Simulation Traffic Flow in Urban Arterial Streets Using CFD (Army Canal Street as a Case Study)

Manar K. Hatem\(^1\), Dr. Gofran J. Qasim\(^2\)

\(^1\)Highway and Transportation Department, College of Engineering, Mustansiriyah University, Baghdad, Iraq

\(^2\)Dr. Highway and Transportation Department, College of Engineering, Mustansiriyah University, Baghdad, Iraq

*Corresponding author: manarkhalil464@gmail.com and gofranqasim@gmail.com.

Abstract: computation fluid dynamics (CFD) is a branch of fluid mechanics that has been widely and successfully applied to different fields of engineering researches by using the numerical analysis and data structures to analyze and solve problems that involve fluid flows. The aim of this paper concentrated on simulation process of traffic flow of particular section of Army Canal Street by using CFD approach. It is found that the tremendous potential to analyze the traffic flow by the CFD approach by providing an estimation of the unknown value of traffic flow in the exits or entries based on collected data of the upstream and downstream or vice versa. This process will help in reduce the time, effort and cost in the traffic count calculation process for a specific segment. The CFD has the capability to clarify visually the high and low regions of the traffic flow on the roadway section based on the colored contour legend.

Keywords: CFD; Euler’s equation; Traffic flows; Arterial Street.

1. Introduction
In the daily life of urban communities, the urban arterials roads network plays the important role through trips of all kinds (Chu, 2014). In Iraq, especially in Baghdad, Army Canal Street is a strategic arterial that connecting south and north of Baghdad province and facilitates the traffic movement of Al Rusafa side. Furthermore, it provides access to neighborhoods with heavy dense population such as Talbiyah, Al-Sadar City, Palestin Street, Zayona and Baghdad AL-jaddeda and to government departments, businesses, hospitals, universities and other essential services. This route also considers as commercial and trading corridor that connects the southern provinces with middle and northern provinces. The selected route connects some areas located on it without the need to enter the center of the capital, shorting the time and distance.

Since half century and more, the theory of fluid dynamics had been incorporated into the transportation researches. In 1950, Lighthill and Whitham [1] developed the famous one-dimensional approach which introduced first attempts to use the methods of fluid dynamic in studying and analyzing the transportation problems. Subsequently, Richards [2] developed a simple traffic flow under the precondition that the movement of a group of discrete vehicles could be treated as a continuous flow and the equation of the conservation of matter was given as:
That equation frequently called the one-dimensional Euler equation. The research and use of the one-dimensional Euler’s equation in traffic flow theory has continued to be a subject of interest since these early pioneering works.

CFD is a branch of fluid mechanics that has been widely and successfully applied to different fields of engineering researches by using the numerical analysis and data structures to analyze and solve problems that involve fluid flows. CFD is a very powerful tool and acceptable in wide range of industrial and non-industrial area including aerodynamics and aerospace analysis, electrical and electronic engineering, civil engineering, and environmental engineering. However, CFD has not yet been introduced as an effective numerical tool for solving the traffic flow problems [3]. Recently, In 2011, Dazhi SUN, Jinpeng LV, and S. Travis WALLER [4] develop CFD implementation methodology for modeling traffic problems such as queue/platoon distribution, shockwave propagation, and prediction of system performance. It is found that CFD approach can facilitate a superior insight into the formation and propagation of congestion, thereby supporting more effective methods to alleviate congestion. In addition, CFD approach is found capable of assessing freeway system performance using less ITS detectors, and enhancing the coverage and reliability of a traffic detection system.

For Euler’s equation implementations in transportation engineering, there are traditionally two viewpoints. The first is referred to as the Lagrangian description. This method concentrates on individual particles in a fluid flow, or individual vehicles in traffic study. Lagrangian description has been applied for studying certain traffic flow problems, such as car-following studies. When utilizing the Eulerian viewpoint, instead of individual vehicles in a flow, traffic is viewed as a simple continuously distributed flow, with consistent gaps between the cars constituting various levels of density, with more emphasis on given road segments [3].

This research concentrated on the simulation process of traffic flow in urban arterial roads by using Computation Fluid Dynamics (CFD), a particular section of Army Canal Street was adopted as a case study for this research. The methodology presented in this study is based on the Eulerian description, and thus emphasis was placed on the flow as a whole or a system and not on the individual vehicles. However, any CFD simulation track the same procedure, as NASA published on their website [6], which can summarized as nine steps. These steps illustrated in Figure (1) schematically. ANSYS simulation is the worldwide used software to analyze real-world engineering problems. Its present the previously shown procedure in a systematically organized way called “Analysis systems”. Fluent analysis system has been utilized to perform the traffic flow. That because of Fluent is one of the most CFD packages using and its solutions have high fidelity and validity in a such a problems as that treated here in this study [4, 6].

![Figure 1. CFD Procedure (A) ANSYS CFD (B) Schematic Steps.](image)
2. Methodology:
The selected section of Army Canal Street is (650) meters in length and has a total width of 10.5 m. The selected section is divided into two parts the first extends for 320 m before the center line of Sindbad Land City Bridge and 330 m after it. Figure (2) presents the satellite image of the study selected section. The first part of the selected section has one exit with 7m total width diverging at angle of 10 degree from the horizontal surface toward Al-Shaab International Stadium. While the second part has an inlet merges from Muthanna Al-Shaibani Street and exit diverging go back to Muthanna Al-Shaibani Street. Both the inlet and the exit branch out of the horizontal surface at angle of 35 degree and total width of 7 m. Figure (3) illustrate the inlets and exits of the selected section.

![Figure 2. The satellite image of the selected section](image1)

![Figure 3. Map Consider of Selected Section](image2)
2.2 Formulating the Traffic Flow Problem

Simply, instead of individual vehicles in a flow, traffic presented as a simple continuously distributed flow as shown in Figure (4).

![Figure 4. Geometry simplification for CFD simulation.](image)

In general, the CAD view usually use for aerodynamics and environmental studies. However, due to the objectives of this study, the 2D presentation was adopted to estimate traffic flow through branches of the considered roadway. This formulation of our computational domain is sufficient as simple as our targeted results [4, 6]. Fig.4 CAD models was taken four type of vehicles (Motorcycles model, private car and taxi model, Pick-up, van, and buses up to 24 passengers” model, heavy truck model) taken in real dimensions, but in a simplified shape for demonstration purpose only. However, the problem considered here solved by mass conservation equation only. To be more clarified, the model considered (as explained in the next section) has two traffic flow inlets and three outlets. The data available is for only one inlet and two outlets. Therefore, the CFD simulation objective is to estimate the data for the remaining boundaries.

2.3 Geometry Modeling

The 2D computational domain was used to represent the crossroad. The model designed by ANSYS DM to be 2D (without thickness). All dimensions considered measured actually from the field and validated with e-maps by Google services. These dimensions have been taken in meters (m) units and fixed on the model edges as in Figure (5).
2.4 Mesh Generation

The purpose of mesh generation in finite volume analysis is to subdivide a domain into a set of subdomains, these subdomains where the governing equations solved discretely. Mesh generation process has been done by using ANSYS Mesh. A total number of 127,010 quadrilateral elements were used with 3 levels of refinement (Max. refinement level). Figure (6) shows the mesh of the study model that was

![Mesh Generation Diagram](image)

**Figure 5.** The 2D computational Domain

**Figure 6.** Generated Mesh; (A) 1st Section (B) 2nd Section (C) 3rd Section
that was subdivided into three segments from left to right of the whole geometry; moreover, the figure portrays the domain meshed by quad cells. Accurately, the governing equations will be illuminated at focuses that lie on cell boundaries that called nodes. The number of elements (cells) and nodes that shaped the framework is listed in Table (1).

| Mesh Statistics | Mesh Details |
|-----------------|--------------|
| Nodes           | 129983       |
| Elements        | 127010       |

2.5 Boundary Conditions and Simulation approaches
To establish the previously mentioned simulation process conditions, a set of boundary conditions should be applied. At first, a highly rich data collected on the Land of Sindbad City crossroad traffic flow to specify the peak hour of traffic flow. The (700-8:00) A.M. was chosen as the peak hour The counting of traffic volume using the recorder videos was based on 15 minutes time intervals based on the type of vehicle for a total of 8 hours. After data extraction from video recordings stage the obtained result of traffic flow counts for every 15 min. for the peak hours was illustrated in the Figures (7) to (9).

Figure 7. Traffic Flow Count of Sindbad Land City Bridge (General inlet and Branch Outlet (1))

Figure 8. Traffic Flow Count of Sindbad Land City Bridge (Branch Outlet (2) and Branch Inlet (1))
Figure 9 Traffic Flow Count of Sindbad Land City Bridge (General Outlet)

The inlets and outlets of the domain renamed to more simple names as shows in Figure 10.

Figure 10. Boundary Conditions Specifications

However, these data simplified to data listed in Table 2. The inlets and outlets of the domain renamed to more simple names as shows in Figure (10).

Table 2 Simplified Traffic Flow Data for CFD Simulation

| Time (AM)   | General Inlet “BC” | Branch Outlet (1) “BC” | General Outlet “BC” |
|-------------|--------------------|------------------------|----------------------|
| 7:00 to 7:15| 1177               | 201                    | 1130                 |
| 7:15 to 7:30| 1059               | 185                    | 1166                 |
| 7:30 to 7:45| 1199               | 254                    | 1250                 |
| 7:45 to 8:00| 1090               | 173                    | 1178                 |
As in Table (2), the target of CFD simulation is to estimate traffic flow at Branch outlet (2) and Branch inlet (1). Thus, the ANSYS has been solved the mass conservation equation to evaluate mass flow rate values at targeted locations. Because of that, the objective is mass flow; any fluid can be used as long as it satisfied the mass conservation law.

Air flow approach considered to be instead of car flows. The 2D model defined as a fluid (Air) domain. The fluid is submitted to some of boundary conditions shown in Figure (10).

In fluid dynamics, mass flow and fluid motion related generally to pressure differences. Thus, the pressure outlet and pressure inlet have been used for Branch outlet (2) and Branch inlet (1) respectively.

In addition, as the flow enters the domain at high mass flow rate, the flow considered as a turbulent flow. Therefore, the \( (k-\varepsilon) \) turbulence model used to solve flow problem. This model used to achieve turbulent parameters simulation that effect directly on mass flow as kinetic energy and energy dissipation [7].

The data explained in Table (2) represent four simulation cases. Each case covers 15 minutes from the peak hour. The values presented in the previously table are independents (not a functions or equations). Thus, they have been simulated in a steady state analysis. The data then collected from these 4 steady states to make transient equations for each inlet and outlet in the domain by mathematical regressions.

ANSYS generally „By default” Solve the whole governing equations (mass and momentum in x and y directions) where our investigation again focused only on mass conservation. The solution approach used by ANSYS Fluent called the iterative solutions. In mathematics, each iterative solution has residual values. The residuals are the error magnitudes for equations as iterations progress. The equations include the governing equations; i.e. the Navier-Stokes momentum equations for each direction (x and y), the continuity equation (conservation of mass), and equations of the turbulence model defined under models-viscous. The residual is the difference between the previous result and the current result. As these errors are decreasing, the equation results are reaching values that are changing less and less. This is what known as convergence. That is the solutions are converging. For the value studied, a residual of \( 10^{-3} \) assumed to be sufficient for solution convergence. Moreover, the plotting of residual vs. iterations is shown in Figure. (11).

![Figure 11. Solution Convergence Plot](image-url)
The simulation setting summarized in Table. 3.

| Table 3. Simulation Settings |
|-----------------------------|
| No. of Cases                | 4 cases                         |
| Solver                     | ANSYS fluent                    |
| Solution Type              | Steady State (4 cases)          |
| Flow Behavior              | Turbulent.                      |
| Model of Turbulence        | k-ε (Standard)                  |
| Wall Function              | Standard wall treatment.        |
| Fluid zone                 | Street                          |
| Equations to solve         | Kinetic energy transport equation |
|                           | Energy dissipation transport    |
|                           | equation.                       |
| Solution Method            | Iterative                       |
| Max. No. of iteration      | 1000                            |
| Convergence Reached        | After only 80 iterations        |

3. Result Discussion:

Figures (12) through (16) illustrate the CFD simulation result of the traffic flow distribution of the Sindbad Land City cross road. The traffic flow distribution has been presented by using a contour legend as shown in the Figure (12). The blue color represents the (Low traffic flow), the green color presents the (Medium traffic flow) while the red color illustrates the (High traffic flow). These colors give an impression of the traffic volume in each section of the street also the approximately distribution of the traffic flow in each lane of the roadway.

![Contour Legend of Traffic Flow](image)

Figure 12. Contour Legend of Traffic Flow.

Figure (13) represents the CFD simulation result of the case 1 (7:00-7:15) A.M. from the figure, It can be noticed that the general inlet has a dark orange color which represent high traffic flow that more than 4400 veh/hr, the blue color represent the branch outlet (1) with traffic flow approximately 800 veh/hr. After that the through section between the branch outlet (1) and branch outlet (2) has light orange with traffic flow less than 4000 veh/hr after the diverging section then the light orange color concentrated in the middle lane while the left and right lane with green color. The branch outlet (2) has a dark blue color with traffic flow less than 400 veh/hr. then the section between the branch outlet (2) and the branch inlet (1) was colored with green for all lanes. finally, the traffic flow at the general outlet after the branch inlet (1) was colored with red because the traffic flow increased from the traffic flow of branch inlet (1) to more than 4500 veh/hr.
Figure 13. CFD Simulation Result of Case 2

The result of case 3 and 4 that presented in the Figures (14) and (15) respectively are same as the result of case 1 as shown below.

Figure 14. CFD Simulation Result of Case 3

Figure 15. CFD Simulation Result of Case 4
4. Validation of CFD Model
In this section the comparing process between the numerical simulations with the survey data is performed. The analysis operation was repeated four times for each case with the same boundary conditions and different traffic flow to predict the traffic flow of branch outlet (2) and branch inlet (1). Table (4) illustrates the prediction value of traffic flow of branch outlet (2) and branch inlet (1) that is converged with the surveyed values that presented early in the Figure (8).

Table. 4 Predicted Traffic Flow Data of CFD Simulation

| Time (AM)     | Branch Outlet (2) | Branch Inlet (1) |
|---------------|-------------------|------------------|
| 7:00 to 7:15  | 47                | 241              |
| 7:15 to 7:30  | 10                | 302              |
| 7:30 to 7:45  | 22                | 327              |
| 7:45 to 8:00  | 35                | 296              |

This performance gave suitable prediction for the inlet and outlet branches, for example, the first case was taken from (7:00 - 7:15) A.M. and the correctness of the solution was verified, when the general inlet was 1177 veh/hr and branch outlet (1) was 201 veh/hr, so to achieve the convergences must: General inlet - Branch outlet (1) - Branch outlet (2) + Branch inlet(1) = General outlet.

The comparative study of the simulation result and collected data of the traffic flow for branch outlet (2) and branch inlet (1) have been illustrated in the Figure 16, 17 which show the convergence between the model results and the data collected for the four cases of the study area.

![Figure 16. CFD Model Results vs. Data Collected of Branch Outlet (2)](image-url)
5. Conclusions:

In this study, CFD simulation has been carried out to simulate the traffic flow in an arterial streets (Sindbad Land City Bridge) segment of Army Canal Street was used as a case study, in Baghdad City. The geometrical conditions and traffic flow, vehicles speed and all data associated with field survey was utilized. The main conclusions came up with this a broach can be summarized in the following point.

1- The result that obtained from this study demonstrated the tremendous potential to analyze the traffic flow by the CFD approach.
2- The CFD has a flexibility to provide an estimation of the unknown value of traffic flow in the exits or entries based on collected data of the upstream and downstream or vice versa. This process will help in reduce the time, effort and cost in the traffic count calculation process for a specific segment.
3- The CFD has the capability to clarify visually the high and low regions of the traffic flow on the roadway section based on the colored contour legend.
4- In addition, CFDs can be adopted to process data from a small number of traffic detectors, thereby estimating the location and time of congestion (or incidents) without need to use detectors for such road sections.

Acknowledgements

The authors are grateful to the Mustansiriyah University, College of Engineering, Highway and Transportation Department for their support and help to accomplish this work contained in this study. This research did not receive any specific grant from funding agencies in the public, commercial, or not-for-profit sectors.

5. References:

[1] Lighthill, M.J. and Whitham, G.B. (1955) On Kinematic Waves, I, Flood Movement in Long Rivers. Proceedings of the Royal Society of London A, 229, 281-316.
[2] Richards, P.I. (1956), Shock waves on the highway, *Operations Research*, 4(1): 42-51.
[3] Kachani, S., & Perakis, G. (2006). Fluid dynamics models and their applications in transportation and pricing. European journal of operational research, 170(2), 496-517.
[4] Sun, D., Lv, J., & Waller, S. T. (2011). In-depth analysis of traffic congestion using computational fluid dynamics (CFD) modeling method. Journal of Modern Transportation, 19(1), 58-67.

[5] https://www.grc.nasa.gov/www/wind/valid/tutorial/process.html

[6] Catalin, C., Jorge, B., Genevieve, D. T., & Aziz, N. (2013, August). Bond graph and computational fluid dynamics in traffic flow. In 2nd International Conference on Systems and Computer Science (pp. 246-251). IEEE.

[7] Launder, B. E., & Spalding, D. B. (1974). The numerical computation of turbulent flows, Comp. Meth. Appl. Mechanical engineering, 3.