SECTION 7

NUMERICAL METHODS
CONTROL OF FLOW SEPARATION ON A HUMP MODEL USING A DIELECTRIC BARRIER DISCHARGE PLASMA ACTUATOR

Mahdi Hasan
North Carolina A&T State University
Greensboro, North Carolina, USA

Michael Atkinson
North Carolina A&T State University
Greensboro, North Carolina, USA

ABSTRACT
A numerical investigation was carried out to investigate the effect of a dielectric barrier discharge (DBD) plasma actuator to control flow separation over a hump at a Reynolds Number of 936,000, based on chord length, and a freestream Mach number of 0.1. In this study, three-dimensional Partially Averaged, Navier-Stokes (PANS) calculations were carried out to simulate the experiments. The baseline code, CALC-LES, developed by Chalmers University of Technology was modified to include the effects of the plasma actuator [1]. The phenomenological DBD plasma actuator was modeled as a steady-state, constant spanwise source term in the momentum equations. The simulations were carried out for both a baseline (no control) and a plasma control case. Comparisons of time-averaged skin friction, coefficient of pressure, and velocity profiles showed good agreement with the experiment for the baseline case. The control case showed that the separation was removed, giving improved control characteristics compared to the previous numerical investigations with similar actuator model.

KEY WORDS: Plasma actuator, flow control, hybrid turbulence modeling, numerical simulation, baseline code

INTRODUCTION
Interest in plasma-based flow control is motivated by its ability to reduce or eliminate flow separation, maintain a low-profile when not active, and operate under varying frequencies to adjust for flow unsteadiness. The potential for plasma actuators to improve the performance of aerospace vehicles has been explored through experimental and computational fluid dynamic (CFD) investigations [2, 3]. However, accurate prediction of separated flow using CFD remains challenging. Flow separation is an unwanted phenomenon for aerodynamic applications, as it results in poor aerodynamic performance (lift, drag etc.).

The effects of flow separation on aerodynamic performance has been studied in numerous works [4-6]. Effective control of flow separation has shown to provide drag reduction and lift augmentation through the redistribution of boundary layer momentum from high-to-low regions [2]. This is achieved by the formation of streamwise counter-rotating vortices [2, 3], and can operate as either in a passive or active mode [7]. The most common passive flow control devices are vortex generators, turbulators, zigzag tapes, flaps, and surface dimples. Whereas, plasma actuators are an active flow control device that require an energy source and often have feedback control loops. A plasma is a quasi-neutral gas of charged and neutral particles which exhibits collective behavior [8]. Active flow control devices (i.e. plasma actuators) are advantageous compared to passive devices in which they have a minimal impact on aerodynamic performance when not in operation [9]. Additional benefits of plasma actuators include: no moving parts, operation in active modes, ease in switching (i.e. on/off) and low maintenance cost [10].

Different types of plasma actuators are used to control fluidflow: glow discharge; dielectric barrier discharge; magnetically accelerated surface discharge are some of the most popular types. DBD actuators are among the most promising plasma actuators for active flow control at low-speed applications. There have been many publications showing effective use of DBD plasmas on airfoils, hump models, bluff bodies, compressors, rearward facing steps, and flat plates. Suzen et al. [11] studied the effect of a DBD plasma actuator on PAK-B blade flow separation. Khoshkoo et al. [12] and Gaitonde et al. [13] used DBD actuators to control flow separation of a NACA 0015 airfoil at a high angle of attack. Suzen et al. [14] simulated low-pressure turbine flow with large flow separation in a quiescent environment. Corke et al. [15] showed that DBD plasma actuators can effectively reduce drag and increase lift. Rizzetta et al. studied the effects of DBD plasma-based control on turbulent separation on a rear-facing ramp [16].

A DBD plasma actuator consists of two electrodes separated by a dielectric material that works as an insulator. The electrodes are typically asymmetrical, with one electrode exposed to the freestream air. The region exposed to the open electrode becomes ionized when a high voltage and frequency is applied. This ionized gas (plasma), is directed towards the covered electrode. In the presence of the electric field, the ions produce a body force due to momentum transfer to the flow close to the wall surface. This causes a suction above the exposed electrode and works similarly like a wall jet. Enloe et al. [17] provided the details of
The schematic of a classical DBD plasma actuator is shown in Figure 1. Plasma actuators can operate in a steady or unsteady (pulsed) mode. A steady mode is the one where the body force is produced continuously, whereas in an unsteady mode, the actuator can be pulsed at a selected frequency. Corke et al. [18] showed that there is no appreciable temperature difference between the steady and unsteady actuator operations.

In the case of a DBD plasma actuator, two approaches are commonly used for numerical simulation: 1) first principle-based model and; 2) phenomenological models [12]. Although the former is more accurate in capturing the details of the flow physics, the latter approach is more popular due to its simplicity and amenable performance. However, many researchers have studied the first principle-based actuator model [19, 20]. A simplified model of DBD plasma actuator is presented by Shyy et al. [21] and known as the Shyy model. Shyy proposed that the electric field lines produced by the electrodes shown in Fig 1. can be simplified and taken as a linearized pattern, shown in Figure 2. The advantages of the Shyy model include: quick response, low cost computation and good accuracy compared with other numerical methods [12]. The details of this model are presented in the next section.

In this investigation we numerically simulated two cases: 1) baseline (no control); 2) control with a DBD plasma actuator. The test article was the NASA wall-mounted hump model. This model was designed to assess the ability of turbulence models to predict separation and reattachment in boundary layer flow due to an adverse pressure gradient. The computations were completed using CALC-LES, an incompressible CFD code developed by Davidson, coupled to a hybrid (RANS/LES) turbulence closure model termed PANS [22], where RANS is the Reynolds’ Average Navier Stokes, LES is Large Eddy Simulation and PANS is Partially Averaged Navier Stokes. Details of the CFD code and turbulence model are described by Davidson in [23, 24]. CALC-LES was modified to simulate with plasma actuator. The simplified DBD plasma actuator has been placed on the hump based on the proposed Shyy model to investigate the effects of plasma body force onto the flow separation. The computational methodology and the corresponding results are presented in the subsequent sections.

NUMERICAL METHOD

CALC-LES is an incompressible finite volume Navier Stokes solver. Spatial discretizations were carried out using a hybrid central-upwind scheme and temporal discretization was achieved using the Crank-Nicolson scheme [24]. A multigrid Poisson solver is used to calculate pressure. The baseline code was modified to include the effects of the plasma by adding source terms to the x and y components of the momentum equation and was modeled as a non-thermal plasma. Moreover, both Gaitonde et al. [13] and Khoshkhoo et al. [12] stated that the plasma effect in energy equation can be neglected because the work done by the plasma is very small. The incompressible Navier stokes equations with the plasma body force are shown below:

\[
\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} = 0
\]

\[
\frac{\partial u'}{\partial x} + u\frac{\partial u'}{\partial x} + v\frac{\partial u'}{\partial y} + w\frac{\partial u'}{\partial z} - \frac{1}{Re} \left( \frac{\partial^2 u'}{\partial x^2} + \frac{\partial^2 u'}{\partial y^2} + \frac{\partial^2 u'}{\partial z^2} \right) = F_{x,\text{plasma}}
\]

\[
\frac{\partial v'}{\partial x} + u\frac{\partial v'}{\partial x} + v\frac{\partial v'}{\partial y} + w\frac{\partial v'}{\partial z} - \frac{1}{Re} \left( \frac{\partial^2 v'}{\partial x^2} + \frac{\partial^2 v'}{\partial y^2} + \frac{\partial^2 v'}{\partial z^2} \right) = F_{y,\text{plasma}}
\]

\[
\frac{\partial w'}{\partial x} + u\frac{\partial w'}{\partial x} + v\frac{\partial w'}{\partial y} + w\frac{\partial w'}{\partial z} - \frac{1}{Re} \left( \frac{\partial^2 w'}{\partial x^2} + \frac{\partial^2 w'}{\partial y^2} + \frac{\partial^2 w'}{\partial z^2} \right) = F_{z,\text{plasma}}
\]

Equations (1)-(4) are the non-dimensional incompressible Navier Stokes equations. Equation (1) represents the continuity equation and equations (2), (3) and (4) represent the x, y and z components of momentum equations. u, v and w are the velocity components, \( \rho \) is the density, \( p \) is the pressure and \( Re \) is the Reynolds number. The right-hand side of the momentum equation indicates the body force produced by the plasma. These plasma forces are expressed as,

\[
F_{\text{plasma}} = D_c qE
\]

where, \( E = (E_x, E_y, E_z) \) is the electric field vector and \( q \) is the charge density. The parameter \( D_c \) is the non-dimensional
parameter used to set the magnitude of the body force. It can also be represented as a scaling of the electrical to inertial forces [13].

\[ D_c = \frac{\rho_c e E_{	ext{ref}} L_{	ext{ref}}}{\rho_{	ext{ref}} U_{	ext{ref}}^2} \]  

(6)

where, \( \rho_c \) is the charge density, which is constant.

**PLASMA ACTUATOR MODEL**

The phenomenological DBD plasma model proposed by Shyy [21] was used for this work. The plasma is assumed to be non-thermal and steady. Figure 2 shows the plasma effect in a linearized distribution shown in the triangular region AOB. Outside of this region, the Electric field \( E \) is considered to be zero, thus producing zero body force. The linear variation of \( E \) in AOB region is formulated as:

\[ |E| = E_0 - k_1 x - k_2 x \]  

(7)

\( E_0 \) is the maximum electric field strength, close to the region near the two electrodes. The constants \( k_1 \) and \( k_2 \) can be found from the boundary values of plasma fluid, considering a linear variation in the electric field.

\[ k_1 = \frac{E_0 - E_b}{b} \]  

(8)

\[ k_2 = \frac{E_0 - E_b}{a} \]  

(9)

where, \( a(OA) \) and \( b(OB) \) the height and width of the plasma, respectively. Point A and B are the breakdown points with breakdown voltage \( E_b \), beyond which there is no plasma effect. The electric field components are calculated as,

\[ E_x = \frac{E k_2}{\sqrt{(k_1^2 + k_2^2)}} \]  

(10)

\[ E_y = \frac{E k_1}{\sqrt{(k_1^2 + k_2^2)}} \]  

(11)

In this effective plasma region, the active body force generated can be determined using the following relation:

\[ \vec{F'} = D_c \theta \Delta t \alpha \rho \vec{E} \delta \]  

(12)

where, \( \theta \) is the input voltage frequency, \( \Delta t \) is the time for plasma discharge, \( \alpha \) is the factor for collision efficiency, and \( \delta \) is the restriction function. The electric field strength is formulated in a way such that the plasma force reduces with an increase in both \( x \) and \( y \) position.

The parameter used in the Shyy model is similar to the experimental values described in ref [25]. The value of \( \theta \) is 5 kHz, the charge density \( \rho_c \) is \( 10^{11} \) cm\(^{-3} \), and the discharge time \( \Delta t \) is 67 \( \mu \)s. The length and height of triangular area (i.e. the plasma affected region) are \( a = 0.018 \) and \( b = 0.024 \). Both the length and height are normalized by the hump chord length \( c \). The nondimensional parameter \( D_c \) is chosen to be 110 based on \( D_c \) formulation stated in equation 6 and the nondimensionalization of electric field strength, \( E \) [26].

**COMPUTATIONAL DOMAIN**

The phenomenological/simplified Shyy model is a two-dimensional formulation. Although, this is nominally a two-dimensional flow field, the computational domain used in this study is three dimensional with z-length (spanwise) of 0.2. The grid was taken from the ATAAC project website, created by Dr. Lars Davidson, Chalmers University of Technology [27]. The domain and the grids on the \( x-y \) plane are shown below in figures 3 and 4. Figure 3 shows the whole entire domain; however, for this study the grid was modified to reduce the computational cost. Figure 4 shows the computational mesh for the current simulations. Here the results are focused on the flow field just upstream of the incipient separation.
NUMERICAL RESULTS OF BASELINE CASE

In this section we will compare the baseline computational results to the Greenblatt experimental results [2]. The purpose of the baseline computation is to assess the ability of CALC-LES to accurately predict the baseline flowfield. The time step for the baseline case is 0.002. Time averaged quantities were obtained after completing 7500 iterations (approximately three “flow through” time). The time averaged values were taken over another 7500 time steps.

Figures 5 and 6 show the velocity streamlines and velocity vectors for the baseline case, respectively. It is evident from the figures that there is a recirculation region at the hump’s trailing edge. The recirculation starts at location $x/c = 0.66$ and the flow reattaches at a location close to $x/c = 1.2$.

Moreover, the coefficient of pressure shows the adverse pressure created by the hump and the resulting separation. The coefficient of pressure, denoted by $C_p$, is a non-dimensional parameter of pressure normalized by the dynamic pressure of the free stream and formulated as,

$$ C_p = \frac{P - P_{ref}}{\frac{1}{2} \rho \bar{V}_\infty^2} $$

(13)

Figure 7 shows the pressure coefficient along the hump wall. The figure shows that the numerical solution of $C_p$ has good agreement with the experimental results. The pressure coefficient increases until $x/c = 0.66$, followed by a sharp decrease in $C_p$ at $x/c = 0.66$ which indicates flow separation at that location. Figure 6 also highlights the flow reattachment at the hump’s trailing edge. There is a slight discrepancy between the numerical and the experimental results in the separated region; however, the numerical results accurately capture the reattachment location at $x/c = 1.2$ and show good agreement with the experiment.

Similarly, the coefficient of skin friction also provides additional flowfield information. The skin friction coefficient, as part of the drag force, is created due to the surface friction between the fluid and the solid surface. This non-dimensional parameter is given by:

$$ C_f = \frac{\tau_{wall}}{\frac{1}{2} \rho \bar{V}_\infty^2} $$

(14)

Figure 8 shows the skin friction coefficient for the baseline case is compared with the experimental values. There is very good agreement between the two results, which exhibits the solver’s ability to solve the flow field using hybrid turbulence modeling. The largest discrepancy in this comparison has been found at location $x/c = 0.50$ from to $x/c = 0.66$, where the separation begins.

Time-averaged U-velocity boundary profiles are compared to the experimental results at three different locations: $x/c = 0.8,$
$x/c = 1.0$ and $x/c = 1.2$ and are shown in Figure 9. The extracted U-velocity at $x/c = 0.8$ and $x/c = 1.0$ falls within the separation zone and hence provides a highly accurate numerical prediction of velocity inside the separation zone.

**Figure 9. Time averaged velocity at $x/c = 0.8$, $x/c = 1.0$ and $x/c = 1.2$**

**NUMERICAL RESULTS WITH PLASMA ACTUATOR**

As discussed in the numerical method section, the plasma actuator was modeled as source terms in the momentum equations. These added forces push the existing fluid through the separation zone to reduce or eliminate the flow separation. For this plasma model case, an important consideration was made by introducing the non-dimensional body force parameter $D_c$ to the applied electric field. All parameters used for this simulation were described in the plasma actuator model section.

Figures 10 and 11, shown below, show the velocity vectors and streamlines at the leeside of the hump for the plasma case. In Figure 10, a fully attached boundary layer is observed and compared to the baseline case, in which a large separated region was present (in Figure 5). Both Figure 10 and Figure 11 indicate a clear effect of the plasma forces on the flowfield. The flow separation is eliminated by these plasma body forces observed in Figure 11.

**Figure 10. Velocity vectors for plasma actuator model**

**Figure 11. Velocity streamlines for plasma actuator model**

Figure 12 below shows the surface pressure coefficient with the plasma on. It is clear that the pressure drops by a significant margin at the position where the separation starts (i.e. at $x/c = 0.66$), hence demonstrating the increased velocity. This large pressure drop indicates the induced body force on the hump flow.

The plasma actuator effects on the skin friction coefficient are shown in Figure 13. For both the pressure coefficient and the skin friction coefficient, comparisons are made with the baseline case results to demonstrate the experimental and numerical results difference clearly.

U-velocity contours are presented in this work to provide additional details on the plasma-induced flow. Figures 14-17 show the instantaneous U-velocity at different timesteps. Figure 14 is the baseline flowfield. Figures 15, 16 and 17 show the flowfield after 300 timesteps, 1500 timesteps and 3300 timesteps, respectively. The U-velocity for the baseline simulation in Figure 14 shows the flow separation on the leeside
of the hump. Figure 15 shows the effect of the plasma actuator after 300 timesteps. The separated flow region was pushed downstream by the plasma induced external body forces. Figures 16 and 17 provide additional details into the flowfield. After 3300 timesteps, the plasma-induced body force eliminates the separation completely.

CONCLUSIONS
A Hybrid RANS-LES (PANS) turbulence model was used to solve the flow over a hump with and without a DBD plasma actuator flow control device. The simulation was carried out at a Mach number 0.1 and Reynolds number of 936000. The baseline case (i.e., no control) showed very good agreement with the experimental results. Comparison of the pressure coefficient skin friction, and velocity profiles were made, thus validating the solver’s ability to accurately predict flow separation and reattachment in an incompressible flowfield. The baseline code and simulations were modified to investigate the effect of a DBD actuator on an incompressible separated flow region. The DBD
ACKNOWLEDGEMENTS

We would like to convey our special thanks to Dr. Lars Davidson of the Chalmers University of Technology for providing us to use his solver CALC-LES, which is capable of turbulence LES and RANS-LES Hybrid modeling.

REFERENCES

[27] Jakirlic, S., 2011, "Test case ST02: 2D Wall mounted Hump."
SOLVING KNIGHTS COVERING PROBLEM: BACKTRACKING, PERMUTATION, BIPARTITE GRAPH, AND INDEPENDENT SET

Serkan Güldal
University of Alabama, Birmingham
Birmingham, Alabama USA

Michael Lipscomb, Murat M. Tanik
University of Alabama, Birmingham
Birmingham, Alabama USA

ABSTRACT

Problems that do not have analytical solutions require tricky algorithms to be solved. Backtracking intuitively solves such problems with brute force. The time complexity of these types of algorithms is polynomial or exponential. Thus, they lose a practical advantage even for small systems. In this study, we found all or some solutions of the Knights Covering Problem (KCP). We compared the time efficiency of four algorithms against each other. The first, unsuccessful attempt involved the backtracking algorithm. The second algorithm used was based on permutations to fix the limitations of backtracking. This solution became resource-expensive quickly. The third algorithm solved the KCP problem by converting it to a bipartite graph. The bipartite graph solution found only 2 solutions for the KCP problem. Yet this method can find solutions for the very large number of KCPs. Our last algorithm used maximum independent sets of a graph. This approach finds all solutions and is more time and resource-efficient.

KEYWORDS: Knight; Set; Backtracking; Algorithm; Permutation; Bipartite Graph; Independent Set; Mathematica; Efficiency; Optimization

INTRODUCTION

In the Knights Covering Problem (KCP), knights protect themselves and the cells around them with respect to the knights’ movement capability. KCP solutions are free of necessary conditions in that the related number of Knights to board sizes are not required. The generalized notation is shown as N-KCP. N represents the board size to be covered by knights. The knight moves in chess are limited (a maximum of 8 options), but unique. The unique moves of knights have been used for cryptography applications [1-3]. Knights are also well known for the knight's tour problem because of its inimitable behavior [4-6].

There is a very limited number of studies about KCP [7-11]. The problem has the capability to be used to demonstrate various algorithms such as permutation generation, divide-and-conquer paradigm, and neural networks. Its usefulness in computer science is a result of the absence of an analytical method.

The KCP solutions have a strong potential to generate test data for engineering applications. First, every KCP solution could represent a stable state of the system, so with every other solution a dynamic of the system could be presented. Board size could be chosen respect to the degrees of freedom. Second, solving N-KCP requires numerical techniques surrounded by computer programming. Thus, it is a rich source of teaching material for computer engineering and mathematics education.

This paper presents different attempts to solve the KCP problem. The first attempt employed the backtracking algorithm. A pure backtracking algorithm will quickly fail to solve KCP, as explained in the section, “Backtracking Algorithm to Solve KCP”.

The second attempt involved using permutations, which is far from practical. The Permutation approach has the flavor of a backtracking algorithm but succeeds where a backtracking algorithm fails. That being said, the permutation is computationally too expensive to be utilized.

The third attempt involved the bipartite graph method. After the board is converted to a graph with respect to the knight’s move, a bipartite form of the graph reveals two KCP solutions. The fourth attempt involved an Independent Set (IS) solution. ISs of the generated graphs correspond to KCP solutions.

The following section begins a brief discussion of the available solutions. Some possible studies are suggested. In the final two sections, concluding remarks will be made.

BACKTRACKING ALGORITHM TO SOLVE KCP

Among the many available algorithms, the backtracking algorithm was the first to be utilized because it is capable and simple. This naive approach fails quickly because knights are blind to the missed previous cells during the iteration. However, it is worth mentioning to present a fuller picture of the solution space.

The backtracking algorithm is applied to a wide variety of problems. The backtracking algorithm places knights on the board one at a time, tests the considered cell, and places a knight or leaves an empty space, before repeating the same steps for the next position. As can be seen, backtracking is based on the
resource-hungry, trial-and-error method. The more errors that are confronted, the more inefficient it becomes.

For example, in Figure 1, a 4 x 4 board is shown by indexed cells. The 4-KCP problem requires the use of knights to cover the 4 by 4 board.

The backtracking algorithm begins by placing the first knight in cell 1. Since there is no attack, the first knight is placed in cell 1. After that, cells 7 and 10 are marked as unavailable to place a knight. Similarly, in the following steps, cells 2, 3, and 4 will be used to place knights. Thus, cells 5 to 12 will be marked as unavailable. After failed backtracking attempts to check these cells, the next knights will be placed in cells 13 to 16. Therefore, the first 4-KCP solution is \{1, 2, 3, 4, 13, 14, 15, 16\}.

To find the second solution, the last knight (in cell 16) will be removed. This is not a solution, for although no knight can attack another, the board is covered incompletely (cell 16 is neither occupied nor attacked). The next move for the backtracking algorithm is to remove the knight in cell 15 and to place a knight in the next cell (16 at this point). This leaves an empty space in cell 15. This is another faulty solution. As the algorithm continues moving for new solutions, similar faulty solutions will be listed.

This issue could be solved by a checker that would test every generated solution to learn whether all cells are occupied or threatened. Yet this would add an additional cost. As can be seen, a pure backtracking algorithm is not able to solve N-KCP, but it could be adapted very easily.

**PERMUTATION SOLUTION OF N-KCP**

The backtracking algorithm fails because of the unique knight moves on the board. Even though the permutation solution has the flavor of the backtracking algorithm, it forces a check on every cell. The permutation algorithm has different steps than the backtracking algorithm, so it is fair to call it an independent method to solve N-KCP.

The permutation algorithm starts by generating all permutations of the indexed board. For example, 4-KCP requires permutation of 16 numbers, 16! = 20922789888000. To generate and store the permutations is beyond the capability of regular computers, and the difficulty increases by \(n^n\). This expensive algorithm is not feasible even though it solves N-KCP correctly.

In summary, the permutation algorithm does not have a blind spot since it checks every cell. Because different orders lead to different solutions, it generates all of the solutions. Thus, this algorithm is better than the backtracking algorithm.

However, the resource-hungry behavior makes it impossible to utilize the permutation algorithm for solving N-KCP.

**BIPARTITE SOLUTION OF N-KCP**

A graph is classified as a bipartite graph if the vertices of the graph are separated two disjoint sets \(V_1\) and \(V_2\) in such a way that each edge joins the nodes from \(V_1\) to \(V_2\). A representation of a bipartite graph is presented in Figure 4 [12]. A graph representation of the N-KCP board corresponds to the bipartite graph. For the KCP problem, each set of the bipartite represents a solution for the KCP.

In the first step, the board is converted to a graph based on the knight moves on the board. Each cell represents a node in the graph. Undirected edges are drawn between the nodes on which knights can move from one node to the other. For example, 4-KCP is the problem of placing knights on a 4x4 board in such a way that no knight can attack another, but every empty cell is attacked by at least one knight.

For a graph of a 4-KCP, it can be seen that cell 1 is able to attack cells 7 and 10. Thus, the edges are defined between the nodes (See Figure 2).

Figure 2. Cell 1 can attack to Cells 7 and 10 in the 4-KCP Problem

When the same test is applied to every cell, and related nodes are connected, the graph shown in Figure 3 is obtained.

Figure 3. Graph Form of 4-KCP

The bipartite form of the graph shown in Figure 3 is presented in Figure 4 below.

Figure 4. Bipartite Graph of 4-KCP
Thus, 2 of the 4-KCP solutions are \{1,3,6,8,9,11,14,16\} and \{2,4,5,7,10,12,13,15\}. This method uses very moderate computational power, so it could be applied to greater boards. Some solutions are listed in Table 1. Only solutions for small boards are presented in the interest of parsimony. These solutions were found virtually in no time.

The bipartite algorithm requires very minimal resources and time. However, it is limited to finding only 2 solutions for the introduced board.

### INDEPENDENT SET SOLUTION OF N-KCP

An Independent Set (IS) is a set of vertices of a graph that are not adjacent. An IS solution is a very elegant solution for N-KCP. The conversion of the board is introduced in the bipartite graph solution. ISs of the representing graph correspond to all N-KCP solutions. For example, the 4-KCP graph has the IS listed in Table 2.

### DISCUSSION OF THE N-KCP SOLUTIONS

In this section, we discuss N-KCP solutions. N-KCP solutions are presented in the histograms. Newly defined graph representations for N-KCP are listed together with the solution distributions.

One cell board, 1-KCP, is a good place to begin. 1-KCP has 1 solution, which only covers the board without attacking. Thus, it satisfies the necessary conditions. A similar state exists for 2-KCP. Four cells cannot attack each other, so each cell must be occupied. The representing graph shows 4 independent nodes. The 2-KCP solution is the Independent Set of the graph, and the set of all nodes, which is also the Maximum Independent Set. From the graph point of view, smaller sizes boards (N=7 or less) are richer in the sense of a variety of solutions. The solution distributions.

From 3-KCP to 9-KCP, as shown in Table 3, solutions are richer in the sense of a variety of solutions. The solution distributions are represented by histograms. In general, N-KCP solutions are found in the lower half of board sizes. For example, 8-KCP has 64 possible cells to place the knights, but solution lengths are below 32. The highest number of solutions are found in the length of 20.

The increasing number of solutions resembles Gaussian distribution in the histograms. Thus, the distribution of the solutions could be approximated by a Gaussian function. Although this cannot guide one to find N-KCP solutions, it can represent the number of solutions and the distribution of the solutions. Since finding the appropriate Gaussian distribution is beyond the purpose of this paper, it will not be discussed further herein.
Table 3. Number Of N-KCP Solutions with the Representing Graphs

<table>
<thead>
<tr>
<th>N-KCP</th>
<th>Number of solutions respect to solution length</th>
<th>Graph form of N-KCP</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td><img src="image1" alt="Graph" /></td>
<td><img src="image2" alt="Graph" /></td>
</tr>
<tr>
<td>2</td>
<td><img src="image3" alt="Graph" /></td>
<td><img src="image4" alt="Graph" /></td>
</tr>
<tr>
<td>3</td>
<td><img src="image5" alt="Graph" /></td>
<td><img src="image6" alt="Graph" /></td>
</tr>
<tr>
<td>4</td>
<td><img src="image7" alt="Graph" /></td>
<td><img src="image8" alt="Graph" /></td>
</tr>
</tbody>
</table>
Table 3. Number Of N-KCP Solutions with the Representing Graphs (Continued)

<table>
<thead>
<tr>
<th>N-KCP</th>
<th>Number of solutions respect to solution length</th>
<th>Graph form of N-KCP</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td><img src="image" alt="Graph for N=5" /></td>
<td><img src="image" alt="Graph for N=5" /></td>
</tr>
<tr>
<td>7</td>
<td><img src="image" alt="Graph for N=7" /></td>
<td><img src="image" alt="Graph for N=7" /></td>
</tr>
<tr>
<td>8</td>
<td><img src="image" alt="Graph for N=8" /></td>
<td><img src="image" alt="Graph for N=8" /></td>
</tr>
<tr>
<td>9</td>
<td><img src="image" alt="Graph for N=9" /></td>
<td><img src="image" alt="Graph for N=9" /></td>
</tr>
</tbody>
</table>
The other point to be made is that the graph form of the N-KCP resembles the board with the appropriate positioning for the N=8 and greater boards. This regularity in the graphs clearly indicates the existence of an analytical solution. This solution utilizes the divide-and-conquer method, such that they are grouped with respect to the number of adjacencies as first corners (FC), second corners (SC) first edges (FE), second edges (SE), and insiders (I) (see Figure 5). These classifications are made based on attack capability of the knights. As our purpose is to create a guide to future studies by the analysis, we will not attempt to find an algorithm which utilizes this property.

![Figure 5. 7-KCP Zone Definitions for Mathematical Divide and Conquer Algorithm](image)

RESULTS AND DISCUSSION

Four attempts are introduced in this study. The first algorithm is backtracking and uses brute force: placing a knight and testing the appropriateness. However, as was shown, it fails to quickly solve because of the unique nature of knights’ movements. The second algorithm, the permutation method, is a robust method but computationally unbearable. The third algorithm uses bipartite graphs to solve N-KCP. This algorithm finds two solutions but is computationally free. The fourth and last algorithm uses an Independent Set (IS) method to find all solutions. Each given algorithm has advantages and disadvantages. Based on the considered N-KCP, at least one solution could be obtained.

In addition to the algorithms, a brief discussion was had about the N-KCP solutions, and this “cracks” possible analytical methods. The first methods have the potential to show the distribution of N-KCP solutions by Gaussian function. The second algorithm introduces a divide-and-conquer method, which is a groundbreaking approach for the combinatorics problem that has inseparable pieces.

CONCLUSION

In this paper, we have demonstrated four attempts to find N-KCP solutions. The backtracking algorithm failed to solve. The permutation algorithm finds all solutions but is an expensive algorithm. The bipartite graph method finds only two solutions but is computationally free. Independent Set solution computes all solutions in a relatively moderate amount of time. There is no single algorithm that satisfies all needs. Yet with the right trade of N-KCP, it could be solved.

Future work will include solving the N-KCP by the methods which proposed in the discussion of N-KCP solutions. The first proposal offers the use of Gaussian function to extract information about the solution distribution. The second suggestion is for a solution algorithm that uses a divide-and-conquer method, which itself utilizes the regularity of the graph.

ACKNOWLEDGMENT

The authors declare no conflict of interest.

REFERENCES


ABSTRACT

Liquid metals are characterized by their higher densities and thermal conductivity compared to non-metallic liquids; these liquids are used in nuclear reactor cooling systems. For this purpose, the investigation and modeling of turbulent heat transfer of low-Pr is needed for a better understanding of the turbulent thermal transport. This paper investigates and discusses the effect of different flow conditions on turbulent thermal transport. On the first part of the study, channel flow simulations are performed for turbulent channel flow with $Re_{τ} = 640$ for low and high Prandtl number (Pr), Pr = 0.025 and 0.71 with and without buoyancy effects, while including both forced and natural/forced convection. Results from the channel flow case results showed that near wall turbulent structures are enhanced and diminished for unstable and stable stratification, respectively. The buoyancy affects the turbulent statistics on the aiding and opposing sides, and it affects the thermal diffusion more in high Reynolds flows than low Reynolds flows. Since the flow through a reactor involves complex flow regimes as separation/reattachment, for this purpose backward facing step simulations were performed for Pr = 0.71 and 0.0088 to understand the effects of the cited flow regimes over the heat transfer characteristics. Results showed that the LES model predicts the overall flow characteristics well compared to available experimental and numerical results [24,26]. LES results from this study help with understanding the turbulence diffusion for the complex convection cases.

KEY WORDS: Low-Pr flows, Turbulence modeling, heat transfer, backward facing step, channel flow

INTRODUCTION

Liquid metals are characterized by their stretchable mechanical properties, excellent electrical and thermal conductivities and biocompatibility [1]. Recently, power plants have used liquid metals for heat removal from fast breeding nuclear reactor cores [2]. Those liquid metals are low Prandtl number Pr ~ 0.002-0.03. Low Pr flows are challenging to model, since modeling involves a scale gap in momentum and thermal turbulent diffusion and dissipation. At high Pr flow, the fluid turbulence is driven by the high diffusive temperature field more strongly than the more tendrill plumes. The near wall turbulence is more enhanced for the unstable buoyant conditions, which results in both momentum and temperature accumulation away from the wall. For the stable condition it results in a sharp gradient in the momentum and temperature variations. [3, 4]

Complex flow regimes, such as flow separation and reattachment, and natural and forced convections are present in the flow through reactor cores. Reattaching flows results in large variations of the local heat transfer coefficients. In order to investigate the effect of convection effects and flow separation/reattachment, available experiments and high fidelity LES/DNS computations have been studied.

LITERATURE REVIEW

DNS channel flow simulations for passive scalar transport were conducted to analyze and understand flows that involve heat and mass transfer. Kasagi et al. [5,6] performed detailed DNS simulations for plane channel flow for mixed convection in high-Pr flows at Re =2700 and Pr =0.71 for both stable and unstable stratification and also for vertical oriented channel including buoyancy aided and opposed flow near high and low temperature plates, respectively. Their discussion of results indicated that the transport of near wall turbulent coherent structures is enhanced and diminished for unstable and stable stratifications cases, respectively. The buoyancy affects the turbulent statistics and the quasi-coherent structures on the aiding and opposing sides. Silano et al. [7] and [8] performed DNS simulation for Rayleigh-Benard convection for Prandtl number between 0.1 and 10000 and Ra between 10^5 and 10^9; their study focused on developing a relationship between Nusselt number, Reynolds number, Rayleigh number and Prandtl number to estimate the heat transfer rate. Richard et al. [9], also conducted a study where they discussed the Prandtl and Rayleigh number dependence of heat transport in high Rayleigh number. Their study discussed results for Ra=2x10^{12} and 0.5 < Pr < 10; they reported that the thermal and kinetic boundary thickness obeys Prandtl-Blasius scaling and that the effect of Pr on the heat transfer decreases with increasing Ra (Rayleigh number). Cioni et al. [10] experimentally investigated the temperature structure functions in turbulent Rayleigh Benard convection at low Prandtl
structures have been investigated by Abe et al. [17] for \( Pr = 0.71 \). This is unlike low \( Pr \) flows where the fluctuations are strongly correlated with the streamwise velocity component. Also, they reported that \( Pr \) strongly affects the velocity-temperature correlations. The affect of \( Pr \) on the thermal transport is still investigated because it involves many flow features. The low \( Pr \) turbulent flows thermal structures have been investigated by Abe et al. [17] for \( Pr = 0.025, 0.71 \) and 1 for \( Re \), up to 640. They reported that the very large-scale structures of the temperature fluctuations appear clearly in the outer layer for \( Re=640 \) at each \( Pr \) number, and there exist very large structures of temperature fluctuations \( \theta^2 \) in the outer layer for \( Re=640 \) with \( Pr=0.025 \) and 0.71. Abe et al. [4] examined the Reynolds and \( Pr \) number effects on the wall variables for \( Re=180~1020 \) with \( Pr=0.025 \) and 0.71, they concluded that the large scale effect increases with increasing \( Re \). The flow under mixed forced and natural convections was studied using both LES and DNS, mostly for low \( Re \) flows as De Santis et al. [19] who performed channel flow at \( Re=3300 \) for both \( Pr = 0.01 \) and 1 with Richardson number of \( Ri = 0.5 \). Their work focused on determining the role of forced and natural convections on heat transfer. For mixed convection the \( Pr \) number has a larger effect on results than is the case of forced convection adjacent to the backward facing step, and they discussed the effect of \( Re \) and aspect ratio on the flow and heat transfer characteristics. In this work the maximum local \( Nu \) (Nusselt number) is observed in the region where spanwise velocity component is maximum. The location of the maximum \( Nu \) is obtained at one step height upstream of the reattachment point. Increasing Reynolds numbers result in higher magnitude of the near wall spanwise velocity component that results in high local \( Nu \). LES of incompressible turbulent flow through a backward facing step with heat transfer for \( Pr = 0.71 \) for different heat fluxes was conducted by Avancha and Pletcher [24]. Their results confirmed the Vogel and Eaton [21] results showing that heat transfer attains its maximum value upstream of the reattachment point. They revoked that downstream of the reattachment, the depression in the wall around it is mainly caused by the shear layer. Another LES simulation for an incompressible flow with \( Pr = 0.71 \) was performed for Keating et al. [25] for a heat transfer in the separated and reattached flow regions downstream of a backward facing step. Their results showed that in heat transfer the peak occurs upstream of the time-averaged mean reattachment location. Correlations of the heat transfer rate and the streamwise coefficient of friction fluctuations are observed to be excellent. Also, they reported that flow near the reattachment region involves large eddies that mostly originate in the shear layer, which explains why the maximum heat transfer coefficient is upstream of the reattachment point.

A direct numerical simulation (DNS) study was performed by Neumann and Frohlich [26]. Their study involved a low \( Pr \) flow past a backward facing step, precisely for liquid metal of \( Pr \) numbers; they concluded that for convection at low \( Pr \) flows (Mercury), the temperature fluctuations in the bulk are transported as a passive scalar, and the difference between high and low \( Pr \) can be interpreted from the different position of Bolgiano’s scale with respect to inertial range. Thus, the small-scale fluctuations behave like a passive scalar in ordinary Kolmogorov turbulence, although the large scales are of course actively driven by buoyancy. For moderate \( Pr \), the temperature appears by contrast to be active at small scales, according to the dynamics of Bolgiano.

Plane channel flows with forced convection DNS simulations were performed for a range of Reynolds numbers up to 2.34x10^4 for low Prandtl number flow (\( Pr = 0.01 \) and 0.025). Most of these simulations agreed that the temperature fluctuations are strongly correlated with the streamwise velocity fluctuations, and there is an independence of \( Re \) for \( Pr = 0.07 \). This is unlike low \( Pr \) flows where the correlation breaks down and the temperature fluctuation tends to be more dependent on \( Re \). Yang et al. [16] studied the effect of the Prandtl number on temperature fields and reported that \( Pr \) strongly affects the velocity-temperature correlations. The affect of \( Pr \) on the thermal transport is still investigated because it involves many flow features. The low \( Pr \) turbulent flows thermal structures have been investigated by Abe et al. [17] for \( Pr = 0.025, 0.71 \) and 1 for \( Re \), up to 640. They reported that the very large-scale structures of the temperature fluctuations appear clearly in the outer layer for \( Re=640 \) at each \( Pr \) number, and there exist very large structures of temperature fluctuations \( \theta^2 \) in the outer layer for \( Re=640 \) with \( Pr=0.025 \) and 0.71. Abe et al. [4] examined the Reynolds and \( Pr \) number effects on the wall variables for \( Re=180~1020 \) with \( Pr=0.025 \) and 0.71, they concluded that the large scale effect increases with increasing \( Re \). The flow under mixed forced and natural convections was studied using both LES and DNS, mostly for low \( Re \) flows as De Santis et al. [19] who performed channel flow at \( Re=3300 \) for both \( Pr = 0.01 \) and 1 with Richardson number of \( Ri = 0.5 \). Their work focused on determining the role of forced and natural convections on heat transfer. For mixed convection the \( Pr \) number has a larger effect on results than is the case of forced convections. Their work also involves generating a validation database for RANS turbulent heat flux models.

Heat transfer is affected by the redistribution of the temperature caused by such complex flow regimes as reattachment. Several numerical simulations and experiments have been conducted to develop an understanding of the effect of the flow separation over the thermal transport. Most of the numerical and experimental studies available were performed for \( Pr = 0.71 \). Those studies revealed that reattaching flows causes variations in the local heat transfer coefficient. The heat transfer rate and the turbulent velocity fluctuations strongly correlates, and the heat transfer peak occurs upstream of the reattachment location.

Armaly et al. [20] conducted an experiment to understand the relation between \( Re \) and the reattachment length for a backward facing step. The results showed that the reattachment length variation is characterized by Reynolds number. For the laminar flow regime the reattachment length tends to increase with \( Re \), then decreases in the transitional region due to the increase of the velocity fluctuations; as the flow fully develops into the turbulent state, the reattachment length remains relatively constant.

Vogel and Eaton [21] report experimental data of measured air flow and heat transfer characteristics downstream of the step. The experiment consisted of a backward facing step geometry where at the bottom wall down the step a constant heat flux of 270W/m^2. The results showed that the reattachment is causing an augmentation of the heat transfer coefficients, although it reaches its maximum upstream of the reattachment at a location that is equal to 6, 66 height step (\( h_s \)) away from the step location. The near wall layer grows as the laminar boundary layer upstream of the reattachment point, and it thickens as it approaches the center of the recirculation bubble. Thus, the thickening of the boundary layer causes the drop of the heat transfer coefficient.

A URANS simulation of a backward facing step corresponding to the Vogel and Eaton [21] experiment was performed by Abe et al. [22]. Their results showed that the heat transfer coefficient depends on near wall turbulence intensity in the separated shear layer near the reattachment point that was found to be equal to 6.69\( h_s \).

Lan et al. [23] performed URANS simulations for a backward facing step. This is the latest performed simulation of turbulent forced convection adjacent to the backward facing step, and they discussed the effect of \( Re \) and aspect ratio on the flow and heat transfer characteristics. In this work the maximum local \( Nu \) (Nusselt number) is observed in the region where spanwise velocity component is maximum. The location of the maximum \( Nu \) is obtained at one step height upstream of the reattachment point. Increasing Reynolds numbers result in higher magnitude of the near wall spanwise velocity component that results in high local wall \( Nu \). LES of incompressible turbulent flow through a backward facing step with heat transfer for \( Pr = 0.71 \) for different heat fluxes was conducted by Avancha and Pletcher [24]. Their results confirmed the Vogel and Eaton [21] results showing that heat transfer attains its maximum value upstream of the reattachment point. They revoked that downstream of the reattachment, the depression in the wall around it is mainly caused by the shear layer. Another LES simulation for an incompressible flow with \( Pr = 0.71 \) was performed for Keating et al. [25] for a heat transfer in the separated and reattached flow regions downstream of a backward facing step. Their results showed that in heat transfer the peak occurs upstream of the time-averaged mean reattachment location. Correlations of the heat transfer rate and the streamwise coefficient of friction fluctuations are observed to be excellent. Also, they reported that flow near the reattachment region involves large eddies that mostly originate in the shear layer, which explains why the maximum heat transfer coefficient is upstream of the reattachment point.
= 0.0088 with a constant wall heat flux at the wall after the step, including buoyancy effects. For the case without gravity force the applied Reynolds stresses mainly occur in the free shear, while in the presence of buoyancy effects it occurs at the wall jet. The buoyancy induced convection significantly alters the flow field and reduces the circulation zone length. Other DNS simulations performed by Guo et al. [27] and Spalart and Strelets [28] investigated the causes of the reduction of the recirculation length; the homogenous heating causes the reduction of the recirculation length that can even lead to the disappearance of the circulation bubble.

OBJECTIVE AND APPROACH

This study is built upon a previous study done by Bhushan et al. (2018) [29], in which they performed computations of the plane channel flow for \( \text{Re}_c = 150 \) and 640 for both \( \text{Pr} = 0.71 \) and 0.025 for RANS, hybrid RANS/LES and LES with and without buoyancy effect, with Grashof number \( \text{Gr} = 0, -1.3 \times 10^6, 4.4 \times 10^6 \) and \( 9.6 \times 10^6 \) (vertical channel) to characterize the capability of the cited turbulence models to predict Low-Pr flows. Their results were compared to DNS data [3, 4, and 30]. And it was shown that Hybrid RANS/LES and DSM predictions for the mean and turbulent flow predictions compare very well with DNS for the range of the flow and stability conditions.

This paper investigates the effect of Prandtl number on heat transfer for flow with and without buoyancy for both natural and mixed convection effects corresponding to available DNS data [3]. Since, the available DNS studies focused on Low \( \text{Re}_c=150 \) for channel flow with heat transfer with mixed forced and natural convection even though, the higher \( \text{Re}_c \) cases are the most challenging cases for turbulence modeling validation. For this purpose, generating and validating DNS data for Higher \( \text{Re}_c \) under different flow conditions for data validation is needed. The summary of available test cases and simulation parameters for both LES and DNS simulations are summarized in (Table 1). The simulations are performed using the open source Nek5000 source code, version 17.0.

1. Channel Flow:

Channel flow simulations were conducted using LES and DNS turbulence models to understand the effect of different flow characteristics on the heat transfer for \( \text{Re}_c = 640 \).

Available DNS data do not provide flow features for higher \( \text{Re} \) with buoyancy under mixed and natural effects. For this purpose, generating DNS data for different flow conditions was initiated.

Duran and Jimenez [31] reported that a domain size \((x \times y \times z)\) with the dimensions of \(2\pi \times 2\pi \times \pi\) is sufficiently large to reproduce the one-point turbulent statistics of the flow. Abe et al. [32] reported that the pressure redistribution is weakened when \( L_x < 2\pi \), since TKE is not redistributed properly to \( \overline{v^+\overline{v}^+} \) and \( \overline{w^+\overline{w}^+} \).

Jimenez and Moin [33] mentioned that the smallest domain that can maintain a near wall cycle of the buffer layer should have a streamwise and spanwise extent to be \( L_x^+ = 250 \) – 350 and \( L_z^+ = 100 \) respectively. Satisfaction of the condition of Moin and Jimenez [33] requires that the two-point correlations of temperature and velocity fluctuations be uncorrelated with the simulation domain [34]. The two points correlation of the temperature, velocity and pressure fluctuations should be negligible for separation distances of half.

Following the conditions from the literature review, to determine the appropriate domain for DNS data requires analyzing the pressure, temperature and velocity correlations in the spanwise and streamwise direction for different \( y^+ \) and \( \text{Re}_c \).

Available DNS data reported velocity and pressure fluctuations for \( \text{Re}_c = 180,395,590 \) and 640 and temperature fluctuations for \( \text{Re}_c = 180,640 \) and 1020.

![Figure 1: two-point streamwise fluctuations and temperature fluctuations from available DNS studies](image)

The domain size shall be determined based on the correlation length for streamwise velocity or pressure in the buffer region \( y^+ \sim 10 \). The correlation length in the buffer region decreases with increasing \( \text{Re}_c \) 150 - 640. While thermal fluctuations correlation are larger than velocity fluctuations correlation for the low \( \text{Pr} \), it still holds comparable for \( \text{Pr} = 0.71 \).

For \( \text{Re}_c = 640 \) and \( \text{Pr} = 0.025 \) the domain size \( L_x \sim 8 \) and \( L_z \sim 4 \), while for \( \text{Pr} = 0.71 \) the domain will be like Moser et al. [30] where they used \( L_x \sim 2\pi \) and \( L_z \sim \pi \).

For the grid resolution some available DNS studies used: \( \Delta x^+ \sim 10 - 13 \) in the streamwise direction and \( \Delta z^+ \sim 5 - 7 \), and for the near wall resolution \( \Delta y^+ \sim 1 \) is used for a better near wall resolution.

Based on the information from literature the DNS simulations for both cases of \( \text{Re} = 640 \) are:

- Pr = 0.71 domain size = \(2\pi \times 2\pi \times \pi\) and grid size = \(300 \times 257 \times 300\)
- Pr = 0.025 domain size = \(8 \times 2 \times 4\) and grid size = \(380 \times 257 \times 380\)
Table 1: Summary of simulations performed for Channel Box

<table>
<thead>
<tr>
<th>Test case</th>
<th>Turbulence model</th>
<th>Flow properties</th>
<th>Walls temperature (Thot and Ttop)</th>
<th>Domain Size x×y×z</th>
<th>Resolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>DNS</td>
<td>DNS</td>
<td>$R_e$ 0.71, $P_r$ 0.025</td>
<td>Equal $2\pi \times 2 \times \pi$</td>
<td>$300 \times 257 \times 300$</td>
<td>$8 \times 2 \times 4$</td>
</tr>
<tr>
<td>DNS</td>
<td>DNS</td>
<td>$R_e$ 0.71, $P_r$ 0.025</td>
<td>$1.4 \times 10^6$, $T_{bottom}$</td>
<td>$300 \times 257 \times 300$</td>
<td>$8 \times 2 \times 4$</td>
</tr>
<tr>
<td>DNS</td>
<td>DNS</td>
<td>$R_e$ 0.71, $P_r$ 0.025</td>
<td>Unequal $2\pi \times 2 \times \pi$</td>
<td>$300 \times 257 \times 300$</td>
<td>$8 \times 2 \times 4$</td>
</tr>
<tr>
<td>LES (DSM)</td>
<td>LES (DSM)</td>
<td>$R_e$ 0.71, $P_r$ 0.025</td>
<td>Vertical $2\pi \times 2 \times \pi$</td>
<td>$96 \times 96 \times 96$</td>
<td></td>
</tr>
</tbody>
</table>

Figure 2: DNS contour plots for equal wall temperature cases for $R_e = 640$ for case of $Gr = 0$ (a) $Pr = 0.71$ and (b) $Pr = 0.025$

Figure 3: Mean velocity, temperature and velocity turbulent fluctuations for $R_e = 640$ for both $Pr = 0.71$ and $0.025$ compared with Kaaga (1992) available DNS data.

The flow characteristics comparison between the available DNS data, in Figure 2, and generated data in Figure 3, agrees well.

With higher $Pr$ values the velocity and temperature show similar turbulent structures, since the velocity scale is proportional to the temperature advection, opposed to low $Pr$ case where the temperature is quite laminar due to molecular effect that dominates the thermal transport. (Figure 4 and 5).

Comparison of different cases with and without buoyancy using both DNS and LES shows that the unstable case enhances near wall turbulence while for the stable case the near wall turbulence is reduced. (Figure 6)
The vertical case with opposed gravity force results in generating an aiding flow near the bottom wall, which is caused by the accumulation of the momentum and temperature near the wall, that results in diminishing of the velocity fluctuations. The effect of low Pr is observed for the temperature profiles that becomes more laminar for all the cases because the thermal transport is due to molecular effects of channel flow. (Figure 5)

2. Backward facing step:

The backward facing step simulations were performed using RANS, LES and hybrid RANS/LES turbulence models for Reₜ = 5300, for both Pr = 0.71 and Pr = 0.0088 with Richardson number Ri = 0 and 0.338. For these simulations a backward facing step geometry is used with an expansion ratio of ER = H/(H-hₛ) = 1.5, where hₛ is the step height and H is the outlet channel height. (Figure 7)
reducing the size of the recirculation bubble, since the hot fluid is acting against the rotation in its recirculation. This effect causes the flow to detach from the wall, which keeps the skin friction coefficients positive along the wall. This study provides profiles of different flow features compared to the results from the available numerical and experimental data.

Simulation setup:

Backward facing step geometry consists of creating four rectangular blocks. The overall size of the domain on the streamwise direction is 42\(h_s\), the spanwise direction is 4\(h_s\) and for normal direction 3\(h_s\), where \(h_s\) is the step height. (Figure 7)

The boundary conditions imposed on the domain are: inflow boundary condition imposed at the far left boundary on the streamwise direction while imposing an internal boundaries between the blocks on both streamwise and normal direction; outflow boundary conditions imposed on the far right wall for both block 4; periodic boundary conditions on the spanwise direction; wall boundary conditions for the top and bottom walls and the step, with inducing a constant heat transfer from the bottom wall downstream of the step. (Table 2)

Recycling boundary conditions are applied at the inflow region, the recirculation region is equal to \(dx = 7h_s\) (from -10 to -3). In order to be able to generate the appropriate flow conditions for the step.

**Table 2: Boundary conditions used for the backward facing step**

<table>
<thead>
<tr>
<th>Block #1</th>
<th>Block #2</th>
<th>Block #3</th>
<th>Block #4</th>
</tr>
</thead>
<tbody>
<tr>
<td>(V)</td>
<td>(T)</td>
<td>(V)</td>
<td>(T)</td>
</tr>
<tr>
<td>X-min</td>
<td>(V)</td>
<td>(T)</td>
<td>(E)</td>
</tr>
<tr>
<td>Y-min</td>
<td>(W)</td>
<td>(I)</td>
<td>(W)</td>
</tr>
<tr>
<td>Z-min</td>
<td>(P)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Z-max</td>
<td>(P)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*V: velocity, \(T\): Temperature, \(E\): internal BC, \(W\): Wall BC, \(O\): outflow BC, \(I\): zero-gradient BC and \(f\): flux

Line and contour plots for different simulations are analyzed and discussed to understand the effect of flow separation on heat transfer, as summarized on (Table 3) our study involved studying two different cases for high and low \(Pr\) flows.

The lines plots were extracted at 6 different locations along x direction \(x/h = 0,3,6,9,12,15\).

The wall temperature for low \(Pr\) is quite laminar due to effect of low \(Pr\), and it diffuses away from the wall, while for the higher Prandtl number case the temperature near the step is higher and has a turbulent nature of diffusion. (Figure 8)

**Table 3: Summary of simulations performed for Backward facing step**

<table>
<thead>
<tr>
<th>Simulations parameters</th>
<th>Flow conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bulk velocity (U_b) (m/s)</td>
<td>1</td>
</tr>
<tr>
<td>Initial/inflow (T_{in}) (C)</td>
<td>150</td>
</tr>
<tr>
<td>Step height (h_s) (m)</td>
<td>1</td>
</tr>
<tr>
<td>Dynamic viscosity (\mu) ((kg/m.s))</td>
<td>(2.083 \times 10^{-4})</td>
</tr>
<tr>
<td>Density (\rho) ((kg/m^3))</td>
<td>1.0</td>
</tr>
<tr>
<td>Kinematic viscosity (\nu) ((m^2/s))</td>
<td>(2.083 \times 10^{-4})</td>
</tr>
<tr>
<td>Fluid specific heat (C_p) ((KJ/Kg. \ K))</td>
<td>1.363</td>
</tr>
<tr>
<td>(\rho C_p) ((KJ/m^3 \ K))</td>
<td>1.00</td>
</tr>
<tr>
<td>Heat flux (q) ((W/m^2))</td>
<td>0.0236</td>
</tr>
<tr>
<td>Thermal expansion (\beta)</td>
<td>0</td>
</tr>
<tr>
<td>Thermal conductivity (k) ((W/m. \ K))</td>
<td>(0.00028169)</td>
</tr>
</tbody>
</table>

\(\Delta T/T_{in} = q* \times h_s\)

\(Re_b = U_b \times h_s/\nu\)

\(Pr = \mu \times \rho C_p \times U_b \times T_{in}\)

\(q^* = q / (\rho C_p \times U_b \times T_{in})\)

\(Ri = (g*\beta \times \Delta T \times h_s)/U_b^2\)
LES predicts unsteady flow well; the resolved turbulence is limited to the downstream region near the step since the inflow lacks resolved turbulence, causing the shear layers to break down later than is predicted. TKE is well predicted downstream of the step for LES, and as for the PANS model, predictions are quite good and agree well with experimental data from literature [24,26]. (Figure 9)

Figure 8: LES backward facing step predictions for (a) Velocity and temperature for Pr = 0.71 and (b) temperature for Pr = 0.0088

LES predictions for wall temperature at low Prandtl number compare well with data represented by Niemman and Frohlich [26]. (Figure 10)

The effect of higher thermal conductivity is observed for the cases of low Pr, as the temperature is more laminar than for higher Pr; this is caused by the molecular effect that governs thermal transport. The comparison between high and low Pr wall temperature plots highlights the low Pr effect, where LES high Pr predictions tend to be more turbulent. (Figure 10)

A decrease in the skin friction coefficient \( C_f \) is observed for both low and higher Pr. That compares well with the results of Neimann and Frohlich [26].

Albeit unsteadiness is observed for the high Pr case, predictions for flow reattachment are well realized. (Figure 11)

Figure 9: comparison of LES and PANS mean streamwise velocity and temperature (b) TKE for LES compared to the available experimental and numerical data

Figure 10: Wall temperature comparison for different Prandtl numbers

Figure 11: Skin friction coefficient \( C_f \) along the vertical wall hot heated wall for low and high Pr
CONCLUSIONS

The channel flow with heat transfer under mixed forced/natural convections simulations results for Re, = 150,640 for both Pr = 0.71 and 0.025 using LES compares well with available DNS data. The lack of a DNS validation dataset for Re, = 640 and low Pr for different convection condition concluded to perform DNS for plane channel flow for Pr = 0.025 and 0.71 for specified Re, as described on Table 2. Detailed discussion of the domain size determination is highlighted. DNS results show a good agreement with available DNS results. Results from the latest model raise the complex interaction between thermal diffusion and momentum, thus it provides a challenging case for turbulence model evaluation. Backward facing-step simulation preliminary results show that the described models predict well the thermal transport of flow with attachment/reattachment nature. LES predicts the resolved turbulence and helps for understanding the turbulence in the reattached flow. The Prandtl effect is observed for different cases. As for future work, improvement of model’s predictions is necessary, with using finer grid, and investigating the effect of buoyancy on low Pr flows for backward facing step geometry and estimating the effect of the variation of Pr, in the flow.

ACKNOWLEDGEMENTS

The project is funded by DOE NEUP R&D Award CFD-17-13179.

REFERENCES


[31] Lozano-Duran, A. and Javier Jimenez, J. 2014. Effect of the computational domain on direct simulation on turbulent channels up to \( Re = 4200 \). Physics of Fluids 26, 011702


AN IMPLEMENTATION OF THE AUTOMOTIVE EXTERNAL AIRBAG TO EXAMINE IMPACT MITIGATION

Dhruvesh Modi
University of Alabama at Birmingham,
Birmingham, Alabama USA

Lee Moradi
University of Alabama at Birmingham,
Birmingham, Alabama USA

ABSTRACT
Internal airbags are used for the safety of passengers in automobiles. This paper examines the implementation of an external airbag for impact mitigation. The study is concerned with simulation, modeling, and analysis of the system using LS-Dyna software. Moreover, use of different pressures in the external airbag was investigated.

Key Words: Airbags, external airbag, LS-dyna, Impact, Pressure

INTRODUCTION
Vehicles are used for travel from one place to another. One of the hazards of using a vehicle is accidents that cause injury or death. To control injuries, automobile manufacturers develop new remedies for passenger’s safety. The airbag, which is a safe element for passengers in a vehicle, is commonly used. It has the capability to open automatically when vehicles have collisions. In addition to airbags, seat belts also play a crucial role for passenger safety. There are several gas-inflated cushions located inside the vehicle with high compression and at different locations such as, on the driver side on steering, on dashboards, on the passenger side, backside, near the leg sides etc. But this paper concentrates on an external air bag system to reduce the impact as well as for the passenger safety in the Automobile world. It is estimated that automobile collisions cause 1.2 million fatalities in the world and 4.8 million injuries annually. In September of 1999, the National Highway Traffic Safety Administration found that using the airbags had saved 4600 lives.

The analysis uses Ls-dyna Software to check the impact with and without an airbag. The model of a metro car was used in this research. The airbag is located on the front bumper of the metro car. The input file used Win SCP for analysis with various pressures.

LITERATURE REVIEW
Arun Kumar and S Madhu published “A research review on an airbag in automobile safety system” examining the airbag contribution in the case of accidents. The main purpose of an airbag in any vehicle is to spare the lives of the drivers as well as the traveler during an accident.[1] The motivation behind an airbag is to pad inhabitants during an accident and give security to their bodies when they strike an object inside the vehicle. In this manner, the utilization of the airbag framework reduces injuries by decreasing the force of impact from the steering wheel, windows and dashboard on the body. The researchers explained the types of airbags used in the automobile and how it works in any accident. According to their research, there are many types of airbags located in a car. By utilizing various types of airbags, casualties from car collision are mitigated. The package is structured such that it can inflate in under a second after the impact. Airbags can be comprised of polyester and nylon. There is a range for nylon which includes nylon 4.6, nylon 6 and nylon 6.6. Nylon 6.6 is much better than other types of nylon in thermal properties, recuperation in the elastic properties, lengthening and moisture substance and density. The textures which have high strength and high quality have great protection from ageing.[2] Another study published by Saeed Barbat and Jiawei Li used bumper and grill airbags for improved vehicle compatibility in the case of side impact.[3] Their research explored deployable guard and grill airbags for improved mitigation during a side impact. The study examined the impact sensors, side geometry and the pre-crash sensors. There are many sensors that play a vital role for an airbag activation system, such as crash sensor, safety sensor, speed sensor, impactor sensor, etc.

If the vehicle impacts an object in front or on any side during an accident, the sensors activate as quickly as possible and the
airbag inflates in 0.005 sec to protect the passenger. There are mainly three chemicals, NaNO₃, KNO₃, and SiO₂ inside an airbag.

Reactions:

\[ 2\text{NaN}_3 = 2\text{Na} + 3\text{N}_2 \]
\[ 10\text{Na} + 2\text{KNO}_3 = \text{K}_2\text{O} + 5\text{Na}_2\text{O} + \text{N}_2 \]
\[ \text{K}_2\text{O} + \text{Na}_2\text{O} + \text{SiO}_2 = \text{Alkaline Silicate} \]

Lotta Jakobssohn, Thomas Broberg, Henrick Karlsson, Anders Fredriksson, Niklas Graberg published a research paper on “Pedestrian Airbag Technology – A Production System”. The paper discusses the evolution of the airbag technology in the automotive world.[5] In some accidents passengers were out of position when their airbag deployed. The fundamental point of their investigation was to assess the performance of side airbag using the Finite Element Method (FEM).[6] In another published paper, Shaikh, Tasmin & Chaudhari, Dr. Satyajit & Rasania, Hiren examined an airbag which worked as a safety restraint system of an Automobile.[7] An airbag system works based on the chemical reaction as well as the seat belt which plays a fundamental role in it.[8]

According to research, an airbag is located at or near the chest area, side and near to legs which protect passengers and driver in the case of a collision.[8] Huizhen Zhan published a research paper in 2003 about injuries during collision, and cited that many times injuries occurred because of the inflated airbag. In some cases, the neck injuries are ignored by scientists and guidelines, yet the injury to the human neck is intense and the rate of debilitation in auto collisions is high.[9] Rachel Casiday and Regina Frey published a paper on “Gas Laws Save Lives: The Chemistry Behind Airbags”. The paper discussed the chemical reaction of an airbag. They noted that sometimes drivers and passengers did not wear a seat belt in a car, so the airbag did not open during the accident. Moreover, they noted that crash tests demonstrated that for an airbag to be helpful as a protective device, the airbag must inflate within 40 milliseconds.

Airbags have certainly saved lives and have dropped the number of severe injuries. Based on the report from the National Highway Traffic Safety Administration (NHTSA), the combination of an airbag and seat belt can decrease the risk of head injuries by 85 percent rather than the 60 percent for belts alone.[10] The Office of Oversight and Investigations Minority Staff published a research paper on “Danger Behind the Wheel: The Takata Airbag Crisis and How to Fix Our Broken Auto Recall Process”. There were many incidents of the Takata airbag malfunctioning in recent history. In one such incident in 2003, the Takata airbag inflator got ruptured in a BMW car in Switzerland. After the incident, the fact-finding committee found that this was a one-off incident and the 17-month-old inflator ruptured because of overloading of propellant in the assembly of the inflator.[11]

In July 1984, cars offered airbags or automatic seatbelts as an automatic occupant protection in the requirement of NHTSA. In 1997 airbags were located inside for the driver and passenger side of the car. Figure1 shows the airbag inflator and its parts. Other problems that faced airbags involved the use of harmful chemicals.[12]

L. A. Wallis and I. Greaves, published a paper on “Injuries associated with airbag deployment”. According to researchers, there is enough evidence around the world that airbags are a lifesaver and provide outstanding protection against severe impairment. Alternately, there is a growing evidence of studies showing damage which is directly linked to these devices. There were 618 injuries, of which 42% affected the face, 33% the upper limb, and 9.6% the chest occurred from 1980 to 1994 according to NHTSA. In all the injuries, 96% were not major. [13]

### MODELING AND SIMULATION

**Modeling of Metro-car with an External Airbag:**

LS-DYNA is generally used to perform for crash simulation. The GEO Metro-car model shown in Figure 2 was used for crash simulation with an external airbag on the front bumper of the car.

![Figure 2 GEO Metro car without an external airbag](image1)

In this simulation a model of a small car was used to run into an airbag attached to a solid wall as shown in Figure 2. The metro-car model uses tons for weight and meters as well as mm for dimensions.

The *AIRBAG_SIMPLE_AIRBAG_MODEL* card was use for an airbag model. The model used nylon 6.6 material.

*MAT_FABRIC* card was used for the fabric properties of the airbag. The model used three different pressures, 5psi, 10psi and 15psi.

![Figure 3 GEO Metro car with an external airbag](image2)

In July 1984, cars offered airbags or automatic seatbelts as an automatic occupant protection in the requirement of NHTSA. In 1997 airbags were located inside for the driver and passenger side of the car. Figure1 shows the airbag inflator and its parts.
5 psi pressure for passenger’s safety purposes. The pressures used are calculated below.

\[
0.06 \text{ Mpa} = 5.8 \text{ psi} \quad \text{(Passenger Airbags)}
\]

\[
35 \times 10^{-3} \text{ Mpa} = 5 \text{ psi}
\]

\[
69 \times 10^{-3} \text{ Mpa} = 10 \text{ psi}
\]

\[
105 \times 10^{-3} \text{ Mpa} = 15 \text{ psi}
\]

\[
138 \times 10^{-3} \text{ Mpa} = 20 \text{ psi}
\]

The material of the metro-car model is steel. The speed of the metro car was 35-40 mph during the crash simulation. In the metro-car model the velocity was specified by \text{INITIAL VELOCITY GENERATION} car.

The scale function for the direction of the airbag and car were used to contact to airbag. The rotational direction used the Rotate function for the angle between the metro-car and the airbag. The translate function was used to put the airbag near the front bumper of the metro-car. Furthermore, the surface contact card, the LS-DYNA, was used to create contact between the airbag on the front bumper. The \text{CONTACT AUTOMATIC NODES TO SURFACE} card was used to make a surface contact between the airbag to the solid wall.

The surface contact card uses a friction value of zero for the friction between the solid wall and the airbag. The metro-car has different meshing style compared to an airbag file. Proper changes in the mesh were performed on the center nodes of the metro-car’s front bumper side and attached to center nodes of an airbag. The airbag is assumed to open automatically with the help of the sensors before collision. This, provided adaptively to the front-end structure of the vehicle and empowered by solid pre-crash detecting, can possibly reduce the impact.

\section*{RESULTS AND DISCUSSION}

Many simulations were performed with an external airbag and without an airbag. The analysis examined airbags with different pressures, and the results are discussed below.

\textbf{Results of a crashing test without an external airbag:}

\begin{figure}
\centering
\includegraphics[width=\textwidth]{figure4}
\caption{Figure 4 (4.1,4.2,4.3,4.4) Results of the Crash test without an external airbag}
\end{figure}

These images show the crash test without an external airbag in the GEO Metro car. In this crash test the combined file of the solid wall and the GEO Metro car file were used. The images show significant damage to the car. The first image which demonstrates the initial position of the vehicle and at time of 0 ms. The car is then going forward in the X-direction with the
velocity of 45 mph. At 0.065 ms the car comes near the solid wall for the crash.

The speed was then rapidly increased in the same direction and the metro-car model crashes into the solid wall. The front bumper and some internal parts are damaged during the crashing test in the third stage at 0.114 ms. Finally, in the fourth stage it displays that the car is damaged at the front bumper as well as on the hood. The analysis also shows that it is damaged in the rear. The displacement of the car without airbag was examined as shown in Figures 5 and 6.

1) Initial stages: results without an external airbag

2) Middle stages: results without an external airbag

3) Final stages: results without an external airbag

Figure 5(5.1,5.1.1,5.2,5.2.2) Initial and middle stages results without an external airbag
6.1.1 Vectors of total displacement at 0.0929ms

Figure 6 (6.1, 6.1.1) Results of displacement without an external airbag

In the second stage it is moved forward in the X-direction with a velocity of 40-45 mph. The displacements are shown with green vectors on the front bumper side of a GEO Metro-car. Node #226139 was selected to check the crushed length of the front of the car without an external airbag. This node crushes inwards at a length of 1.4298 meters.

Graph 1 Displacement graph between the front bumper side to solid wall

Graph 1 shows the distance between the center of the front bumper (node #137356) to the center of the solid wall (node #900047). The graph shows the distance to gradually decrease to 0.0680 m at 0.14ms.

Results of a crash test with an external airbag:

The analysis with the external airbag on the front bumper in the car was the focal point of this research.
At this point the speed was rapidly increased and the metro-car model crashed into the solid wall with the external airbag in between. The third image shows the vehicle crashing into the external bag and solid wall at 0.14 ms. These images of the crash show that the damage is less with the use of external airbag than without.

Figure 13 Inflated airbag thickness

Figure 13 shows an inflated airbag with 0.3 meters depth on the front bumper. At 0.0345 ms the airbag opened properly with a pressure of 5.6 psi. The car was going forward in the X-direction with the velocity of 45 mph.

Figure 14 An external airbag on the front bumper side of a metro-car model

Here, Figure 14 shows the three different stages of an external airbag on the front bumper side. It can be seen that the external airbag which opens rapidly before to hit the solid wall. The first two images show the initial position of an external airbag; after that the car with an external airbag will be hit on a solid wall. In the final stages of the crash test the car will not damage more instead of the car without an external airbag.

Results of the displacement with an external airbag

1) Initial stages results with an external airbag on the front bumper
15.1 Initial stage at 0.014ms

15.1.1 Vectors of total displacement at 0.014ms

Figure 15 (15.1,15.1.1) Initial Stage Results of displacement with an external airbag

The color codes show the displacement of the external airbag and the car. The light blue color in Figure 15 is the starting phase of the crash into the external airbag. The front bumper is obviously displacing less due to the flexibility of external airbag. This displacement is shown by the vectors in the images.

2) Middle stages results with an external airbag on the front bumper

16.1 Initial stage at 0.039ms

16.1.1 Vectors of total displacement at 0.039ms

Figure 16 (16.1,16.1.1) Mid stage of the displacement with an external airbag

In the second stage, the car moves forward with a velocity of 40-45 mph. The displacement are shown with green vectors on the front side of the car.

3) Final stages results with an external airbag on the front bumper

17.1 Final stage at 0.6298ms
17.1.1 Vectors of total displacement at 0.06298ms

Figure 17 (17.1,17.1.1) Final stage of the displacement with an external airbag

In the final stage, displacement are shown in dark green. Node #226139 was selected to check the crushed length of the front of the metro-car with a 5.6 psi external airbag. The results show the crushed length of the front of the car is 0.954 meters.

Figures 15 through 17 show different stages of crash. It can be seen that the external airbag opens rapidly before the car hits the solid wall. Subsequently, the car squeezes the airbag and impacts the wall.

Graph 2 Displacement graph between front bumper side to solid wall with an external airbag

Graph 2 shows the difference in distance between the center of the front bumper (node #137356) to the center of the solid wall (node #900047). The results indicate that the distance gradually decreases until 0.06 ms when it starts increasing until 0.09 ms, and finally decreasing to 0.2 m until the termination of analysis at 0.14 ms.

Results with 15psi of an external airbag:

Graph 3 is the result of the simulation with an airbag at 15 psi internal pressure. The graph shows the distance between the center of the front bumper (node #137356) to the center of the solid wall (node #900047). The graph shows a gradual decrease until 0.06 ms when it increases until 0.09 ms, and finally decreases until the end of, then termination of the run. The total distance from the center of the front bumper to the solid wall with an external airbag is shown to be 0.160 m at 0.14 ms. In addition, Node #224137 was selected to check the crushed length of the front of the metro-car with a 15 psi external airbag. The results show the crushed length of the front of the car is 1.1 meters.

CONCLUSION

This research shows that an external front bumper airbag can mitigate impact damage to cars and passengers. Therefore, it can be concluded that an external airbag will be a good addition to future automobiles. The results with different airbag pressures are useful for impact mitigation. Moreover, it can be summarized that external airbags need more pressure than the internal airbags, since they do not interact with the passengers.

SUMMARY AND FUTURE SCOPE

Automobiles have been and will remain for the foreseeable future a vital instrument of transportation. External airbags can reduce crash damage to vehicle and passengers. Further research is necessary for frontal and side external airbags. In future research, different material properties may be used to find improved impact resistance for the airbag.

REFERENCES


