June 2021

Bibliometric Analysis on Artificial Compressibility Method based CFD Simulations

Bhavna Hemraj Joshi Miss
Symbiosis Institute of Technology, Symbiosis International Deemed University, India,
Joshi.bhavna1@gmail.com

Chandrakant R. Sonawane Dr
Symbiosis Institute of Technology, Symbiosis International Deemed University, India,
crsonawane@gmail.com

Priyambada bada Praharaj Mrs
Symbiosis Institute of Technology, Symbiosis International Deemed University, India,
priyambada.praharaj@sitpune.edu.in

Anand Kumar Pandey Dr
Symbiosis Institute of Technology, Symbiosis International Deemed University, India,
anand.pandey@sitpune.edu.in

Bibliometric Analysis on Artificial Compressibility Method based CFD Simulations

Bhavna Hemraj Joshi[1], Chandrakant Sonawane*,[1], Priyambada Praharaj[1], Anand Kumar Pandey[1]

[1]Symbiosis Institute of Technology, Symbiosis International (Deemed University), Pune, India

* Corresponding Author - crsonawane@gmail.com

Abstract:

This bibliometric analysis in this paper aims to study the quantitative progress done in the artificial compressibility (AC) method-based CFD simulation and analyze its potential in solving incompressible flow simulations in computational fluid dynamics smoothly. The sector of CFD is enhancing more and more maturely due to advancements in computing architecture, numerical methods, and simulation tools. There have been various attempts to solve the pressure-velocity coupling issue in the Navier-Stokes equation. The artificial compressibility method (ACM), as opposed to pressure-correction methods, solves the incompressible equation in a non-segregated manner. With the introduction of the ACM, the system of the equation becomes hyperbolic in pseudo-time and consequently, numerical techniques used for hyperbolic solvers can be used. This paper focuses on the development done in this ACM formulation-based CFD research since 2000 and the following years. The bibliometric analysis and research are performed based on 139 published papers collected from Scopus. The main purpose of this analysis is to systematically and statistically study the data based on various parameters which will help in searching the gap. This survey can be beneficial while analyzing and studying the type and the measure of work done in this area and the future scope of the research.

Keywords: Computational Fluid Dynamics, Artificial Compressibility Method, Incompressible Flow, Numerical Simulation.

1. Introduction

Over the past few decades, Compressible flow techniques gained more attention due to the stringent level of accuracy required in the simulation of aerodynamic performance. But over recent years, as products have become more sophisticated due to the involvement of complicated geometries, dimensional accuracy, surface finish, and high technical and economic efficiency, the demand for high-performance products and machinery for incompressible flow simulation is arising.

To treat the coupled velocity and pressure, Chorin [1] introduced the ACM firstly in 1967. The unsteady artificial compressibility equations have been introduced and solved. In this technique, an artificial continuity equation with the pressure time derivative, the momentum equation, and the unsteady energy equation (if the heat transfer is taken into account). This method accelerates the convergence to a time-steady, nearly incompressible sector [2, 3]. The ACM is a simplified and efficient method. It is easy to implement to solve steady incompressible flow and the characteristics of ACM have been demonstrated in some published works [4].The artificial compressibility method changes the mathematical nature of the incompressible Navier-Stokes equations from elliptic-parabolic type to hyperbolic-parabolic type [5]. This modified formulation looks very similar to compressible gas dynamics equations. The research carried out for compressible flow is much extensive as compared to incompressible flow. Hence, by making the set of equations compressible-like, all advanced methods available for compressible flow can be applied, in principle. The wave-like nature of the solution aids in time marching and obtaining time accurate results much quicker than other primitive variable methods which require the solution of another derived equation at each time step.

As preliminary research, a bibliographic survey clarifies the quantity of research already done in this area [6 -10], the generic idea of the topic, the need to focus on this research, and the future scope of the research. We have performed the analysis using only two important keywords. This paper will present the literature available on the large databaseof Scopus.

The sections are as below:

1. Section 2: a bibliometric analysis
2. Section 3: network analysis
3. Section 4: the future scope of the study
4. Section 5: conclusion

2. Bibliometric Analysis

2.1 PublicationAnalysis:

Table no. 1 represents the year-wise analysis of publications published from 2000 to 2021 in the sector of Artificial Compressibility Method for CFD. We find that the publications have been fluctuating in numbers in reference years. Overall 70 publications were published in the last decade, whereas 69 were published in the 2000s.

Figure 1 presents the bar graph for the publication in the last decades. Maximum publications have been seen and 2018 whereas the least published in 2015 and 2019. In 2021, 2 publications are seen so far.

Table 1: Year-wise Publication trends (2000 – 2021)

(Source: Data Extracted on March 25, 2021, from https://www.scopus.com/)

Publication year	Total No. of publications	Publication year	Total No. of publications
2021	2	2010	8
2020	9	2009	5
2019	3	2008	8
2018	10	2007	9
2017	5	2006	3
2016	4	2005	5
2015	3	2004	5
2014	10	2003	3
2013	8	2002	6
2012	7	2001	8
2011	9	2000	9

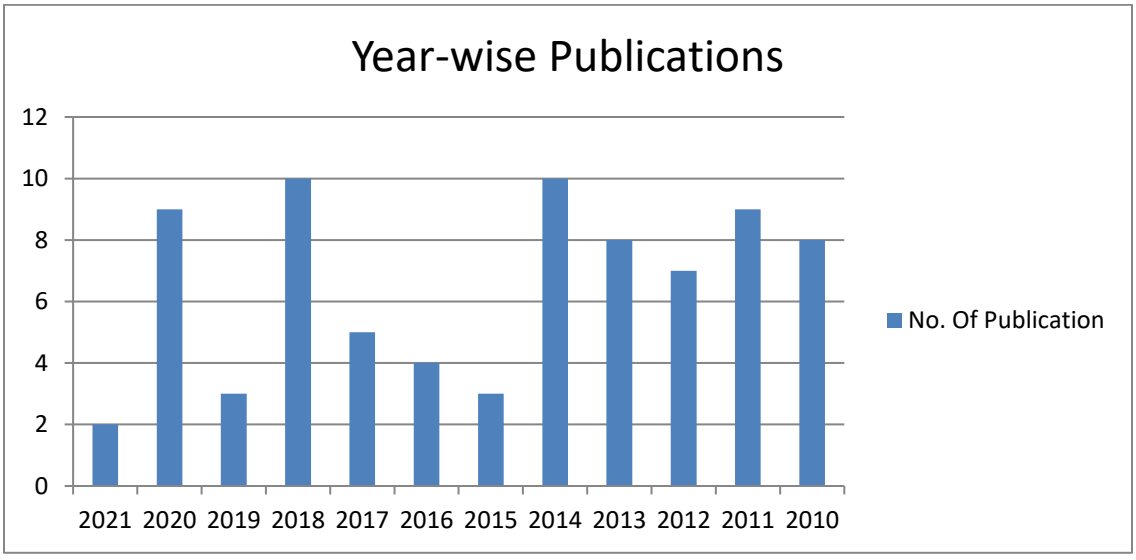

Figure 1: Year-wise publication for the last 10 Years.

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

2.2 Country-wise Publication Analysis:

In Table 2, the Analysis of the number of documents published in the sector of Artificial Compressibility Method for CFD. We can see that maximum research has been done in China, the Czech Republic, and the United States, no. of documents published are 22, 19, and 17 respectively.

Table 2: Country-wise documents published

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

Name of Country	No. of documents	Name of Country	No. of documents
China	22	Switzerland	3
Czech Republic	19	Hong Kong	2
United States	17	Morocco	2
Iran	13	Russian Federation	2
Singapore	13	Spain	2
Pakistan	10	Canada	1
Japan	9	Finland	1
United Kingdom	8	Norway	1
France	7	Poland	1
Greece	5	Portugal	1
India	5	Romania	1
Bahrain	3	Sweden	1
Germany	3	Taiwan	1
Italy	3	Undefined	2
South Korea	3		

The country-wise location cluster can be seen in the world map figure 2. The countries are colored depending on the no. of documents published.

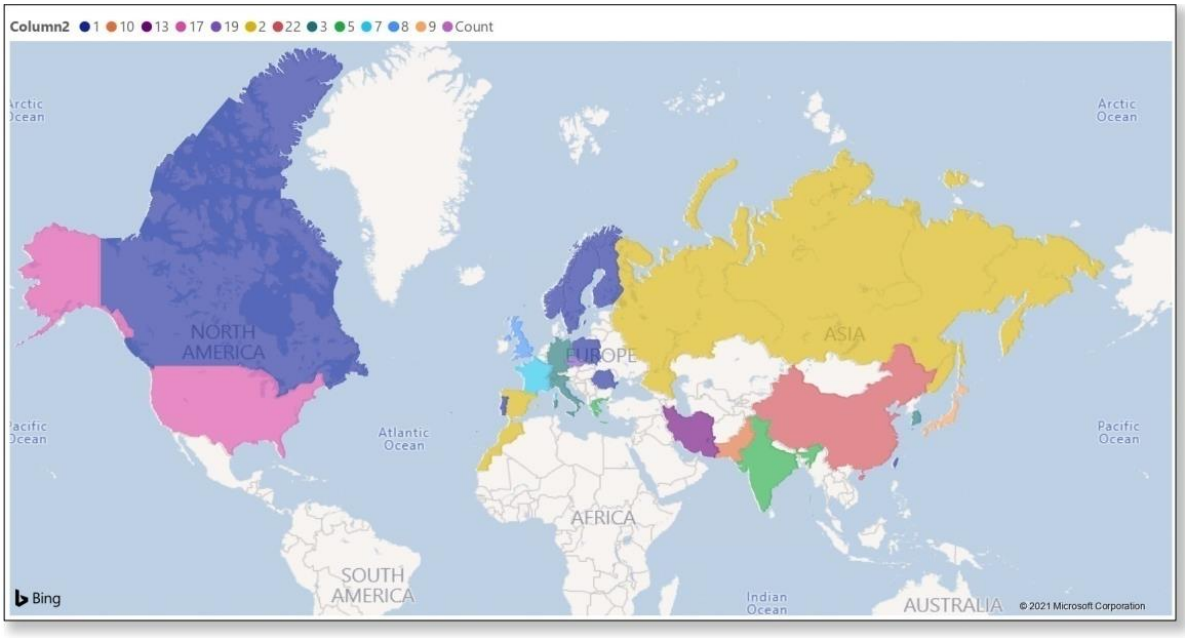

Figure 2: Research on Artificial Compressibility Method for CFD in Several countries

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

2.3. Subject Area Analysis

Figure 3 is a pie chart that informs about the research going on in the streamof Artificial Compressibility Method for CFD. Maximum publications are in the sector of engineering which is about 29%, Mathematics(22%), Physics and Astronomy and followed by computer science (21%). Work done in the other significant areas includes Physics and Astronomy (15%).

2.4. Analysis of Publication category:

In table 3, Category wise document which has been published in the Artificial compressibility method for CFD is presented. From this publication category analysis, we found that the maximum papers are Articles and conference papers. There are very few Book Chapters and Conference reviews. In Figure 4, the pie chart is shown.

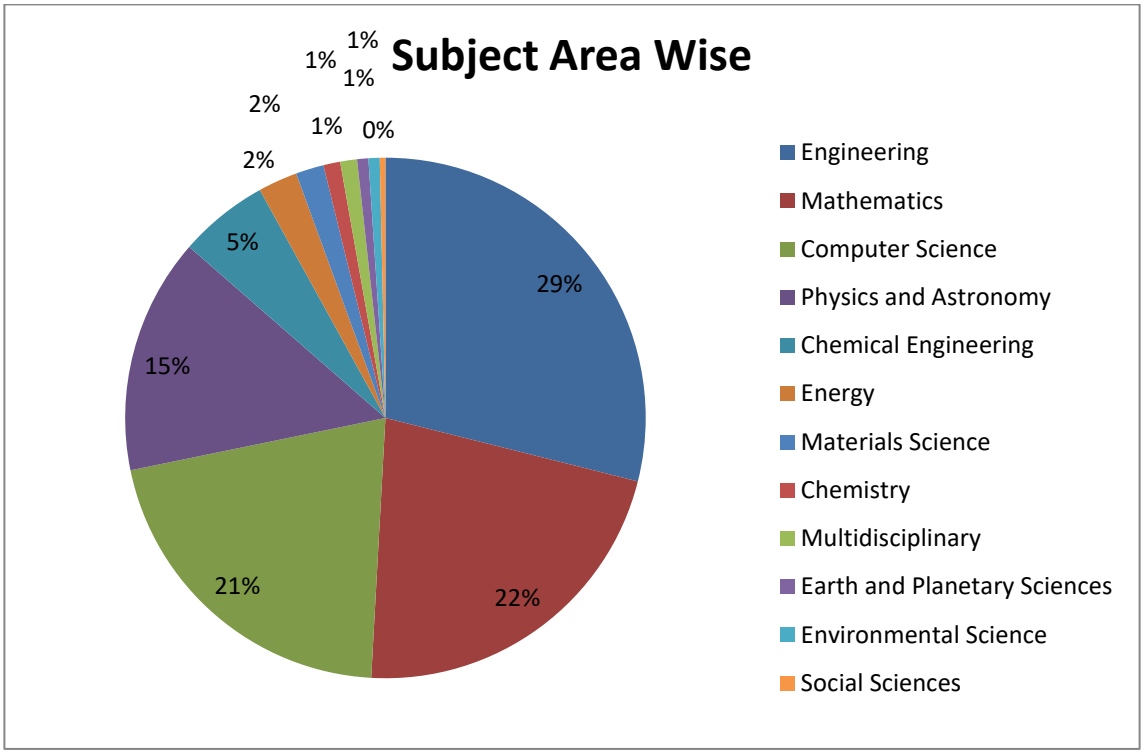

Figure 3: Leading subject areas in Artificial Compressibility Method for CFD

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

Table 3: Document type-wise publications

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

Document category	**Publication**	**Document category**	**Publications**
Article	95	Book Chapters	1
Conference Papers	42	Conference Review	1

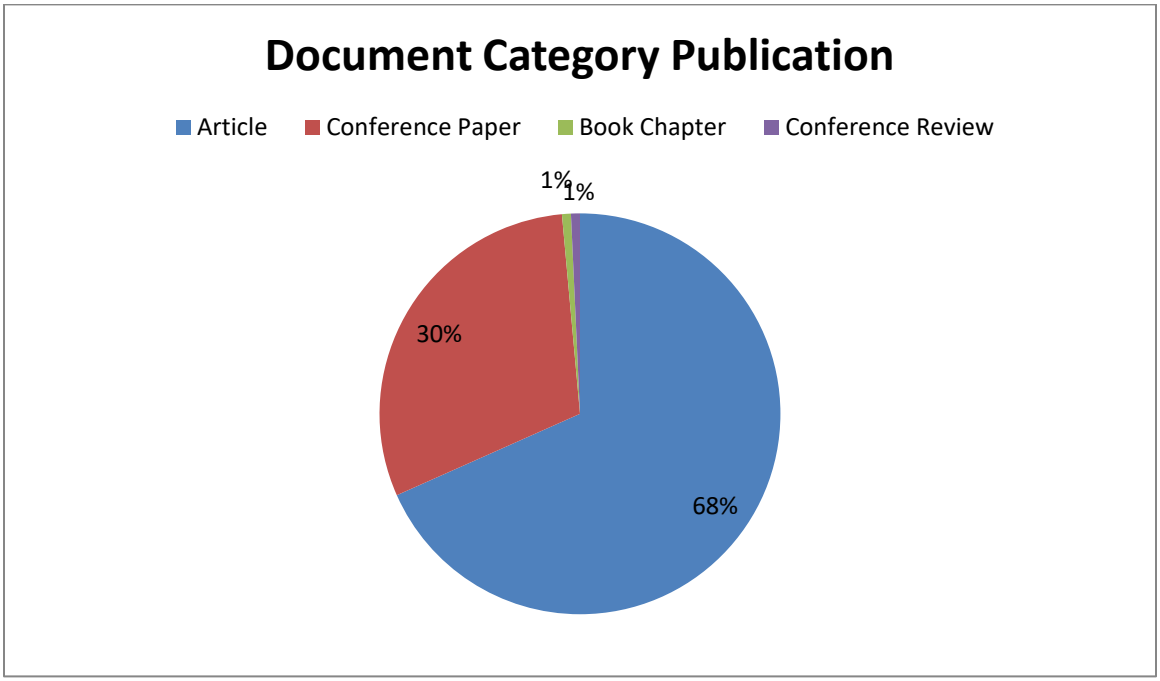

Figure 4: Document Category Publication

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

2.5. Authors based analysis:

The bar graph in figure 5 represents the top 10 authors who have contributed to the sector of Artificial Compressibility Method for CFD. We can see that Kozel, K. (14) has the maximum number of publications followed by researchers KeslerovÃ¡, R. (11) and Zhao, Y. (11) who have a significant contribution to the area.

2.6 Analysis based on affiliated institutions:

In figure 6, the graph represents the top 10 research centers and universities that are contributing to the sector in a higher manner. We can see that CeskÃ© vysokÃ© ucenÃ- technickÃ© v Praze (18) has the most number of publication followed by Nanyang Technological University (12). Other major institutions are the School of Mechanical and Aerospace Engineering (10) and COMSATS University Islamabad (10).

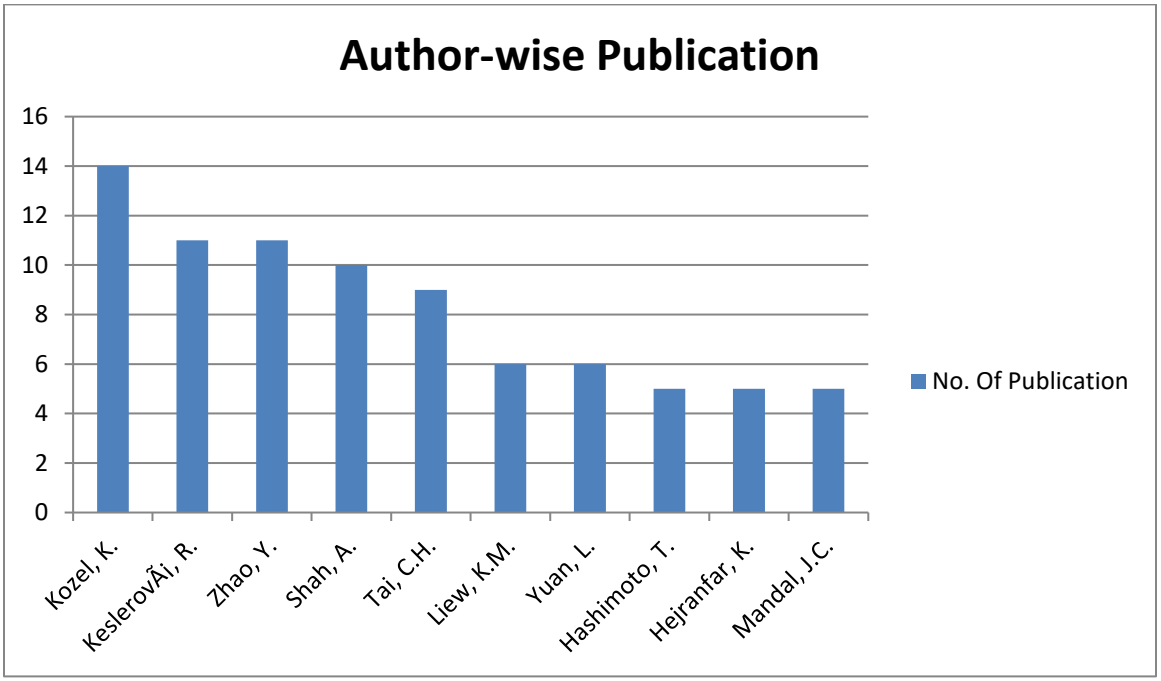

Figure 5: The author wise publications

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

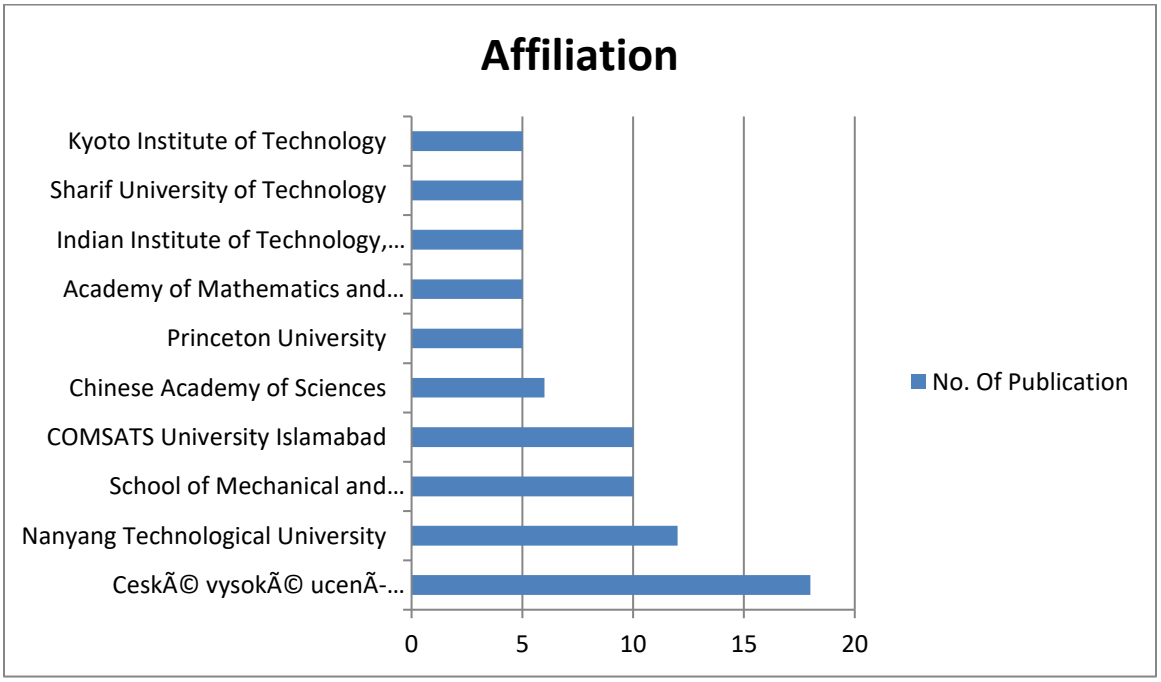

Figure 6: Top 10 Affiliated institutes and universities

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

2.7 Analysis based on the Funding Agency:

In Figure 7, the graph represents the top 10 research centers and universities that are highly contributing funding to help and boost researchers to work more on the sector. As we can see National Natural Science Foundation of China has the largest number of researchers (10) whereas Nanyang Technological University (4), Sun Microsystems (3), and the rest have 2 publications each.

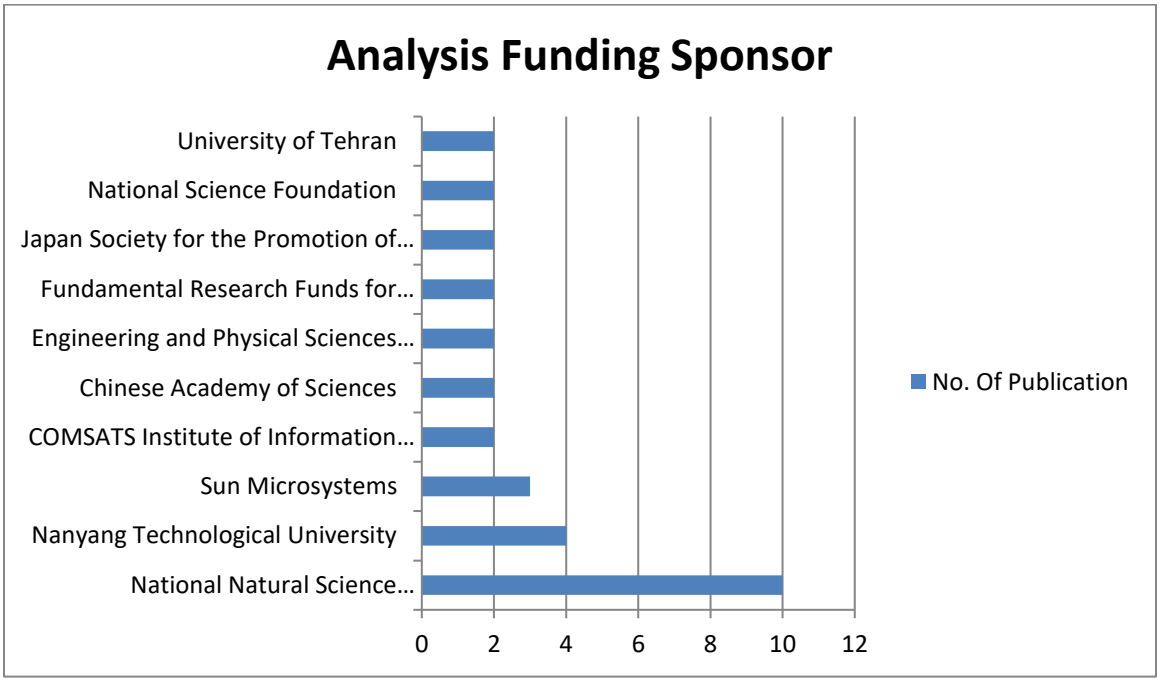

Figure 7: Top funding sponsors

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

2.8 Citations based analysis:

Figure 8 represents the citation analysis of the topic for the past 10 years in the sector of Artificial Compressibility Method for CFD. We can see a decrease in the citations from the graph taken from the Scopus Database. The graph represents the fluctuation but decreasing citations in the last 11 years i.e. 2010-2021.

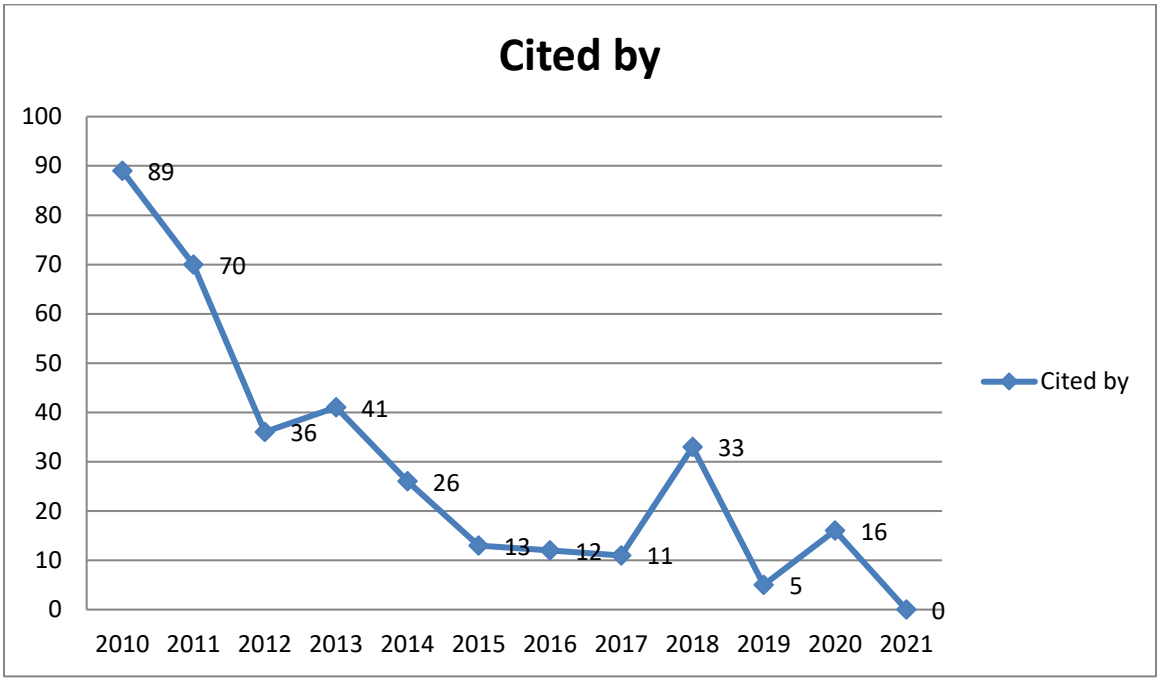

Figure 8: Citations in Last five years

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

3. Network Analysis

Various software can portray Network Analysis to give an understandable picture of the various attributes of the research. In this research, the network analysis of Scopus data is done using the Vosviewer software. Network analysis of source titles is studiedby considering two attributes for comparison. We studied the network analysis between source titles, Author keywords, and index keywords based on co-occurrence and represented in figures 9 and 10. The extent of occurrence is indicated by the size of the graph. In figure 9, The closeness of the relationship among keywords is indicated by the distance between them in the graph.Artificial compressibility shows a very strong connection with other keywords.The color system represents the connection amongst the keywords connected using lines.We observed that 318 keywords crossed the threshold by setting the bar of minimum number 1 of the occurrence of keywords. In figure 10, the authors' collaboration on the keywords is given.

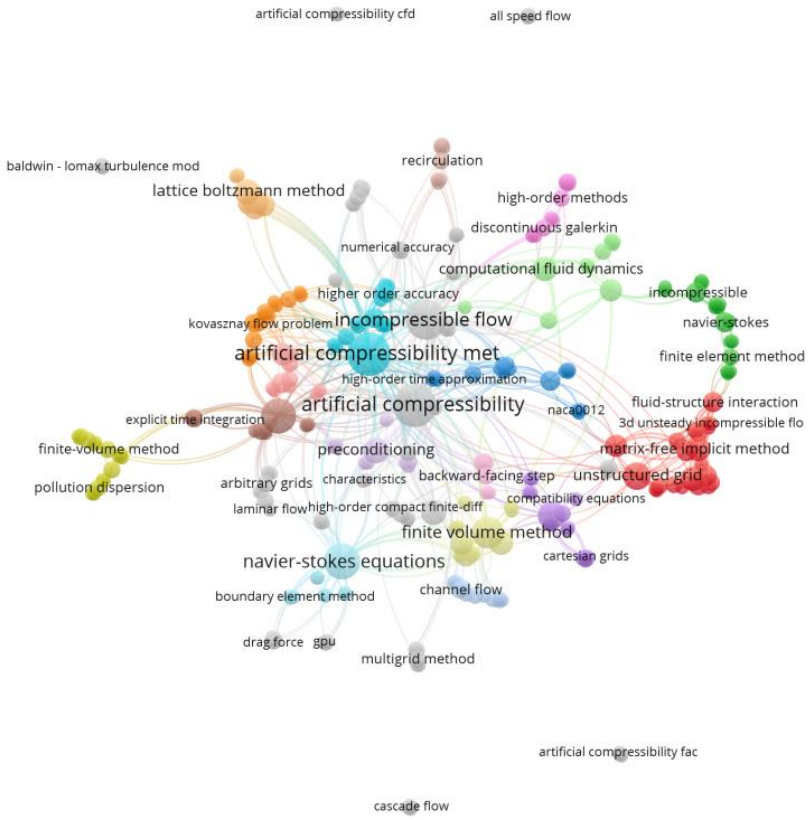

Figure 9: Author keywordsbasedNetwork Analysis diagram

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

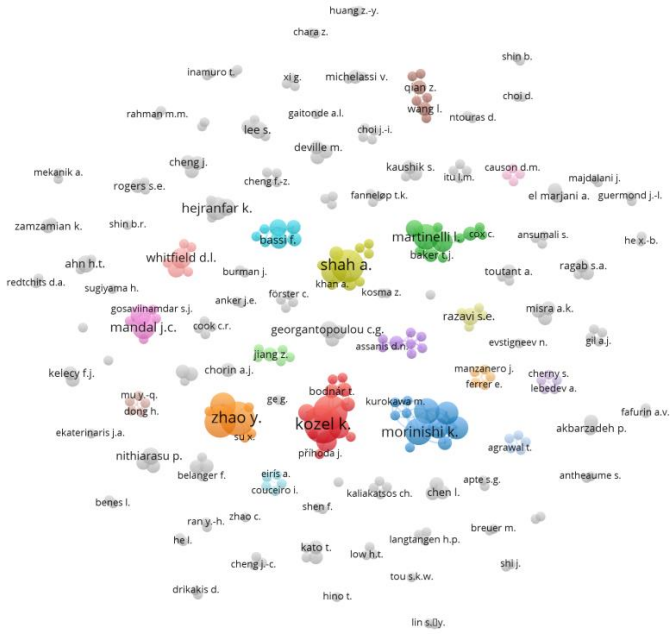

Figure 10: Authors collaboration basedNetwork Analysis diagram

Source: Data Extracted on March 25, 2021, from https://www.scopus.com/

4. Future work:

In this paper, we have analyzed and presented a generalized survey for the Artificial compressibility method for the simulation of incompressible flows. For network analysis, we have usedVOSviewer. We have analyzed the results using Scopus data. Similar analysis can be done using Web of Science, Google scholar data. This survey was generic. Similar analysis can be done by focusing on the specific research area like high order accurate methods for ACM formulation. [11 - 16].

5. Conclusion:

Bibliometric data analysis on CFD simulations using the Artificial Compressibility Method provides us a brief insight into the statistical analysis of the research work being carried out in the particular area. Because of the complexity involved in this area, we can observe that very little research has been done in this area. This representsthe need for research in this sector. From the geographical analysis, we can observe that China, Czech Republic, and the USA have been enthusiastically contributing to the particular research. From this bibliometric survey, we noticed that a particular subject or topic can be researched not only from the engineering domain but also from various other domains like Computer Science, Mathematics, Physics, Astronomy, etc. The author survey helped us in finding out the major contributors that are contributing to these research areas of interest and the quality of work was also analyzed through citations. The major contribution in this area is carried by Kozel, K. (14) has the maximum number of publications followed by researchers KeslerovÃ¡, R. (11) and Zhao, Y. (11).

REFERENCES:

[1] A. J. Chorin. A numerical method for solving incompressible viscous flow problem. Journal of Computational Physics, (Article No. CP975716) Volume 135, Issue 2, 1997, Pages 118-125, ISSN 0021-9991, DOI: https://doi.org/10.1006/jcph.1997.5716.

[2] Fan Zhang et al. A simplified artificial compressibility flux for the discontinuous Galerkin solution of the incompressible Navier-Stokes equations.Commun. Comput. Phys., 25 (2019), pp. 988-1009. Published online: 2018-12DOI:10.4208/cicp.OA-2018-0051

[3] Jing-Kui Zhang et al. A combined method for solving 2D incompressible flow and heat transfer by spectral collocation method and artificial compressibility method, International Journal of Heat and Mass Transfer,Volume 112, 2017, Pages 289-299, ISSN 0017-9310, DOI: https://doi.org/10.1016/j.ijheatmasstransfer.2017.04.051

[4] Christopher Cox et al., A high-order solver for unsteady incompressible Navier–Stokes equations using the flux reconstruction method on unstructured grids with implicit dual time-stepping, Journal of Computational Physics, Volume 314, 2016, Pages 414-435, ISSN 0021-9991, DOI: https://doi.org/10.1016/j.jcp.2016.03.016

[5] Sonawane C.R., Mandal J.C.,Simulation of moderator flow and temperature inside calandria of CANDU reactor using artificial compressibility method,Heat transfer

engineering,Taylor & Francis Group,Volume 35, 2014. DOI: https://doi.org/10.1080/01457632.2013.876802

[6] JC Mandal, CR Sonawane, Incompressible flow computations over moving boundary using a novel upwind method, Computers & fluids, Volume 46, 2011. DOI: https://doi.org/10.1016/j.compfluid.2010.08.011

[7] Shah A et al. Fourth-order central compact scheme for the numerical solution of incompressible Navier Stokes equations. International Journal of Computer Mathematics, 94: 2492-2507, 2017 DOI:https://doi.org/10.1080/00207160.2017.1284315

[8] Suzuki K., Inamuro T., Nakamura A., Horai F., Pan K.-L., Yoshino M., Simple extended lattice Boltzmann methods for incompressible viscous single-phase and two-phase fluid flows, Physics of Fluids, Volume 33, 2021 DOI: https://doi.org/10.1063/5.0041854

[9] Shi X., Agrawal T., Lin C.-A., Hwang F.-N., Chiu T.-H., A parallel nonlinear multigrid solver for unsteady incompressible flow simulation on multi-GPU cluster, Journal of Computational Physics, Volume 414, 2020 DOI: https://doi.org/10.1016/j.jcp.2020.109447

[10] Sonawane C.R., Mandal J.C., Rao S., High-Resolution Incompressible Flow Computations over Unstructured Mesh using SDWLS Gradients, Journal of The Institution of Engineers (India): Series C, Volume 100, 2019DOI:10.1007/s40032-017-0390-x

[11] Georgantopoulou C.G., Vasilikos N.S., Georgantopoulos G.A., Recirculating flows analysis and estimation inside channels, MATEC Web of Conferences, Volume 172, 2018. DOI: https://doi.org/10.1051/matecconf/201817201001

[12] Sonawane C.R., et al Numerical simulation of flow-through heat exchanger having helical flow passage using high order accurate solution dependent weighted least square based gradient calculations, Energy Sources, Part A: Recovery, Utilization, and Environmental Effects:Taylor & Francis, 2021. DOI: https://doi.org/10.1080/15567036.2021.1900457

[13] Sonawane C.R., et al Numerical simulation of vortex induced vibrations of a circular cylinder: isothermal and heat transfer cases, E3S Web of Conferences,2019. DOI:10.1051/e3sconf/201912809001

[14] Sonawane C.R., et al. Numerical simulation of unsteady channel flow with a moving indentation, E3S Web of Conferences,2019.
DOI: https://doi.org/10.1080/01457632.2021.1874661

[15] Mandal, J.C. and Sonawane, C.R. (2013), "Simulation of flow inside differentially heated rotating cavity", International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 23 No. 1, pp. 23-54. https://doi.org/10.1108/09615531311289097

[16] C. R. Sonawane, Y. B. More, Anand Kumar Pandey, “Numerical simulation of unsteady channel flow with a moving indentation using solution dependent weighted least squares based gradients calculations over unstructured mesh”, Heat Transfer Engineering Journal, 2021, (SCI), doi: https://doi.org/10.1080/01457632.2021.1874661