CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

CFD Analysis of Hydrogen Gas Behaviour in a Multi-Zone Flow System: A Parametric Study

Ramesh Kumar R.[1], Yogeesha H.C.[2], Channa Keshava Naik N.[3], Abhishek Bhattacherjee[4], Abhijit Bhowmik[5], Mayur Dondiram Pawar[6], Sameer Algburi[7], Salah Jasim Mohammed[8], Hasan Shakir Majdi[9], Sher Afghan Khan[10,*]

1 Vel Tech Rangarajan Dr.Sagunthala R&D Institute of Science and Technology, Chennai, Tamil Nadu 600062, India
2 Department of Mechanical Engineering, Nagarjuna College of Engineering and Technology, Venkatagirikote Post, Devanahalli, Bengaluru, Karnataka 562110, India
3 Department of Mechanical Engineering, BGS College of Engineering and Technology, Mahalakshmipuram, Bengaluru, Karnataka 560086, India
4 Department of Computer Science Engineering, Lovely Professional University, Phagwara, Punjab 144411, India
5 Chitkara Centre for Research and Development, Chitkara University, Rajpura, Punjab 140401, India
6 Department of Mechanical Engineering, Ballari Institute of Technology and Management, Ballari, 583104, India
7 Al-Kitab University, Kirkuk 36015, Iraq
8 Civil Engineering Department, Dijlah University College, Baghdad 00964, Iraq
9 Department of Chemical Engineering and Petroleum Industries, Al-Mustaqbal University College, Babylon 51001, Iraq
10 Department of Mechanical Engineering, International Islamic University Malaysia, 53100 Kuala Lumpur, Malaysia

ARTICLE INFO

Article history:
Received 3 December 2024
Received in revised form 5 January 2025
Accepted 12 March 2025
Available online 30 April 2025

Keywords:
Hydrogen; Ansys CFD; oxygen; static pressure; temperature; velocity; NH_3

ABSTRACT

The ANSYS CFD analysis made use of hydrogen in this study. The investigation aims at understanding the behaviour of hydrogen gas in such interactions with the inlet air and other fluid zones around it. Such research took advantage of sophisticated simulation tools to offer useful insights into gas flow and mixing patterns, thereby justifying why hydrogen could possibly become an alternative fuel one day (hydrogen is future-oriented). In addition, a model was created to establish the interaction between combustion involving hydrogen as well as tanks of inlet air. This was done on volumes for such tanks. Many factors affect performance and efficiency for different operational conditions of hydrogen gas that were explored through elaborate simulations. These findings should improve our comprehension about physics involved which plays an important role in enhancement of systems using hydrogen for power generation and also help various industries' operations using this element in their processes. It is research that looks at the complicated flow structures and behaviours that arise in fluid domains to uncover key parameters for better designs and more efficient hydrogen systems. Consequently, it is expected that the present study will help future designs of hydrogen technology so that they play a part of sustainable energy source.

* *Corresponding author.*
E-mail address: sakhan@iium.edu.my (Sher Afghan Khan)

https://doi.org/10.37934/cfdl.17.10.76105

1. Introduction

This article describes briefly about Computational Fluid Dynamics (CFD) especially for H_2 gas with the help of simulation tool ANSYS. This will analyse how hydrogen gas behaves and interacts with incoming air (using normal Laminar Inlet) and fluid zones that exist within such specific system. Advanced simulation tools will be utilized in this research to investigate flow dynamics, mixing patterns and performance implications of H_2 gas under various operating conditions. The aim of this study is to provide a short review Computational Fluid Dynamics (CFD) from the perspective of hydrogen gas by using ANSYS software package. The main purpose is to investigate the behaviour and interaction of hydrogen gas with inlet air zone, fluid zone in a real system. The study aims to understand the flow dynamics, mixing patterns and performance aspects of hydrogen gas in a wide range of operational conditions with use of state-of-the-art simulation methodologies.

This article provides an overview in brief about CFD relating primarily to H_2 gas precise through Newtonian simulation tool such as ANSYS. In analysing how does hydrogen react and behave during its contact with incoming air (normal Laminar Inlet) as well as fluid zones that are inherent in particular system, this plays an essential role. Through this research advanced simulations will be deployed so as to establish flow dynamics, mixing trends and performance implications pertaining to H_2 under different operational states. The objective is simply an attempt at providing a basic outline regarding Computer Fluid Dynamics (CFD) from the angle point view of H_2 gas employing package called ANSYS simulation. The research focuses on understanding one aspect of behaviours found when interacting within inlet air zone vis-a-vis real life scenarios representing different fluid zones towards hydrogen gas. Therefore, besides utilizing new simulation methods known in the field, this investigation shall be looking into hydrogen flow dynamics, mixing patterns and performance perspectives under various operating conditions of interest.

The report provides a short background on Computational Fluid Dynamics (CFD) of hydrogen gas by using ANSYS making it a software platform. In this research the main objective is to study the influence of hydrogen gas with inlet air and fluid regions in a particular system. The project intends to apply advanced techniques of simulations so as to obtain findings about flow dynamics, mixing patterns and effects of using hydrogen gas under different working circumstances.

Description this analysis provides insight on fluids flow when hydrogen is used as Computational Fluid Dynamics in which ANSYS has been employed as a Software Platform. This study investigates the interactions between hydrogen gas and inlet air, while also taking consideration of liquid zones in specific systems. Hence, this research focuses on understanding how the presence of molecular hydrogen affects thermal and mechanical parameters of fluids by using advanced simulation techniques at several operating conditions.

Our researchers' undertaken work is concisely presented herein this commemoration of ANSYS using hydrogen gas and Computational Fluid Dynamics (CFD). The principal objective is to examine hydrogen gas behaviour under interactions between inlet air and fluid zones enclosed in a particular system's working volume. The study employs high-fidelity simulation techniques to shed light on flow dynamic, mixing parameters and possible implications for performance with differential operating conditions of hydrogen gas.

Theoretical Computational models of Hydrogen gas: Additionally, Computational Fluid Dynamics (CFD) is used to analyse velocity contours and pressure distributions in air and other fluid areas as shown in Ansys. Thus, the intention of this study is to show the behaviour of hydrogen gas with respect to air and fluid environment around it, so that one can understand how a gas flows or pressure varies. Therefore, this study seeks to provide insight into such complex phenomena occurring at the interface between hydrogen and other fluids that are important for energy applications and safety analysis through advanced simulation techniques.

This paper presents a study on hydrogen gas simulated using Computational Fluid Dynamics (CFD) focusing mainly on velocity contour and pressure distribution within the inlet air zone as well as Fluid Zone modelled in ANSYS. The objective here is to understand how hydrogen behaves when it is mixed with other gases and some liquids in moving conditions because it contributes significantly to the comprehension of gas flow dynamics and pressure variations. Sophisticated simulation methods will address the complicated interactions between H_2-fluid interfaces which have implications for energy systems design and safety assessments.

Introduction When the velocity contours and pressure distributions are examined in ANSYS Computational Fluid Dynamics (CFD) for both inlet air zone and fluid region; it is expected that the evaluation of hydrogen gas will be a well thought out concept. The aim of this study is to give an insight into how hydrogen gas behaves in a combined air and fluid environment, thus providing some understanding on flow dynamics and pressure fluctuations. This study which seeks to explore more about complex behaviours of hydrogen as well as other fluids under dangerous conditions pertinent to energy systems as well as safety involved utilization of sophisticated simulation methods.

Simulations of hydrogen gas with ANSYS are involved in CFD research that outlines flow patterns within the inlet air and fluids examined during this investigation. The ultimate goal is to gain a deeper understanding of complaints that arise when hydrogen encounters external air or submerged fluids which makes it easier to comprehend such phenomena as gaseous dynamics or pressure oscillations. This research emphasizes advanced modelling techniques in order to investigate interfacial processes between hydrogen and other fluids, which have been known to be crucial in the design of energy systems and their safety evaluations.

The purpose of this study is to observe CFD velocity contours and pressure distributions of hydrogen gas in inlet air and fluid zones on ANSYS. The study uses those simulations to understand how a gas such as hydrogen reacts with air, the most familiar fluid in our environment.

Dolan [1] carried out research on CFD simulation of a membrane reactor for hydrogen production from ammonia. The study utilizes current best practices in reactor modelling and examining chemical compounds to understand how we can scale up these types of successful processes for hydrogen generation. Membrane reactor technology is making significant advancements that could help with future sustainable energy production using the same method. The main point of this research is that it highlights the increase in popularity of membrane reactors as an alternative pathway for generating hydrogen from ammonia which may have some implications for the longer-term sustainability of energy generation. In combination with CFD modelling and plasma technology, this has produced a new approach towards further research & development in this area by opening up different directions.

A study was conducted by Hayakawa *et al.,* [2] which investigated the production of hydrogen through decomposition of ammonium using plasma membrane reactors. As the authors have discussed, experimental results show that hydrogen production yields can be improved with plasma-assisted techniques and that varied conditions affect reactor behaviour. The research not only points to the application of plasma technology in examining energy aspects but also indicates alternative means for producing hydrogen. In addition, all previous research indicates that there has been an upsurge in applying membrane reactors for hydrogen from ammonia production, thus making it a viable solution for addressing energy sustainability challenges. This is a huge step forward in the field combining Computational Fluid Dynamics (CFD) modelling with plasma technology for near future.

Solar-driven multichannel ammonia decomposition for potential hydrogen production by Xia *et al.,* [3]. This innovative technique utilizes sunlight to induce the reaction that turns out to be much more efficient for hydrogen production. They also elaborate on the multichannel design of the reactor as well as its operating principle, which increases heat and mass transfer thereby giving rise to greater hydrogen yield. In their study Xia *et al.,* [3]. A novel way in which they used solar energy was driving off the chemical reaction and made this thus making it more efficient in producing for instance hydrogen The theory behind the operation of a reactor is given by authors themselves whereupon its mechanical construction enabling good quality transfer of heat and matter thereby enhancing hydrogen output.

In 2023, Hayakawa *et al.,* [4] did research that ends in advanced ammonia-based hydrogen production systems utilizing a dual catalytic reactor and plasma membrane. This research provides substantial insights into existing advancements in hydrogen production technologies which are vital in achieving sustainable energy solutions. The article explores practical utilization of two types of reactors used for ammonia hydrogen technology by presenting one with an infinite source capable of being synthesized at low energy costs. The purpose of using a catalytic reactor is to trigger ammonia decomposition while plasma membrane reactor enhances the separation purification process for hydrogen; hence both reactors will work towards improving efficiency. Meeting global sustainability targets requires such an integrated approach not only increase hydrogen production but also reduce its energy consumption along with environmental impact.

The research by Fauzi *et al.,* [5] introduced a technique for producing high purity hydrogen from ammonia using membrane reactors. The H_2 molecular sieve made of the alloy MRT24 Type 120 (Fe-70 wt% Pd) was applied as solid wall separating two compartments, while Ru/Cs_2O/Pr6O11 was used for ammonia decomposition specifically. The findings published in the International Journal of Hydrogen Energy indicate this method can revolutionize hydrogen production technologies. A remarkable aspect of this study is presenting a hybrid design that incorporates both membrane technology with high purity and catalytic activity at one place towards improving the yield and purity of hydrogen from ammonia. It is worth mentioning here that upon diffusion through V-10mol%Fe alloy membrane only H_2 gas passes across leaving other gases out. This selectivity is crucial in enabling production of ultra-pure hydrogen which can serve different energy and industrial needs.

Arafat *et al.,* [6]. A multiphase model is established using Computational Fluid Dynamics (CFD) as well as the combination to simulate the complex phenomena occurred during bio-hydrogen production. In this research work, their investigations aimed at looking at behaviours of such substances to enhance efficiency in a new packed bed reactor with continuous flow principle applied. As such, therefore; extensive examinations of reactive multiphase flows were conducted through CFD by authors in order to obtain significant knowledge for better performance on output bio-hydrogen from this reactor. This study should make useful contributions in designing and operating reactors with a view on energy generation incorporating ideas of sustainability. This present study aims to develop a multi-phase model for bio-hydrogen production using Computational Fluid Dynamics (CFD) simulations. The packed bed reactor system is investigated in their study alongside the flow-through operation to learn more about continuous-flow bio-hydrogen production that is self-sustained. Before the work for optimized bio-hydrogen yield was being laid down, it relied to a large extend on Computational Fluid Dynamics in investigating multiphase interactions involved in the reactor. Therefore, the findings of researchers in this work might enhance our knowledge an account of designing and operating of reactors being developed for renewable energies generation.

In another fascinating study by Carpio *et al.,* [7] hydrogen production from ammonia decomposition using a Ru/Al_2O_3 commercially available catalyst in a microchannel reactor. The study also looked at hydrogen generation efficiency and effectiveness one of the increasingly important fields in the world that wants to reduce its energy dependence on clean alternatives. The authors demonstrate a strong concurrence between their results and experimental data as well as themselves validated through CFD simulations which give credence to the findings. This research not only provides experimental data for what could be an innovative technique but through CFD modelling it also analyses some theoretical aspects of this process. Such combined methodologies help develop better understanding of reaction mechanisms governing ammonia decomposition and performance characteristics linked with Ru/Al_2O_3 catalyst operated under different conditions. A notable aspect about this study is its comprehensive integration of experimental and simulation methods expanding its scope within hydrogen energy research.

The study conducted by Shaikh *et al.,* [8] proposes a new method for the production of hydrogen, thereby relying on solar energy in a multi-channel membrane reactor which is not only meant to produce hot gases but also function as an ion conduit. It is important to note that this type of reactor for electricity generation differs from conventional methods in that it does not rely on burning fuels or other processes that create significant amounts of atmospheric carbon dioxide. Hydrogen produced using solar power could easily capture some of the market lost against natural gas or coal, while helping with the development of more environmentally friendly automobile engines among other applications. In general terms, we can say that these results are significant as they provide insight into issues facing future societies in regard to sustainable renewable energy technologies and their advancement. For example, through ammonia decomposition there is another way in which solar-based chemical production could contribute towards hydrogen production. This makes it a must-read for all researchers and professionals involved in this area because it provides important information about how one could design such reactors with solar energy.

According to Shaikh *et al.,* [9] states that a solar driven multichannel membrane reactor for hydrogen production through ammonia decomposition was designed. This paper presents comprehensive efforts towards the synthesis of membrane reactor, which enables efficient operation of existing methodologies and technologies of ammonia decomposition. They examine how this could improve reaction kinetics and product separation thereby optimizing hydrogen yield. Literature review highlights some challenges faced in developing solar based systems like the choice of materials, reactor design and incorporation of solar energy capturing technologies, providing an overview of previously conducted research activities. The paper combines previous research findings to identify major themes in research, gaps in literature and recommend future directions for more effective or cost-effective alternatives. The work contributes to the ongoing academic discourse about renewable energy technologies and serves as a reference document for subsequent inquiries aimed at innovation in H_2 production from ammonia decomposition.

Samylingam *et al.,* [10] conducted comprehensive research on hydrogen production using a silica membrane reactor while using simulation method for the CFD analysis to examine both efficacy and efficiency of such reactors on-site during syngas production which is crucial in sustainable energy applications. With this study, they attempted to increase the hydrogen production ratio by utilizing a silica membrane reactor so as to separate hydrogen from reaction mixture. The importance of this CFD analysis is that it enables simulations of the reactor under various operating conditions thereby serving as a research tool. This approach to methodology can also play a great role in designing the reactors that would yield more H_2. The discoveries are then brought to the wider discussion of green renewable hydrogen generations. Comprehensive Computational Fluid Dynamics (CFD) analysis of water-gas shift (WGS)-based hydrogen production in hybrid sorption-enhanced membrane reaction concept 2017. The methodology for CFD analysis is given in detail since it entails all simulation parameters and physical models that are applicable in running hybrid reactor conditions. Indeed, simultaneous adsorption with membrane separation has several exclusive advantages like a deeper comprehension on reaction kinetics and mass transfer process within it. This study offers a down-to-earth view of how this kind of a device performs which further reinforces getting high amounts of hydrogen at reduced energy expenditure alongside low operational costs through high-end computational instruments.

In Itoh *et al.,* [11], came out with an inspiring study focusing on a tube-wall catalytic membrane reactor based innovative method for low-temperature ammonia decomposition. This review seeks to highlight how hydrogen gas can be generated and why it will be more important than ever before as renewable source of energy source in the near future. It then goes to elaborate on the materials that are needed for operations and how they arrived at a best-case scenario reactor that is expected to produce more hydrogen using less energy. With these conditions, hydrogen purity above 99% volume fractions is realizable in membrane reactor tube-wall catalytic method at lower temperature compared with other existing methods which depend on high heat for excess combustion and poor by-products separation leading to reduced energy dissipation.

In Zakeri *et al.,* [12] examined Tesla microchannel reactor's simulation model for hydrogen generation through glycerol steam reforming. The researchers focus mainly on the efficiency and high production capacity aspects of hydrogen production from glycerol reforming while using microchannel reactors. Therefore, CFD modelling provides more insights into reaction kinetics as well as fluid flow characteristics that fundamentally contribute to the search for optimal operating conditions for maximum quantity of hydrogen. This paper conveys some outcomes that greatly improve our comprehension of techniques used for making renewable hydrogen from biomass-sourced glycerol.

Hydrogen production utilizing ammonia in membrane reactor, as tested through plasma thawing by Kambara *et al.,* [13] has offered a new dimension for the field. This plasma membrane reactor has been explained in terms of drives and performance needed for increasing hydrogen yield while reducing polluting effects on environment. The research outlines information about the best conditions that the reactor should operate under and also discusses experimental design. This is illustrative of the advantages this strategy may have, as well as giving a comprehensive kinetic mapping which can be beneficial for discourse on sustainable hydrogen-producing routes.

As per the study conducted by Nailwal *et al.,* [14], henceforth, Hydrogen Production employing a Packed Bed Catalytic Membrane Reactor for ammonia decomposition based on decarburization is what we talk about. One area that has assumed significance in light of increasing global energy needs and environmental concerns is sustainable hydrogen production. This article provides an overview of how packed bed catalytic membrane reactors can be applied to enhance hydrogen generation through improved efficiency of ammonia decomposition. A detailed discussion on experimental conditions, catalytic materials and operational parameters affecting the reaction process is presented by the authors. This work hopes to combine these diverse elements so as to maximize total hydrogen output at minimal energy input with low emissions. The findings may also assist hydrogen energy researchers and practitioners most especially in designing better catalytic systems.

Nikzad *et al.,* [15] study entails a wide-ranging comparison of three reactor types for ammonia production including RSFR (Radial Spherical Flow Reactor), axial systems that are made of spherical or fully spherical geometric shapes and tuber-like reactors. The authors dive deep into each type by looking at their merits as well as demerits whilst examining hitting flow patterns as well as reaction kinetics on key performance indices such as ammonia output. In order to visualize and explore many phenomena taking place during nitrogen fixation the researchers developed digital mock ups of such reactors using CFD simulations thereby giving an idea of how different designs influence the efficiency. Overall, this systematic review expands the knowledge base chemical engineering and offers recommendations on optimizing reactor designs in industrial applications.

In this research area, as pointed out by Cechetto *et al.,* [16], significant progress has been achieved. This research involves the preparation of ultra-pure hydrogen from ammonia decomposition over a catalytic membrane reactor. Therefore, the article presents their methods and results in terms of experimental procedures followed when applying this novel method, thus making it availed for publication under International Journal of Hydrogen Energy. The study demonstrates that catalytic membrane reactor is one of accessible methods regarding efficiency and efficacy for ammonia decomposition which is quite essential for obtaining pure hydrogen gas. The authors have painstakingly described experimental setups, catalysts used and operational parameters optimized to ensure maximum yield is realized in the production of ultra-pure hydrogen. These findings provide valuable insights into the ammonia-hydrogen carrier concept and could facilitate future advancements in membrane technologies for this purpose.

This's study by Li *et al.,* [17] has emphasized on some vital advances in ammonia decomposition specifically in bimodal catalytic membrane reactor amalgamated arrangement. A new scheme is preferred where the gas industry will be let free to generate hydrogen from carbon dioxide without any hindrance-this being the pure stuff that today you require for producing numerous well-known industrial goods. The article written about catalytic ammonia decomposition for green generation of hydrogen main thrust was made towards supporting very efficacious transformation of gaseous ammonia into hydrogen in terms of their actual speeds-catalysis activity and membrane transport properties are measured by classical catalysts directions. The authors clarify how their solution can improve performance by more than 50% through increasing ammonia breakdown rate and enhancing hydrogen production quality. This bimodal catalytic membrane reactor utilizes both membrane and catalytic features for optimal ammonia decomposition. The results suggest that this reactor showed much better reaction rates as well overall yields such that it could potentially be used instead of existing ones.

The paper by Koyunoğlu *et al.,* [18] discusses the use of a fluidized bed reactor for making dimethyl ether (DME) from syngas. The fluid dynamic environment (CFD) was used to look into and improve the production process computationally. This study aims at investigating gas-solid interaction and elucidating how reactor productivity varies with changes in operating conditions via simulation influences on CFD. Specifically, a detailed Computer Fluid Dynamics (CFD) simulation of an integrated hybrid reactor using solar and electric energy along with methane cracking was provided by Msheik *et al.,* [19]. Such research was based on this CFD simulation employed that allowed for a detailed analysis of how such type of reactor performs during different operating scenarios. The performance of the hybrid reactor is expected to increase considerably when sun light and heat source are combined together facilitating methane cracking a key strategy for clean energy application with guaranteed CO_2 net-zero emissions.

It's possible to use NH_3 to promote sustainability just like for hydrogen, which is why an analysis of the in-situ H production from ammonia decomposition via membrane reactor designs has been included here on their possible feasibility in producing purified hydrogen that would also work well in Proton Exchange Membrane (PEM) fuel cells. Even if these results stem from a research article published in the Korean Journal of Chemical Engineering, it must be stated that such a need is supported by them; so as to come up with efficient technologies for generating hydrogen free of CO_X processes leading to sustainability and operation enhancement through fuel cells. In this review paper, such concerns are elaborated through a case study on ammonia breakdown mechanism and principles giving a complete overview of potentially available advantages using micro-sized membrane reactors. Some operational parameters and design aspects can be optimized for hydrogen production from ammonia focusing particularly on their approach's innovative features. The use of a membrane reactor in this study will enhance the selectivity of hydrogen separation from other by-products which is another significant step towards boosting fuel cell system efficiency as well. Overall, the findings demonstrate that this technique produces H_2 at almost similar rates as electrolysis as but with less energy loss than conventional methods making it appealing for possible future applications in energy.

In this last part of the literature review, will consider the subject of hydrogen production through ammonia decomposition in a membrane reactor. This methodology has gained significant attention as it might be the way forward in producing hydrogen, which is a critical element in achieving clean energy. The review provides an overview of various research initiatives that have studied the kinetics, efficiencies and advancements on ammonia decomposition particularly on membrane technologies as an amplifier to such processes. From a synthesis of various literatures, it can be observed that ammonia breakdown presents great prospects for hydrogen production especially when used together with advanced membrane techniques resolved by Kim *et al.,* [20].

Selectively permeable membranes make great separating barriers between hydrogen and other products for enhancing hydrogen yield. The review discusses challenges such as determining reaction parameters that do not bring thermodynamic constraints unnecessarily (meant to minimize energy consumption). As the world's energy transition into sustainability gets underway, new environmental-friendly ways of producing hydrogen keeps emerging, probably making membrane technologies to be an important aspect of increased commodity demand. Accordingly, we recommend that future improvements illustrated here are of immense value to the existing hydrogen production regime while additional sensitivity optimization techniques be explored as a flow on from prior studies.

2. Methodology

It is a creative and hopeful method of obtaining cleaner renewable power through hydrogen production by ammonia decomposition in a membrane. The process revolves around extracting hydrogen from thermal NH_3 decomposition. Heating ammonia past 500°C cause the chemical bonds between its nitrogen and hydrogen atoms to start degrading. The breakdown of ammonia will release H_2 gas, which can be captured for use. A selective membrane that separates hydrogen while retaining nitrogen is the essential component of this system. This isolation is made more efficiently by using this membrane separation technology, because it separates the products of hydrogen decay reaction, induced by other reactions from the desired final product H_2. A more advanced methodology that involves ammonia makes it easier for these scientists to better manipulate the manner and volume of hydrogen produced through the use of suitable temperatures, pressures, types of membranes etc. Therefore, the hydrogen produced can either be employed directly as clean fuel or preserved and converted into other types of energy hence paving way for sustainable electricity generation free from pollutants. To conclude, the use of membranes in moving ammonia decomposition hydrogen production is an extraordinary breakthrough which may facilitate an overall transition to more environmentally friendly energy sources.

3. Results

The research of how hydrogen gas flows through the air can provide essential information about the complicated systems involved in this process by using Computational Fluid Dynamics (CFD). The first point of analysis is by precisely defining walls of inlet air and fluids on which the simulations depend. While injecting hydrogen gas into the field, we keep track of its behaviour: what's happening with flow paths; turbulence structures formed within them; mixing with air. This investigation delivers an in-depth, data-driven insight to the hydrogen gas distribution, velocity profiles and pressure gradients across this system. This research produces results that are linked to a discussion about hydrogen storage, transport and use based on CFD model analysis for developing clean energy solutions. Thus, engineers are able to produce enhanced designs for hydrogen-based technologies using these outputs. As a result, scientists may refer to the detailed analysis mentioned in the article for identifying their models' weaknesses so that necessary improvements can be made henceforth advancing hydrogen gas management and applications. CFD studies have provided us with important data that can be used to address challenges related to this area which has received much attention when it comes to sustainable energy generation.

When it comes to the electrode surface, a model ANSYS represents what goes on deep inside hydrogen gas flowing through porous media using CFD simulation approach. Below are combined results of GE and GAP mixtures based on major aspects related to hydrogen in the incoming model geometry fluid zone away from where air is circulating. By using quantitative results from ANSYS software together with visualization methods research scholars have been able to reveal flow structures, pressure drop distributions and turbulence properties in ventilation systems. Such elaborate analyses are indispensable in developing models for optimizing design and operation perhaps hydrogen powered systems help enhance safety during utilization of such eco-friendly energy sources. Virtual testing via CFD simulation is achieved without having to build costly prototypes since it allows changing design parameters like inlet air velocity level, hydrogen concentration and different geometries. By researchers, the insights of these data will be used to establish how they could help in understanding other places and thus determine the opportunities for growth. Consequently, CFD simulation comprises important ground that enables engineers and researchers who want to take claims of a sustainable energy future beyond their current limits, using hydrogen gas as an alternative fuel. Through this extensive Computational Fluid Dynamics (CFD) research, engineers and scientists contemplating going beyond conventional borders with hydrogen gas have been adequately prepared for their search for sustainable energy sources.

Geographical Reactivity of the Hydrogen Gas which has been carried out via a Computational Fluid Dynamics (CFD) simulation using ANSYS Workbench and observing the behaviour of hydrogen gas in inlet air zones as well liquid zones. Another dimension of complexity in these dynamics becomes visible in this study with regards to the results and discussions presented. Presumably, the simulation also considered how hydrogen gas enters into its system and combines with air and liquids. The latest CFD techniques while measuring various patterns of turbulence structures and their relationship with classical mixing theories within controlled areas. Velocity profiles, pressure distributions and concentration gradients measured quantitatively allowed us to come up with an accurate understanding not only on how exactly did hydrogen lead but also on the reasons behind its behaviour. One of the issues that this article could discuss under its discussion section include whether when used as an inflow into a system inlet hydrogen works well, what will be the implications on general performance as well as safety. This way, it gives space for higher-level conversations that allow one to understand how complete or robust actually is in terms of air quality modelling within ASSM as a prerequisite for more complex scenarios to come. A deep analysis of these findings followed by further discussion would indicate important directions regarding optimization strategies aimed at enhancing system performance, safety allowances to safeguard hydrogen systems and cost-effective technology on the energy supplies derived from this resource in this regard for theorists and practitioners alike.

Creating a dynamic pressure simulation, applicable to a certain hydrogen-based CFD model in ANSYS software's framework, is not only a difficult but also an expertly-performing task that requires careful computer skills to be done well together with in-depth knowledge of physics. It further complicates matters because hydrogen is coupled interactively with ammonia thus changing general dynamics of the flow itself and the pressure patterns. Therefore, it is important that these feedbacks are captured correctly because they assist in predicting events such as; earlier-than expected flow separation (whereby the flow comes off an object), vortex shedding (the vacuums left after air foils) or even structural responses caused by loads. Figure 1 is shown the 2D Ansys Workbench Hydrogen Model.

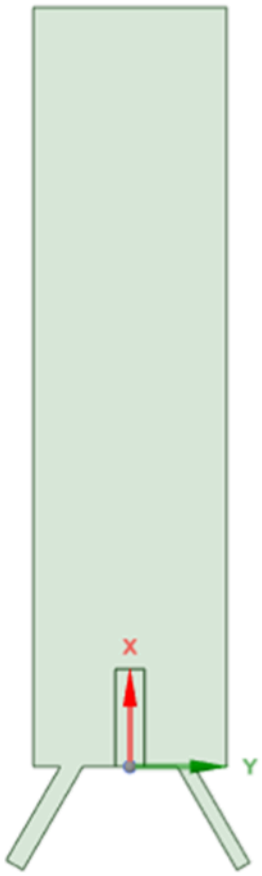

Fig. 1. 2D Ansys workbench hydrogen model

This simulation based on CFD is constructed on the dynamic pressure principle, which means that it refers to the pressure exerted by a liquid during its movement. However, since solvers make many simplifications wanting to approach the reality and initial model representations are often idealized, these regions can end up becoming bottlenecks resulting into non-physical behaviours. Figure 2 is shown Number of Iterations processed based on the simulations. At various speeds, a fluid in a CFD domain flows much in accordance to flow disturbances caused by variations in its density resulting to local high/low pressure regions which can be very important for entire systems macro scale performance/stability.

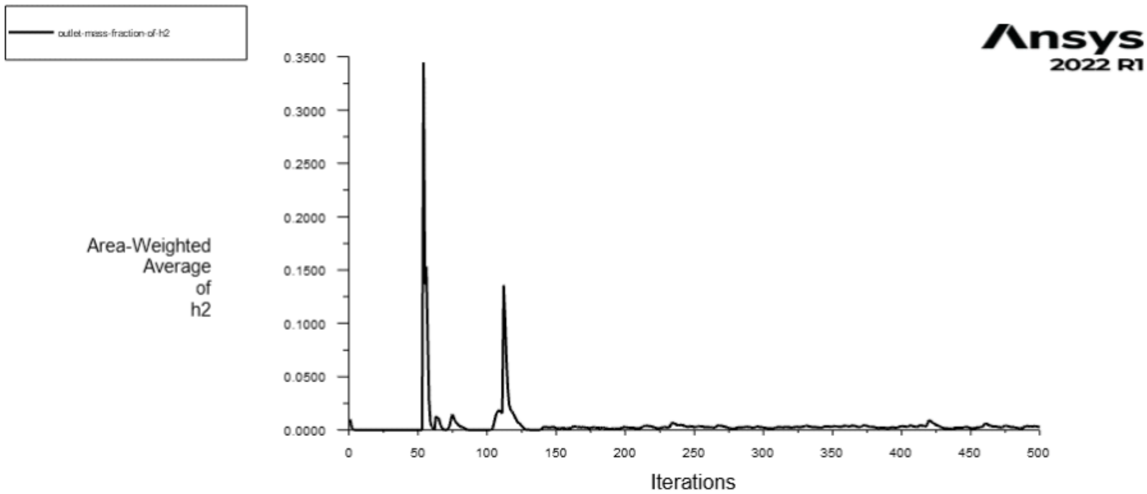

Fig. 2. Number of Iterations processed based on the simulations (Ansys 2022)

Adding ammonia to hydrogen systems results in a plethora of other chemical reactions and phase transitions, creating additional flow complications. It is worth mentioning that dissociation and recombination of ammonia molecules, as well as the development of liquid or solid phases under an ammonia atmosphere could have great impact on its inhomogeneous fluid properties leading to drastic changes in dynamic pressure distribution. This necessitates precise modelling of thermochemical processes and appropriate selection of turbulence models & numerical schemes for CFD simulation. Figure 3 is shown dynamic pressure processed based on hydrogen simulation. In addition to that, the intricate synergy between components of hydrogen and ammonia requires careful consideration for transport phenomena such as diffusion, convection and species interactions which is highly complex. Over time, these local pressure gradients could develop a dynamic pressure field around it, this research work further investigates how system responds under such influences in a dynamic way.

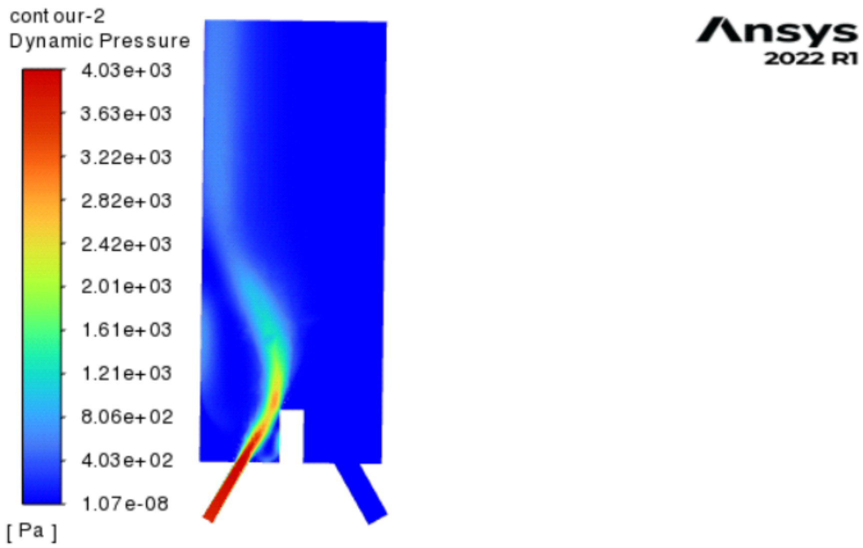

Fig. 3. dynamic pressure processed based on hydrogen simulation (Ansys 2022)

Simulating dynamic pressure using an ammonia-based hydrogen CFD model is a challenging task that needs understanding of Fluid Mechanics, Thermodynamics and Numerical Computational methods in depth. Maintaining dynamic pressure with high precision and predictability is the most essential aspect for designing and optimizing the hydrogen-based energy technologies safely.

Static pressure analysis is one of the key parameters to study the behaviour of these gases in a CFD simulation. The quantity surfaces represent static pressure distribution inside of your simulated domain in ANSYS software and shows why flow dynamics can tell us how likely a pressure-related incident is to occur. Being an element that has the least atomic mass, hydrogen has some special properties with regard to its compressibility and reactivity which have to be rightfully factored into the CFD model. Figure 4 is shown the Static Pressure simulation based on the hydrogen generation path. Ammonia on the other hand, as one of very common industries chemicals has its own physical characteristics that can influence overall pressure distribution.

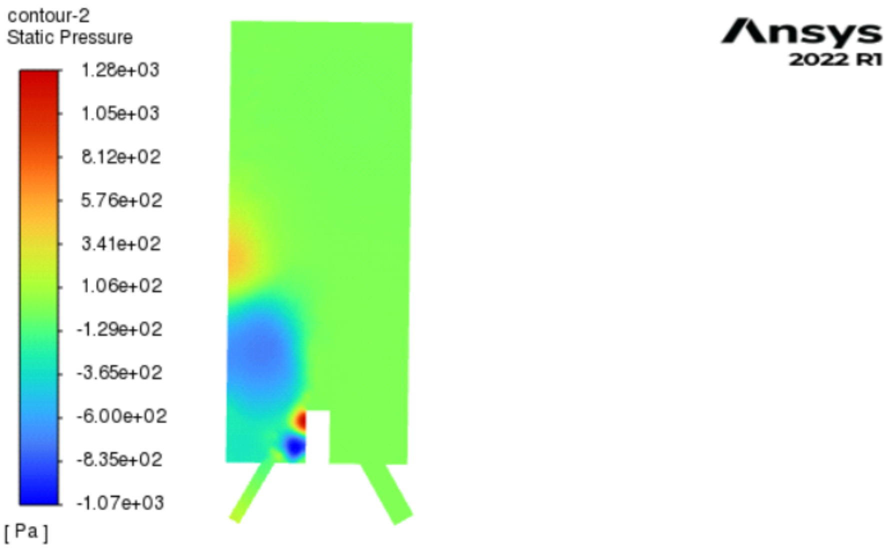

Fig. 4. Static Pressure simulation based on the hydrogen generation path (Ansys 2022)

Thus, simulation for methane and air has been combined in order to clarify this complex interplay between chemical reactions and fluid motion with respect to pressure gradient etc. Static pressure data derived from the CFD analysis can be used to directly optimize pressure variations in an ANSYS environment. This provides an understanding of areas which are prone to hazardous conditions. The further deductions are necessary for plotting important hydrogen-based or ammonia technologies prediction, planning lines for safety, because they require forecasting predictions on their performances. A thorough examination into static pressure during a hydrogen-ammonia CFD simulation gives a solid basis for informed choices about enhancing energy systems or specific applications of these gases in industry due to their versatility in usage.

This paper presents results from a study on Hydrogen Computational Fluid Dynamics (CFD) simulations using ANSYS software package which includes computational kinematics of hydrogen gas flow in a cylindrical tube containing different concentrations of ammonia. A major part of this work includes contour 2 velocity angle which has direct impact on flow behaviour that could be characterized by identifying possible areas where recirculation and stagnation occur through various simulations (contour 2 velocity angle). Figure 5 is shown Velocity angle based on from contour 2 Distribution. In other words, it involves looking at a particular slice through the software's virtual space (Si) along which are plotted all the velocity vectors and their directions.

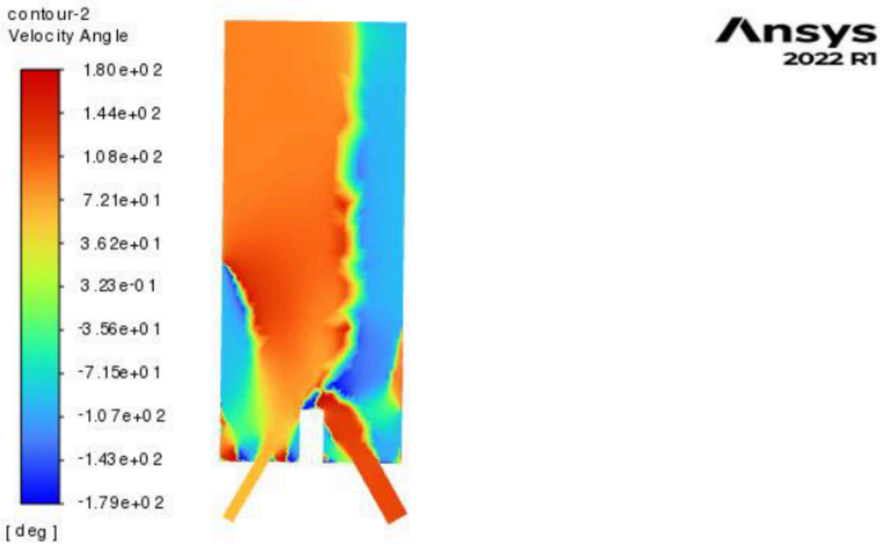

Fig. 5. Velocity angle based on from contour 2 Distribution (Ansys 2022)

With such information, one can start gaining insights into overall flow behaviour and determine if there were any sections experiencing either circulation or stillness. These systems will be designed utilizing this data to make them less egregious in terms of pump performance characteristics. With its capacity for disrupting gas-gas interactions, ammonia poses additional challenges to the realization of gaseous flows and thus its impact should also be taken into account. This is because the ANSYS CFD simulation using contour 2 for calculating velocity angle distribution provides a great opportunity for analysing these kinds of complex fluid dynamics. Therefore, it can serve as a vital tool in a more informed decision-making process which would eventually support the development of innovative hydrogen-based technologies that could form the basis for sustainable energy alternatives in the future.

As a crucial parameter, the velocity magnitude of contour 2 in the hydrogen CFD simulation within ANSYS has to be studied as the hydrogen gas flow behaviour is complex. In this case, hydrogen is combined with ammonia producing a very interesting or rather difficult to simulate multiphase flow situation. Thus, since they provide a color-coded contour showing the velocity magnitude of its components in speed (magnitude), contour 2 velocity magnitude represents how fast these components are able to move about in the simulated environment. The velocity field reflects primarily on hydrogen behaviour, when it comes to moving faster or slower in some places while being stationary in others. You can derive from these velocity contours on such aspects such as mixing dynamics between a mixture of hydrogen and ammonia, points where a reaction could occur (areas of turbulence) like any hot spots or vortices that might be reactive agents as well as general flow patterns affecting overall performance and security. Figure 6 is shown velocity magnitude for generating hydrogen production. Because it offers high-resolution simulations, ANSYS allows for a very detailed investigation of hydrogen, ammonia interaction. Consequently, modification of injection rates and nozzle geometries with specific patterns of flow paths can be done in order to achieve the preferred performance indicators. This understanding is critical in order to advance hydrogen-related energy technologies by comprehending this extremely intricate, multi-phase fluid dynamics problem.

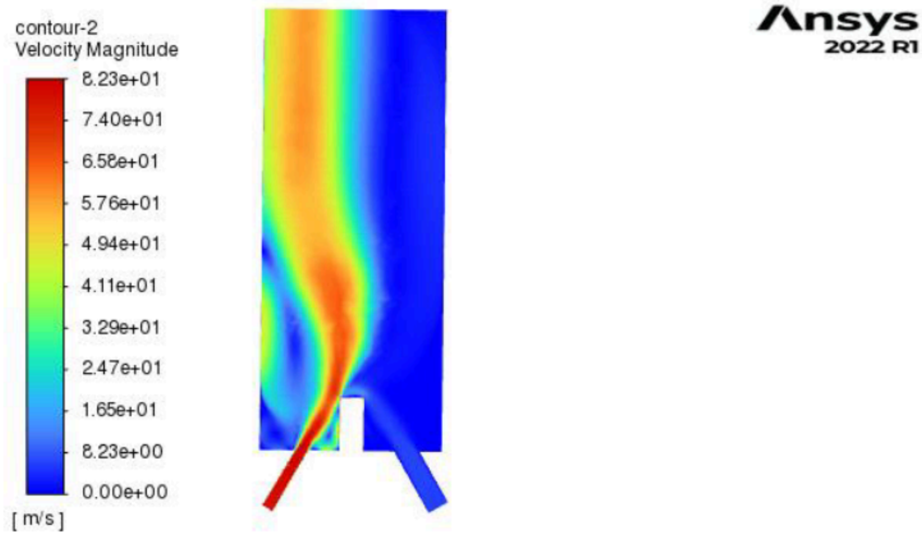

Fig. 6. velocity magnitude for generating hydrogen production (Ansys 2022)

The study of multi-phase flow dynamics has greatly benefited from the development of simulation software, particularly ANSYS, which is one of the best software packages for Computational Fluid Dynamics. This case study examines how water moves through an environment filled with hydrogen while treating ammonia as one among other key constituents; The colour contours display the mass fraction (mass content from the total fluid) in simulated domain that is represented by this figure also involves. Figure 7 is representing the Mass fraction of H_2O for simulation oh hydrogen. Different types of mixing, turbulent bursting height and concentration differences due to different sources (e.g., pollution) can easily be visualized as important patterns and gradients using it because they help us understand if turbulence has any role to play followed by chemical reactions possibly accompanied by phase changes. In the future, making super-efficient and clean hydrogen systems is going to require that we take into account that minimizing the amount of water as a by-product during hydrogen production processes design and combustion dynamics. Such detailed ANSYS simulation allows us to capture many phenomena from hydrogen and ammonia chemistry to coupled/devolved steam mechanisms within one CFD model frame thereby improving our basic knowledge of this crucial energy technology.

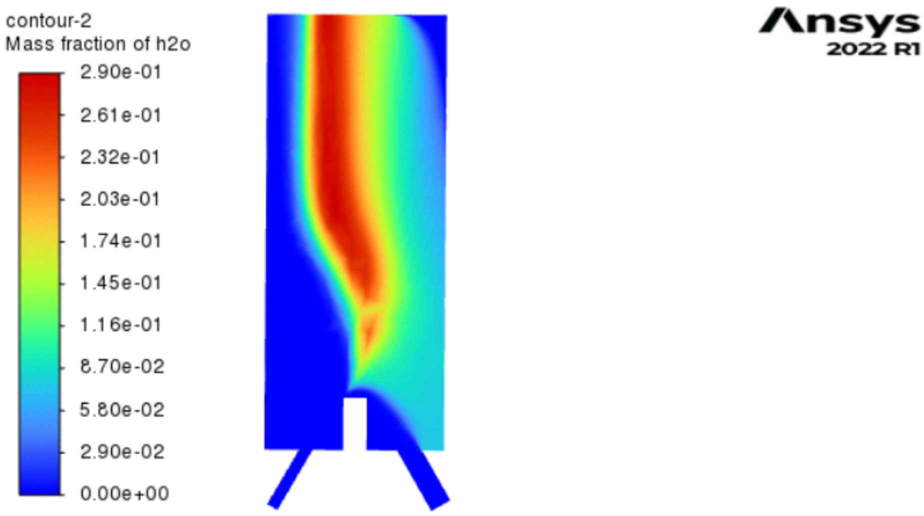

Fig. 7. Mass fraction of H2O for simulation oh hydrogen (Ansys 2022)

In a hydrogen development system, the mass fractions of O_2 obtained from ANSYS CFD simulation involving ammonia is one of the major factors which provide information on the complicated chemical reactions and fluid dynamics which are interconnected. Researchers in this specialized simulation are investigating behaviour of such a hydrogen-based system together with ammonia, a common compound in coal and stockyards. The Contour 2 provides distribution of oxygen molecule (O_2) through the simulated environment, where oxidation occurs. This mass fraction data allows scientists to observe in detail how addition of ammonia affects hydrogen fuel combustion and heat transfer processes. This information is vital for optimizing hydrogen-powered systems as the world phase out less sustainable sources of energy even if it is critical for fuel efficiency and safety. Figure 8 is representing the Mass Fraction of O_2 for simulation on hydrogen. Based on the training through which ANSYS is able to simulate the aforementioned fluid flow patterns and chemical solutions, the purpose of this study is to shed more light on how oxygen composition impacts hydrogen systems design. Engineers can use Computational Fluid Dynamics (CFD) simulations to fine-tune their hydrogen-ammonia system designs by identifying problems early, before they invest in costly physical prototypes. Hence, a simple interpretation will tell you that another crucial aspect for our future use of hydrogen fuel cell as well as vehicles is represented by the mass percent of oxygen.

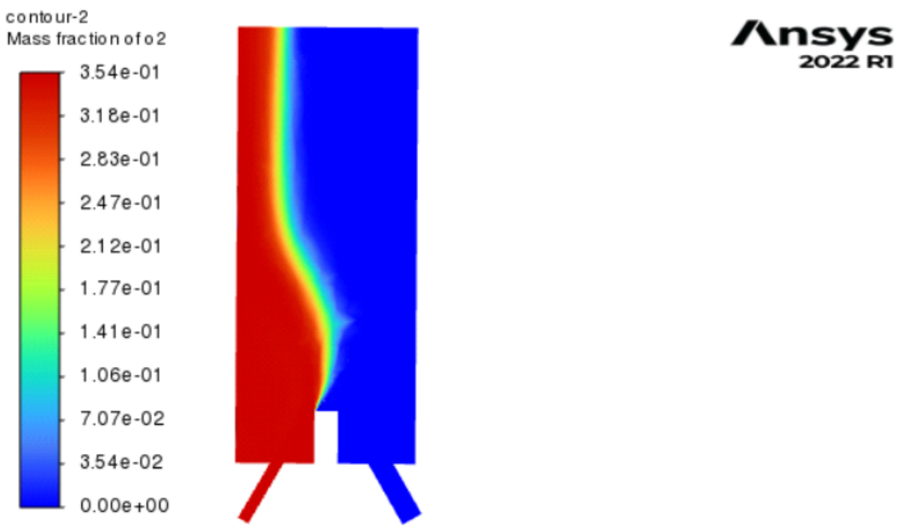

Fig. 8. Mass Fraction of O_2 for simulation on hydrogen (Ansys 2022)

In Computational Fluid Dynamics (CFD), the rotation speed of a hydrogen system has to be taken into consideration as indicated by amazing simulations of ammonia flow. Utilizing the powerful ANSYS software, these researchers are able to delve deep into these complicated five variable systems that correspond to 3D formation illustrated in Figure 9. The liquid velocity either side of central axis direction was called the radial velocity; this means that such heating device is horizontally evaporative or condensing heat pipe working on horizontal thermosiphon designs. However, both hydrogen and ammonia are connected through complex phenomena taking place in process directionally. This information helped us to examine the relationships among pressure, temperature and molecular behaviour which led us to select our system parameters that would produce optimum results. The unrivalled computational strength of ANSYS together with state-of-the-art modelling techniques allow researchers to investigate this hydrogen-ammonia system's complexities at levels not done before leading to unparalleled progress towards sustainable energy solutions and beyond.

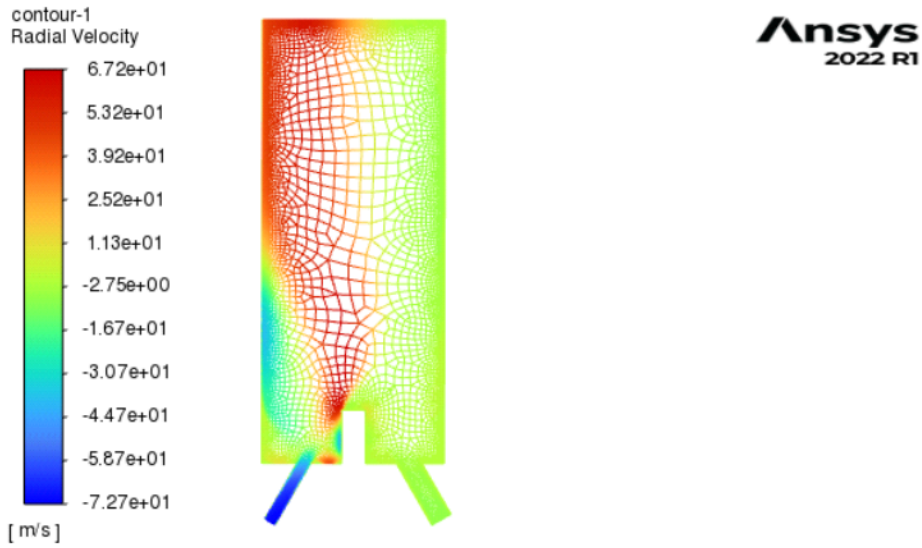

Fig. 9. Mass Fraction of O_2 for simulation on hydrogen (Ansys 2022)

The 1x contour of hydrogen was found to be significant in CFD simulations hence substantial amount of ammonia profiles should be considered as well. A significant role is played by the x-axis, a particular spatial dimension, in observing and interpreting the complex flow patterns of chemical species during simulation. It reveals contours or lines at which points within this coordinate space have the same value that informs on how hydrogen and ammonia interact. This contour is used to display level temperature and velocity profile variations inside the simulated environment so that one can derive a comprehensive understanding about different physical and chemical processes. Figure 10 is shown the Contour grade from X-coordinate of O_2 for simulation on hydrogen. By examining specific peak shape, gradient and any deviations from predicted contour it is possible to obtain knowledge about efficiency of hydrogen interaction with ammonia or if some by-products appear as a result of certain interaction events - it's basically like peeping into machinery of such system. To sum up, investigation of the 1 x-coordinate contour in hydrogen CFD ANSYS simulation together with ammonia serves as an effective way through which performance in terms of safety as well as environmental aspects related to this important energy process are improved.

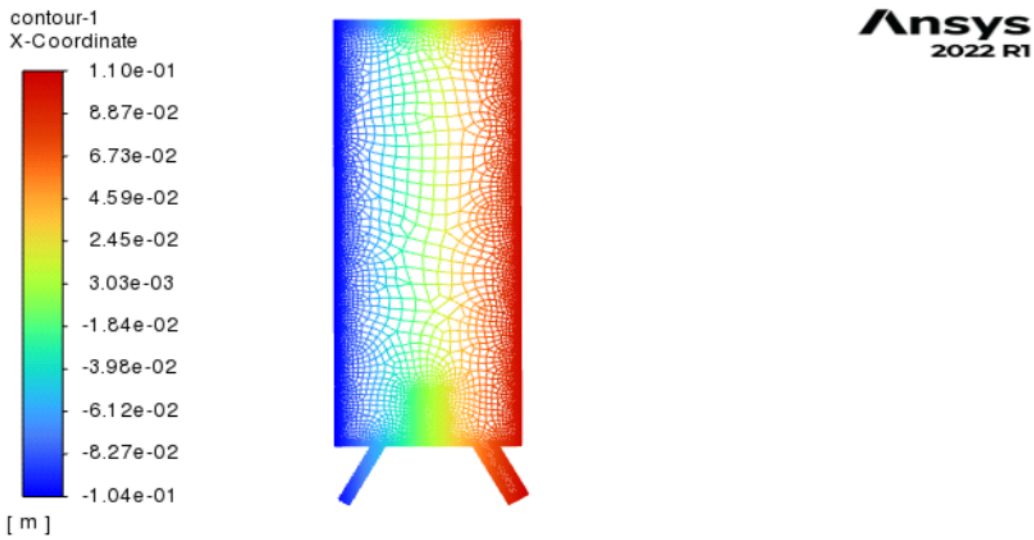

Fig. 10. Contour grade from X-coordinate of O_2 for simulation on hydrogen (Ansys 2022)

Hydrogen (H_2) is commonly chosen In CFD simulation as system while ammonia-fuelled or other critical component distribution & intensity examples are taken as case studies. From the specific case presented, we observe that H_2 mass fraction reaches up to 0.964 (roughly one half of total hydrogen concentration). These wide disparities show that there are various rates of chemical reactions mixings and flows with respect to hydrogen. The contour plot of H_2 mass fraction helps researchers and practicing engineers comprehend better about fluid flow dynamics during these processes, as well as complex thermochemical behaviours occurring therein. Figure 11 is shown Contour -1 Mass fraction of O_2 for simulation on hydrogen. This can be important when performance, safety (e.g., runaway reactions) or unwanted by products at localized regions of high/low H_2 content are concerned. Identification of hydrogen spatial distribution and quantification is a powerful support tool when designing or improving any technology based on this molecule such as fuel cells combustion engines or chemical processing plants. ANSYS simulation allows for gradual multi-scale analysis with such a fine mesh thus designing more efficient hydrogen-powered systems that are also environmentally friendly thereby contributing towards an energy future that is sustainable.

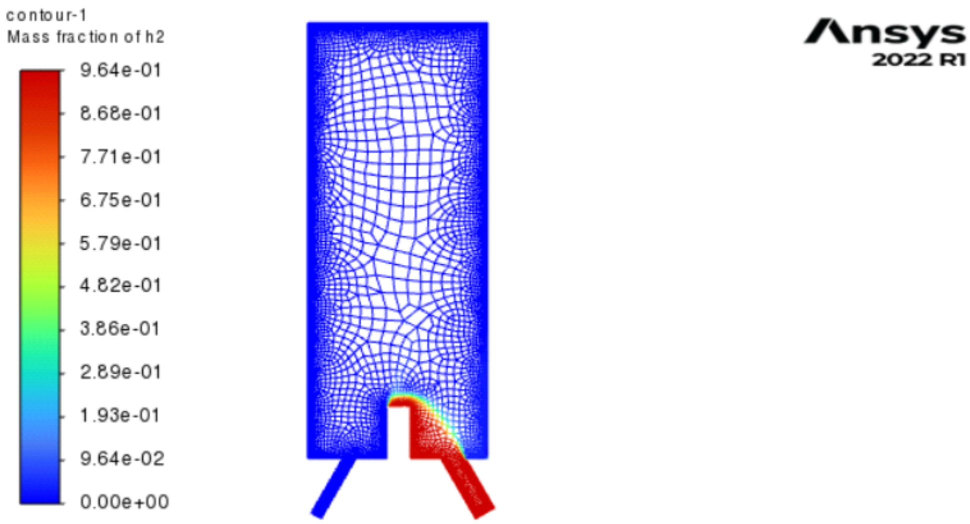

Fig. 11. Contour -1 Mass fraction of O_2 for simulation on hydrogen (Ansys 2022)

In a Computational Fluid Dynamics (CFD) simulation with ANSYS software package, contour 1 denotes velocity magnitude which signifies a key parameter that helps obtain essential insights concerning the dynamics of fluid motion inside systems. The velocity magnitude at contour 1 for this particular simulation results in (8.23, $4.11e^{1}$, $7.40e^{1}$, $8.23e^{-1}$). These scalars indicate how these flows vary from one point on contour 1 to another in their high and low speeds. Figure 12 is representing the Contour -1 Mass fraction of O_2 for simulation on hydrogen.

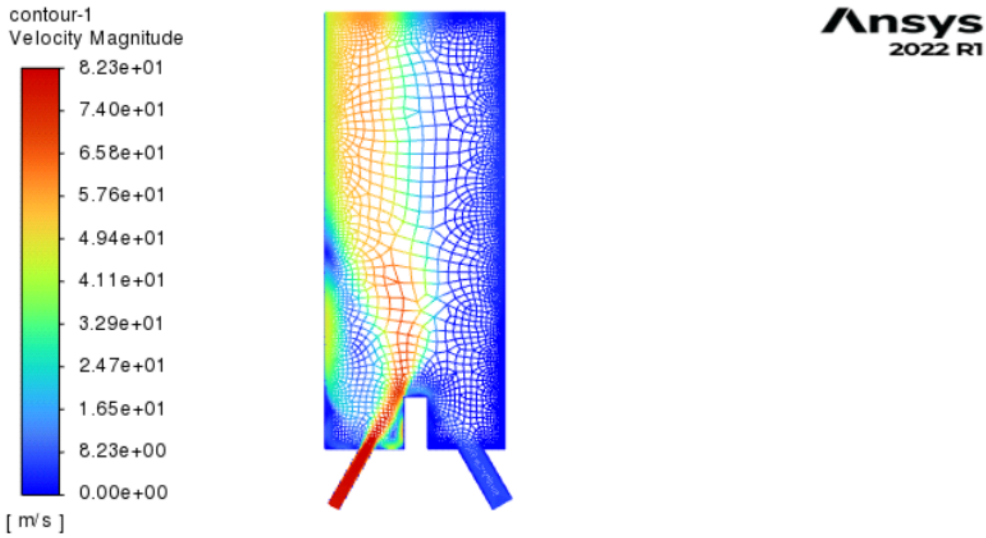

Fig. 12. Contour -1 Mass fraction of O_2 for simulation on hydrogen (Ansys 2022)

An ammonia tonal mixture with a concentration of 8.23 already has a remarkable distinction by CCN standards, thus complicating the flow dynamics further since hydrogen and ammonia can interact resulting into intricate chemical reactions and changing overall flow behaviour at such levels that mainly depend on the rate of mixing. These numbers (10's) associated with $4.11e^1$, $7.40e^1$ and $8.23e^0$ represent a wide range of possible velocities which tells us about a significant variation in the speed each fluid particle moves at different locations inside contour area number one due to model dependence on ammonia actions as well as geometry restraints/boundary/initial conditions within the simulation. The way velocity changes may strongly affect many aspects of system performance and efficiency including heat transfer, mixing and turbulence building in hydrogen-based systems.

Contour 1 velocity magnitude data provide more information about hydraulic flow characteristics but it also gives hints as to how hydrogen containment systems can be designed and operated in a way that prevents accidents. Engineers and researchers working on future designs will find this knowledge useful in improving performance and reliability. So far, detailed computations for fluid dynamics using computers (CFD) have helped with building new systems for generation of energy from alternative sources through controlling complex fluid systems in an efficient way.

This ANSYS model of hydrogen and ammonia fluid dynamics (CFD) simulation represent the contour plots showing contour lines for static pressure in the system. There are three easily visible regions with average static pressures of $–1.07e^3$ Pa and a couple hot spots; $1.28e^3$ Pa, -13kPa. In addition, this swirling contour map reveals some peculiarities related to flow dynamics. When hydrogen and ammonia gases may lose momentum as well as show turbulence, negative pressure regions correspond to flow separation/recirculation zones. On the contrary, positive pressure region maybe represents flow stagnation points or areas with fluid collision, whereby fluid speeds reduce while static pressures increase. Designing hydrogen-ammonia systems requires knowledge on their pressure contours which could also be used to offer better design placement, flow path geometry among other engineering aspects. Figure 13 is shown the Contour-1 Static Pressure Simulation on hydrogen Production. Consequently, through CDF analysis that captures complex differentiable pressure distributions, engineers can have deeper understanding and make hydrogen-based energy systems more reliable in visual terms. This type of huge computational modelling represents an accurate tool to make strides in our understanding of hydrogen and ammonia as alternative clean fuels.

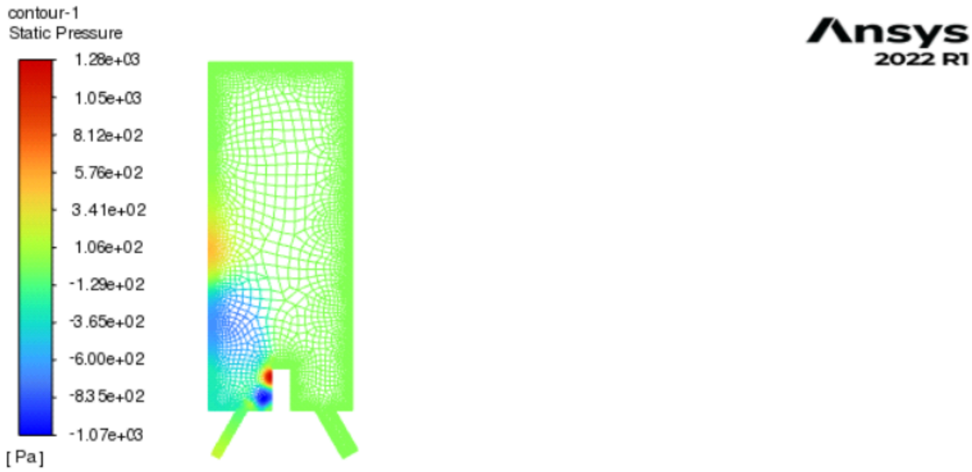

Fig. 13. Contour -1 static pressure simulation on hydrogen production (Ansys 2022)

H_2O, O_2, H_2 and hydrogen are accelerations that should be compared as well as ammonia is a working fluid in ANSYS Software that reports the information. The main goal of this study is to analyse and compare the acceleration characteristics of the above-mentioned substances in the controlled CFD environment. The ANSYS simulation platform enables researchers to model and visualize the dynamic behaviour of water (H_2O), oxygen (O_2) and hydrogen at different speeds. Furthermore, adding ammonia by means of choice of working medium increases the complexity because it possesses a lot of chemical properties impacting on its thermodynamic cycles. First time ever these CFD simulations provide an extensive insight into such contrasting substances and show how they differ in terms of their relative capacity for accelerating themselves. Figure 14 is shown the Acceleration vs input gas Simulations on hydrogen Production.

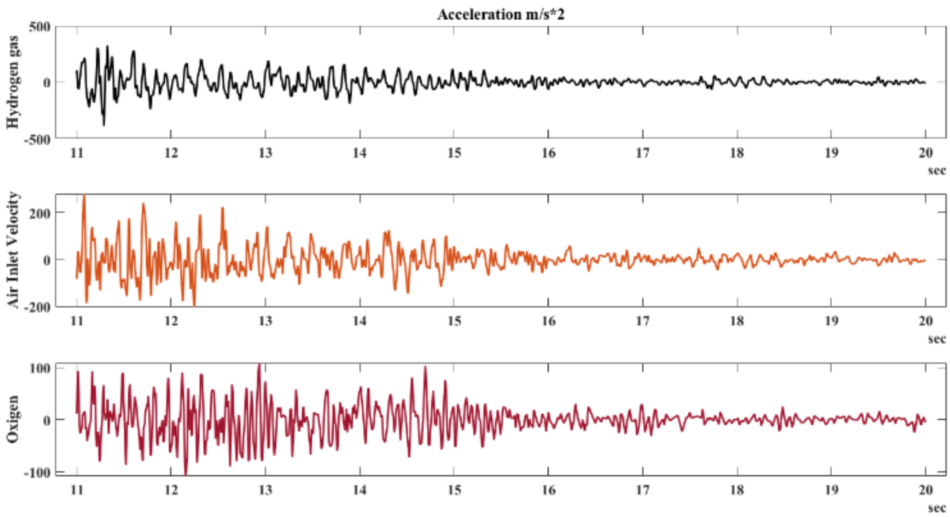

Fig. 14. Acceleration vs input gas simulations on hydrogen production

An important source of information is contained in the accurate models and acceleration profiles comparisons with different materials which aid in the improvement of performance hence broadening the understanding. Thus, for CFD modellers and simulations it is now beyond 2 times of real-time compared to water, O_2 as hydrogen evolution reaction (HER) or ammonia hypothesized reversible mixture which distinguishes an important frontier both in improved technology and science capabilities.

The simulation outputs show complex relationships between forces and temperature that exist in hydrogen-ammonia system. The changes on radial stress range from -4.31 MPa to 6.69 Mpa which represents a huge amount of compressive strain at the same time large amount of tensile stress for different parts of this system. However, hoop stress undergoes extensive variations ranging from -24.81 MPa to 41.6 MPa (maximum), illustrating dynamic nature of circumferential forces during ingesting process. The von misses stress, an overall distortion energy measure has its peak value at 42.28 MPa being concentrated in regions with loaded material thus indicating possible structural deformation problems. It can be inferred that the simulation is capable of estimating extremely high temperatures, the likes of which can hardly be conceived, up to around 114.04°C or above 1000 K. Hydrogen gas shares a simulated volume with ammonia vapours that are released from ten areas on thawed ocean as such complex fluid dynamics blend and interplay, evolving at different rates. To harness the tremendous potential held by these new forms of fuel in energy systems mostly powered either fully or partially by hydrogen technology as long as there is need to know about controlling this rather more complex one distinct in many layers of physical processes following properly crafted strategies for instance cost effective materials. Figure 15 is representing the Stress Comparison results for 4s of hydrogen Production.

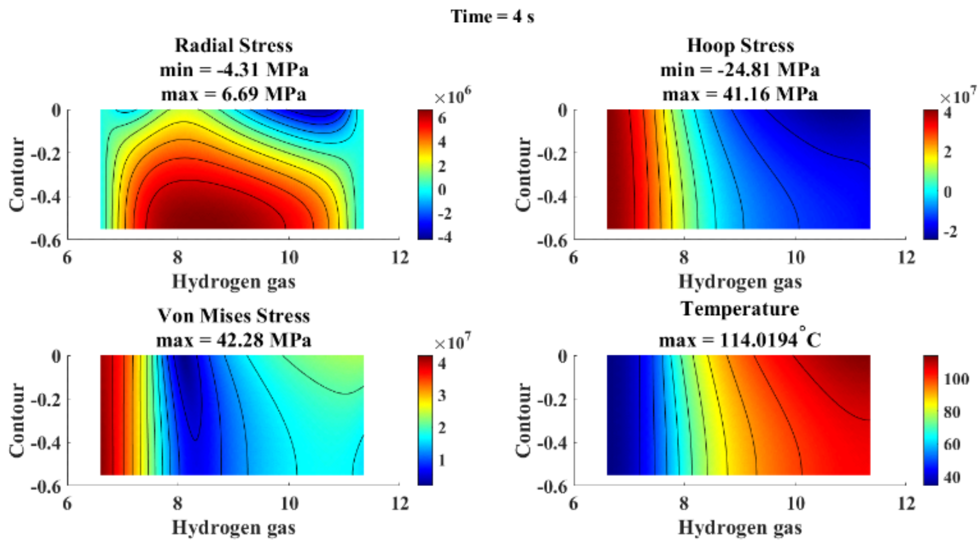

Fig. 15. Stress comparison results for 4s of hydrogen production

Ammonia was simulated using ANSYS Computational Fluid Dynamics (CFD). The most notable radial stresses revealed by these largest pull and push forces were -13.22 MPa and 9.71 Mpa, respectively. However, the hoop stress depicts even wildest fluctuations ranging between -29.86 MPa and 36.69 MP. Such extensive hoop stress ranges indicate that ammonia passing through the material is subject to considerable bending and twisting moments. The von Mises value undoubtedly ranks first among all other stress measures; it amounts to a maximum of 37.84 MPa- essentially serving as reference point in relation to maximum material strength and possibility of its failure due to various reasons mainly related to manufacturing defects or occurrence of excessively brittle nature. Figure16 is represent the Stress Comparison results for 2.5s of hydrogen Production. This predicted maximum temperature equals about 103.86^{O}C, therefore it will compound mechanical stresses with those brought about by heat build-up caused by the simulation above-mentioned earlier. In this way, materials selection and thermal management and efficiency of an ammonia-based system are all influenced by this increase in temperature which can affect its safety. Consequently, it is through this ANSYS CFD analysis that engineers designing and characterizing the demanding fluid dynamics that are encountered in applications powered by ammonia will benefit extensively; hence this facilitates optimization of designs for superior performance and reliability.

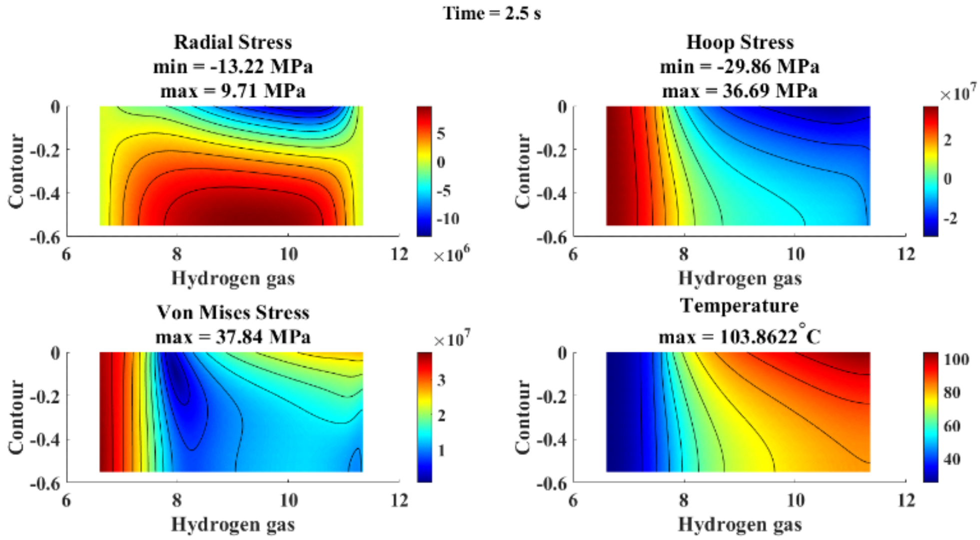

Fig. 16. Stress comparison results for 2.5s of hydrogen production

This topic description refers to temperature and pressure stress analysis at system or structural levels. The so-called radial stress, which is an indicator of the tensile or compressive forces acting perpendicularly on the surface has minima (-24.20 MPa) and maxima (11.93 MPa). The fact that there are irregularities with regard to this radial stress suggests that a state of loading may be dynamic, where the material undergoes both compression and stretching probably due to external loads applied or variation in internal pressure. Figure 17 is shown the Stress Comparison results for 0.5s of hydrogen Production.

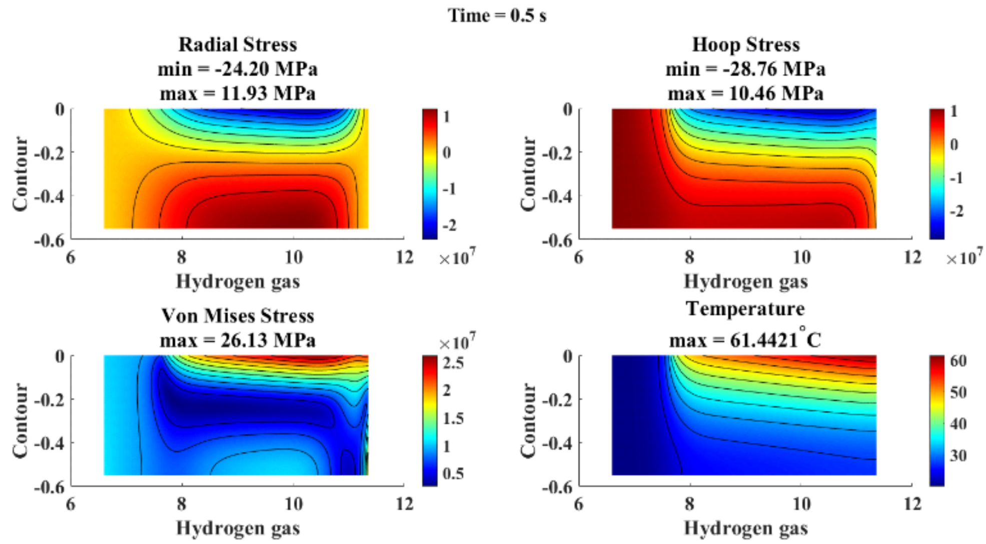

Fig. 17. Stress comparison results for 0.5s of hydrogen production

Just like in radial stresses, there is opposite behaviour in hoop stresses that range from -28.76 MPa (minimum) to 10·46 MPa (maximum) that represent cyclic largest circumferential tensile forces applied on circular cross sections of such kinds of systems. Our product lines include cylindrical structures or sometimes spherical ones and it is these hoop stresses which might be more pronounced only in terms of pressure state plans rather than the unit axials ones. It means that the system experiences both compressive and tensile forces tangentially, possibly due to bending or torsion or other loading conditions that are complicated.

In the proposed model, detailed analysis indicates that maximum von Mises stress is 26.13 MPa. Since tire responses have more than one principal residual stress, von Mises is an effective measure characterized by yield in terms of three-dimensionality of a state and an infinite series of planes in space, the shown maximum stress remains up till so often. Figure 4 shows that high von misses stress value (probably greatest) implies significant distortions or yielding of material leading to failure.

As for this case description, there is a subject having maximum temperature of 61.44°C that has been added. Such as external heating from the system itself (ohmic losses), other heat sources outside the circuit and even some chemical reactions occurring within it cause such high values. On account of temperature variations; material properties changes occur leading to thermal expansions thus compromising systems' performance and reliability.

In mass fraction versus chart graph, hydrogen inlet, air inlet fluid zone and peak variation graphs of a complex engineering or scientific analysis these components are more significant. Mass fraction, for example, represents the composition or distribution of different parts in a mixture; usually shown as a graph. Additionally, it provides an excellent means for tracking and enhancing processes that involve measuring the entrance of chemicals like hydrogen gas into certain zones through some valve. One key area of interest is the air inlet fluid region which is one of the most dynamic and interacting parts that can strongly influence system performance. The peak variation graph indicates how variations in a measurable property can be designed and also offer important insights into system steadiness and reliability. Figure 18 is shown the Mass Fraction results for O_2 of hydrogen Production. Comprehensive insights on system design plus operations will also require further interrogation parents this intricate interaction between flows rates at inlets, mass fractions, fluid domains and their peak variations.

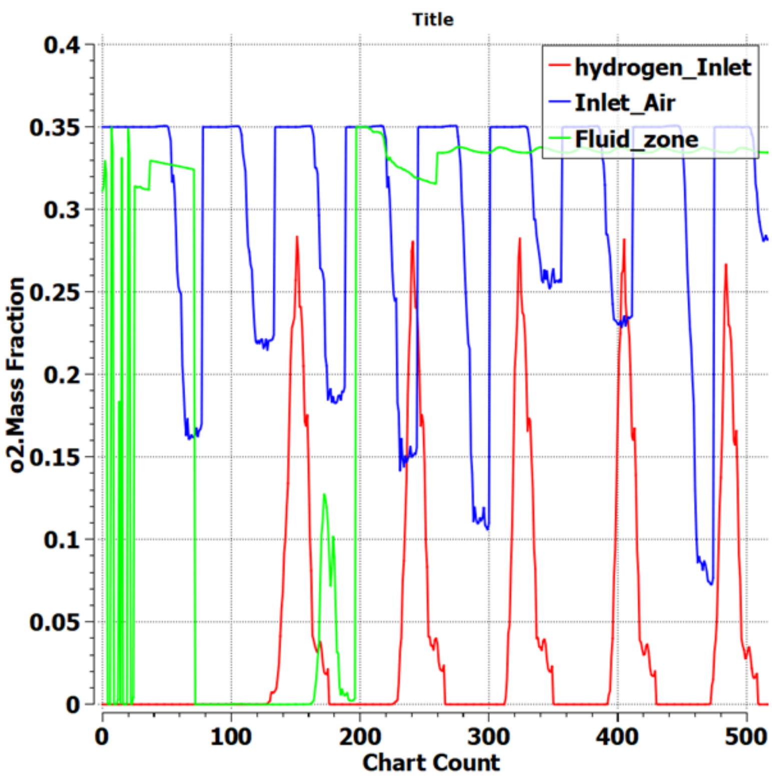

Fig. 18. Mass fraction results for O_2 of hydrogen production

Composition fraction and its graph arrangement will be mentioned as the most fundamental factor within fluid or thermo-dynamics. Here mass fraction subjected the percentage concentration of water vapour in a mixture that may be significant for selected attributes. Understanding that covariate properly is essentials to all fields where moisture content allows forecasting climate changes accurately, processing records to see which processes are going well or evaluate effects on environment of pollutants chemistries become more appropriate. Introduction of hydrogen is an importance part in systems where this element presents a fuel source or reacting agent. Connotatively this inlet permits hydrogen gas stream to flow into the same zone within system where optimal mixing and reaction rate can be attained. For ensuring correct stoichiometric ratios for different reactions which are usually involved in using fuel cells and combustion engines proper composition and positioning of hydrogen inlet is necessary. Figure 19 is shown the Mass Fraction results for H_2O of hydrogen Production. Controlling the hydrogen inlet properly makes it a crucial point in providing efficient operation, sustainable energy technologies development through improving systems with low emissions and achieving maximum performance recyclability.

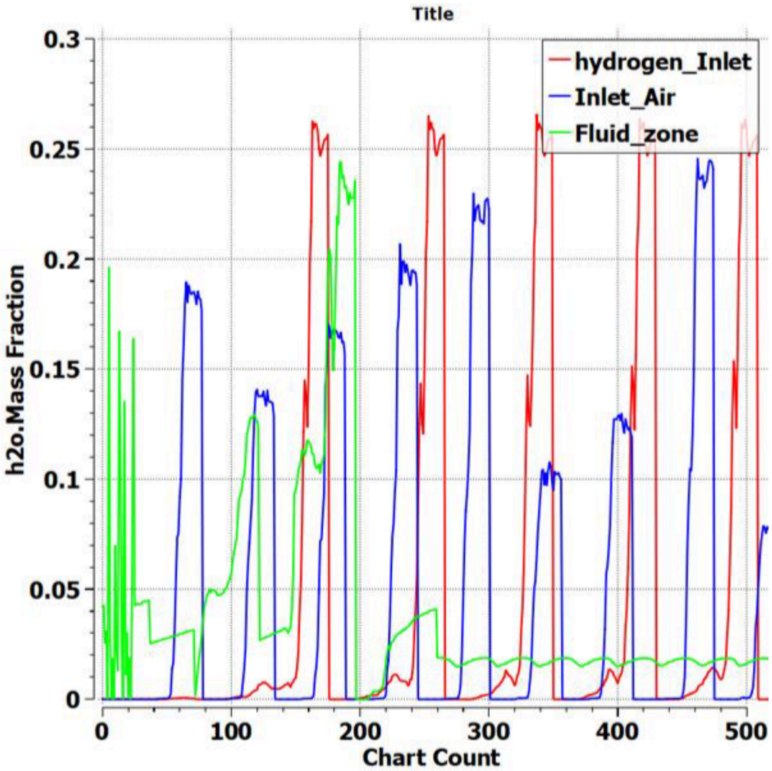

Fig. 19. Mass fraction results for h_2o of hydrogen production

The region where air enters and leaves fluid layers is another significant part of fluid mixtures, but it does not perform as well. It is a zone for introducing in to system with regard to mostly, temperature, pressure velocity etc. so much affect every other component that make fluid mix. An air inlet design should therefore consider turbulence occurrence therein and flow patterns besides determining flow transformations that could influence processes like combustion or heat exchange. Knowledge on these characteristics at air inlet fluid region is vital for performance maximization and ensuring safe operation across various industrial applications.

This difference between x-direction and the chart captures how system works. The chart visualizes data and describes relationships between the hydrogen inlets, air inlet fluid zone. However, the most important part and that which is totally responsible for introducing hydrogen into the specified region is simply known as a hydrogen inlet. This air inlet also plays an equally important role because it allows you to enter the system when combined with hydrogen bed at a fluid zone which influence overall fluid dynamics and efficiency of reactions. Figure 20 is shown the X direction of Axis Input flow process for hydrogen Production.

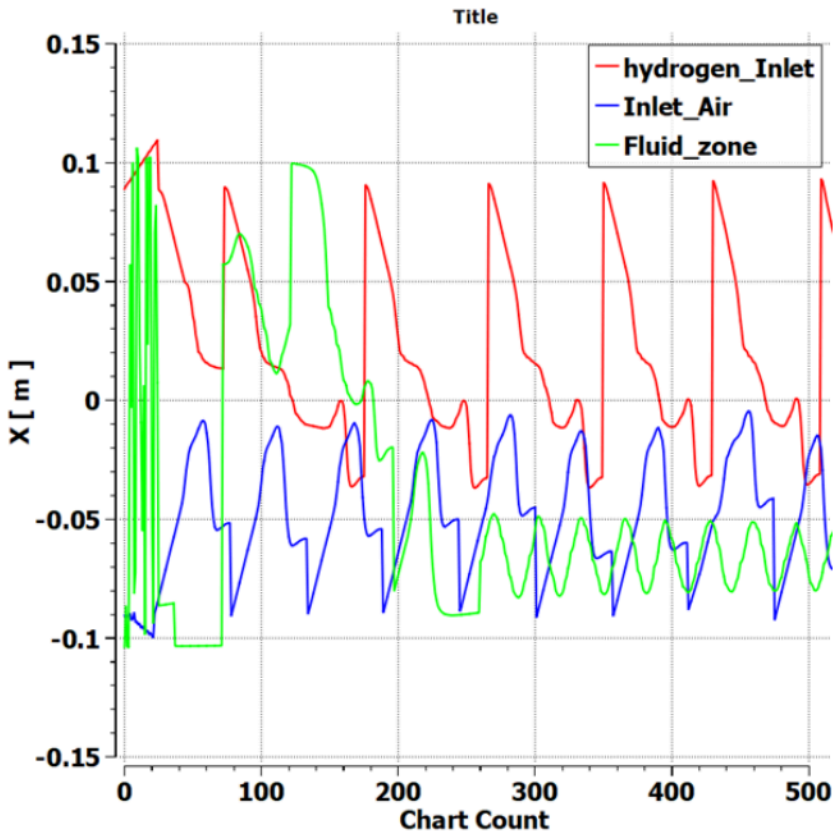

Fig. 20. X direction of axis input flow process for hydrogen production

Hydrogen inlet x-direction and Air inlet are interacting each other to achieve their best performance optimization. The x-direction offers a structure for understanding how such components behave in the vicinity while a graph allows visualization and quantification. Moreover, closer examination of these elements will reveal some insights on system operation, possibly resulting in improved design and performance.

In fluid dynamics, particularly hydrogen systems that require high pressure and air inputs graphical representations highlight how different processes performance and efficiency depend on fluid velocity. Such plots show how speed changes affect behaviour of fluids flowing into new regions (e.g., hydrogen and air), which has a huge influence on combustion rates as stated before.

The overall dynamics of system are more connected to the fluid mechanics in that hydrogen inlet and air inlets are important for determining where this originates from. They have been designed & configured such that they can improve flow characteristics and deliver the required velocity needed during operation. By looking at the turbulent zones, pressure drop zones and mixing efficiency zones within which dab inlet's function, an engineer can understand what might go wrong with a system's performance. Figure 21 is shown the Velocity results of fluid process for hydrogen Production.

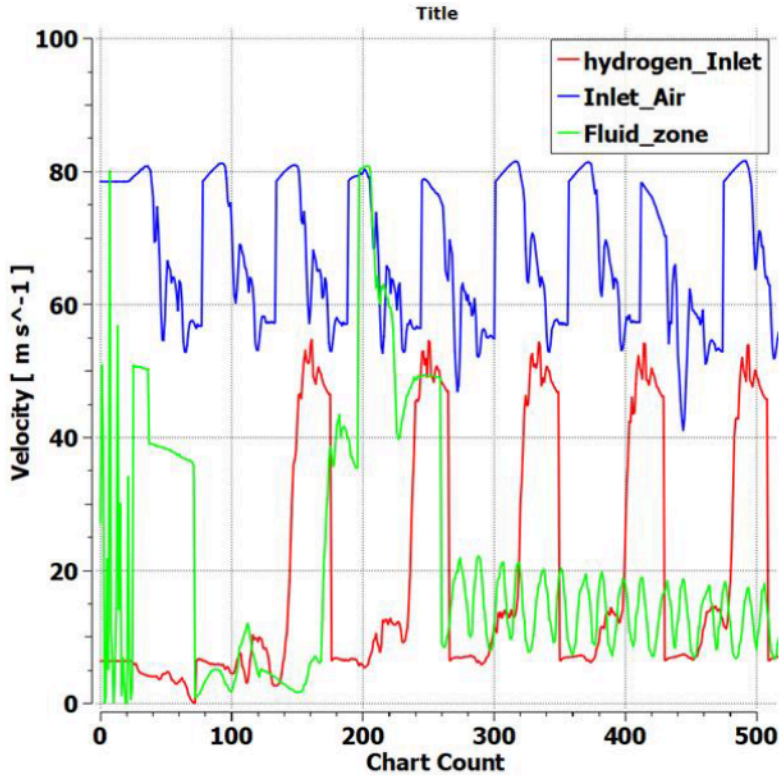

Fig. 21. Velocity results of fluid process for hydrogen production

In this case speed and different ducts work in tandem to ensure hydrogen-air systems are functioning optimally. That section of flow where they are contained is regulated so as to have laminar flow and velocity profiles that suit application purposes. Thus, using these modern modelling and simulation tools, engineers can predict fluid motion at various velocities before the design is constructed which means more efficient designs starting with higher performance.

Modelling the mixing of hydrogen particles can be severely impacted by temperature in Computational Fluid Dynamics (CFD) simulations: basic parameters play an important role in analysis and behaviour prediction process inside hydrogen-based systems. In fact, as it is a simple and lightest element, hydrogen reacts with other materials very easily and also diffuses more quickly at relatively higher temperatures. The distribution of temperature among hydrogen particles in turn becomes one of the most crucial parameters considered for numerical simulations with CFD designed to predict the effectiveness and safety of hydrogen powered applications such as fuel cells, internal combustion engines (ICE) or hydrogen storage systems. Hence there is no doubt that higher temperatures will cause the hydrogen particles' kinetic energy to increase resulting in greater degree of randomness among them i.e. Figure 22 is shown the Temperature effects for fluid process for hydrogen Production.

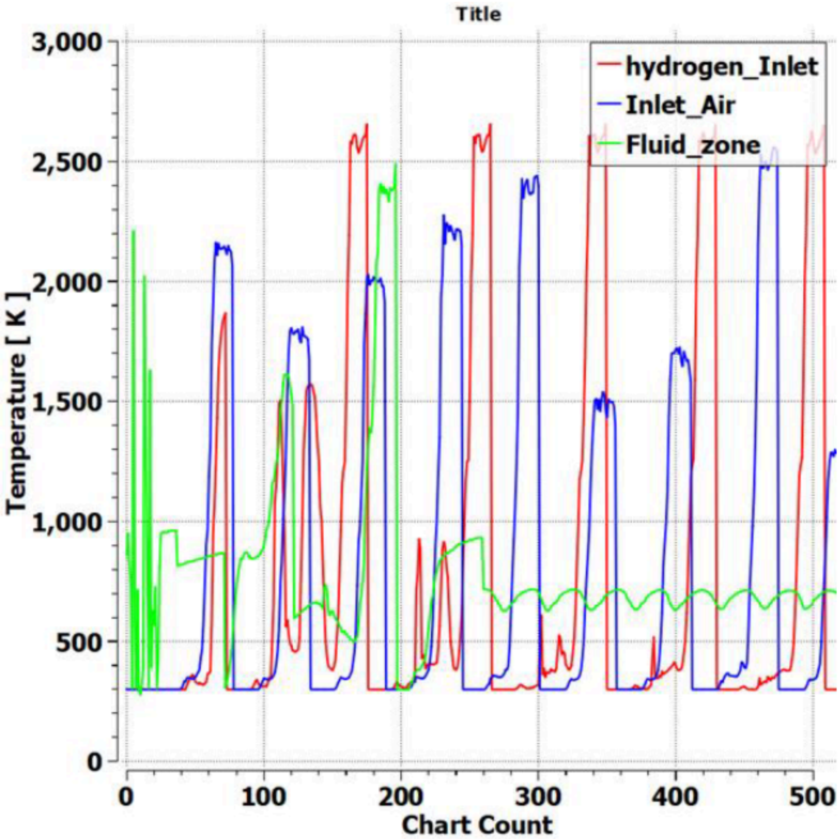

Fig. 22. Temperature effects for fluid process for hydrogen production

Such high molecular movements enable quicker diffusion and more homogenous mixing which can entirely change how a system works. In a hydrogen fuel cell, for instance, molecular hydrogen must mix properly with oxygen to initiate the electrochemical reactions responsible for producing electricity. However, on the other hand, temperature induced mixing in hydrogen storage tanks may result to pressure build-up risking safety. For engineering practitioners an exciting avenue has been brought about through the use of temperature-dependent hydrogen-particle behaviour simulations in CFD that elaborate upon this delicate dualism between thermal consequences and transport of hydrogen; hence shifting their attention to safer and better-performing systems developed on this promising energy vector.

This is significant in fluid dynamics because it governs the correlation between a certain rate of energy dissipation and its related chart, especially when introducing hydrogen into air systems. It is an important factor in determining how efficient different processes are when it comes to loss of energy from fluid systems. Thus, by looking at the chart that captured these conditions, we can track what addition of hydrogen and air contributes to overall energy dynamics within specified boundaries of such type hydrodynamic behaviour. Figure 23 is shown the Specific Dissipation rate effects for fluid process for hydrogen Production.

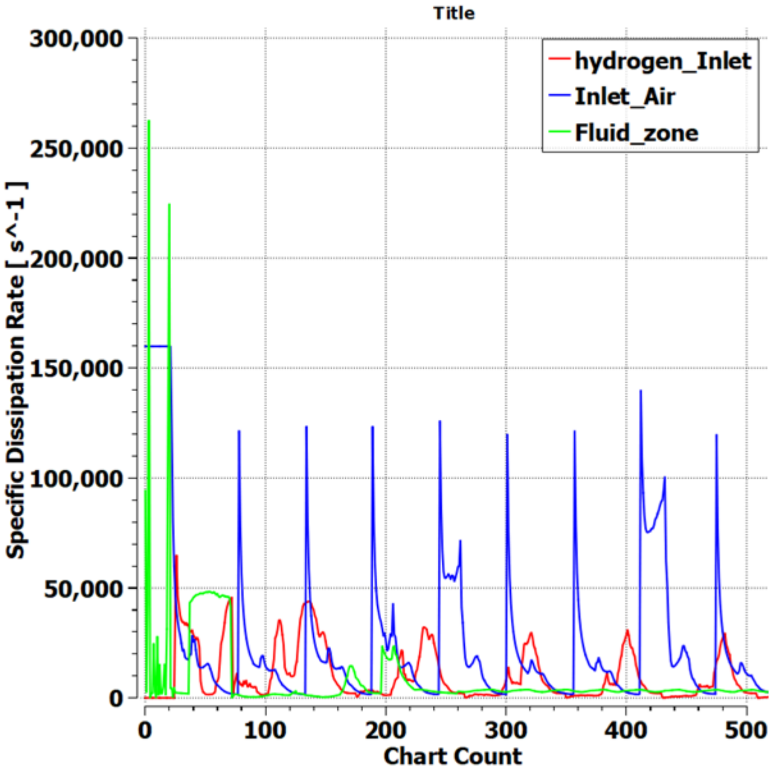

Fig. 23. Specific dissipation rate effects for fluid process for hydrogen production

The hydrogen inlet and an air inlet are important for defining the interactions among the composition species in fluid zones. For instance, a lighter gas is introduced through hydrogen inlet that may alter densities as well as viscosities of some types of liquids by several orders of magnitude whereas your air inlet gives general zonal aeration as well to ensure mixing. These two inlets can have mutual interactions which may lead to varying specific dissipation rates, with the specific details depicted by the above picture that gives very good insight into how these various operational features interact.

Also, it is fundamental for processes such as hydrogen-air fluidic blending this rate of dissipation since it will be represented now in per cent of chart information. This chart has data on various inlet designs so that engineers and scientists can explore the theory and find optimal conditions with a view of their targeting an energy dissipation but at low loss levels. It will enhance even further system performance and lead to better strategies for manipulating fluid dynamics across multiple industrial applications.

In many engineering and scientific applications, pressure can be represented by charts which can also lead to map developments. Charts are ideal for the pedagogical representation of pressure variation with respect to other parameters making them useful in their analysis. These diagrams may reveal the behaviour of gaseous and liquid substances under various conditions hence enhancing the overall performance of the system while at the same time allowing for monitoring.

hydrogen inlets need to know about their properties and circumstances at its entry into a system. Therefore, hydrogen inlets should take into account pressure differences, flow rates and as well as appropriateness features in order to avoid leakage or hazards. Well-designed hydrogen inlets are able to supply this gas productively and securely leading to its diverse applications which include fuel cells or industrial purposes. Figure 24 is shown the Pressure effects for fluid process for hydrogen Production.

In the same way, in most systems which use drawing air for combustion or other processes, the air inlet fluid zone is critical. And this area had to be carefully designed optimizing the air flow with least amount of turbulence and proper mixing with rest of fluid such as fuel or hydrogen. Engineers can enhance engine performance and HVAC systems through close scrutiny of airflow in an air inlet zone together with other technologies that require that processes are kept under tight control.

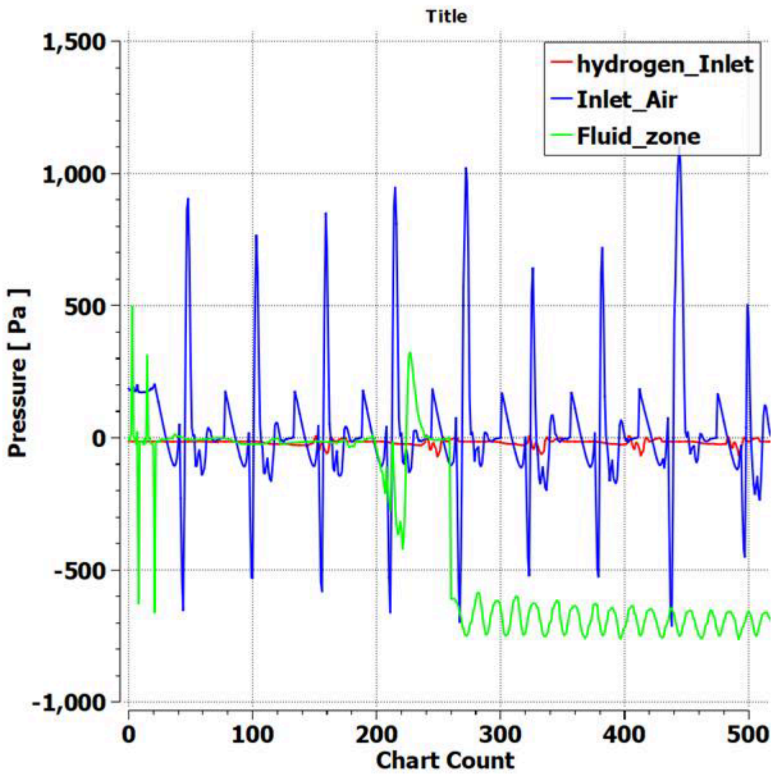

Fig. 24. Pressure effects for fluid process for hydrogen production

4. Conclusions

4.1 Dynamic Pressure Verification

Conclusion The CFD simulation results of hydrogen in ANSYS with ammonia gives an understanding on performance of these substances under different conditions. The analysis revealed a lot many things about hydrogen-ammonia relationship including dynamic pressure oscillations. This is important for further developments or utilization of hydrogen-based systems, particularly in ammonia related systems, since they can either act as transporters or participants.

In summary, this CFD simulation study has brought to light the fact that dynamic pressure should be one of the parameters considered when determining the performance of hydrogen-ammonia mixtures. It has been observed that dynamic pressure varies significantly with change in flow conditions, temperature and concentration of reactants among others. Consequently, engineers and researchers need to consider this variability in their design processes involving hydrogen and ammonia systems due to its potential impact on overall system efficiencies as well as safety issues associated with them. Future directions in research and development within areas such as these fields could thus use this as a basis.

The evaluation of dynamic pressure in CFD simulations of hydrogen using ANSYS, especially in contrast to ammonia, gives an overview of what is happening from the Aerodynamics perspective. This research not only widens the theoretical base about hydrogen and ammonia interaction but also indirectly affects the design and optimization of power systems. With growing demand for renewable energy options, these types of simulations can help hasten advanced technologies that will reveal the potential benefits of hydrogen and ammonia as clean energy shuttles. Beyond the stress variability, within interactive gas dynamics problems with a view towards production of eco-friendly fuels as indicated by Campbell *et al.,* [21], one remarkable aspect that is less understood is how Radial velocity influences combustion. Furthermore, little is known about how radial velocity affects combustion process especially when hydrogen and methane interaction is considered.

The analysis of radial velocity in a CFD simulation (using ANSYS) about hydrogen and ammonia interactions also shows interesting views on the stress distribution, thermal characteristics etc. The radial stress values measured vary from a minimum of -4.31 MPa to maximum 6.69 MPa suggesting strong variations in the internal forces acting on the fluid. On the other hand, hoop stress is found in the range of values from -24.81 MPa to 41.6 MPa again showing that fluid dynamics considerably contributes complex state of stresses within real vessel design condition under simulation environment and calculation basis. Finally, it’s noted that the von Mises stress (a global criterion for yielding of material) has a maximum value around 42.28 MPa which can be used to identify failure areas under specific operational conditions.

The maximum temperature inside simulation results concluded to be 114.04^{O}C and thus, temperature profile variation is important in order to achieve a definite w/h. For example, higher temperatures (p) have significant impact on fluid behaviour react differently and change their phase at much easier rate. The mechanical & thermal stresses induced by this radial-hoop straining and thermal loading illustrate the complete view of thermal intrusion. Such findings are quite fundamental in hydrogen and ammonia-based systems’ future design prophecies particularly those for safe operation with optimal efficiency.

Moreover, the complete simulation encompasses mass fraction distributions depicted in graphs. The graph indicates how hydrogen/ammonia occurs within fluid zones as air and H_2 inlets aim at introduction purpose only. The mass fraction is significant because it allows us to infer about what happens to a specific mixture while also telling us whether it could be effectively utilized in reactions or processes that may take place inside the system. As a result, these ANSYS simulation outcomes will guide engineers along with researchers on hydrogen and ammonia applications generally for both power generation systems as well as chemical processes.

Acknowledgement

This research was funded by a grant from Ministry of Higher Education of Malaysia (FRGS Grant R.J130000.7824.4X172).

References

[1] Dolan, Michael D. "CFD modelling of a membrane reactor for hydrogen production from ammonia." In *IOP Conference Series: Materials Science and Engineering*, vol. 297, no. 1, p. 012027. IOP Publishing, 2018. https://doi.org/10.1088/1757-899X/297/1/012027

[2] Hayakawa, Yukio, Shinji Kambara and Tomonori Miura. "Hydrogen production from ammonia by the plasma membrane reactor." *International Journal of Hydrogen Energy* 45, no. 56 (2020): 32082-32088. https://doi.org/10.1016/j.ijhydene.2020.08.178

[3] Xia, Qi, Zihan Lin, Congwei Wang, Zhouxin Pan, Weiya Jin, Chen Chen and Qibin Liu. "Solar-driven multichannel membrane reactor for hydrogen production from ammonia decomposition." *Fuel* 356 (2024): 129591. https://doi.org/10.1016/j.fuel.2023.129591

[4] Hayakawa, Yukio, Tomonori Miura, Kota Shizuya, Shintaro Wakazono, Kenya Tokunaga and Shinji Kambara. "Hydrogen production system combined with a catalytic reactor and a plasma membrane reactor from ammonia." *International Journal of Hydrogen Energy* 44, no. 20 (2019): 9987-9993. https://doi.org/10.1016/j.ijhydene.2018.12.141

[5] Fauzi, Farahanie, Abdul Rahman Mohd Kasim, Siti Farah Haryatie Mohd Kanafiah, Syazwani Mohd Zokri, Adeosun Adeshina Taofeeq and Siti Hanani Mat Yasin. "Advancing Heat Transfer: Exploring Nanofluids and Regression

analysis on Lower Stagnation Point of a Horizontal Circular Cylinder for Brinkman-Viscoelastic fluid." *Semarak Engineering Journal* 7, no. 1 (2024): 1-10.
[6] Arafat, Mohammad, Izuan Amin Ishak, Muhammad Aidil Safwan Abdul Aziz andrew Wee Shong Soh, Woei Ting Tiong, Nur Rasyidah Roziman, Nur Amiza Mohd Hairul *et al.,* "A Hybrid RANS/LES Model for Predicting the Aerodynamics of Small City Vehicles." *Journal of Advanced Research in Experimental Fluid Mechanics and Heat Transfer* 17, no. 1 (2024): 1-13. https://doi.org/10.37934/arefmht.17.1.113
[7] Carpio, Ian Lev Arriaga, Jasuo Sayán, Julio Ronceros, Mirko Klusmann, Carlos Raymundo, Wilder Namay and Gustavo Ronceros. "Study of Internal Flow with Deceleration in a Cryogenic Chamber Composed of Jet-Type Atomizers." *CFD Letters* 17, no. 6 (2025): 57-68.
[8] Shaikh, Javed Shoukat, Shamitha Shetty, Khizar Ahmed Pathan, Sher Afghan Khan, Qazi Fasihuddin and Krishna Kumar. "Numerical Simulation of Surface Pressure and Temperature Distribution along a Cone at Supersonic Mach Numbers using CFD." *Journal of Advanced Research in Numerical Heat Transfer* 28, no. 1 (2025): 1-26.
[9] Shaikh, Javed Shoukat, Shamitha Shetty, Khizar Ahmed Pathan, Sher Afghan Khan, Qazi Fasihuddin and Krishna Kumar. "Numerical Simulation of Surface Pressure and Temperature Distribution along a Cone at Supersonic Mach Numbers using CFD." *Journal of Advanced Research in Numerical Heat Transfer* 28, no. 1 (2025): 1-26.
[10] Samylingam, Lingenthiran, Navid Aslfattahi, Kumaran Kadirgama, Devarajan Ramasamy, Kuang Kok Chee, Norazlianie Sazali, Wan Sharuzi Wan Harun, Nor Atiqah Zolpakar and Mohd Fairusham Ghazali. "Thermal and Flow Characteristics of Alumina Nanofluids in Microfluidic Systems: A Low-Concentration Study." *Journal of Advanced Research in Numerical Heat Transfer* 28, no. 1 (2025): 131-144. https://doi.org/10.37934/arnht.28.1.131144
[11] Itoh, Naotsugu, Yosuke Kikuchi, Takeshi Furusawa and Takafumi Sato. "Tube-wall catalytic membrane reactor for hydrogen production by low-temperature ammonia decomposition." *International Journal of Hydrogen Energy* 46, no. 38 (2021): 20257-20265. https://doi.org/10.1016/j.ijhydene.2020.03.162
[12] Zakeri, Reza and Ali Fazeli. "CFD modeling of hydrogen production from glycerol steam reforming in Tesla microchannel reactor." *Fuel* 357 (2024): 129646. https://doi.org/10.1016/j.fuel.2023.129646
[13] Kambara, Shinji, Yukio Hayakawa, Yu Inoue and Tomonori Miura. "Hydrogen production from ammonia using plasma membrane reactor." *Journal of Sustainable Development of Energy, Water and Environment Systems* 4, no. 2 (2016): 193-202. https://doi.org/10.13044/j.sdewes.2016.04.0016
[14] Nailwal, B. C., P. Chotalia, J. Salvi, N. Goswami, L. Muhmood, A. K. Adak and Soumitra Kar. "Ammonia decomposition for hydrogen production using packed bed catalytic membrane reactor." *International Journal of Hydrogen Energy* 49 (2024): 1272-1287. https://doi.org/10.1016/j.ijhydene.2023.09.229
[15] Nikzad, Amin, Davood Iranshahi and Mohammad Ranjbaran. "Comparative evaluation of spherical radial flow reactor, spherical axial flow reactor and tubular reactor in ammonia production by using CFD simulation." *Fuel* 350 (2023): 128772. https://doi.org/10.1016/j.fuel.2023.128772
[16] Cechetto, Valentina, Luca Di Felice, Rocio Gutierrez Martinez, Alba Arratibel Plazaola and Fausto Gallucci. "Ultra-pure hydrogen production via ammonia decomposition in a catalytic membrane reactor." *international journal of hydrogen energy* 47, no. 49 (2022): 21220-21230. https://doi.org/10.1016/j.ijhydene.2022.04.240
[17] Li, Gang, Masakoto Kanezashi and Toshinori Tsuru. "Highly enhanced ammonia decomposition in a bimodal catalytic membrane reactor for COx-free hydrogen production." *Catalysis Communications* 15, no. 1 (2011): 60-63. https://doi.org/10.1016/j.catcom.2011.08.011
[18] Koyunoğlu, Cemil, Hüseyin Karaca and Hakan Serhad Soyhan. "Modelling DME production from synthetic gases with a fluidized bed reactor: A CFD approach." *Fuel* 304 (2021): 121331. https://doi.org/10.1016/j.fuel.2021.121331
[19] Msheik, Malek, Sylvain Rodat and Stéphane Abanades. "CFD simulation of a hybrid solar/electric reactor for hydrogen and carbon production from methane cracking." *Fluids* 8, no. 1 (2023): 18. https://doi.org/10.3390/fluids8010018
[20] Kim, Sehwa, Jiseon Song and Hankwon Lim. "Conceptual feasibility studies of a CO X-free hydrogen production from ammonia decomposition in a membrane reactor for PEM fuel cells." *Korean Journal of Chemical Engineering* 35 (2018): 1509-1516. https://doi.org/10.1007/s11814-018-0037-5
[21] Campbell, Colin J., and Alexander Wöstmann. *Campbell's atlas of oil and gas depletion*. New York: Springer, 2013. https://doi.org/10.1007/978-1-4614-3576-1