ASME 2012 5th Annual Dynamic Systems and Control Conference joint with the
JSME 2012 11th Motion and Vibration Conference
DSCC2012-MOVIC2012
October 17-19, 2012, Fort Lauderdale, Florida, USA

DSCC2012-MOVIC2012-8857

MODEL AND HARDWARE DEVELOPMENT FOR PREDICTIVE PLUME CONTROL IN PIPE LINES

Boyun. Wang*
Anna G. Stefanopoulou
Department of Mechanical Engineering
University of Michigan
Ann Arbor, Michigan 48109
Email: bywang@umich.edu

Nikolaos D. Katopodes
Civil and Environmental Engineering
University of Michigan
Ann Arbor, Michigan, 48109
ndk@umich.edu

ABSTRACT

The problem of model reference predictive control for eliminating contaminant cloud from a pipe fluid system by boundary control action is addressed. A lab-scale pipe fluid system prototype is developed for studying the control of fluid system. Experimental results validate the possibility of eliminating the contaminant cloud by boundary control. A model reference control architecture is constructed, in which a parameterizable reduced order mathematical model for simulating fluid particle path-lines is developed. Compared to traditional Computational Fluid Dynamics (CFD) method, this reduced order model can be solved within very short time by common Ordinary Differential Equation (ODE) solver which enables the implementation of iterative optimal control.

INTRODUCTION

Eliminating accidental hazardous release in buildings, transportation tunnels or water supply system can reduce the threat to human life. Solving such a problem requires an automatic hazard elimination process, including contaminant detection, prediction of spreading and fast-effective control action, such as neutralizing or capturing the contaminant cloud.

Contaminant spreading problems are typically formulated as Partial Differential Equations (PDE) problems. In order to relate PDE with control theory, many mathematical methods were developed. In 1991, an adjoint based optimization method for heat conduction partial differential equations was discussed by Y. Jarny et al [1]. M. Piasecki and N. Katopodes applied this method to control of contaminant release in a river in 1997 [2]. Direct Numerical Simulation (DNS)-based method for optimal feedback control was introduced by T. Bewley in 2001 [3]. In 2007, for the purpose of making use of efficient linear system theory, linearization on PDE system was illurstrated by J. Kim and T. Bewley [4].

These methods, although computational intensive, theoretically prove the possibility of manipulating PDEs by traditional control theory. Control of mixing in 2-Dimensions (2D) channel by boundary feedback was demonstrated by Aamo et al in 2003 [5]. Because their problem was studied via mathematical simulation, the entire fluid region information was known at every simulation time step. For example, the fluid velocity field was used to calculate cost function. That means an underlying assumption exists that, infinite number of ideal sensors or perfect models exist for the fluid field. In addition, the boundary control representation was a continuous function on space, which was another assumption of infinity number of virtual boundary actuators. Later in 2005, Balogh et al expended the problem to 3-Dimensions (3D) [6]. In 2001, Bewley et al extended predictive control architecture to include turbulent fluid [3]. All assumed infinite sensors and actuators.

In 2006, fluid control with finite sensors and finite boundary actuators was illustrated by N. Katopodes and R. Wu. [7]. Recently, a method for fluid predictive control by finite sensors and actuators was developed by N. Katopodes in 2009 [8]. Their simulation results of eliminating contaminant cloud from open channel flow were numerically illustrated. With all these simulations and theoretical analysis, together with the fast growing computational power, unlimited applications of fluid control in real world physical systems are becoming feasible. However, because of the computational intensity involved in solving fluid PDEs, tra-

*Address all correspondence to this author.

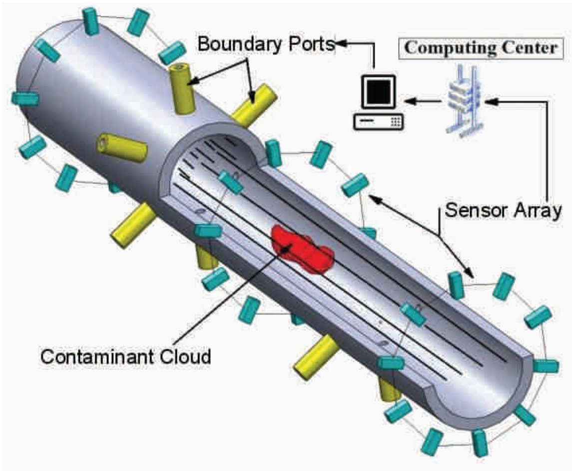

Figure 1. CONTAMINANT CLOUD ELIMINATNG PROBLEM

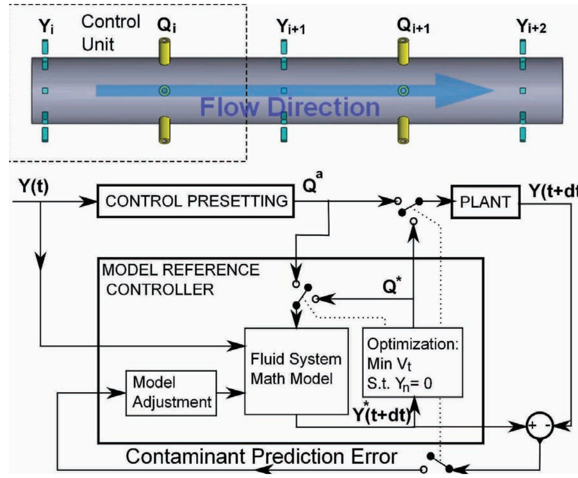

Figure 2. MODEL REFERENCE CONTROL ARCHITECTURE

ditional CFD solver cannot be used when fast control response is needed due to the computational expense of these algorithms. Even though many advanced CFD methods generate very precise solutions, delays caused by computing time cannot be avoided. Due to this reason, in this article, we develop an alternative fluid modeling method that is simple enough to implement predictive control on physical prototype real-time control system.

A simplified contaminant cloud elimination problem is described in Fig. (1). The control system is composed of sensor arrays, boundary actuator ports (at strategically located points on fluid boundary), target contaminant cloud and computer. With position of the contaminant cloud captured by sensor arrays, the optimal control strategy is calculated and control action is then assigned to each boundary port. The objective is to eliminate all of the contaminant cloud from the fluid system through these ports. The model reference predictive control architecture is illustrated in Section 1. In order to predict the trajectory of contaminant cloud, the fluid system mathematical model used in the optimal control iteration process must be solved in very short time to ensure the system response. A method for building such model solving for flow steady state path line is discussed in Section 3. In Section 2, we describe the lab-scale prototype that was constructed for experiments.

NOMENCLATURE

Y_i Contaminant cloud location information provided by i^{th} sensor array; i = 1,2...n; n = number of sensor arrays

Y Array containing all Y_i; Y = [Y_1 Y_2 ... Y_n]

Y^* Reference contaminant cloud location, predicted by math model

Q_i Control command (Volumn flow rate) assigned to i^{th} boundary port array; i = 1,2...m; m = number of boundary port sets

Q Array containing all Q_i; Q = [Q_1 Q_2 ... Q_m]

Q^a Initial control command generated by pre controller

Q^* Optimal control generated by model reference controller

V_t Volume of uncontaminanted fluid that is drawn away by boundary ports

V_c Volume of contaminant that is drawn

1 MODEL REFERENCE CONTROL ARCHITECTURE

As shown in Fig. (2) top, the complete physical plant is composed of several control units which are connected in series. Each unit is assumed to have identical dynamics and contain one set of sensor array and one set of boundary ports. The contaminant cloud elimination process is carried out step by step through the entire plant (From left to right in Fig. (2)). For example, contaminant cloud is detected at Y_i, but Q_i ports are not capable of eliminating all of it. The remaining contaminant is then confirmed and eliminated by Y_{i+1}, Q_{i+1} and continue at Y_{i+2}, Q_{i+2}.

In Fig. (2) control loop, when the contaminant cloud is detected and captured in Y at time t (denotS Y(t)), the CONTROL PRESETTING generates control command, Q^a, by searching through an existing database which will be most likely get generated in future work by processing the model presented here in Section 3. This feed forward control command may not be optimal but aims at providing fast response to the occurrence of a contaminant cloud. At the same time, $Y(t)$ and Q^a are send to a predictive optimal control algorithm in the MODEL REFERENCE CONTROLLER.

An optimization process is carried out to find optimal control strategy in the model reference controller. The optimization objective is to maximize the removal of contaminated fluid while minimizing the amount of uncontaminated fluid that is drawn away, under the condition that no contaminant is detected at last sensor array, $Y_n = 0$. The initial starting point is set by Q^a. Then fluid system math model predicts future contaminant cloud location and calculates the optimization objective and constraints. At

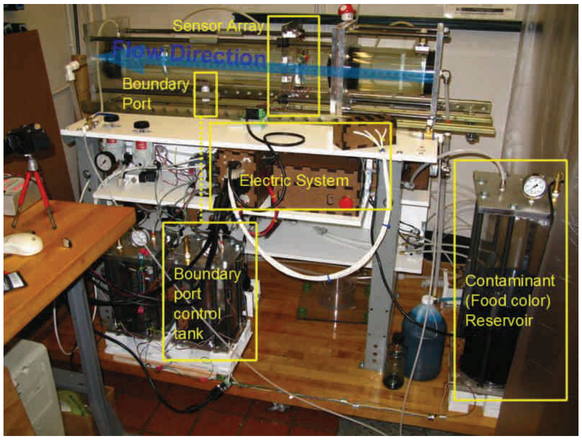

Figure 3. COMPLETE PROTOTYPE SYSTEM (Computing Center Not Shown In Figure)

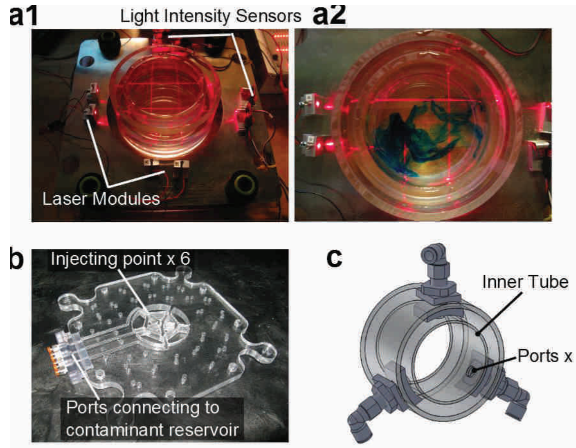

Figure 4. SENSOR ARRAY (a1-no contaminant, a2-with contaminant); CONTAMINANT INJECTION (b); BOUNDARY PORTS STRUCTURE (c)

the end of the optimization process, optimal control action Q^* is sent out to correct the original control action generated by pre-controller; Predicted contaminant location Y^* at next time step is compared with actual measured value and is used to adjust the math model parameters. Further more, adjusted model and optimal control strategy can be stored to enrich the database used by the CONTROL PRESETTING, so that future system response is continuously improved.

2 PHYSICAL SYSTEM

In order to study and construct a mathematical model for each control unit in Fig. (2), we start the prototype with one set of sensor array and one boundary port. Bulk fluid media and contaminant is selected as tap water and blue food color respectively. The prototype is designed for laminar flow, which is characterized by Reynold number. In fully developed pipe flow, Reynold number smaller than 2300 indicates laminar flow. Based on Reynold number equation, $Re = \frac{V \times D}{\nu}$, where V is flow velocity [m/s] and D is pipe diameter [m], ν the kinematic viscosity, prototype testing tube diameter is selected as 4 inch (0.1016 m) with nominal bulk flow rate below 2 cm/s. The calculated Reynold number is below 2000.

2.1 Prototype

The complete prototype is shown in Fig. (3), where the transparent round tube at top is the main testing region with 4 inch (101.6 mm) in diameter and 36 inch (914.4 mm) in length. In which, clear tap water flows from right to left; Light intensity sensors are installed outside of the pipe, as shown in Fig. (4)a, so that the sensors do not influence fluid dynamics. Low power laser beam pass through transparent tube wall and tap water. The presence of color cloud blocks laser beam, resulting in voltage drop from light intensity sensors; In order to minimize turbulence, boundary ports are carefully designed and situated outside of the flow path as shown in Fig. (4)c. Figure. (4)b shows 6 individual injection points, which enable us vary the color cloud's initial condition.

During experiment, the nominal flow rate is kept low and constant to ensure laminar flow for simplicity; color is injected and detected by sensor array; control action is then assigned to the boundary port by controlling pressure in a water tank, which is connected to the port as shown in Fig (3) (Boundary Port Control Tank). For example, if the pressure assignment is lower than the tube operating pressure, fluid flows out from the main tube.

2.2 Extreme Case Experiment

Currently, one open loop control case study was performed corresponds to the worst case scenario when the color cloud has spread through out entire tube cross section. Snapshots of color cloud elimination process are shown in Fig. (5). At the beginning, clear water flows leftward. At $t = 5$ sec, the color cloud was injected and dispersed. After the detection of the color cloud, boundary port (located at the bottom of each snapshot) started drawing fluid. In this extreme case experiment, because color cloud cover entire cross section, the only reasonable control action is to draw all fluid from the tube to eliminate all color cloud. The boundary port flow rate required to achieve this objective was pre-determined during the process for sizing our prototype actuators.

Although this is an extreme case, it proves that boundary control is capable of eliminating all contaminant by removing all bulk fluid. So, in the complete multi boundary control problem, there always exists at least one solution to the elimination problem: that is, use only one port to draw all bulk fluid at the cost of eliminating uncontaminated fluid together with contaminant cloud. Thus an optimization control strategy for multi boundary control can be implemented and targeted to minimize the volume of eliminated uncontaminated fluid.

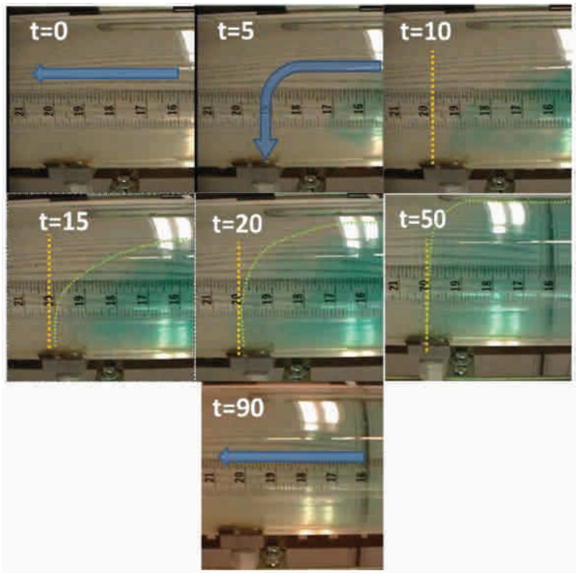

Figure 5. EXTREME CASE EXPERIMENT

3 MATHEMATICAL MODEL DEVELOPMENT

An efficient mathematical model is crucial for implementing predictive and iterative optimal control. The lab-scale prototype was simulated using Ansys Fluent with medium meshing size. The CFD solver requires more than 10 minutes for one solution. However, in the real-time prototype problem, if the fluid velocity is only 1 cm/s with 40 cm distance between boundary port and sensor array, the contaminant passes the controlling location within one minute and no optimal control can be calculated in this time period. As a result, the model reference controller losses all its functionality because of the long computing time. Thus, iterative method requires a reduced order model which can be solved very quickly and provide relatively accurate solutions. In this section, we illustrate a method to construct such reduced order model. The model solves for flow path-lines, which is a good representation of contaminant cloud spacial trajectory.

One further assumption is made for simplicity that the flow follows the steady state path lines, which is a good approximation for transient path line. The condition for this assumption are: 1) Boundary port flow rate changes do not have long-distance effects on path line shape; As shown in Fig. 6, in each CFD simulation (Done by Ansys Fluent), path lines are almost identical at left-most region no matter how we change fluid initial condition and boundary control flow rate. We denote above region as unaffected region. 2) The time period between contaminant detection and flow steady state settling is short, so that contaminant cloud stays in the unaffected region within this time period. Based on above assumptions, the ideal reduced order model should be able to generate fluid steady state path lines and be calibrated by CFD simulations. Six CFD simulation cases are shown in Fig. 6 under typical fluid conditions. The first 4 CFD simulations are boundary drawing cases and 5, 6 are injections cases. 1, 2, 3 and 5 have

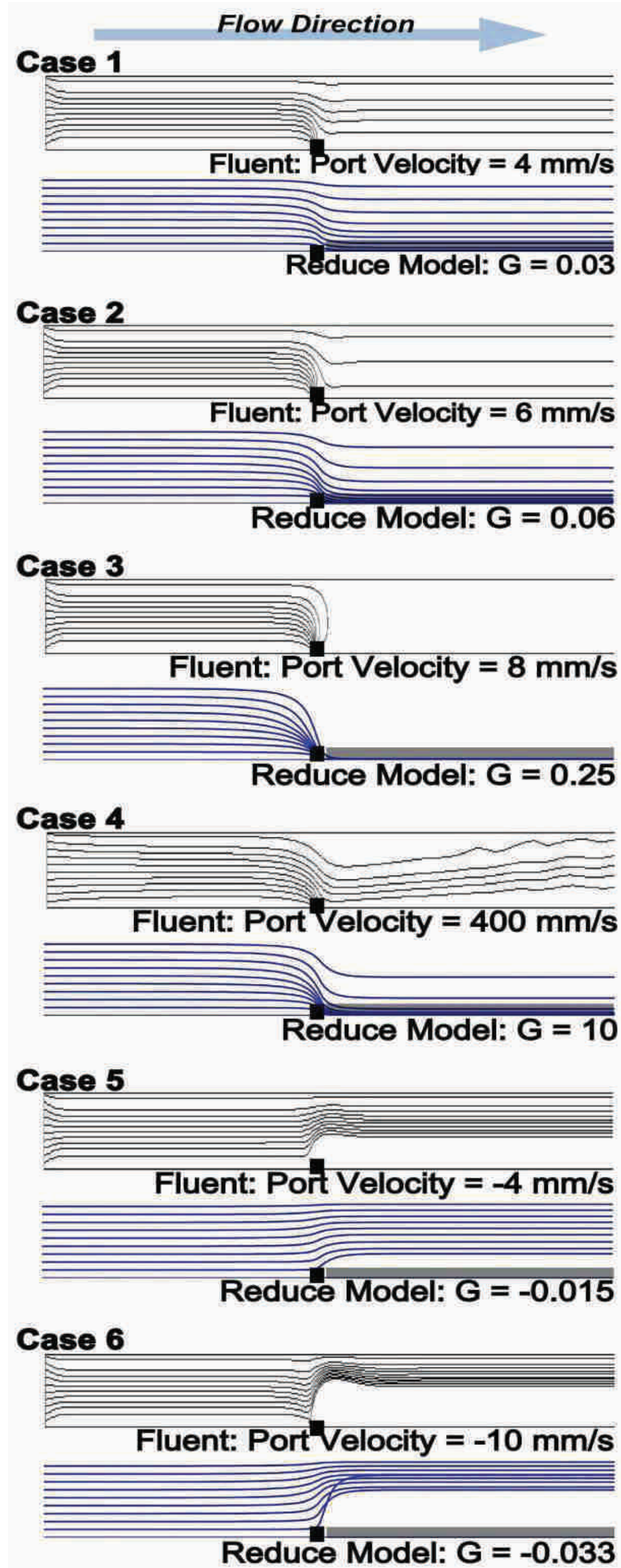

Figure 6. PATH-LINE SIMULATION COMPARISON, $c_1 = 10, x_{pen} = 0.1, c_2 = 10, a_1 = 1, a_2 = 1$. Positive setting represents drawing and negative for injecting.

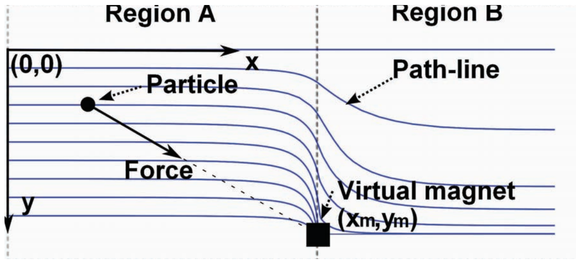

Figure 7. PHYSICAL INTERPRETATION OF MATH MODEL

identical initial and boundary conditions with different boundary control flow rate. In each of these 4 cases, flow is laminar. However, in the fourth case when bulk fluid flow rate and control flow rate is increased, flow become turbulent. Turbulence also occur in case 5 when the boundary injection rate is increased. These simulation tell us: 1) Boundary injection (negative port velocity) is more likely to produce turbulence compared to boundary drawing (positive port velocity); 2) Although boundary drawing is mostly safe, i.e, not causing turbulence, bulk fluid velocity need to be taken into account. Although we calculated laminar condition in Section 2, flow may be interrupted by boundary control and bulk flow velocity should be slower than the calculated value. 3) Flow may still stay laminar with very small injection flow rate. In summary, the reduced order model should reproduce path line for mostly boundary drawing and some slow injection.

3.1 Constructing Reduced Model

The idea behind this model is to remove all the process of solving fluid PDEs on-line, which is usually computational intensive. Further more, measured contaminant cloud location is compared with the predicted one only at the locations of sensor array. Thus, the model accuracy for predictive control is very important at sensor locations and the model structure should enable model adaptation using these measurements. With these thoughts, a two-mode-hybrid ODE system with tunable parameters is constructed. System (1) in Eq. 1 solves path-line in region A in Fig. 7, while system (2) in Eq. 4 solves the rest. System states x_1 to x_4 are, respectively, particle x coordinates, velocity x component, y coordinates and velocity y component. Initial conditions are the detected contaminant cloud location for x_1, x_3, flow velocity for x_2 and zero for x_4. G is the reduced model control input.

The system equations can be interpreted in a physical way: A virtual magnet replaces the boundary port and contaminant cloud is replaced by particles that reacts with the magnet as shown in Fig. 7. In second and fourth line of Eq. 1, $\frac{G}{r^{a_1}}$ represents force between magnet and particle. G can be viewed as the strength of magnet and r be the distance between magnet and particle. In analogy to boundary control, positive G represents drawing fluid and negative represents injecting fluid. The magnitude of G is related to boundary port flow rate. Although the relation between G and flow rate has not been fully determined, nonlinearity is observed.

SYS.1 :

$$
\begin{aligned}
\dot{x_1} &= x_2 \\
\dot{x_2} &= \frac{G}{r^{a_1}} \times \frac{x_m - x_1}{r^{a_2}} \times x_{pen} \\
\dot{x_3} &= x_4 \times z_1 \\
\dot{x_4} &= \frac{G}{r^{a_1}} \times \frac{y_m - x_3}{r^{a_2}}
\end{aligned} \tag{1}
$$

In this system equations, a_1 is analogous to the distance square term in gravity equation denominator, defining how force increases when particle getting closer to magnet. The variables, x_m and y_m, are the location of magnet. The terms. $\frac{x_m - x_1}{r^{a_2}}$, $\frac{y_m - x_3}{r^{a_2}}$ and x_{pen} are added as penalties to limit particle acceleration. The variables a_1, a_2, x_m and x_{pen} are set as constant and y_m described by Eq. 2 is the magnet y coordinates written as a function of particle location:

$$
y_m = x_{3ini} + \frac{D - x_{3ini}}{x_m} \times x_1, \tag{2}
$$

where x_{3ini} is the initial condition set for x_3. To interpret this equation, magnet gradually moves from particle initial location to tube boundary as particle moves rightward. Thus, particle does not have y acceleration when it is far away from magnet. In real flow system, boundary port action does not affect contaminant cloud trajectory when the distance is far.

The variable, z_1, is used to ensure that particle does not penetrate fluid boundary

$$
z_1 = 1 - exp\left[c_1 \times \left(\left|x_3 - \frac{D}{2}\right| - \frac{D}{2}\right)\right], \tag{3}
$$

where c_1 is a constant and D is the pipe diameter. This equation decays the y velocity of the particle when the particle gets closer to fluid boundary. This is analogous to boundary zero slip condition in laminar steady state flow. It is important to notice that, in CFD simulation, path-lines converging to port correspond to fluid escaping through the boundary. But in the reduced model, particles can never penetrate the boundary. Thus, a method to define an escaped particle must be added to the model. One potential method is create an 'escaping region', shown as the shaded region in Fig. 6. All particles go into this region are labeled to be 'escaped'.

Now we have a solvable ODE system. However, because a magnet alway applies as attracting force, the particle turns back and reverses its direction in region B, which can never happen in real pipe flow. So we need another model which reduces the

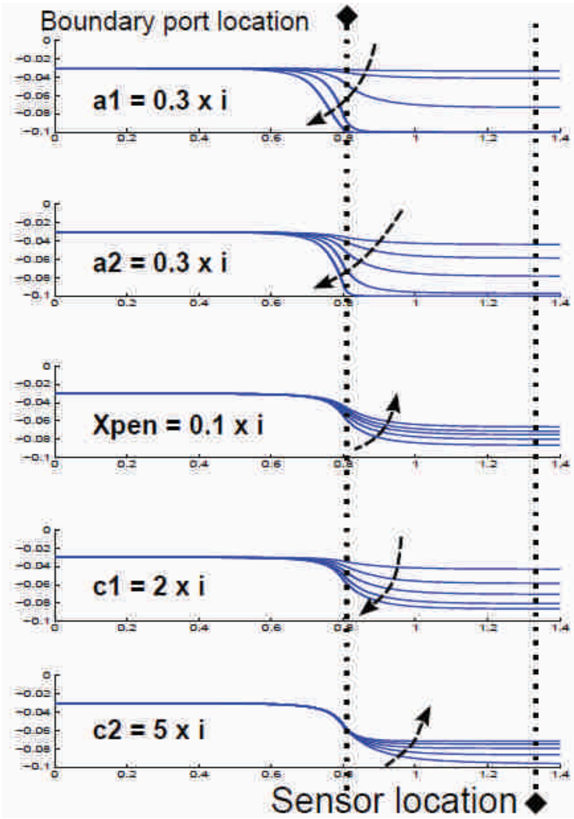

Figure 8. MODEL PARAMETER INFLUENCE ON SOLUTION, $G = 0.1, IC = [0,1,0.03,0]$, $i = 1 \cdots 5$, Arrow direction = i increasing

effect of the magnet when the particles go into region B. A simple way is to switch to a new ODE system. In the new ODE system, acceleration term $\dot{x_2}$ and $\dot{x_4}$ are forced to zero and we let y-velocity decay over time. The new system is described by Eq. 4:

SYS.2 :

$$\begin{aligned} \dot{x_1} &= x_2 \\ \dot{x_2} &= 0 \\ \dot{x_3} &= x_4 \times z_1 \times z_2 \\ \dot{x_4} &= 0 \end{aligned} \tag{4}$$

where y velocity decays by z_2:

$$z_2 = exp(-c_2 \times (x_1 - x_m)) \tag{5}$$

where c_2 defines the decay rate.

The fluid system path line approximated by the above system equations has six tunable parameters, which makes this math model very flexible. The simulations shown in Fig. 8 are carried out for the same particle trajectory under same initial condition and G with different model parameter setting. By adjusting a_1 and a_2, we can manipulate when path line starts bending towards boundary port. c_1, c_2 and x_{pen} are used to adjust path line shape to the right of boundary port. As mentioned at the beginning of this subsection, the accuracy of the predicted path-lines should be high at the sensor locations but not necessarily near the boundary ports area. Theoretically, the predicted path line value at sensor locations can be freely assigned by careful parameter tunning, thus accurately predicting the contaminant cloud location at sensor points.

3.2 Reduce Model Parameter Study With CFD Simulations

In cases 1, 2 and 3 shown in Fig. 6, CFD simulations indicate that more path-lines converge to boundary port as the flow rate at boundary port increases. The reduced model can accurately predict this behavior. In case 4, when laminar is retained, reduced model can still generate solutions. However, in case 5 and 6, the model failed to approximate turbulence fluid conditions. These two cases indicate thresholds when turbulence occurs and the reduced model fails.

Fig. 9 illustrates a quantized comparison between one CFD simulation and reduced model results in 2D case when laminar is retained. In the figure, solid lines are CFD simulation results when bulk flow velocity is 20 mm/s at left and boundary port flow velocity is 2 mm/s outwards. Stars are used to show the reduced model ODE numerical solution points. Relation between reduced model parameters and flow conditions are still under investigation, however good results proves the potential of such model for control application.

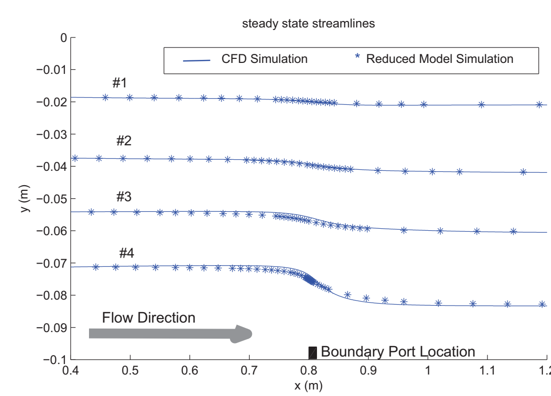

Figure 9. Quantized comparison between CFD simulation and Reduce model.

In summary, the reduced model is solved by common ODE solver in less than 1 second while CFD model took more than 15 second on the same computer for 2D problem. Further more, the reduced model can be easily expended to 3-dimensional by adding 2 more ODE equations to the system without increas-

Table 1. REDUCED MODEL PARAMETERS AND INITIAL CONDITIONS FOR SIMULATIONS IN Fig. 9

#	Initial Condition	Parameters
	[x Vx y Vy]	[a_1 a_2 $x_p en$ c_1 c_2 G]
1	[0 1 -0.019 0]	[0.876 0.427 0.254 3.702 13.60 0.12]
2	[0 1 -0.037 0]	[0.874 0.441 0.246 3.540 13.61 0.12]
3	[0 1 -0.054 0]	[0.889 0.448 0.374 3.598 13.60 0.17]
4	[0 1 -0.071 0]	[0.854 0.702 0.000 2.024 13.07 0.76]

ing computation time by a large amount. But in CFD method, solving a 3-dimensional problem is a lot more difficult than 2-dimensional one.

In addition, for laminar flow problem, the proposed modeling method provides high flexibility and the reduced model can be tuned by CFD simulation and prototype experiment. Because of its short computing time, the reduced model has high potential to serve optimal predictive control purpose.

On the other hand, flow conditions under which this math model fails need to be further investigated. For example, boundary port flow rate, bulk flow velocity and system physical dimensions. Prototype experiments can be better designed based on above study and problem of higher complexity can be solved.

4 SUMMARY AND FUTURE WORK

A reduced order math model for simulating fluid path-lines is constructed. Thus, contaminant cloud spatial trajectory can be predicted in relatively short time compared to traditional CFD approach.

This model is highly adaptable and can be tuned using prototype testing data and CFD simulation. However, the accuracy and limitation of this model need to be further studied to expand to other laminar flow boundary control application. Based on this reduced order model, a model reference predictive control architecture was established. Future experiments are planed to validate model and control algorithm by using lab scale prototype. Additional sensors and boundary ports will be added to study optimal utilization of multiple boundary ports.

ACKNOWLEDGMENT

Thanks go to National Science Foundation. This material is based in part upon work supported by the National Science Foundation under Grant Numbers CMMI 0856438.

REFERENCES

[1] Y. Jarny, M. N. Ozisik, J. P. B., 1991. “A general optimization method using adjoint equation for solving multidimensional inverse heat conduction”. *Heat Mass Transfer,* **34**, pp. 2911–2919.

[2] Michael Piasecki, N. D. K., 1997. “Control of contaminant release in rivers. i: Adjoint sensitivity analysis”. *Journal of Hydraulic Engineering,* **123**, pp. 486–492.

[3] T. R. Bewley, P. Moin, R. T., 2001. “Dns-based predictive control of turbulence: an optimal benchmark for feedback algorithms”. *Journal of Fluid Mechanics,* **447**, pp. 179–225.

[4] Jhon Kim, T. R. B., 2007. “A linear system approach to flow control”. *Fluid Mechanics,* **39**, pp. 383–417.

[5] O. M. Aamo, M. Krstic, T. R. B., 2003. “Control of mixing by boundary feedback in 2d channel flow”. *Automatica,* **39**, p. 15971606.

[6] A. Balogh, O. M. Aamo, M. K., 2005. “Optimal mixing enhancement in 3-d pipe flow”. *IEEE Transactions on Control Systems Technology,* **13**, pp. 27–41.

[7] R. Wu, N. D. K., 2006. “Control of chemical spills by boundary suction”. In Proceedings of the 2006 WSEAS/IASME International Conference on Fluid Mechanics.

[8] Katopodes, N. D., 2009. “Control of sudden releases in channel flow”. *Fluid Dynamics Research,* **41**(065002), p. 24.