International Invention of Scientific Journal

Available Online at http://www.iisj.in

- eISSN: 2457-0958

Volume 04 | Issue 08 | August, 2020 |

Mathematical Engineering in addressing complex systems with computational simulation tools

Correspondence Author : Eduardo Gago, Lucas D'Alessandro, Matías Romero, CarenBrstilo, Marcelo Zurbriggen

Computer and Multidisciplinary Laboratory of Basic Sciences.
Rosario Regional Faculty, National Technological University
Zeballos 1341, Rosario, República Argentina

Article Received 30-06-2020, Accepted 30-07-2020 , Published 28-08-2020

Abstract

The proposal presented corresponds to a class experience in the subject Advanced Calculation appealing to perform a multi-discipline activity through the use of the technologies applied within the framework of a line of research in mathematical education that has among its objectives the integration of different design and simulation tools. The learning teaching process is carried out by analyzing a dynamic system for developing the topic Partial Differential Equations. A model is designed to simulate the operation of a fluid circulating through a pipe that has a contraction. The purpose of the class is to analyze fluid flow patterns relative to the influence that velocity and pressure exert on that fluid.

Keywords: Simulation, fluid flow, multiphysics.

1. Introduction

In the development of new trends in Education, Mathematical Engineering is an innovation line for learning in undergraduate degrees in Engineering that works on new teaching paradigms with the intention of achieving, a greater development of intellectual capacities, using resources that we recommend students to acquire skills, replacing outdated techniques for more efficient and faster means, which lead to a better integration of knowledge in the teaching-learning process. The inclusion of computational resources is essential to carry out these changes already requires the use of new pedagogical methodologies and various teaching strategies that develop multidisciplinary and collaborative work. This work presents an experience in the Computer laboratory and Multidisciplinary of Basic Sciences of the National Technological University Regional Faculty

Rosario to develop an application of the theme Partial differential equations within the curriculum of the subject Advanced Calculation, where for this is modeled a fluid flow system.

2. Theoretical framework

The proposal presented is a classroom experience that aims to show an integrated work to analyze the circulation of a fluid through a pipe that has restriction in its cross area. A simulation is performed with the COMSOL Multiphysics platform using CFD (Computational Fluid Dynamics) technology. The study of fluids by appreciating techniques that contemplate computer-assisted dynamics is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems with fluid flow. The Creeping Flow module enabled in the program works with a participation of the constituent surfaces of the system through the application of the finite element method and uses Navier-Stokes equations for non-isothermal flow and which also consider the constant density in the convective term. The equations of COMSOL uses for non-isothermal flow are:

$$\rho(u.\nabla)u = \nabla.\left[-\rho I + \mu(\nabla u)^T - \frac{2}{3}\mu(u.\nabla)I\right] + F \quad (1)$$

$$\nabla.(\rho u) = 0 \quad (2)$$

Where, ρ:fluid density (kg/m^3); u: velocity field(m/s); I: identity matrix; μ: dynamic viscosity($Pa.s$); T:absolute temperature (K); F: constant.

This phenomenon of the transport of mass and momentum of the fluid is described with the Forms this phenomenon of mass transport and amount of fluid movement with Navier-Stokes equations using the midd form of Reynolds number (RANS Reynolds-averaged Navier-Stokes), which considers constant density in the convective term: ($(u\nabla)u$) and the effects of turbulence are appreciated . Navier-Stokes equations are a set of equations in nonlinear partial derivatives. While equations (1) and (2) do not have a solution by generalized methods, except for certain particular conditions that meet some types of flow and very specific situations, it is not possible to find an analytical solution; so on many occasions it is necessary to resort to numerical analysis to determine an approximate solution.

3. Objectives and Methodology

The applied mathematics software at the University should be oriented so that the activities that the student develops in class can allow him to build cognitive structures capable of achieving the skills required in academic programs. Presenting simple cases that address problems with engineering applications helps strengthen the learning teaching process to make it functional and meaningful.
The actions that are implemented to develop scheduled activities aim to perform a lab experience where students perform self-managed and collaborative work to analyze the flow model of a fluid when transiting a pipe that has a restriction.

The implementation of a learning system in the area of Higher Mathematics that privileges the use of visualization as a means of interpreting the parameters and concepts related to the variables involved in the circulation of a fluid is done through a computational tool that by its interactivity and versatility allows to understand the characteristics of different physical systems. Like also, you want to generate new work styles to change the prevailing model in many chairs where theory, practice and its applications are separated. These changes are incentivized if learning spaces are achieved that enable the use of numerical and symbolic information during the development of the topics and thus reinforce conceptualization through graphic designs whose complexity requires skills and analytical capacity. The methodological proposal for the presentation of mathematical content is based on the search for models that simulate the situation to be formulated or the technical situation in mathematical terms, for which a simplified situation is presented, this situation is translated into mathematical terminology, and works with that model. This methodology allows to stimulate interest in discovery and gain confidence in the use of the formative aspects of Mathematics, related to other areas of knowledge, as in this case in the analysis of the dynamics of a certain fluid.

4. Results and Discussion

Students are asked to model and simulate a system of the flow of a fluid in laminar regime that presents in its geometry a surface constraint. The students will solve this activity by applying the COMSOL Multiphysics simulation platform. There are several application modules specific to COMSOL, in this case the system is analyzed using the CFD module, since its interfaces allow to model laminar flow in one or multiple phases. In this case students will work with a flow of glycerol on a laminar basis, circulating at$0,01\ m/s$ at$320\ K$ de absolute temperature. Due to the complexity of interacting with the program and to facilitate the management of students, 2D axisimetric geometry is used. They first design the system geometry as shown in Figure 1.
The next step students take is to select a fine mesh, which is the option that the program suggests by default as the most suitable for this system.

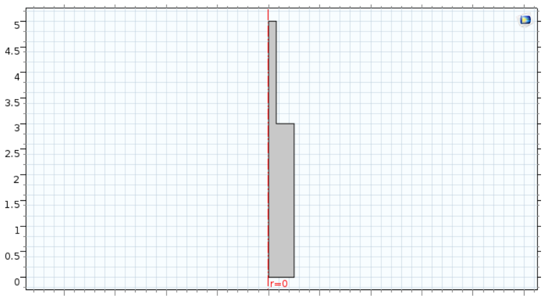

Figure 1. System geometry

Figure 2 shows the meshing of the system that is done by applying the finite element method, a tool that has COMSOL and consists of dividing the domain into a finite set of generally unstructured discrete elements, and for our case that works in 2D, triangle or quadrilateral elements are used. An advantage of this method is that partitioning into small surface units makes it possible to study complex geometries and break them down into very simple study sections.

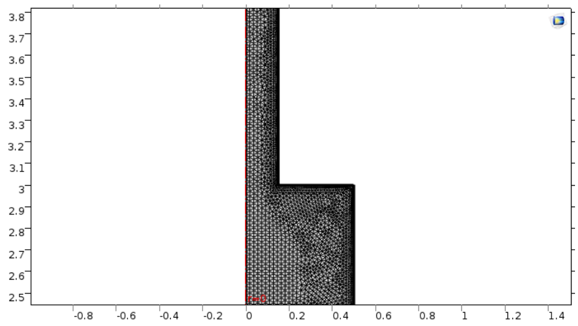

Figure 2. Meshing system

Figure 3 shows the graph that students obtain from the evolution of the fluid velocity along the pipe, in terms of its behavior, they conclude that initially the speed remains constant, with the speed profile developing slowly until the restriction, which causes the acceleration of the fluid and the development of another velocity profile. In the blue area students observe and analyze that the flow is fully developed with a constant velocity along the geometry.

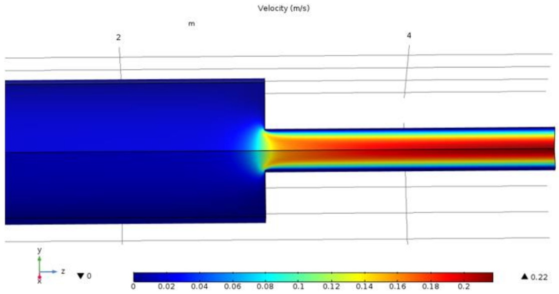

Figure 3. Fluid velocity

Students acknowledge that because of the non-slip condition, the fluid layers against the wall of the entire pipe are at zero speed (same as the speed of the pipe in the blue zone). When analyzing the speed profile after the constraint, they notice that it has an increase in radial direction speed, i.e. the software allows them to see that the speed increase occurs from $r = R$to $r = 0$. Also from the speed analysis they interpret that it varies from a value that they consider null $(v = 0)$ on the walls of the constraint to its maximum value in the center, which can be seen in the dark red area. Again students point out that this is due to the condition of non-slip against the walls and that in this place there is maximum shear effort. While in the area of maximum speed, in the center of the tube where $r = 0$ (dark red color), you can observe that the shear stress is minimal.

After they analyze the graphics that show the speed contours and pressure contours

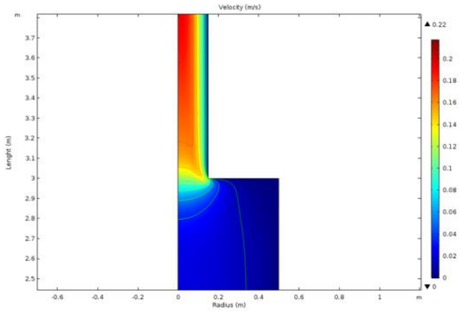

Figure 4. Velocity contours

In Figures 4 and 5, students verify how the system experiences increased velocity and a decrease in pressure in response to disturbance caused by reduced pipe surface.They also comment that this situation is consistent with the principle of energy conservation.

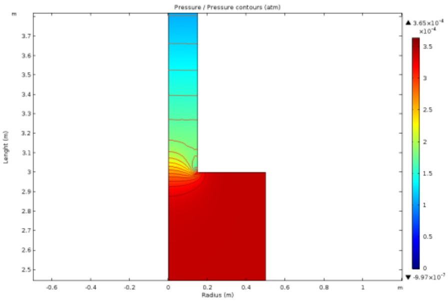

Figure 5. Pressure contours

In the analysis presented by the students refer that along the pipe the pressure decreases (passes from red zone to celestial zone) and the speed increases (from blue zone to red zone), while when performing a radial analysis, at the restriction, the speed increases to reach its maximum value of $0{,}2\ m/s$ in the center of it and the pressure remains radially constant and the pressure remains radially constant.

Regarding the velocity profiles, they comment that the speed profile develops as the fluid travels through the pipe in laminar regime. In Figure 6, students note that from $4{,}5\ m.$ the speed profile is fully developed, and they also find the existence of a flat profile with constant speed before the restriction, and a parabolic profile once the input region is crossed.

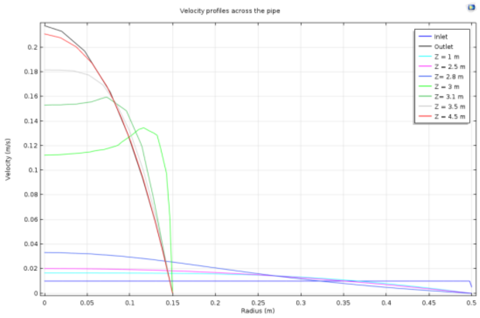

Figure 6. Velocity profiles

Finally, they conclude that when the speed profiles are analyzed in some systems it is extremely important, to move away from the inlet region of the pipe because it would work with erroneous profiles.

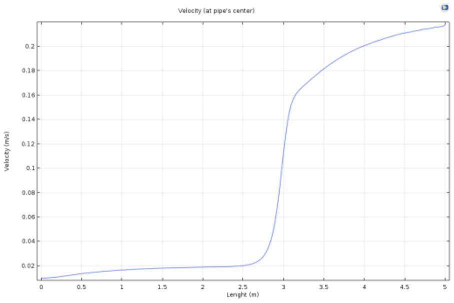

Figure 7. Fluid velocity in the center of the pipe

In Figure 7 you can see that as the fluid travels through the pipe, it increases velocity in the center due to the consequences caused by the restriction, becoming stabilized at $5\ m.$ of pipe.

Students describe that the fluid experiences a disturbance as it passes through the hole. In response to this disturbance, it increases its velocity until it reaches a steady state again at $5\ m.$ of pipe.

5. Conclusion

It is important to note that visualizations made with the multiphysical simulation platform in the development of the class collaborated to perform a detailed analysis of the velocity and pressure of a simple fluid flow system These types of practices help to promote the share of abstraction and generalization of knowledge that is often associated with learning concepts of Higher Mathematics. The advantage of having the Multidisciplinary Laboratory of Basic Sciences promotes research activities that allow the student to infer the next and independently manage their own knowledge and work in a collaborative environment that reproduces their future professional work environment Simulation tools allow us to save time in solving engineering problems that being solved analytically would take long hours of work and even depending on the geometry could not be solved. In addition to having simulation software makes this type of experience not risky for students since there is no physical contact with industrial materials and equipment. The approach of multidisciplinary activities from the Basic Sciences, through the exploration of new knowledge, encourage creativity from the analysis and management of information, resulting in an innovative learning teaching process.

Referencias

- Bird, R., Stewart, W., Ligthfoot, E. (2001) Transport phenomena, John Wiley & Sons Inc, EEUU.
- Cengel, J., Cimbala, J. (2006) Fluid Mechanics: Fundamentals and applications. McGraw- Hill, Mexico.
- Comsol. Introduction to CFD (2011) Module, version 4.3.
- James, G. (2002) Advanced Mathematics for Engineering. Prentice Hall, Mexico.
- Tinnirello, A., Gago, E. (2013) Interdisciplinary activities to improve the learning methodology performed in mechanical engineering degree studies. Proceedings of EDULEARN 2013 (pp. 5407-5416). Spain.
- Tinnirello, A., Gago, E., Valentini, M. (2015) Design and Simulation of Mechanical Equipment by Design Tools and Multiphysics Platforms, Proceedings of 8th International Conference of Education, Research and Innovation, pp. 789-797. Spain.
- Torres, R., Grau, J. (2007) Introduction to fluid mechanics and heat transfer with Comsol Multiphysics. Addlink Media. Spain