International Journal of Research Publication and Reviews, Vol 4, no 6, pp 1794-1801 June 2023

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Computational Fluid Dynamics Analysis of a Propeller

[1] S. Rajesh, [2] S. Surendra Naidu, [3] T. Appanna, [4] V. Vamsi, [5] Y. Pavan Akshaynaidu.

U.G Scholars, Dept. of Mechanical Engineering, Raghu Engineering College(A), Dakamarri, Bheemunipatnam Mandal, Visakhapatnam, AP, India

Assistant Professor, Dept. of Mechanical Engineering, Raghu Engineering College(A), Dakamarri, Bheemunipatnam Mandal, Visakhapatnam, AP, India

ABSTRACT

Computational Fluid Dynamics (CFD) is a useful technique for studying fluid flow around objects. CFD can be used to simulate the flow of air or water over the blades of a propeller, yielding significant insights into its performance. The propeller is a revolving device that produces thrust by accelerating a fluid in the opposite direction of its rotation. The movement of fluid around the blades of a propeller is complex and three-dimensional, making standard approaches difficult to analyse.

CFD allows you to simulate this flow and forecast how the propeller will operate under various scenarios. Computational fluid dynamics (CFD) is a strong tool for studying the intricate flow of fluid around a propeller, providing vital insights into its performance and assisting in the design process.

INTRODUCTION

Computational fluid dynamics (CFD) is a branch of fluid mechanics that deals with the numerical analysis and simulation of flow phenomena. This involves using computer algorithms and mathematical models to simulate and solve complex flow problems that cannot be solved analytically. CFD applications span a wide range of fields, including aerospace, automotive, energy, environmental and biomedical engineering.

CFD is also widely used in the aerospace industry to design and optimize aircraft and spacecraft. CFD simulations are used to study airflow over aircraft wings, determine air resistance, and optimize wing designs to minimize drag and maximize lift. CFD is also used to simulate airflow around a spacecraft during launch and re-entry, and to design heat shields that protect the spacecraft from high temperatures.

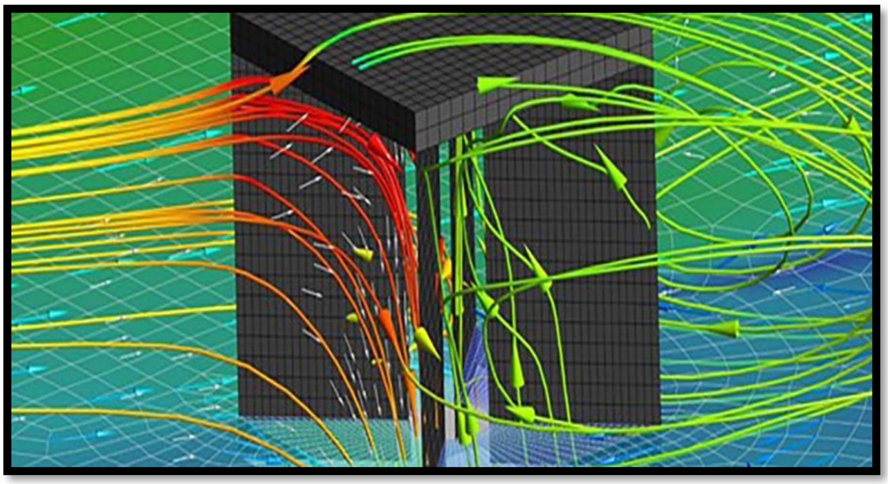

FIG.: CFD usage on daily basis

CFD ANALYSIS ON A PROPELLER

A propeller is a device that converts rotational motion into thrust and creates fluid flow in the direction opposite to the direction of rotation. Propeller performance is affected by many factors, including blade design, rotational speed, and fluid properties. Computational Fluid Dynamics (CFD) can be used to simulate fluid flow around a propeller and study its performance under various conditions.

To perform a CFD analysis of the propeller, a 3D model of the propeller and his surroundings is created. The model is then divided into a mesh of smaller elements and the equations describing the fluid behavior are solved for each element. These equations take into account the effects of factors such as viscosity, turbulence and pressure.

Important aspect of simulating fluid flow around a propeller is the boundary conditions. Boundary conditions specify velocity, pressure, and temperature values at the boundaries of the computational domain. For propeller analyses, boundary conditions are typically defined based on propeller rotational speed, blade geometry, and fluid properties.

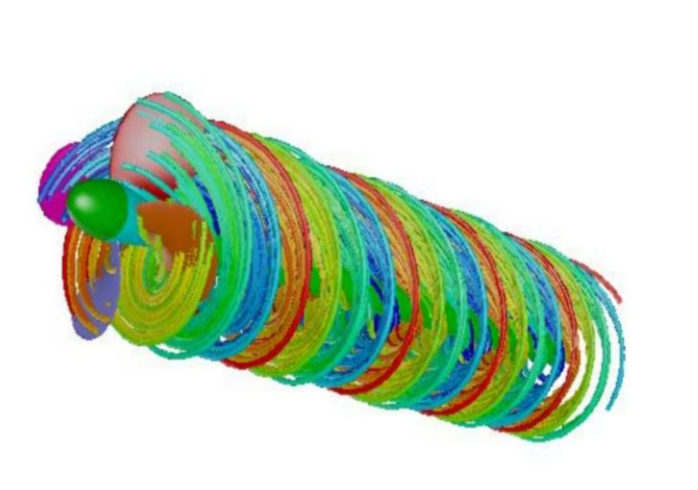

FIG.: CFD analysis on a propeller

ANALSIS ON ANSYS

ANSYS is a widely used commercial software package for performing computational fluid dynamics (CFD) simulations. It provides a comprehensive suite of tools for modeling, meshing, and analyzing complex fluid flow problems, including those related to propellers.

ANSYS is a powerful and versatile software package, but it does have some limitations. One limitation is the steep learning curve associated with the software. Using ANSYS effectively requires a significant amount of training and experience. Additionally, ANSYS is very expensive, which may limit its accessibility for some users. ANSYS provides a comprehensive suite of tools for modeling, meshing, and analyzing fluid flow around propellers that can be used to optimize propeller designs for maximum performance. Although ANSYS has some limitations, its rich feature set makes it popular with many researchers and engineers working in the field of fluid dynamics.

FIG.: ANALYSIS ON ANSYS

OBJECTIVE

The objective of Computational Fluid Dynamics (CFD) is to use numerical methods and algorithms to simulate the behavior of fluids, such as liquids and gases. CFD aims to solve the equations governing fluid flow, such as: B. Provides insight into Navier-Stokes equations and fluid flow patterns, pressure distributions, and other properties.

The main purposes of CFD include:

1. Prediction and analysis of liquid flow behavior under complex geometries and various conditions.

2. Optimize the design of fluid systems and components such as aircraft wings, turbines and heat exchangers to improve performance and efficiency.

3. Understanding the physical phenomena associated with fluid flow, such as turbulence, heat transfer, and chemical reactions. Development and refinement of numerical methods and algorithms for simulating fluid flow.

4. Provides a cost-effective and time-efficient alternative to experimental testing.

PROJECT FUTURE SCOPE

The future scope of computational fluid dynamics (CFD) is vast and promising due to continued advances in technology and research. Some of the potential future developments and applications of CFD are listed below:

1. Artificial intelligence (AI) and machine learning (ML) techniques are being increasingly used to optimize CFD simulations and improve accuracy and efficiency.

2. Integrate CFD with other simulation tools such as structural mechanics, electromagnetics, and acoustics for a more comprehensive analysis of complex systems.

3. Developing his CFD models and simulations of new technologies such as renewable energy systems, electric vehicles and additive manufacturing.

4. Advances in high performance computing (HPC) and cloud computing are enabling faster and more complex CFD simulations. Extending CFD applications in biomedical engineering and healthcare, such as B. Simulation of human blood flow and drug delivery systems.

5. CFD design optimization and decision-making tools that enable engineers to quickly evaluate and compare different design options.

6. Integrate CFD with Internet of Things (IoT) and sensor technology to enable real-time monitoring and control of fluid system.

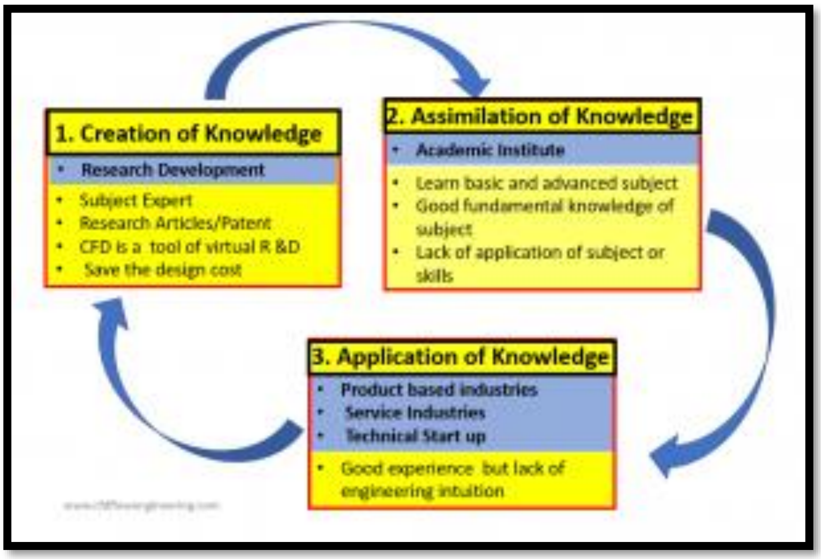

FIG.: picture exhibiting the future scope of computational fluid dynamics analysis

MATERIAL USED IN PROPELLERS

The materials from which propellers are made today can broadly be classed as members of the bronzes or stainless steels. The once popular of cast iron has now virtually disappeared, even for the production of spare propellers, in favor of materials with better mechanical and cavitation resistant properties. The propeller in this project is made from aluminium alloy. The properties required in a propeller material will depend to a very large extent on the duty and service conditions of the vessel to which propeller are being fitted.

The once popular of cast iron has now virtually disappeared, even for the production of spare propellers, in favor of materials with better mechanical and cavitation resistant properties. The propeller in this project is made from Aluminium alloy. The properties required in a propeller material will depend to a very large extent on the duty and service conditions of the vessel to which propeller are being fitted. However, the most desirable set of properties which it should possess as follows:

1. High corrosion fatigue resistance in water.

2. High resistance to cavitation erosion.

3. Good resistance to general corrosion.

4. High strength to weight ratio.

5. Good repair characteristics including weldability and freedom from subsequent cracking.

6. Good casting characteristics.

TYPES OF PROPELLERS

Propellers can be roughly divided into the following two types:

1. Fixed Pitch Propeller: As the name suggests, the blades have a fixed pitch or angle, and the propeller produces constant thrust at a given speed. These types of propellers are simple in design, inexpensive and reliable. They are commonly used in small aircraft, boats, and some industrial applications.

2. Controllable Pitch Propeller: These propellers have adjustable blades that allow you to change the angle of attack to optimize performance at different speeds and altitudes. Pitch can be adjusted manually or automatically to increase or decrease thrust. These propellers are commonly used in high performance aircraft, helicopters and some marine applications.

Within these two broad categories are several sub-types of propellers, including:

- Fixed Pitch Wooden Propeller: These propellers are made of wood and have a fixed pitch. Commonly used on small planes and microlights.

- Fixed Pitch Metal Propeller: These propellers are made of aluminum or steel and have a fixed pitch. Often used on larger aircraft.

-Ground Adjustable Propeller: These propellers have a fixed pitch, but the blades can be adjusted on the ground to optimize performance for a particular aircraft or application.

- Constant Speed Propeller: These propellers have a mechanism that automatically adjusts the blade pitch to maintain a constant RPM. They are commonly used in high performance aircraft.

- Blade Propeller: These propellers have blades that can rotate with the airflow, reducing drag and increasing efficiency. These are commonly used on multi-engine aircraft and helicopters.

- Propeller with Controllable Pitch: These propellers have blades that can be adjusted in flight to optimize performance in different flight conditions. They are commonly used in high performance aircraft and military applications.

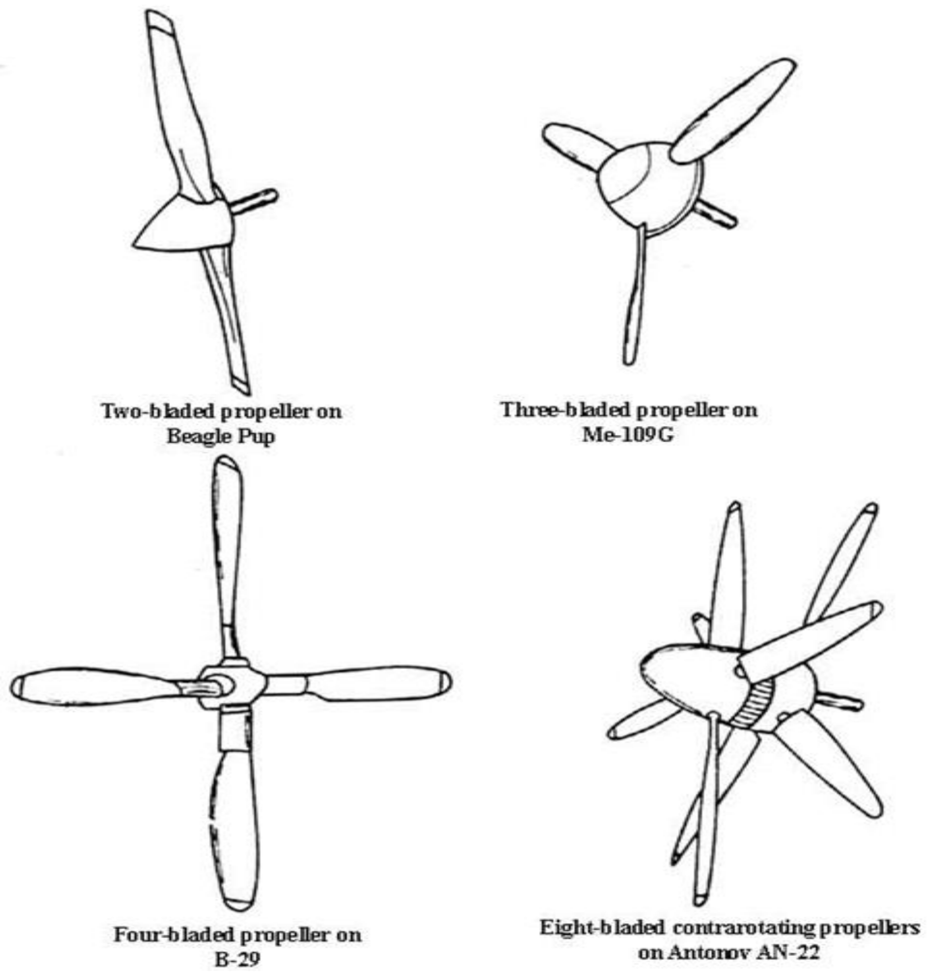

FIG.: different types of propellers

PROPELLER PERFORMANCE CHARACTERISTICS:

Propeller performance characteristics refer to the various factors that affect the efficiency and effectiveness of a propeller in generating thrust and propelling a vehicle through a fluid medium, such as air or water. These characteristics include:

1. Pitch: The angle of the propeller blades relative to the plane of rotation, which determines the amount of thrust generated per revolution.

2. Diameter: The size of the propeller, which determines the amount of thrust generated per unit of power input.

3. Blade shape: The design of the propeller blades, which affects the efficiency of the propeller by minimizing drag and maximizing lift.

4. RPM: The speed at which the propeller rotates, which affects the amount of thrust generated and the fuel consumption of the engine.

5. Blade area ratio: The ratio of the total blade area to the disc area swept by the propeller, which affects the efficiency of the propeller at different speeds.

6. Cavitation: The formation of bubbles or voids in the fluid around the propeller blades, which can reduce the effectiveness of the propeller and cause damage to the blades.

7. Blade loading: The amount of force exerted on each blade of the propeller, which affects the durability and performance of the propeller.

8. Thrust coefficient: The ratio of the thrust generated by the propeller to the product of the blade area and the fluid density, which is a measure of the propeller's efficiency.

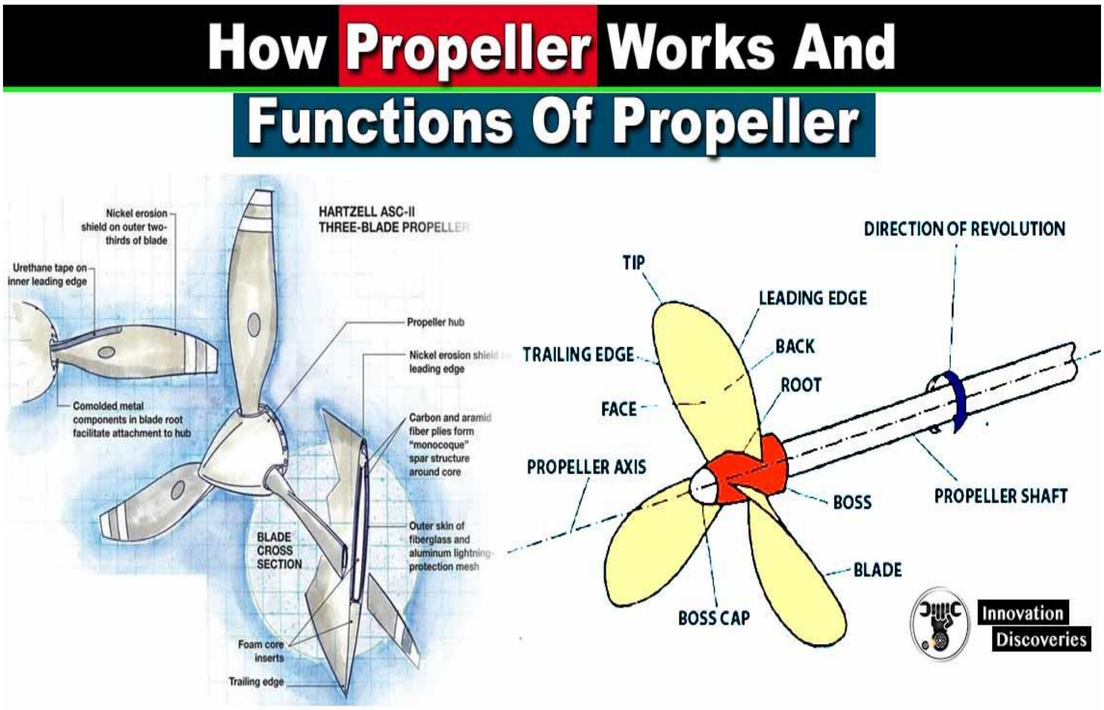

FIG: functioning of a propeller

PROCEDURE INVOLVED

The following are the steps involved in the process of analyzing the computational fluid dynamics (CFD) of a propeller:

FLUID FLOW FLUENT: Fluent is a popular computational fluid dynamics (CFD) software developed by ANSYS that is widely used to simulate fluid flow and heat transfer in a variety of engineering applications.

GEOMETRY: Geometry is a key aspect in ANSYS for computational fluid dynamics (CFD) simulations. This involves creating or importing the 3D geometry of the object or system to be analyzed.

TOOLS: ANSYS offers a wide range of powerful tools for a variety of applications including, Pretreatment tools, Solver tools, Post-processing tools, Optimization tools, Multiphysics tools, Customization tools.

AIR FLOW: ANSYS Computational Fluid Dynamics (CFD) software offers powerful capabilities for simulating and analyzing airflow in a variety of applications. ANSYS Fluent, the industry-leading CFD solver, allows users to create or import geometry, generate high-quality meshes, define boundary conditions, and solve dominant fluid equations.

CREATE: Various approaches can be used to create box geometries in ANSYS, depending on the ANSYS software used and the level of complexity desired. Here is an overview of how to create box geometries in ANSYS.

Import Box Geometry: Another option is to import box geometry into ANSYS from external CAD software such as SolidWorks, CATIA or STEP files. ANSYS supports a variety of file formats for geometry import, and you can use ANSYS Design Modeler or ANSYS Space Claim geometry import functionality to insert pre-built box geometry into your ANSYS model.

MESH CONSTRUCTION: Meshing is a key step in the numerical simulation process using ANSYS software. This is because the geometry is discretized into smaller elements or cells to solve the equations underlying fluid flow, heat transfer, structural analysis, and other physics.

RESULTS

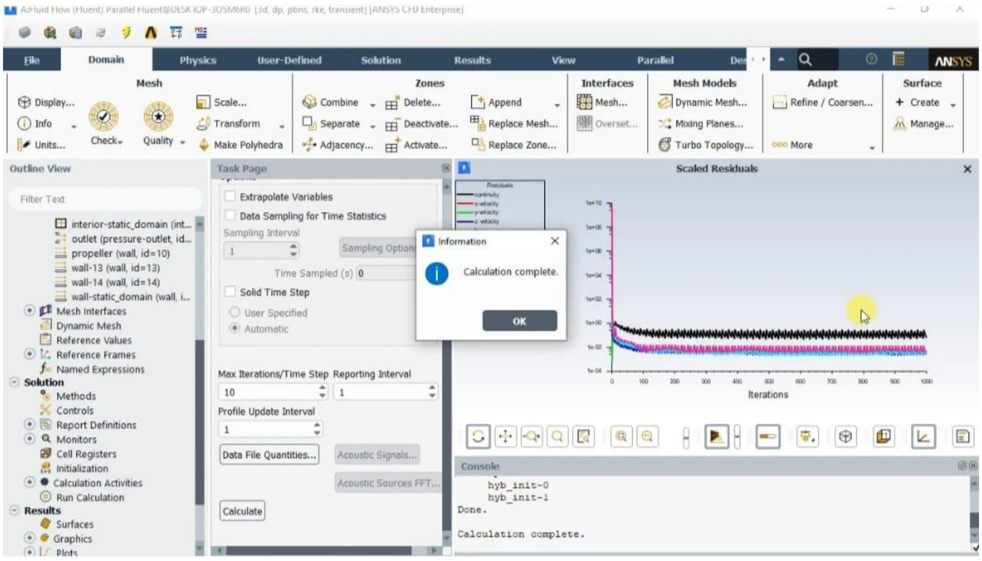

FIG.: picture exhibiting the graph of completion of analysis

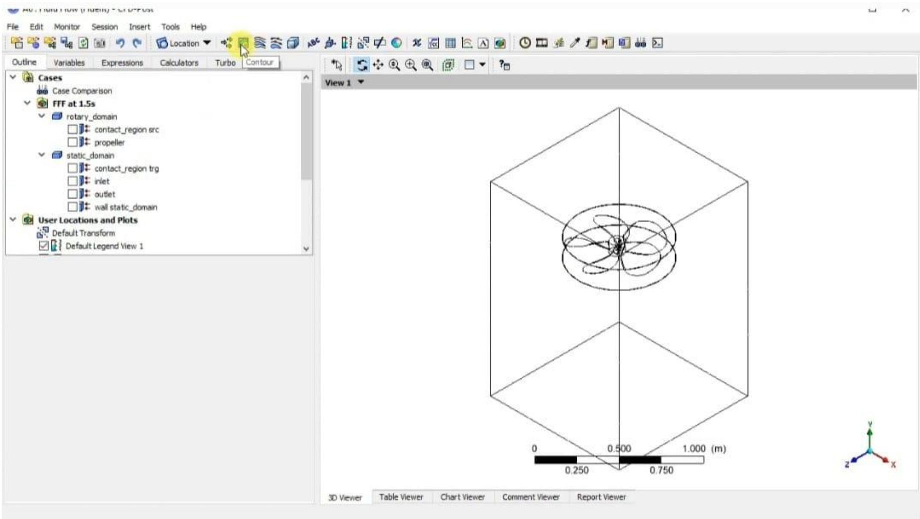

FIG.: picture exhibiting the completion on semi closed mesh analysis

CONCLUTION

In today's rapidly evolving field of engineering and aerodynamics, computational fluid dynamics (CFD) has become an essential tool for propeller analysis. This report highlights the reasons for using his CFD in propeller analysis, including accuracy, cost-effectiveness, flexibility, safety, parametric studies, insight into flow physics, and time efficiency.

CFD also offers great flexibility in optimizing propeller design. Design parameters such as blade shape, pitch, and number of blades can be easily changed in CFD simulations, enabling rapid iteration and optimization of propeller designs. This flexibility allows engineers to explore a wide range of design options and choose the propeller design with the best performance for a particular application, resulting in improved performance and efficiency.

Safety is another important aspect of propeller analysis, especially for large or high-speed propellers. Physical testing of propellers can be dangerous due to rotating machinery and complex flow patterns. CFD simulation eliminates the risks associated with physical testing and allows engineers to analyze and optimize propeller designs in a safe and controlled virtual environment. This ensures the safety of engineers and technicians involved in the analytical process.

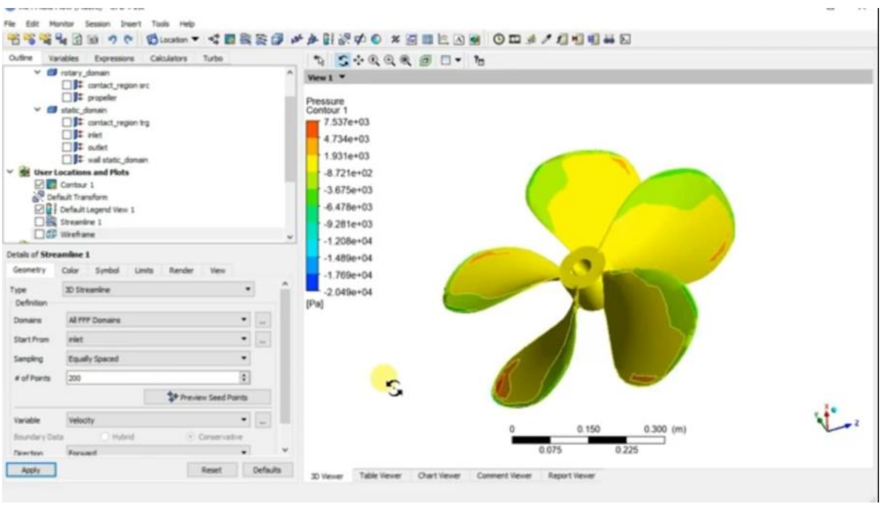

FIG.: completion of analysis with thrust acting on the propeller