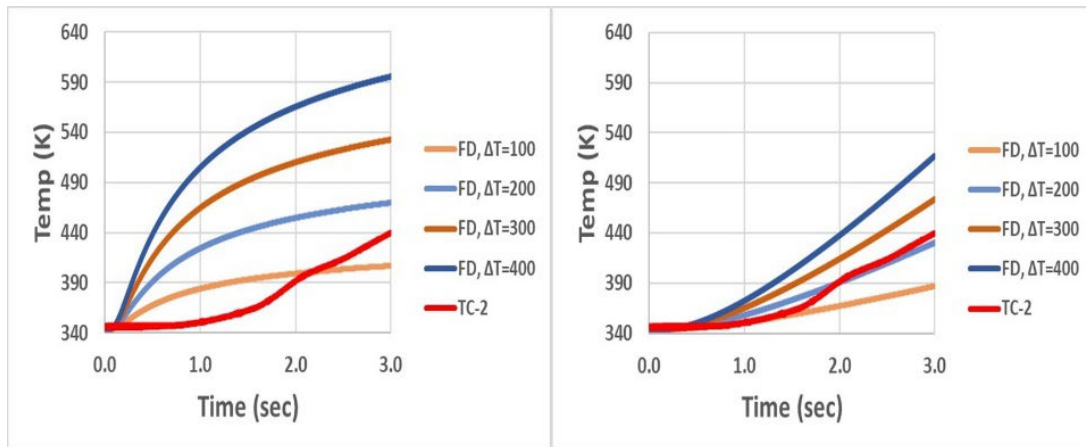


Figure 3.8: Comparison of unsteady temperature calculations with constant temperature BC's and ramp temperature BCs at thermocouple 2

A direct comparison of the heat transfer coefficients that were calculated for the cases with constant temperature BC's and the cases with ramped temperature BC's is shown in Figure 3.9. The heat transfer coefficient is normally assumed to be constant in time for all axial locations but show variation along the axis. Because this coefficient decreases with time for all of the constant temperature BC cases and increases in time for all the ramped BC cases indicates that for each different ΔT the unsteady temperature distribution on the boundary that would produce a constant H would be some hybrid combination of the two.

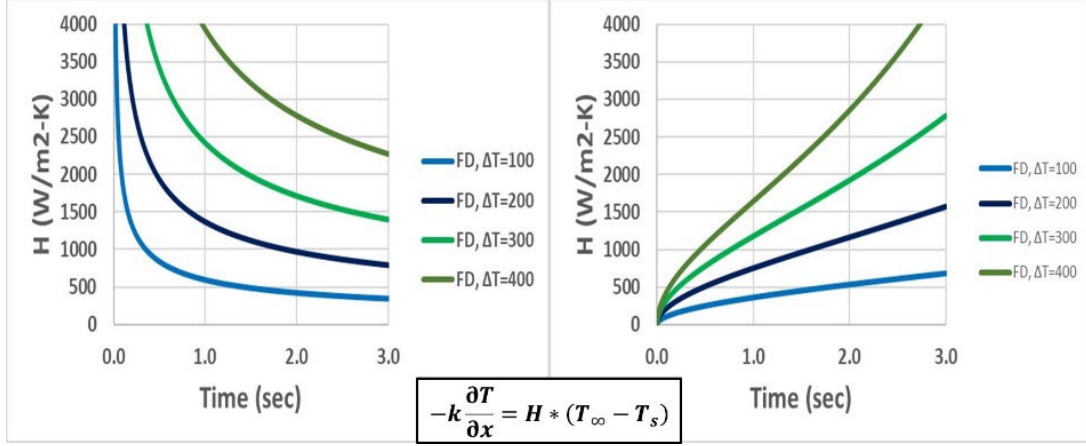


Figure 3.9: Comparison of the convection heat transfer coefficient for the inner wall of the RDE outer body resulting from constant temperature BC's and ramp temperature BC's

Hybrid Variable Temperature BC:

After analysis of the results for both the constant temperature BC cases and the ramped temperature BC cases a hybrid variable unsteady temperature BC is formulated for the inner wall that would more closely represent the actual temperature rise on the boundary and thus be more likely to generate a constant convective heat transfer coefficient. To do this a parabolic curve is generated over the region ($0 \leq \xi' \leq 1$) using the parameter $\bar{\eta} = 0.4$ according to the equation:

$$\eta' = 4\bar{\eta}\xi'(1 - \xi') \quad (3.1)$$

The parabola is then rotated by an angle θ generating a curve relating the normalized time to the normalized temperature profile.

$$\frac{(t-t_{min})}{(t_{max}-t_{min})} = +\xi' \cos\theta - \eta' \sin\theta \quad (3.2)$$

$$\frac{(T-T_{min})}{(T_{max}-T_{min})} = -\xi' \cos\theta + \eta' \sin\theta \quad (3.3)$$

Equations (3.2) and (3.3) generate the hybrid unsteady temperature BC on the inner surface using only two parameters, (i.e., $\bar{\eta}$ and θ), instead of the complex 6th order polynomial fit which produces a slow oscillation. When this curve is mapped into four different temperature rises on the inner wall boundary using different values for the temperature increase, (i.e., $\Delta T = T_{max} - T_{min} = 100$ (K), 200 (K), 300 (K) and 400 (K)), and using the 3 second time interval of the experimental data, the unsteady temperature distributions on the inner wall of the RDE outer body are shown in Figure 3.10.

Using these boundary conditions the resulting unsteady temperature distributions within the wall obtained from the finite difference equation (2.2) at the locations of thermocouples 4 and 2 are shown in Figure 3.11 where they are compared with the actual temperature measured by the thermocouple gauges.

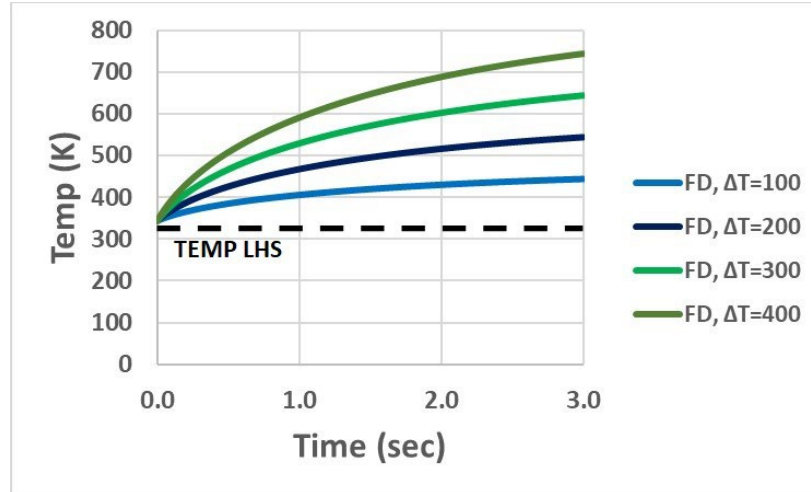


Figure 3.10: Hybrid temperature BCs for the inner wall of the RDE outer body

It is clearly seen that the hybrid temperature BC's produce better results for both thermocouples than either the constant temperature BC's or the ramped temperature BC's. The best fit to the data for these cases would be a ΔT of approximately 200 (K) for thermocouple 4, and a ΔT of approximately 100 (K) for thermocouple 2. The overall best fit would therefore be produced using a ΔT of approximately 150 (K).

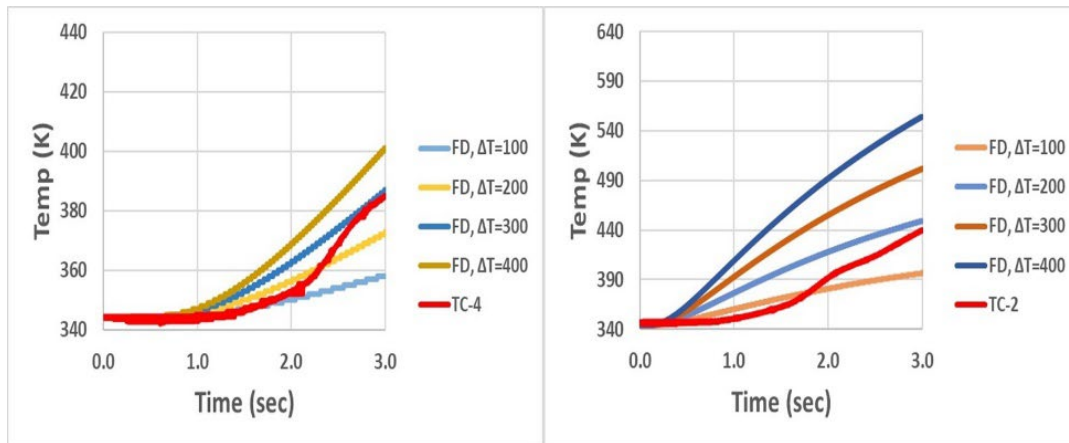


Figure 3.11: Comparison of unsteady temperature calculations with hybrid temperature boundary condition with thermocouple gauge data

Calculations of the resulting convection heat transfer coefficient for the four ΔT 's are shown in Figure 3.12. The most obvious thing that is observed is that for each ΔT the convective heat transfer coefficient achieves an approximately constant value very quickly within the first second. This signifies that the shape of the unsteady temperature boundary condition is much closer to physical reality than either the constant temperature boundary condition or the ramped temperature boundary condition. It can also be seen from Figure 3.12 that if the best fit for the temperature data, i.e., $\Delta T = 150$ (K), would be used in the calculations it would produce a convection heat transfer coefficient certainly less than 1000 ($\text{W}/\text{m}^2\text{-K}$) with a good estimation of approximately

800 (W/m²-K). But these calculations were carried out assuming a gas temperature of $T_{\infty} = 1000$ (K).

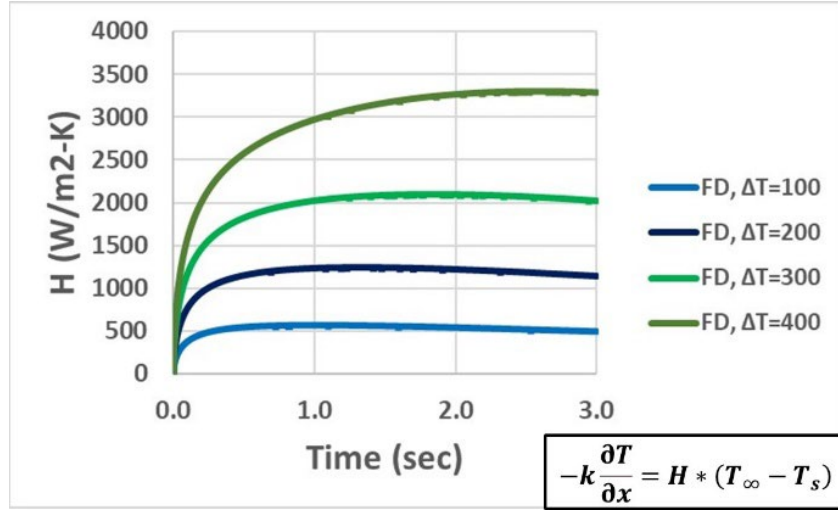


Figure 3.12: Convection heat transfer coefficient for the inner wall of the RDE outer body resulting from hybrid temperature boundary conditions

The calculations produced by Randall et.al. [2] show three different gas temperatures that might be used to estimate the temperature of the hot gas in the annulus ranging from 50%, 75% and 100% of the detonation temperature. These were cited as $T_{\infty} = 1550$ (K), $T_{\infty} = 2300$ (K), and 3100 (K) respectively. When these temperatures are used in the calculation of the convection heat transfer coefficient the results are shown in Figure 3.13.

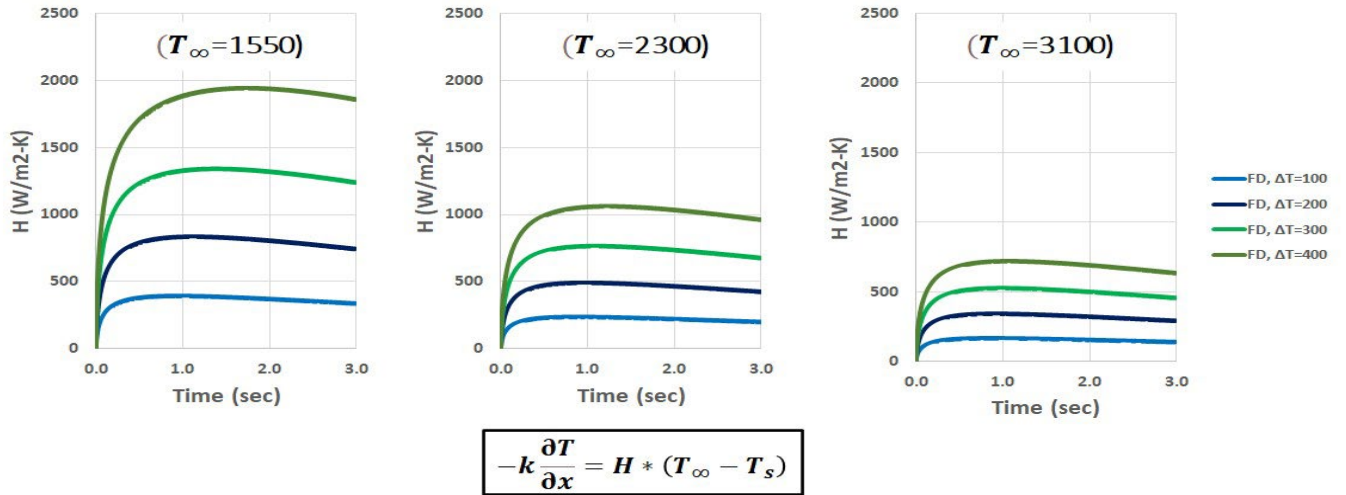


Figure 3.13: Convection heat transfer coefficient for the inner wall of the RDE outer body resulting from hybrid temperature boundary conditions and different free stream temperatures

The hottest portion of the combustion region would occur at the inlet of the annular combustion chamber which is shown on the left side in Figure 1.2 to be near row 1. The placement of thermocouples 4 and 2 were in row 4 which is downstream of the hottest portion of the reaction

zone so the flow there would be cooler. If the bulk temperature of the hot gas at this point is only 50% of the detonation temperature, then the convective heat transfer coefficient that is calculated using the hybrid unsteady temperature BC with a ΔT of approximately 150 (K) would be between 400 and 750 (W/m²-K). However, if the bulk temperature of the hot gas at this point is 75% of the detonation temperature, then the convection heat transfer coefficient would be between 250 and 450 (W/m²-K). Lastly, If the hot gas retains most of its detonation temperature, then the convective heat transfer coefficient would be between 100 and 250 (W/m²-K). These results show that it is possible that the convective heat transfer coefficient for RDE's may be consistent with long standing conventional data for the forced convection of a gas which has been reported by many authors over time. [10] Ranges of data for this are summarized in Figure 3.14 where the Biot number is plotted as a function of the convective heat transfer coefficient for a series of wall thicknesses.

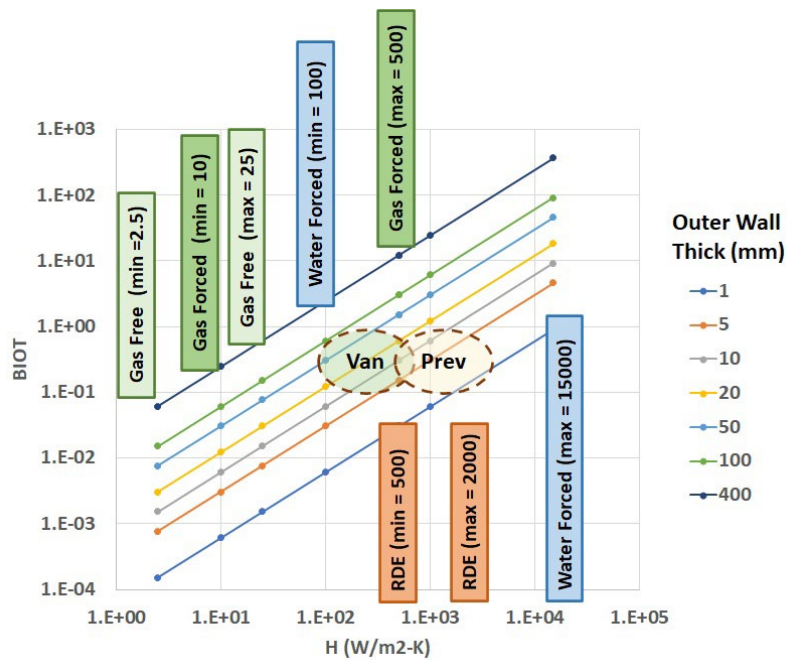


Figure 3.14 Biot Number vs. Convection Heat Transfer Coefficient for Various RDE Outer Wall Thicknesses

For the 10 (mm) thickness of the RDE outer body with a hot gas temperature of 75% of its detonation temperature the calculated convection heat transfer coefficient falls within the normal range of 250-450 (W/m²-K). This range is loosely marked as VAN in Figure 3.14. The upper end of this range is near the 500 (W/m²-K) maximum for the forced convection of a gas and is significantly lower than what has been reported in previous work. Previous estimates are loosely marked as PREV in Figure 3.14. With an RDE wall thickness of (10 mm) the Biot number for this is greater than 0.1 signifying that a thermal gradient will certainly exist in the outer wall as the experimental data and calculations bear this out.

Better calculations are always needed but if the calculations presented here are reasonably accurate then an assumption can be made that the mere presence of shock waves moving around the anulus

of the RDE might not be a sufficient reason to make the convection heat transfer coefficient significantly higher than it would be for the normal forced convection of a gas. This is simply because the shock wave only propagates through the relatively slow-moving gas changing its bulk properties as it passes in the angular direction, but it does not significantly stir the fluid or alter or increase the cross-stream diffusion. If this is the case, then the convection heat transfer coefficient should retain a more conventional value.

4.0 Summary and Conclusions:

Calculations of the convection heat transfer coefficient for the inner surface of the RDE outer body were done using a finite difference approximation to the one-dimensional unsteady conduction heat transfer equation. The finite difference calculations were validated using two different analytical solutions each having a different mathematical form, and each having different boundary conditions. Finite difference calculations were then carried out using several sets of unsteady temperature boundary conditions in order to see which set of boundary conditions best matched the experimental temperature data of Randell et al. [2], and also which set of boundary conditions produced a convection heat transfer coefficient that was constant in time. From these results the local convective heat transfer coefficient on the inner wall at the location of Row 4 was approximately determined. The results show that the convective heat transfer coefficient might probably be within the normal range which is typical for most forced convection gas flows.

In order to improve the calculation results more experimental data is needed. Data should be taken over a longer period of time so that an asymptotic steady temperature can be achieved within the containment walls. This should be done perhaps with wall cooling. Care must be taken when this data is generated so that the initial starting temperature is the same for all runs. The original data showed some variation in this initial temperature which made it difficult to directly compare data from different runs. Multiple measurements at multiple depths in each row of the experimental apparatus should be taken. This would allow for the axial variation of the local convective heat transfer coefficient to be determined.

There is also a need for 2-D unsteady CFD calculations with appropriate source/sink terms added to the equations in order to model the momentum and energy loss due to friction and radial heat transfer. This calculation should then be averaged according to the methods presented here in order to produce the quasi-unsteady gas temperatures, velocities and pressures as functions of time and the axial distance along the annulus. Having the axial distribution of these averaged values would allow a direct coupling of the gas flow to the conduction heat transfer calculation in the containment wall using equation (3). This calculation would have to be done according to the iterative procedure discussed in this work in order to properly couple the conduction heat transfer in the walls with the hot gas in the annulus to make both the temperature and the heat transfer rate on the interface boundary match. In the absence of experimental data which would directly measure the average properties of the gas flow, a converged calculation of this kind might be the only way to obtain the local convective heat transfer coefficient, the local Nusselt number and the local Stanton number.

5.0 References and Citations:

1. Johnson May, Kristyn B., "Combustion Feature Characterization using Computer Vision Diagnostics within Rotating Detonation Combustors" (2022). Graduate Theses, Dissertations, and Problem Reports. 11216.
<https://researchrepository.wvu.edu/etd/11216>
2. Steven Randall, Andrew St. George, Robert Driscoll, Vijay Anand, and Ephraim J. Gutmark. 2015. "Numerical and Experimental Study of Heat Transfer in a Rotating Detonation Engine." AIAA 2015-0880
3. Arnab Roy, Clinton Bedick, Peter Strakey, Todd Sidwell, Don Ferguson, Andrew Sisler, Andrew Nix. 2015. "Development of a Three-dimensional Transient Wall Heat Transfer Model of a Rotating Detonation Combustor,"
4. Arnab Roy, Peter Strakey, Todd Sidwell, Don Ferguson, 2015, "Unsteady Heat Transfer Analysis to Predict Combustor Wall Temperature in Rotating Detonation Engine," AIAA 2015-4191
5. Fourier, J. (1955), "The Analytical Theory of Heat", (Translation by A. Freeman). Dover Publications, Inc., New York.
6. Incropera, Frank P. and DeWitt, David P., "Fundamentals of Heat Transfer", John Wiley & Sons, New York, 1981
7. Bird, Byron R., Stewart, Warren E., Lightfoot, Edwin N., "Transport Phenomena", 2nd Edition, New York, John Wiley & Sons, Inc.
8. Anderson, John D., Hypersonic and High Temperature Gas Dynamics, New York, McGraw Hill Book Company, 1989
9. Arpaci, Vedat S. Conduction Heat Transfer, Reading Massachusetts, Addison-Wesley Publishing Company, 1966.
10. Arpaci, Vedat S. Conduction Heat Transfer, Reading Massachusetts, Addison-Wesley Publishing Company, 1966.