

# Computational Fluid Dynamic Solver Based on Cellular Discrete-Event Simulation

Michael Van Schyndel, Gabriel Wainer, Mohammad Moallemi

*Dept. of Systems & Computer Engineering,*

*Carleton University, Ottawa, ON, Canada*

*mvschynd@connect.carleton.ca, {moallemi, gwainer}@sce.carleton.ca*

**Keywords:** Cell-DEVS, Computational Fluid Dynamics.

## Abstract

Computational Fluid Dynamics (CFD) deals with computing the equations of fluid flows using numerical methods. The Discrete-Event System specification (DEVS) theory has been used to approximate the continuous systems by applying a quantized state system approach. In this research, we employ Cellular DEVS theory (Cell-DEVS) – originally proposed for modeling and simulation of spatial environments – to create a uniform set of rules for CFD. This harmonized set of state changes can effectively render the fluid dynamics, by applying the accurate rule that represents the behavior of the fluid. The combination of the simplicity and the mathematical backbone allows for constructing models computable on an average computer or an array of cluster computers.

## 1. INTRODUCTION

Computational Fluid Dynamics (CFD) solving is referred to the research on numerical methods and algorithms to solve and analyze the movement and interactions of fluid flows (Anderson, 2009). In general, no analytical solution exists for non-linear fluid models; hence, the numerical approximation methods also called “computational models” come to play. CFD Solvers are required to process a large number of computations, which makes the use of computer-based approaches inevitable. In a computerized processing of CFD, a boundary for the problem is defined and the environment is divided into cellular spaces, each of them representing a physical volume. The laws of motion are defined based on equations of motion, enthalpy, radiation, and species conservation. The behavior of the fluid at the boundaries is also defined, which is called boundary conditions. These specifications construct a model of the fluid, which can be simulated on a computing device. Finally, visualization and analysis of the results can render a meaningful and sensible outcome of the computations. Different cellular methods have been proposed to solve these problems. In particular, Cellular automata (CA) theory (Ilachinski, 2001) is a branch of discrete dynamic systems, in which space is represented by a cellular grid, with each cell being a

state machine. In CA the time advances in a discrete manner, triggering state changes in the cells, based on the value of their neighbor cells. CA have been used in physics, complexity science, theoretical biology, microstructure modeling, etc.

The Cell-DEVS formalism (Wainer, 2009) is an improved derivative of CA, which solves the problem of unnecessary processing burden in cells and allows efficient asynchronous execution, using a continuous time-base, and without losing accuracy. In this methodology, each cell is represented as DEVS atomic models (Zeigler, Praefer Kim, 2000) that changes states in an event-driven fashion. In this research, we propose using Cell-DEVS to implement CFD equations to simulate fluid dynamics. The rule-based nature of cellular model behavior definition provides a platform for area-wise behavior definition, leading to easier and faster experimentation of CFD solvers. The other advantage of this method is its fast computing apparatus working asynchronously on the cellular grid, increasing the execution speed. The continuous time-advance nature of Cell-DEVS can contribute to the seamless simulation of CFD, in comparison with the discrete timing in CA that lacks the smoothness of fluid flow. The model can be able to provide realistic results with reasonable speed. Finally, the formal I/O port definitions in the formalism permits producing output signals based on specific condition satisfaction in the cell lattice, allowing for data transfer between different spatial components.

The CD++ tool (Wainer, 2009) provides a development environment to create and navigate through the process of Modeling and Simulation (M&S) of a Cell-DEVS model. CD++ is an open-source framework that has been used to model environmental, biological, physical and chemical models as well as many other real-life simulations. The toolkit includes a high-level scripting language keyed to Cell-DEVS, a simulation engine, a testing interface and 2D and 3D graphical interfaces.

The solver proposed here provided results that are realistic and achieve the goals stated. We will

discuss how the framework can be used and how to export the generated data to graphical environments.

## 2. RELATED WORK

Fluid dynamic solvers are used for a wide variety of purposes. Their goal is to create a realistic representation of a naturally occurring fluid system such as smoke rising or dust blowing. The flow of fluids can be viewed as solid particles interacting with velocity fields or as densities. There are different methods to solve some CFD; Lattice-Gas method (Chen and Doolen 1998), Navier-Stokes Equations (Stam, 2003) and Riemann Solvers (Currie, 1974).

Over the years there have been many of the methods used for solving fluid dynamics have been implemented using CA. In general, CFD methods are categorized into two groups; a) Discretization methods and b) Turbulence models. Discretization methods are a subset of divide and conquer method in solving difficult computational problems, in which the computational domain is discretized and “each term within the partial differential equation describing the flow is written in such a manner that the computer can be programmed to calculate” (Frisch et al, 1986). In Turbulence models, the focus is in computing the interest factors in the Fluid dynamics. A range of length and time scales of the fluid movement are modeled, in which, the more scales that are resolved, the better the granularity.

Navier-Stokes equations were the first physical description of fluid motion by applying Newton’s second law of motion “with the assumption that the stress in the fluid is the sum of a diffusing viscous term (proportional to the gradient of velocity) and a pressure term” (Sukop, 2006). The first comprehensive simulation of the d-dimensional Navier Stokes equations appeared in (Frisch et al, 1986). In (Sukop, 2006), the author provides a method for creating a basic model of 2D fluid flow, by mapping the possible collisions that can occur and the outcomes that are determined by a set procedure. The randomness generated by these procedures that is essential to its ability to simulate flows. This procedure does provide results; however, with the standard of ever realism increasing, its ability to provide a realistic model is substantially limited.

A similar model was made to model the effect of polymer chains on fluid flow in (Koelman 1992) where a lattice-gas automata was used to provide a 2-dimensional model. It was noted that further work must be done to develop a method of using the lattice-gas method to provide a 3-dimensional model that was able to provide realistic results with a rea-

sonable computational effort. In (Koelman and Nepveu 1992) the authors demonstrate how it is possible to use a CA to model flow through a porous material. They were able to model a one-phase Darcy automaton based on a Navier-Stokes automaton, however when they implemented a two-phase Darcy automaton they had to implement much simpler local transition rules. In a research presented in (Stam, 2003), the Navier-Stokes equations are used to model the fluid dynamics. While the algorithms implemented do not meet the formalism of CA, they do share several key characteristics. A cell lattice is spanned over the simulation window with each cell holding unique information regarding that particular area. The first difference is that each cell space stores a density value and the horizontal and vertical velocity components (the z component for a 3-dimensional model). The cell spaces are updated simultaneously at discrete time intervals. While in a true CA, each cell can be updated independent of other cells, and the algorithms must solve multiple steps for all the cells before the final value is obtained. Nevertheless, the algorithm provided very realistic results with a limited computational effort by utilizing a rather basic set of rules, and has potential to be adapted as for Cell-DEVS.

In this paper, we will use the algorithms presented by (Stam, 2003) to create a CFD solver that falls within the Cell-DEVS formalism and will be implemented using the CD++ toolkit. This method was chosen because the technique used was already similar to that of a Cell-DEVS model and therefore would most likely be the easiest to implement. Besides, the results generated by the algorithms seemed to generate the best/most realistic results. The most significant hurdle that will have to be overcome is changing the updating of the cells from synchronous to an asynchronous process.

## 3. MODEL DEFINITION

The model in (Stam, 2003) was based on the Navier-Stokes equations for solving simple fluid flow. Equation 1 is the Navier-Stokes equations for velocity and density moving through a velocity field.

**Equation 1**

$$\frac{\Delta u}{\Delta t} = -(u \cdot \nabla)u + \nu \nabla^2 u + f$$

$$\frac{\Delta p}{\Delta t} = -(u \cdot \nabla)p + k \nabla^2 p + f$$

The model treats the fluid space as a 2D grid space. The fluid is projected as a movement of densities instead of particles and therefore each cell contains the density for the given cell area. In Cell-

DEVS, each cell must contain all the additional information as well as the set of rules that are used to determine the cell values in the future. The model solves the density in a 3 step process as seen in Figure 1. The diffusion of the densities is first calculated using Equation 1. Then the densities are “moved” by examining the forces from the vector field and determining their new locations.

To do this correctly and realistically, the “forces” or the velocity fields must be evolving as well. The model must create realistic eddies and swirls in the appropriate places. The process of implementing this is even more complicated.

The change in the velocity vectors are due to three main reasons; the addition of forces over time, the diffusion of the forces and the self-propelling nature of the forces. The diffusion of the forces is calculated similar to the densities, as well as the advection/movement stage. The new stage is called the projection. The projection stage allows for the velocities to be mass conserving. Additionally, this step improves the realism of the model by creating eddies that provide realistic swirling flows.

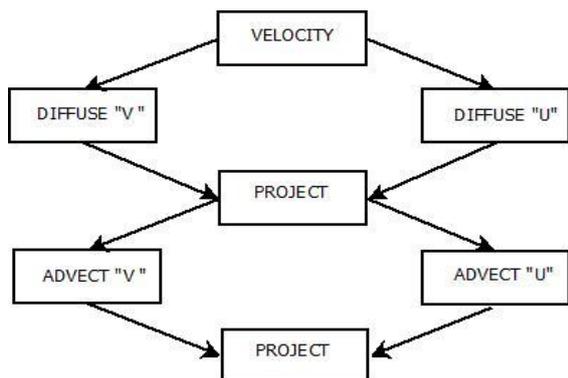


Figure 1. Velocity Solver steps

While the framework and execution of the model may vary from its predecessor, the results are ideally the same. At the end of every cycle, the densities have been diffused and moved and the velocity fields have been updated. The external forces and densities are added and ready to begin with the next frame.

#### 4. RULES IN CELL-DEVS

A Cell-DEVS model is a lattice of cells holding state variables and a computing apparatus, which is in charge of updating the cell states according to a local rule. This is done using the current cell state and those of a finite set of nearby cells (called its neighborhood). Cell-DEVS improves execution performance of cellular models by using a discrete-event approach. It also enhances the cell’s timing definition

by making it more expressive. Each cell is defined as a DEVS atomic model, and it can be later integrated into a coupled model representing the cell space.

Cell-DEVS models in CD++ are built following the formal specifications of Cell-DEVS and using a built-in language is provided to describe the behavior of each cell. The model specification includes the definition of the size and dimension of the cell space, the shape of the neighborhood and borders. The cell’s local computing function is defined using a set of rules with the form POSTCONDITION DELAY {PRE-CONDITION}. These indicate that when the PRECONDITION is satisfied, the state of the cell will change to the designated POSTCONDITION, whose computed values will be transmitted to other components after consuming the DELAY. If the precondition is false, the next rule in the list is evaluated until a rule is satisfied or there are no more rules.

Since the cell states are calculated asynchronously, each cell must contain the following information: the density of the fluid for that cell space, the velocity vectors  $u$  and  $v$  as well as all the intermediate calculations. The cell space will be layered with each layer holding a different piece of information for its corresponding cell as shown in Table 1.

Table 1. Cell Space Layers and Range of Values

Name	Function	Values Used
<b>Temp u</b>	Handles the advection of "u" component of the velocity vector.	Range: (-2, 2) Positive = Left Negative = Right
<b>Temp v</b>	Handles the advection of "v" component of the velocity vector.	Range: (-2, 2) Positive = Up Negative = Down
<b>U</b>	Handles the final projection step & stores the final values for "u" of the velocity vectors.	Range: (-2, 2) Positive = Left Negative = Right
<b>V</b>	Handles the final projection step; stores the final values for the "v" of the velocity vectors.	Range: (-2, 2) Positive = Up Negative = Down
<b>Div</b>	Handles the initial projection step defined as <i>div</i> in the algorithms	N/A
<b>P</b>	Handles the second projection step, defined as <i>p</i> in the algorithms	N/A
<b>Diff'</b>	Handles the diffusion of the densities	Range: (0, 1) values in gradient, 1 = solid & <1 is a density
<b>Source</b>	Stores densities during the density calculations	Range: (0, 1) as above
<b>Final</b>	Handles advection of the densities and represents the solution to the density	Range: (0, 1) as above

	solver	
--	--------	--

Each distinct part of the algorithm, which is defined later, will make use of one or more layers, and therefore it is very important that the layer information be exact.

#### 4.1. Diffusion

The diffusion can be calculated by taking the initial density value of the cell and adding the scaled sum of the densities that could enter that cell from the surrounding cells and then calculating the average. The result is a flow of density from higher to lower concentration. By looping this function, we are able to extend the diffusion to cells outside the neighborhood; however, with a low value for a cell, these values are negligible at  $n \pm 2$  from the cell. The diffusion step is defined with the following equation (Stam, 2003):

##### Equation 2 Density Formula

$$x(i,j) = \frac{x(i,j) + (x(i-1,j) + x(i+1,j) + x(i,j+1) + x(i,j-1))}{1 + 4a}$$

The implementation of this step is relatively easy. The values of  $x$  are stored in the Source layer and the values of  $x'$  are stored in the Final layer. For each step the function is run for 20 cycles and on the 20th cycle the value is stored and the cycle is reset. The “if” statement in CD++ operates as expected, however by looking at the timing information we were able to change its behavior to that of a loop, where  $n=20$  and after each cycle it restarts at zero. The resulting code looks like the following:

```
rule:{ if(remainder(time,20)=0,
      (0,0,2), ((0,0,2)+0.1*
      ((1,0,1)+(0,1,1)+(0,1,1)+
      (1,0,1))) /1.4 ) } 1 { t }
```

As it can be seen, every time the *time* variable reaches a multiple of 20 (i.e. 20 cycles passed), it is reset. Otherwise, the new density is recalculated based on the current density, and the weighted average of the surrounding cells. The amount the average is weighted is by the variable 'a' which in this situation is 0.1.

#### 4.2. Advection

The advection step is responsible for the movement of densities and velocity fields. The most obvious method of determining where a *density* will end up is to trace it forward based on the velocity field. However, the method described by (Stam, 2003) suggests starting in the center of the cell space and trace backwards to find the origins, based on the ve-

locity field. Then, at this point take the weighted average of the four closest cells to determine the source density. This is done because the source will most likely not fall directly in the middle of the cell and therefore the surrounding densities would affect the new densities. The advection step as it appears in the original algorithm is as follows:

```
void advect ( int N, int b, float *
d, float * d0, float * u, float * v,
float dt ) {
  int i, j, i0, j0, i1, j1;
  float x, y, s0, t0, s1, t1, dt0;

  dt0 = dt*N;
  for ( i=1 ; i<=N ; i++ ) {
    for ( j=1 ; j<=N ; j++ ) {
      x = i-dt0*u[IX(i,j)];
      y = j- dt0*v[IX(i,j)];
      if (x<0.5) x=0.5;
      if (x>N+0.5) x=N+0.5;
      i0=(int)x; i1=i0+1;
      if (y<0.5) y=0.5;
      if (y>N+0.5) y=N+ 0.5;
      j0=(int)y; j1=j0+1;
      s1 = x-i0;    s0 = 1-s1;
      t1 = y-j0;    t0 = 1-t1;
      d[IX(i,j)] = s0*(t0*d0[IX
        (i0,j0)]+t1*d0[IX(i0,j1)])
        + s1*(t0*d0[IX(i1,j0)] +
          t1*d0[IX(i1,j1)]);
    }
  }
  set_bnd ( N, b, d );
}
```

This step is used to trace the origins of the current density by looking at the vector field. Since the origin is not likely to be at a cell center a weighted average of the surrounding four cells is taken, with their weight dependent on their proximity to the origin location.

In order to model the advection step is in Cell-DEVS we are required to include any of the cells where the density can originate, in the neighborhood. The first and most important part is ensuring that the possible source cells are included within the neighborhood. The neighborhood of the advection step is defined 6 by 6 cells therefore; the maximum distance a particle can travel is 2 cells from the center. Hence, the velocity vectors cannot exceed the range of (-2,2). This can be done by scaling the time step to ensure that the velocities remain within the acceptable limits. For example, if the velocity is 4, it can be scaled down to 2 and the new time step would be half of the original. The values of the cells are truncated to discrete values therefore, there are 5 potential values: -2,-1,0,1 and 2, for  $u$  and  $v$  and therefore 25 different

combinations of the two. For each combination, the ratio of the four source cells is calculated. The following is a portion of the code that determines the value if the truncated values of the  $u$  and  $v$  velocities are 1 and 1, respectively.

```

if( trunc((0,0,-6)) = 1,
if( trunc((0,0,-5)) = 1,
  ((1-remainder(abs((0,0,-6)),1))
  *((1-remainder(abs((0,0,-5)),1))
  *(-1,-1,-2)+ remainder(
  abs((0,0,-5)),1)*(-1,0,-2))+
  remainder(abs((0,0,-6)),1)*
  ((1-remainder(abs((0,0,-5)),1))
  *(0,-1,-2)+remainder(
  abs((0,0,5)),1)*(0,0,-2) )
)
) * this is 1 of 24 possibilities

```

In this code, it is checking to see if the  $u$  and  $v$  vectors fall within the range of 1.0 to 1.999. If this is the case then by the weighted averages are calculated and summed.

The code contains 25 iterations of the above code segment to cover the possible outcomes. This function is used 3 times in each cycle; the advection of the density, the advection of  $u$  and the advection of  $v$ . However, since the offsets of the required planes are the same for both  $u$  and  $v$  (the offset is 0), the function can be recycled to solve for both. The advection of the density step, however, requires access to a different plane with a different offset (2) and therefore must be rewritten with its corresponding neighbor values.

### 4.3. Projection

The projection step can be broken into three sub-sections: solving for  $div$ ,  $p$ ,  $u$ , and  $v$ . The original algorithm is implemented using the following code:

```

void project ( int N, float *u,
float *v, float *p, float *div) {
  int i, j, k; float h;
  h = 1.0/N;
  for ( i=1 ; i<=N ; i++ ) {
    for ( j=1 ; j<=N ; j++ ) {
      div[IX(i,j)] = -0.5*h*
      (u[IX(i+1,j)]- u[IX(i-1,j)]+
      v[IX(i,j+1)]-v[IX(i,j-1)]);
      p[IX(i,j)] = 0;
    }
  }
  set_bnd(N,0,div); set_bnd(N,0,p);
  for ( k=0 ; k<20 ; k++ ) {
    for ( i=1 ; i<=N ; i++ ) {
      for ( j=1 ; j<=N ; j++ ) {
        p[IX(i,j)] = (div[IX(i,j)]+
        p[IX(i-1,j)]+p[IX(i+1,j)]+
        p[IX(i,j-1)]+p[IX(i,j+1)])/4;
      }
    }
  }
}

```

```

set_bnd ( N, 0, p );
}
for ( i=1 ; i<=N ; i++ ) {
  for ( j=1 ; j<=N ; j++ ) {
    u[IX(i,j)] -= 0.5*(p[IX
    (i+1,j)]-p[IX(i-1,j)])/h;
    v[IX(i,j)] -= 0.5*(p[IX(i,
    j+1)]- p[IX(i,j-1)])/h;
  }
}
set_bnd(N,1,u); set_bnd ( N, 2,
v ); } // (Stam 2003)

```

The implementation of  $Div$  is straightforward. It takes the two  $u$  and two  $v$  values from their respective  $temp$  layers and is implemented with the following code snippet in CD++ Model file:

```

rule : { if( remainder( time, 20 ) = 0,
-0.05*((1,0,-4)-(-1,0,-4)+(0,1,-3)
- (0,-1,-3) ), (0,0,0) ) } 1 {t}

```

As can be seen, the “ $if$ ” function will work as a loop that is reset after each iteration. The calculations are essentially the same with the only difference on where and how the information is accessed. The  $-4$  at the end of each neighbor cell means those values are taken from the temporary layer for the  $u$  vectors while the cells with  $-3$  are taken from the temporary  $v$  vectors.

To solve for  $p$ , we use the same method as solving for the diffusion. The code segment is exactly the same as mentioned before, however the values of  $a$  are adjusted to reflect the viscosity instead of the diffusion coefficient.

The final step for the projection is the separating of the vector fields into component form. The separating of the horizontal and vertical components in our model is performed as it in the algorithm, presented below:

```

[u]
rule : { if( remainder(time,20)=0 ,
if( time=0, (0,0,-2), (0,0,-2)-
0.05*((-1,0,-6)-(1,0,-6))),
(0,0,0) ) } 0 {t}
[v]
rule : { if(remainder(time,20) = 0,
if( time = 0, (0,0,-2),
(0,0,-2)-0.05*((0,-1,-7)-
(0,1,-7))), (0,0,0) ) } 0 {t}

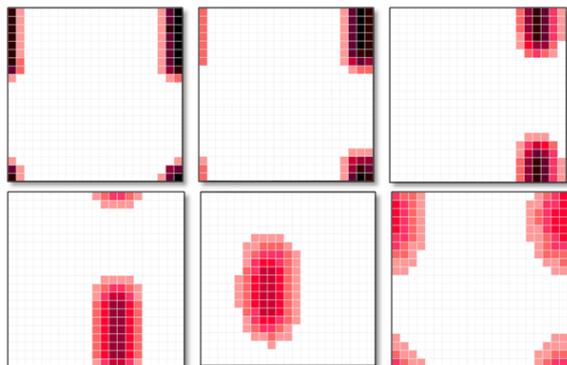
```

During the projection stage, we had added the velocity vectors together to make a single velocity field. However, for the rest of the algorithm we like to have the velocities in separate fields. These steps will be used twice for each cycle of the model.

## 5. SIMULATION RESULTS

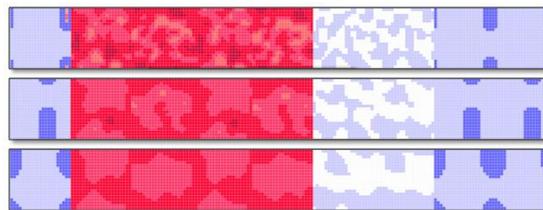
To test the model, we have executed several simulation scenarios. The first simulation was initialized as single foci of density with the velocity vectors being randomly generated to have an approximate value of 1, i.e. a velocity in the upwards diagonal to the left. The diffusion coefficient and the viscosity coefficient were both set to a low value in the range of (0.1). The expected result is that the density foci will spread to more of a cloud with the highest densities being on the leading edge of the cloud as it proceeds to the top left corner.

Figure 2 shows the results of simulation scenario using a cell space of 21 by 21 cells. The coefficient of diffusion ( $a$ ) was set to be 0.1. The density field was exposed to a velocity field whose  $u$  and  $v$  values were randomly set to a range of 0.9 to 1. The viscosity was set as 1. The results illustrated in Figure 2 are what we would expect to see in a real situation. The density cloud traveled up and left at an angle of approximately 45 degrees, which corresponds to the field applied to it. Additionally, the limited dispersion of the cloud reflects the low diffusive coefficient used. The slight teardrop shape that the cloud took which occurs when the density cloud is moving can be noticed. The concentration will be slightly higher on the leading edge and taper out at the end.



**Figure 2. Progression of Diffusion. Coefficient  $a=0.1$**

Figure 3 demonstrates how over time the velocity fields become more regular. Since the initial value assignment was random the field would not be stable. As the values did not vary too much ( $\max < 10\%$ ) the field soon become more evenly distributed and settled between the range of 0.94 to 0.96. This is expected since it was a uniform distribution between 0.9 and 1 and with a relatively larger viscosity, the fields would settle quickly.



**Figure 3. Demonstrating the evolving velocity field**

## 6. CONCLUSIONS

Fluid dynamic solvers are used in a wide variety of application ranging from video games and entertainment to modeling of environmental events. In this research, a CFD solver is proposed that reuses the parameters of a CA in Cell-DEVS. The asynchronous and more efficient computing grid of Cell-DEVS with the continuous time-base allowed for more realistic simulation of the fluid dynamics. We showed how CD++ toolkit was used to implement the Cell-DEVS model of the Navier-Stokes equations for CFD. we were able to create a fluid dynamic solver that met the requirements of a Cellular Automata, demonstrating that it is possible to create models of vary complex phenomenon using a relatively simple technique. While the model required significantly longer time to generate results, it provided a more detailed description of what is happening at every stage of the simulation and stored massive amounts of detail. As an initial implementation, this may not be a desired characteristic; however such a high level of detail would allow the model to be integrated easily to generate a more complex visualization of the fluid movement.

## REFERENCES

- Anderson, J., 2009. Basic philosophy of CFD. In: Computational Fluid Dynamics.. pp. 3-14.
- Chen, S. and Doolen. G. 1998. Lattice Boltzman Method for Fluid Flows. Annual Review of Fluid Mechanics, Volume 30, pp. 329-364 .
- Currie, I. G., 1974. Fundamental Mechanics of Fluids. McGraw-Hill, Inc.
- Frisch U, Hasslacher B, Pomeau Y. 1986. Lattice-gas automata for the Navier- Stokes equation. Phys Rev Let 56:1505-1508G
- Ilachinski, A., 2001. Cellular Automata: A Discrete Universe. World Scientific Publishing Co.
- Koelman, J., 1992. Cellular-Automata-Based Computer Simulations of Polymer Fluids. Lecture notes in Physics, Volume 398, pp. 146-153.
- Koelman, J. & Nepveu, M., 1992. Darcy flow in porous media: Cellular Automata Simulations. Lecture notes in Physics, Volume 398, pp. 136-145.
- Saleh, J. M., 2002. Fluid flow handbook. New York: McGraw-Hill.
- Stam, J., 2003. Real-Time Fluid Dynamics for Games. Proceedings of the Game Developer Conference.
- Sukop, M. C. & Thorne, D. T. J., 2006. Lattice Boltzmann Modeling: An Introduction for Geoscientists and Engineers. :Springer.
- Toro, E. F., 2009. Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction. 3rd Edition ed. Berlin Heidelberg: Springer-Verlag .
- Wainer, G., 2009. Discrete-event modeling and simulation: a practitioner's approach. :CRC.
- Zeigler, B. P.; Praehofer, H.; and Kim, Tag-Gon. 2000. Theory of Modeling and Simulation. Academic Press