International Journal of Advanced Research in Engineering Technology & Science

ISSN: 2349-2819

Impact Factor: 7.10

(Peer-Reviewed, Open Access, Fully Refereed International Journal)

Email: editor@ijarets.org Volume-11 Issue-5 May-2024 www.ijarets.org

NUMERICAL METHODS FOR SOLVING FLUID DYNAMICS PROBLEMS

Name Dr. Manju Bala

Associate Professor, Department of Mathematics

S V College Aligarh

ABSTRACT

In this article, we investigate a variety of numerical approaches that are applied for the purpose of addressing issues pertaining to fluid dynamics, which are governed by the Navier-Stokes equations. Because of the non-linearity and complexity of the equations that control fluid dynamics, it is necessary to have computational methods that are both reliable and precise. Fluid dynamics is an important field that serves both the engineering and physical sciences. In this article, we shall concentrate on finite difference techniques (FDM), finite element methods (FEM), and finite volume approaches (FVM), and we will explore the advantages and disadvantages of each of these methods. Spectral approaches, lattice Boltzmann methods, and particle-based methods such as Smoothed Particle Hydrodynamics (SPH) are some of the advanced techniques that we investigate in this article. We take a look at how these techniques can be utilized to simulate incompressible and compressible flows, as well as turbulence and multiphase flows. In addition to this, we discuss the significance of grid generation, stability, and convergence, as well as the role that high-performance computing plays in improving the effectiveness of various numerical methods. There are case studies offered that illustrate the practical application of these strategies in solving fluid dynamics problems that occur in the real world setting. The findings highlight the need of selecting appropriate numerical approaches based on the particular characteristics of the fluid flow problem in order to attain the best possible outcomes.

Keywords: *numerical method, fluid dynamics.*

INTRODUCTION

Within the realms of engineering and the physical sciences, fluid dynamics is an essential field of research that encompasses the investigation of fluids (both liquids and gases) in motion dynamics. The Navier-Stokes equations are a set of nonlinear partial differential equations that describe the conservation of mass, momentum, and energy. These equations govern the behavior of fluids and are responsible for governing their physical properties. The analytical solution of these equations is frequently problematic due to the complexity of the equations themselves, particularly for issues that occur in the real world and involve turbulence, multiphase flows, and complicated geometries. As a consequence of this, numerical methods have developed into instruments that are absolutely necessary for the study and application of fluid dynamics.

The discovery and improvement of numerical methods have made it possible for us to make great progress in our understanding of fluid behavior and in our ability to forecast behaviors of fluids. Methods such as the finite difference method (FDM), the finite element method (FEM), and the finite volume method (FVM) are among the numerical techniques that are utilized the most frequently. Every single one of these approaches comes with its own set of benefits and can be utilized to solve a certain kind of fluid dynamics problem. Additionally, in order to address particular difficulties in fluid dynamics, additional specialized methods have been created. These methods include spectral methods, lattice Boltzmann methods, and particle-based approaches such as Smoothed Particle Hydrodynamics (SPH).

The choice of an appropriate numerical approach is of the utmost importance and is contingent upon a number of parameters. These factors include the characteristics of the fluid flow, the level of precision that is required, the computational resources that are available, and the complexity of the problem domain. Additionally, high-performance computing (HPC) has been an essential contributor to the development of fluid dynamics simulations, which has made it possible to solve issues that are becoming increasingly difficult and extensive in scope.

This paper presents an in-depth analysis of the most important numerical methods that are utilized in the field of fluid dynamics. We are going to investigate the underlying ideas that underlie each method, as well as their implementation and how they might be applied to a variety of fluid dynamics issues. The purpose of this investigation is to bring to light the advantages and disadvantages of each approach, as well as to offer direction for the suitable application of these approaches in a variety of contexts.

The application of these numerical methods to the resolution of real-world fluid dynamics issues will be demonstrated through the examination of case studies and practical implementations. The purpose of this endeavor is to provide researchers and practitioners with a more in-depth understanding of the computational tools that are currently available and the possible ways in which these tools might improve the precision and effectiveness of fluid dynamics simulations.

OBJECTIVES

1. To study numerical method for fluid dynamics.
2. To study fluid dynamics.

Numerical method for fluid dynamics.

A variety of methods exist for solving partial differential equations, each tailored to a certain type of equation. So, to have a feel for the language used to describe the many forms of partial differential equations, it's helpful to review the basics. An equation containing partial differential terms is said to have an order that matches the order of the highest-order partial derivative included inside the equation. It is common practice to develop numerical techniques for tackling time-dependent issues with the express goal of solving systems of first-order partial differential equations. Partial differential equations with higher-order temporal derivatives can be solved using these numerical approaches. A new unknown function, defined as the lower-order temporal derivative of the previous unknown function, is defined to achieve this. Next, the outcome is represented as a set of first-order partial differential equations. Take, as an example, the second-order partial differential equation.

$$\frac{\partial^2 \psi}{\partial t^2} + \psi \frac{\partial \psi}{\partial x} = 0$$

Alternatively, one may refer to this as the system of the first order.

$$\frac{\partial \upsilon}{\partial t} + \psi \frac{\partial \psi}{\partial x} = 0,$$

$$\frac{\partial \psi}{\partial t} - \upsilon = 0.$$

The formulation of first-order-in-time equations using this method is not always required in geophysical applications since it is not always needed. In most cases, appropriate first-order-in-time systems may be derived from fundamental physical principles. This is the reason why this is the case. When a solution develops significant perturbations on spatial scales that are close to the shortest scale that can be resolved by the numerical model, it becomes more difficult to acquire an accurate numerical solution to equations that describe wavelike movement. This is because the numerical model requires the solution to be on the smallest scale possible. There is a larger possibility that waves will develop small-scale disturbances from smooth beginning data as the partial differential equation that drives the system becomes more nonlinear. This is because waves use smooth initial data to generate disturbances. In order for a partial differential equation to be deemed linear, it must be linear in both the unknown functions and their derivatives. This includes the equation's derivatives. In this particular circumstance, the coefficients that multiply each function or derivative are only reliant on the variables that are independent of one another. Using this as an example,

$$\frac{\partial u}{\partial t} + x^3 \frac{\partial u}{\partial x} = 0$$

When the issue at hand is a linear partial differential equation of the first order, consider the following:

$$\left(\frac{\partial u}{\partial t}\right)^2 + \sin\left(u \frac{\partial u}{\partial x}\right) = 0$$

presents itself as a first-order partial differential equation that is nonlinear.

It is possible to generalize the analytical methods and solution processes developed for linear partial differential equations to the subset of nonlinear PDEs that are quasi-linear with minimal effort. The independent variables and all derivatives of the unknown function up to order p 1 can be used via the coefficient that multiplies each pth derivative. If a partial differential equation with an order of p is linear in the derivatives of the same order, we say that it is quasi-linear. Here are two examples of quasi-linear partial differential equations:

$$\frac{\partial u}{\partial t} + u^3 \frac{\partial u}{\partial x} = 0$$

in addition to the equation of vorticity that is connected with flow that is nondivergent in two dimensions

$$\frac{\partial \nabla^2 \psi}{\partial t} + \frac{\partial \psi}{\partial x}\frac{\partial \nabla^2 \psi}{\partial y} - \frac{\partial \psi}{\partial y}\frac{\partial \nabla^2 \psi}{\partial x} = 0,$$

Within the context of this equation, the notation represents the stream function for the velocity field that does not exhibit any variations. This is x, y, and t.

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2}.$$

Dynamic wave equations for geophysical fluids

Wave-like motions are of primary interest in geophysical fluid dynamics. These wave-like motions include the physical transport of scalar variables through the motion of fluid parcels, oscillatory motions associated with buoyancy perturbations (gravity waves), and oscillatory motions associated with potential vorticity perturbations (Rossby waves). Additionally, acoustic waves, commonly known as sound waves, are able to travel through all geophysical fluids. However, in many applications, these disturbances are of a modest amplitude, and the exact structure of these particles is not of relevance. The mathematical description of the propagation of sound waves and the transport of inviscid tracer materials is accomplished through the use of hyperbolic partial differential equations. However, some of the fluid properties that are necessary for the support of these waves are represented in the governing equations by terms that involve the zero-order derivatives of the unknown variables. Furthermore, Rossby waves and gravity waves are also solutions to hyperbolic systems of partial differential equations. These zero-order terms do not play any part in the classification of the governing equations as hyperbolic. Furthermore, it is possible to derive simpler nonhyperbolic systems of partial differential equations, such as the Boussinesq equations, whose solutions are very similar to the gravity-wave and Rossby-wave solutions to the hyperbolic system that was initially used. The term "filtered equations" will be used to refer to these more straightforward systems.

Numerical methodologies: Fluid Mechanics

The surface tension of a liquid is nothing more than the tendency of the surface of the liquid to contract into the smallest possible surface area. Take, for instance, the fact that a needle will float in water if it is placed there with care. The gradual withdrawal of an inviscid fluid with a finite depth into a line sink has been explored by Hocking and Forbes, with the primary emphasis being placed on the surface tension that is acting on the free surface. A numerical solution to this problem was found by the authors by the application of the boundary-integral-equation approach. It should also be brought to your attention that the flow is determined by the Froude number. Using numerical methods, Tuck and Vanden-Broeck have successfully found a "cusp solution" for a line sink in water of infinite depth (i.e., $H \to \infty$). This solution, which has been considered for a considerable amount of time, pertains to the solutions of steady flow that correspond to the critical drawdown value. They discovered a one-of-a-kind answer, which was FS = 12•622. Hocking has recently presented compelling data that demonstrates the importance of this solution for this particular instance of 'infinite' depth. The solutions that Hocking had determined were comparable to those that Tuck and Vanden-Broeck had found, except there was a boundary beneath the sink that sloped away without any limits. The occurrence of these solutions once more took place at a different Froude number for every angle. In addition, the authors made use of a complex potential portion and Cauchy's integral formula in order to guarantee that the equation is analytic in its flow domain and to satisfy the requirement that there is no flow across boundaries. An intriguing new understanding of the nature of these withdrawal issues is provided by the findings. From the perspective of the subcritical region, it would appear that all regions are capable of having solutions of the stagnation-point type, with the exception of those that are located along a single limiting curve. It would be impossible to discover single-layer flow solutions if the cusp solution on the limiting curve corresponds to critical drawdown values. This would make it impossible to locate solutions.

Continuous flows and unstable flows are the two categories of flows that can be found. Therefore, flows that are unsteady or non-steady are those whose properties fluctuate over time, whereas steady flows are those whose properties do not change over time with respect to the flow. Researchers Colicchio and Landrini have investigated the Mixed Eulerian-Lagrangian Methods (MEL) for free-surface potential flows. These flows were addressed by employing boundary-integral equations (BIEs), and the researchers also investigated the diffusion and dispersion errors that were present in the discrete linearized problem. Additional topics that have been covered include the stability analysis of the Runge-Kutta and Taylor-expansion schemes. MEL approaches that are based on first-order and second-order explicit Runge-Kutta and Taylor-expansion schemes have been demonstrated to be unstable. On the other hand, higher-order Runge-Kutta and Taylor-expansion schemes have resulted in conditionally stable forms. It was confirmed by the authors that the theoretical estimates of the errors for two alternative boundary-element approaches were accurate. The problem of determining the velocity potential was solved by employing a high-order panel approach that was founded on BSplines. In order to solve the velocity field calculation, an Euler-McLaurin summation formula was utilized. The equation of the body motion approach in a free-floating vessel has been introduced by Longuet-Higgins and Cokelet for recurrent problems. Faltinsen has independently introduced this equation for floating-body problems.

The majority of the time, the free surface is calculated using a Lagrangian approach, and the method in question is known as the Mixed EulerianLagrangian technique (MEL formulation). For the free-surface equations, a van Neumann analysis was carried out without taking into account the impact of spatial discretization, and stability conditions were shown to be present. In, Nakos et al. had demonstrated the influence of spatial discretization based on third-order splines, in addition to generalizing the spectral analysis that they had previously developed. Buchmann, who followed in his footsteps, had also adopted the same strategy, and in that, he had examined the stability aspects of an algorithm that was based on three-dimensional B-Spline discretization. In this study, the matrix method was utilized to demonstrate the stability features of MEL approaches that can be utilized for the linearized issue when stated in a general manner by utilizing the properties of the impact matrices. This was accomplished despite the fact that the technique that was utilized to solve the boundary-integral equations was not utilized. Through the utilization of Runge-Kutta and Taylor-Expansion time-integration techniques, this analysis has been developed to a level of accuracy that is equivalent to the fourth order. The processes of regridding and interpolation, which are frequently utilized in non-linear simulations, have also been described, along with the impact that these operations have on the theoretical growth rate, which is linear.

Taking into consideration the contact angle πβ, Fraenkel and Keady resolve a question that had been unresolved. This angle is provided in conjunction with the wedge angle, also known as the vertex angle, which is defined as 2πα. An integral equation of boundary-layer type is also discussed in this study. This equation allows for numerical calculation without the need to extrapolate the limiting solution as α approaches zero, and it also provides the value β0 that corresponds to α equal to zero. It was Wagner who came up with the idea of an infinite wedge that would enter the water and proceed vertically downwards at a constant speed. Wagner also discovered a similarity transformation that removes time from the equation when viscosity, surface tension, and gravity are not present. This transformation eliminates time from the equation. This study addresses the free-boundary problem of an infinite wedge that is introduced into the water, moves vertically downwards at a constant velocity, and eventually reaches the horizontal free surface of the water at time zero. This topic was described in the first half of this paper. There are also two portions or supplements to two others that are included in it. These include a variety of procedures and estimations, as well as the verification of answers that are provided in detail.

One of the most significant contributions made by this work was the proof for the set of solutions that had been established in. This proof proves that any wedge angle 2πα in the open interval (0,π) happens at least once. This is one of the key contributions of this paper. There is a strong requirement that the supremum πβ¯ of the contact angle πβ be less than π/4. There would be a run of solutions ((βn, hn))∞n=1 for which α(βn, hn) → −1/4 as n → ∞, which highly opposes the statement that 0 <α< 1/2 for an answer. If it were demonstrated that πβ¯ were equal to π/4, then in that particular scenario, there would be a sequence of solutions for which α(βn, hn) → −1/4. As the value of α approaches zero, this elucidates the reason why the contact angle πβ does not tend to π/2. In the process of constructing a limiting solution for β → 0 and α → 1/2, the boundary-layer equation, which was utilized in the process, also played a vital role under the assumption that β → 1/4 for a sequence of keys.

The existence theory was investigated by Fraenkel, who utilized it to generate approximations by insuring the flow for burnt wedges, which are wedge angles that are close to the symbol π. In the case where 0 is less than or equal to 1/4, the integral equation that is used to reduce the issue, which is similar to the equation that is used for numerical calculation, is in agreement with the solutions. This investigation was mentioned and taken into consideration in. The extraction process of two layers of fluids with differing densities that are separated by an interface in a porous medium is referred to as supercritical withdrawing or coning. A coupled integral equation was utilized by G.C.Hoking and H.Zhang in order to examine this phenomenon. Formulation and resolution of the equations were accomplished by the utilization of coupled integration equations and boundary integration methods. Methods of analysis were utilized by Muskat and Wyckoff in order to investigate the phenomenon of coning. When conducting their research, Bear and Dagan used the unbounded media as a point of reference and looked into critical and single flow phenomena. The flow that happens when the flow velocity is equal to the wave velocity is referred to as critical flow. The comparable problem of supercritical withdrawal in two-layer surface water bodies was taken into consideration, and an integral-equation approach was utilized in order to locate numerical solutions that are accurate. By employing finite difference approaches, Yu et al. and Hendersen et al. were able to model isothermal, nanophasic, and incompressible flow in a supercritical situation.

Result and discussion

For the purpose of representing viscous and incompressible fluid flow at a low Reynolds number, Oseen equations are an extremely important tool in the field of fluid dynamics. It is common knowledge that the Reynolds number is a dimensionless quantity in the field of fluid mechanics that is utilized for the purpose of determining the flow pattern. An investigation on the viscous, laminar, and divided flow that occurs downstream of a rapid expansion in a pipe has been carried out. An Oseen-type equation is used to describe the flow in this case; nevertheless, the nonlinearity that is present in the swirl is preserved. In this case, the precise answers for a High Reynolds number limit are established. It is possible to acquire an arbitrary Reynolds number by employing the Eigen function-expansion process with a swirl, which in turn results in a non-standard eigenvalue issue. Additionally, the author has explored the impact that pressure gradients have on the velocity profiles for the vehicles. It has been discussed by Ramakrishnan and Shankar that model equations that are comparable to the ones that are utilized in this article have outcomes that satisfy those of the N–S equations when the Re value is low. As a result, this demonstrates qualitative characteristics that are identical to those of the N–S equations for which Re goes to infinity. A further observation that was made was that the corner recirculation zone is compressed when the swirl amplitude is increased or decreased. The observations made by Abujelala and Lilley are likewise in agreement with these observations. One of the most fascinating characteristics of these swirling flows is the production of a center recirculation bubble, particularly when the swirl amplitude is sufficiently large. This phenomenon is analogous to the breakup of the vortex that occurs when the swirl surpasses the critical value, as in. The purpose of this research is to develop a model that is applicable to all Reynolds numbers and makes a seamless transition to the limiting form. All of the arbitrary values that are appropriate for the general case of the Reynolds number will be taken into consideration here. The eigenvalues of the flat case, which were presented in, serve as useful starting values for the computation of the current ones in this article since they were given in. They continue to be refined using the same Newton–Raphson approach that was used in the past. For the purpose of ensuring that no eigenvalues are overlooked, the authors have utilized the principle of the argument in order to determine the total number of eigenvalues that are contained within a specific region in the right half of the complex plane. In a general sense, Moffatt has stated that there would be an unlimited number of corner eddies that are either very small or diminishing in size and intensity. It is difficult to identify anything more than qualitative resemblance between the two types of flows since the mixing properties of turbulent flows are different from those of laminar flows. This is because the majority of the experimental data deal with turbulent flows. Considering that this model is only applicable to laminar flows, this is one of the most significant limitations it possesses. As is the case with all turbulent flows when the Reynolds number is sufficiently high, this model is also turbulent when the Reynolds number is quite high. It is presumed that this is an approximation of the probable laminar solutions of the Navier–Stokes equations, which may, at best, represent only the qualitative characteristics of natural flows. It is also important to note that the model is erroneous under near-field conditions close to the rapid expansion because of the quasi-linearization of the convective factors that pertain to the Oseen equations. In addition, this offers insight on the complicated phenomenon that is involved in comprehending the significance of swirl.

Software simulations are yet another fascinating discipline within the field of mechanical engineering. In order to investigate the effects of subgrid-scale turbulence on incompressible flow turbulence, a square duct was used for investigation. During the process of capturing this flow pattern, they utilized a numerical simulation database. The earliest SGS dynamic model, which was referred to as the DSM model, was developed by Germno and friends. It was the second one that Sarveti and Banerji came up with, and they called it the DTM model. When it came to these models, there were both positive and negative aspects. As an illustration, the DSM model makes an overestimation of the subgrid-scale dissipation on average, but the DTM model demonstrated the significant result of SGS dissipation. Simulations of stream-wise corner flows were used by the current authors to evaluate both models using large-eddy simulations. The difference between the two is that in DSM, they utilized a Fourier cut-off filter in addition to a modified Gaussian filter, but in DTM, they utilized only the modified Gaussian filter. Generally speaking, the SGS filtering process ought to be carried out in a mixture of complex flows without the use of homogenous directions. Their plan was to use the methods that are already available in order to put the SGS filter concept into practice with all three directions. It was the authors Salvetti and Banerjee, Zang, and Najjar and Tafti who worked in the same field that served as a source of inspiration for them. During the process of filtering in wall-normal directions, there was an issue with second-order communication failures. Vasilyev and his colleagues had already created solutions to this issue, which these writers subsequently adopted.

Finding the many numerical approaches by FEM, FDM, FVM, and BVM is discussed in this review, which gives the knowledge necessary to do so. In addition, these techniques assist us in comprehending the benefits and drawbacks associated with one another, as shown in Table 1.

Table 1. Advantages and Disadvantages of Numerical Methods.

Methods	Advantages	Disadvantages
Finite Element Method (FEM)	• Suitable for Symmetrical and sparse matrices. • Integration of simple functions can be easily made.	• Can't be done for infinite problem cases, also domain meshing is needed. • Its computation is a timeconsuming process.
Finite Difference Method (FDM)	• Simplest method among FEM, FVM, BEM to implement. • Doesn't require any numerical integration.	• Very fine grids are required to solve problems. • Requires domain meshing and is time-consuming
Finite Volume method (FVM)	• Ability of adaptive mesh, and can be utilized for unstructured grids. • Appropriate for turbulence	• Especially while solving non-conservative laws, this method can be considered less efficient. • False diffusion and is biased towards edges.
Boundary Element Method (BEM)	• Here it is suitable for infinite problems and the computation process is less time-consuming compared to other methods.	• Integral relations can be complicated. • Non-symmetric matrices.

	• Discretization of boundary	

The Finite Difference Method (FDM) is believed to be more suitable for addressing fluid flow and heat transfer issues than the Finite Element Method (FEM), the Finite Difference Method (FDM), the Finite Value Method (FVM), and the Boundary Element Method (BEM). This is owing to the fact that the FDM is simpler, more efficient, and requires less computational time. Since regular grids are suitable for very-large-scale simulations on supercomputers, FDM is primarily easy to acquire higher-order schemes on regular grids. This is because regular grids are frequently employed in simulations of meteorological, seismological, and astrophysical phenomena, as was described before.

Conclusion

When it comes to the science of fluid dynamics, numerical methods have become a vital tool since they make it possible to solve difficult fluid flow issues that would otherwise be insurmountable using analytical methods. The fundamental principles and applications of several important numerical methods have been discussed in this article. These methods include the finite difference method (FDM), the finite element method (FEM), and the finite volume method (FVM), as well as more advanced techniques such as spectral methods, lattice Boltzmann methods, and particle-based approaches such as Smoothed Particle Hydrodynamics (SPH). Each numerical method has its own set of benefits, and there are particular kinds of fluid dynamics issues that are best suited to its application. On the other hand, finite element method (FEM) is highly versatile and effective for solving problems with complicated geometries, whilst finite difference modeling (FDM) is praised for its simplicity and ease of application in structured grids. Because of its conservation properties, FVM is widely utilized in a variety of applications, including those in the academic and industrial sectors. Spectral methods offer a high level of accuracy when used to problems that have smooth solutions. On the other hand, lattice Boltzmann methods and SPH are especially helpful when it comes to simulating multiphase and particle flows. The selection of an appropriate numerical approach is of the utmost importance and ought to be led by the particular characteristics of the fluid flow problem, the level of accuracy that is sought, and the amounts of computer resources that are available. The power of numerical simulations has been substantially improved thanks to high-performance computing (HPC), which has made it possible to solve fluid dynamics issues that are becoming increasingly sophisticated and on a larger scale. The application of these numerical approaches to problems that occur in the real world of fluid dynamics has been demonstrated by us through the use of case studies and practical examples. These examples highlight how important it is to use the appropriate computing technique in order to produce the best possible outcomes in terms of accuracy, efficiency, and the minimum amount of computational resources required. In conclusion, numerical methods are currently undergoing continuous improvement, which is being pushed by the continual development of new algorithms as well as advancements in computer capacity. The continuing integration of these numerical approaches with cutting-edge computational technology will lead to simulations that are more accurate and efficient, which will be the future of study and application in the field of fluid dynamics. In the future, as academics and practitioners continue to refine and create these methodologies, the potential to tackle complicated fluid dynamics problems will expand. This will contribute to advancements in engineering, environmental science, and a variety of other sectors.

References

1. Nayak RK, Bhattacharyya S, and Pop I, “Numerical study on mixed convection and entropy generation of Cu water nanofluid in a differentially heated skewed enclosure”, Int J Heat Mass Transf 85:620–634, (2015).
2. Tyvand, P. A., & Storesletten L, “Degenerate onset of convection in vertical porous cylinders”, Journal of Engineering Mathematics,133, 633-651, (2018).
3. Talebi F, Mahmoudi AH, and Shahi M, “Numerical study of mixed convection flows in a square lid-driven cavity utilizing nanofluid”, Int Commun Heat Mass Transf 37:79–90, (2020).
4. Muthtamilselvan M, Kandaswamy P, and Lee J, “Heat transfer enhancement of copper-water nanofluids in a lid-driven enclosure” ,Commun Nonlinear Sci Numer Simul 15(6):1501–1510, (2020).
5. Moumni H,Welhezi H, Djebali R, and Sediki E, “Accurate finite volume investigation of nanofluid mixed convection in two-sided lid-driven cavity including discrete heat sources”, Appl Math Model 39(14,15):4164– 4179, (2015).
6. Malik, Sumit & Nayak, A., “Effect of moving walls on heat transfer and entropy generation in a nanofluidfilled enclosure”, Journal of Engineering Mathematics, 110 (6), 1-19,(2017).
7. Nayak AK, and Bhattacharyya S, “Double-diffusive convection in a cubical lid-driven cavity with opposing temperature and concentration gradients”, Theor Comput Fluid Dyn 26(6):565–581, (2019).
8. Aizikovich SM, Alexandrov VM, Kalker JJ, Krenev LI, and Trubchik IS. “Analytical solution of the spherical indentation problem for a half-space with gradients with the depth elastic properties”, Int J Solids Struct, 39 (10) 2745-2772, (2020).
9. Krenev L, Aizikovich S, Tokovyy Yu, and Wang Y-C., “Axisymmetric problem on the indentation of a hot circular punch into an arbitrarily nonhomogeneous halfspace”, Int J Solids Struct, (2015).
10. Tilley, B. S., Yang, V. C., Baiense, J. C., & Evans, S., “Frequency-dependent thermal resistance of vertical U-tube geothermal heat exchangers”, Journal of Engineering Mathematics,102, 131-150, (2017).
11. Krenev, L. I., Tokovyy, Y. V., Aizikovich, S. M., Seleznev, N. M., & Gorokhov, S. V., “A numericalanalytical solution to the mixed boundary-value problem of the heat-conduction theory for arbitrarily inhomogeneous coatings”, International Journal of Thermal Sciences, 107, 56-65,(2016).
12. Sun, Y., & Zhang, X., “Analysis of transient conduction and radiation problems using lattice Boltzmann and finite volume methods”, International Journal of Heat and Mass Transfer,97, (2016).
13. González-Santander, and Juan., “Series expansion and asymptotic formulas for heat transfer of an inclined moving heat source”, Journal of Engineering Mathematics, (2016).
14. Chapko R,”On the numerical solution of the Dirichlet initial boundary-value problem for the heat equation in the case of a torus”. J Eng Math 43:45–87,(2020).
15. Chapko R,”On the combination of some semi-discretization methods and boundary integral equations for the numerical solution of initial boundary value problems”, PAMM: Proc. Appl Math Mech 1:424–425, (2020).

16. Chapko, R., & Johansson, B. T. "Numerical solution of the Dirichlet initial boundary value problem for the heat equation in exterior 3-dimensional domains using integral equations". Journal of Engineering Mathematics, 103 (1),(2016).
17. Fu ZL, Zhang YX, Cheng H, Ma YJ. "The a posteriori Fourier method for solving ill-posed problems". Inverse Probl. ,28,168–194, (2021).
18. A. Al-Amiri, K. Khanafer, and I. Pop, "Steady-state conjugate natural convection in a fluid-saturated porous cavity", International Journal of Heat and Mass Transfer, 51,(17-18), 4260-4275, (2018).