Three Meter Antenna Structural Design and Analysis. Part 1: Wind Analysis

Om Gadhave[1], Mr. Manish Patil (Guide)[2]

[1]Department of Mechanical Engineering,
[2]Goverment college of engineering and research Avasari (kh),
[3]Giant Metrewave Radio Telescope (GMRT) Khodad, (NCRA)-(TIFR).

* * *

Abstract - *This research focuses on the wind analysis of a Three-meter Antenna, specifically examining the dish component. Utilizing ANSYS software, wind tunnel simulations were conducted to determine the drag and lift forces acting on the antenna dish. A detailed model of the antenna dish was created using SolidWorks software. The primary objective of this analysis was to quantify the drag and lift forces under varying angles of the antenna and different wind velocities. The findings from this study provide valuable insights into the structural behavior of the antenna under wind loading conditions, aiding in its optimization and performance enhancement.*

***Key Words*: Wind Analysis, Drag Force, Lift Force, Wind Tunnel Simulation, Three-meter Antenna dish.**

1.INTRODUCTION

Wind analysis stands as a cornerstone in the realm of structural engineering, particularly concerning the design and evaluation of critical components like antenna dishes. This project delves into a comprehensive wind analysis of a Three-meter Antenna Dish, utilizing advanced computational tools such as ANSYS Fluent software. The primary objective is to quantify the drag and lift forces acting on the dish under diverse wind velocity (20 kmph to 90 kmph) and at different positions of dish in tunnel (0 degree to 90 degree), crucial for ensuring its structural integrity and operational reliability.

Antenna dishes, integral to communication systems, endure continuous exposure to environmental forces, with wind being a significant factor. Understanding the intricate interplay between wind and antenna dishes is imperative for guaranteeing their longevity, reliability, and optimal performance.

One of the key parameters studied in wind analysis is pressure distribution across the surface of the antenna dish. Wind exerts pressure on the dish, creating variations in pressure across its surface. These pressure differentials can lead to localized stress concentrations, potentially compromising the dish's structural integrity. By mapping out pressure distributions, engineers can identify critical areas prone to high stress and develop strategies to mitigate potential failure modes. Additionally, wind analysis involves evaluating aerodynamic forces such as drag and lift. Drag force, acting parallel to the direction of the wind flow, opposes the motion of the dish through the air. Lift force, perpendicular to the wind flow, results from differences in air pressure above and below the dish's surface. These aerodynamic forces play a pivotal role in determining the dish's stability and structural response to wind loading.

By comprehensively studying the aerodynamic forces, engineers can anticipate potential structural vulnerabilities and devise strategies to mitigate them. Furthermore, wind analysis aids in optimizing the dish's design to enhance its performance and longevity under varying wind conditions. This comprehensive approach ensures that antenna dishes not only meet safety standards but also exhibit robustness and reliability in their operational environments.

1.1 Purpose of the project

The purpose of this project is to conduct comprehensive structural design and analysis of a three-meter antenna and its accompanying dish. This antenna holds significant potential for a multitude of applications, particularly in the realm of astronomical observations and Radio Frequency Interference (RFI) measurements. By harnessing its capabilities, we can delve into the depths of space, capturing invaluable data for astronomical research. Additionally, its precise design enables accurate RFI measurements, contributing to the mitigation of interference in radio astronomy observations. In summary, this project seeks to develop a versatile antenna system capable of facilitating groundbreaking astronomical observations while effectively addressing challenges posed by RFI, thereby advancing our understanding of the universe and enhancing the efficiency of radio astronomy endeavors

1.2 Geometry details of Antenna Dish

Antenna dish is designed with following parameters: -

1. F/D ratio: 0.35
2. Diameter (D): 10.00 feet = 3048.00 mm
3. Base-plate diameter at dish-center: 1.00 feet = 304.80 mm
4. Number of support arms in the dish: 12

5. Diameter of the shadow region at the center: 4 inch = 101.60 mm

6. Width of the square tube to be used for arms: 2 cm = 20.00 mm

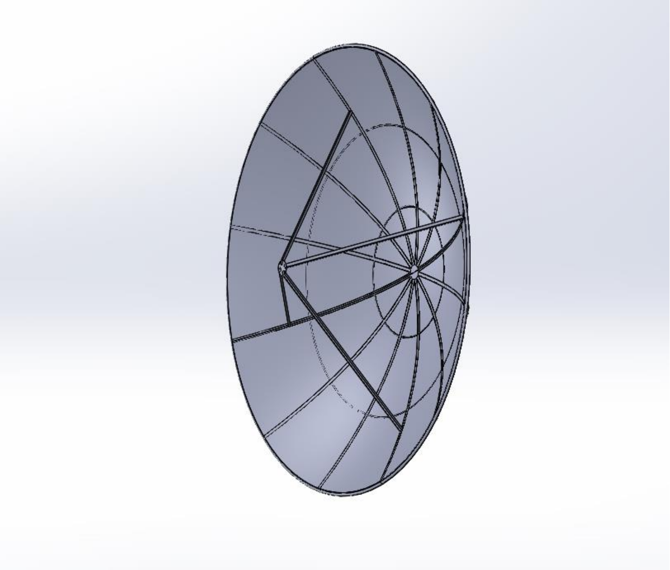


Fig-1: Model of Antenna Dish

2. PHYSICAL DESCRIPTION OF THE WIND ANALYSIS

2.1 Geometry

In the wind analysis project, the initial step involved crafting a detailed model of the antenna dish using SolidWorks software. Once the model was meticulously designed, it was saved as a .stp (STEP) file format, facilitating seamless integration into ANSYS Fluent for subsequent analysis. Within ANSYS Fluent, the imported dish model served as the focal point for the analysis, representing the physical geometry upon which various simulations were conducted. To replicate real-world conditions and ensure accurate results, an enclosure, akin to a wind tunnel, was meticulously constructed around the dish model using techniques such as Boolean subtraction as shown in fig. 2. This enclosure provided the computational domain within which the wind flow interactions with the dish were examined. The enclosure, measuring 8 meters in length, served as the computational domain for the analysis, allowing for precise examination of wind flow interactions with the dish.

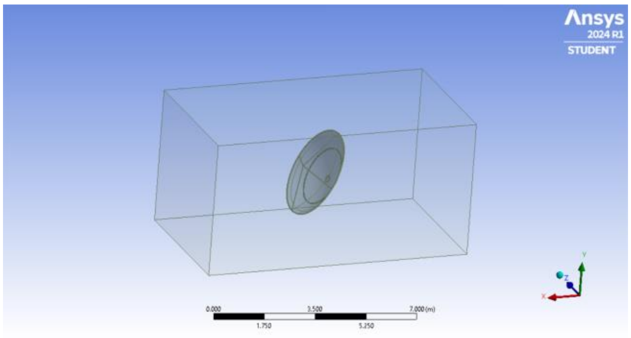


Fig-2: Enclosure surrounding the antenna dish, providing clear view of computational setup used for analysis

2.2 Meshing

In the meshing phase using ANSYS Fluent, default settings were initially applied, followed by adjustments to the element size up to 100 mm for improved accuracy. Meshing is a pivotal step in computational fluid dynamics (CFD) as it subdivides the geometry into smaller elements, enhancing numerical calculations. Special attention was given to refining the mesh around critical areas such as the antenna dish to capture finer flow details and ensure precise results. Mesh quality parameters like aspect ratio, skewness, and orthogonality were closely monitored to guarantee the generation of a high-quality mesh conducive to robust simulations. This iterative process, aimed at balancing computational efficiency with solution accuracy, is essential for producing reliable and meaningful results in CFD simulations.

Following are the parameters of mesh assigned to obtain the desired mesh characteristics:

i) Mesh element size: 100 mm

ii) Methods: Tetrahedral mesh

iii) No. of Nodes: 21798

iv) No. of Elements: 108303

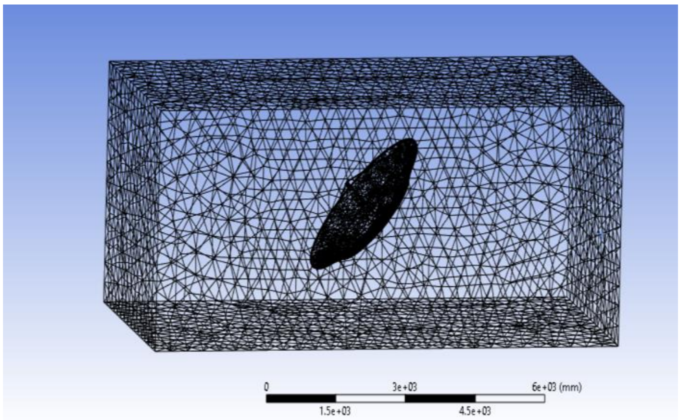


Fig-3 Shows the meshing of model

2.3 Named selections:

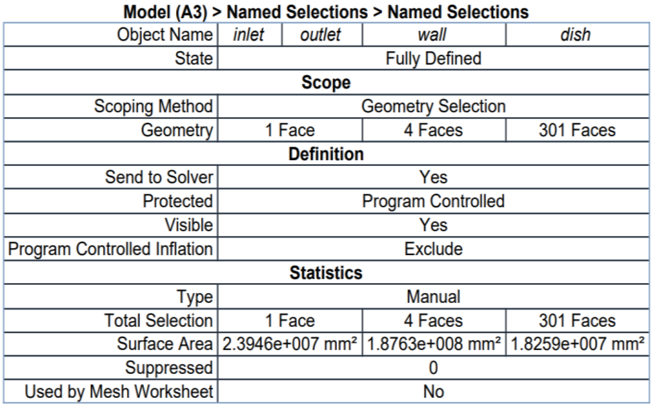
Named selections are crucial in Computational Fluid Dynamics (CFD) simulations, enabling users to apply specific boundary conditions, monitor regions of interest, and perform force calculations within the computational domain. Here's why they are important:

A) Boundary Conditions: Assign unique boundary conditions like velocity, pressure, or temperature to different parts of the geometry, especially beneficial for complex geometries.

B) Region Monitoring: Track and analyze pressure drops, heat transfer rates, or flow rates in specific areas, aiding in focused analysis within CFD simulations.

C) Force and Moment Calculations: Essential for fluid-structure interaction simulations, named selections define areas for accurate force and moment computations, aiding in structural analysis and fluid force predictions.

Table-1 Parameters for Name selection.



Model (A3) > Named Selections > Named Selections

Object Name	inlet	outlet	wall	dish
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face		4 Faces	301 Faces
Definition				
Send to Solver	Yes			
Protected	Program Controlled			
Visible	Yes			
Program Controlled Inflation	Exclude			
Statistics				
Type	Manual			
Total Selection	1 Face		4 Faces	301 Faces
Surface Area	2.3946e+007 mm²		1.8763e+008 mm²	1.8259e+007 mm²
Suppressed	0			
Used by Mesh Worksheet	No			

2.4 Setup

In ANSYS Fluent, the term "setup" refers to the configuration and specification of the various parameters and settings necessary to perform a computational fluid dynamics (CFD) simulation.

A) Model

The model setup in ANSYS Fluent is critical for accurately calculating wind drag and lift forces on the dish for wind analysis. Key parameters such as "Space 3D" capture the three-dimensional geometry of the dish, ensuring an accurate representation of flow behavior. "Time Steady" maintains constant flow variables over time, providing a stable simulation snapshot of the wind's impact. Additionally, the Viscous SST k-omega Turbulence Model accurately simulates turbulent airflow, crucial for capturing intricate flow patterns that affect drag and lift forces. These parameters collectively enable a comprehensive analysis of wind-dish interaction, aiding in informed design decisions and optimizations for enhanced performance and efficiency.

B) Solver

The solver in ANSYS Fluent performs the calculations necessary to obtain a solution for the specified physics and boundary conditions.

i) Type: Pressure-Based

ii) Velocity Formulation: Absolute

iii) Time: Constant

iv) 3D space

C) Boundary conditions:

In the wind analysis conducted in ANSYS Fluent, several boundary conditions were selected to accurately simulate the flow behavior around the dish. These boundary conditions play a critical role in defining the environment in which the simulation operates.

1. Inlet Boundary Condition:

- **Type:** Velocity Inlet

- **Description:** The inlet boundary condition represents the region where air enters the computational domain. By specifying a velocity inlet, a prescribed velocity profile is imposed at this boundary, simulating the incoming wind flow. Various air velocities were applied at the inlet to analyze the dish's response under different wind conditions.

2. Outlet Boundary Condition:

- **Type:** Pressure Outlet

- **Description:** The outlet boundary condition defines the region where air exits the computational domain. By setting it as a pressure outlet, a static pressure is specified, allowing the flow to exit freely without reflecting back into the domain. This condition ensures that the simulation accurately represents the open environment surrounding the dish.

3. Dish Surface Boundary Condition:

- **Type:** Wall

- **Description:** The dish surface boundary condition represents the solid surface of the dish. By setting it as a wall boundary condition, the airflow is constrained at this surface, simulating the no-slip condition where air velocity is zero relative to the dish surface. This boundary condition is essential for accurately capturing the interaction between the airflow and the dish geometry, enabling the calculation of drag and lift forces.

4. Enclosure Side Boundary Condition:

- Type: Wall

- Description: Defines the walls of the computational domain, confining airflow to prevent artificial effects from flow escaping domain boundaries, ensuring realistic wind flow simulations.

These boundary conditions collectively create a well-defined computational environment for accurately simulating wind flow around the dish, enabling precise calculations of drag and lift forces under realistic wind conditions.

D) Material Properties

Material: air (fluid)

Property	**Units**	**Method**	**Value(s)**
Density	kg/m^3	constant	1.225
Cp (Specific Heat)	J/ (kg K)	constant	1006.43
Thermal Conductivity	W/ (m K)	constant	0.0242
Viscosity	kg/ (m s)	constant	1.7894e-05
Molecular Weight	kg/kmol	constant	28.966

E) Solution Methods:

In our research, we emphasize the selection of appropriate solution methods in ANSYS Fluent, focusing on the second-order upwind scheme for fluid flow and heat transfer simulations. This scheme strikes a balance between accuracy and stability, crucial for accurately capturing wind flow behavior and computing drag and lift forces. The second-order upwind scheme minimizes numerical diffusion, allowing faithful representation of flow features and boundary layers. However, careful mesh refinement and solution monitoring are necessary to address potential oscillations in regions of steep gradients. Through detailed documentation of settings and parameters, our research ensures reproducibility and transparency, supporting the reliability of our findings in wind analysis and force calculations.

These are the Scheme selected for Variable.

- **Pressure:** Second Order
- **Momentum:** Second Order Upwind
- **Turbulent Kinetic Energy:** Second Order Upwind
- **Specific Dissipation Rate:** Second Order Upwind

F) Report Definitions:

In ANSYS Fluent, report definitions are essential for specifying and monitoring quantities of interest during or after a simulation. They enable tracking and recording of various parameters and results to analyze fluid flow or heat transfer behavior conveniently. For our project objectives and flow parameters, we created two report definitions: Drag and Lift coefficient. These were created under the "force report" option in report definitions, specifying output types for drag and lift coefficients. ANSYS Fluent automatically generates these reports at specified intervals during the simulation.

G) Initialization:

In wind analysis conducted using ANSYS Fluent to calculate drag force and lift force, we employ the standard initialization method to set the initial conditions of the flow field. This initialization procedure is fundamental in establishing a starting point for the iterative solution process, ensuring convergence towards a physically realistic solution

H) Run Calculation:

"Run calculation" involves executing the simulation using specified settings and parameters. In our wind analysis with ANSYS Fluent for calculating wind forces, we ran the simulation for 200 iterations to achieve convergence and accuracy. This choice balances computational efficiency with solution accuracy, ensuring the solution adequately converges to a steady state. We monitored convergence criteria such as residual values and solution stability throughout the simulation. Documenting our selection of 200 iterations and the monitoring process ensures transparency and reliability in our analysis, facilitating reproducibility and understanding of the obtained results.

3.POSITION OF DISH IN WIND TUNNEL.

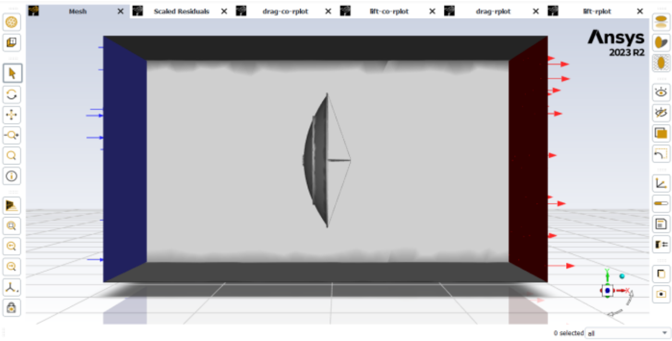


Fig-4: Position of the 0-degree dish in the wind tunnel.

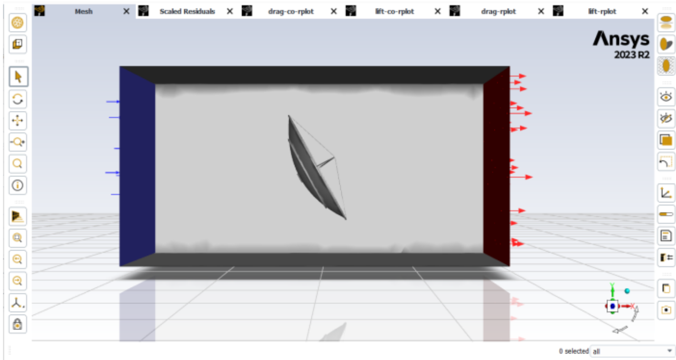


Fig-5: Position of the 30-degree dish in the wind tunnel.

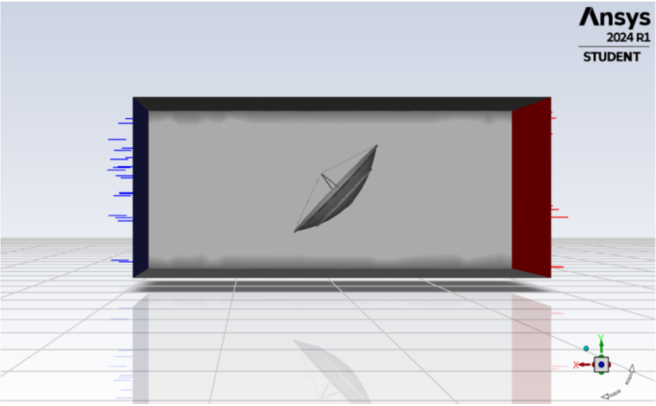


Fig-6: Position of the 45-degree dish in the wind tunnel.

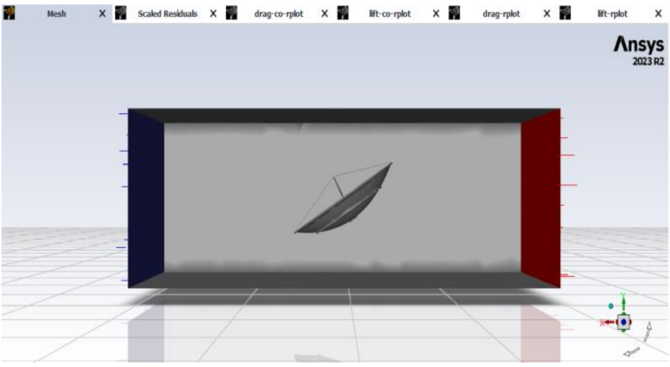


Fig-7: Position of the 60-degree dish in the wind tunnel.

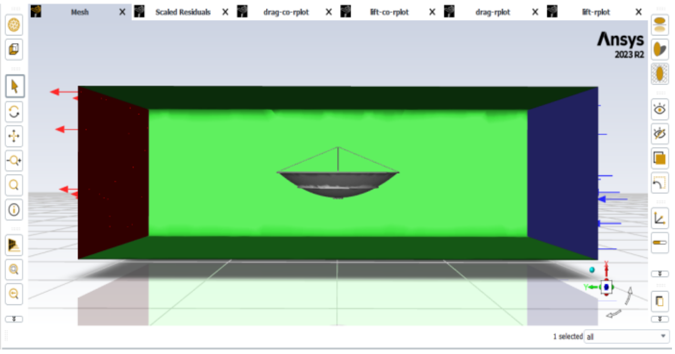


Fig-8: Position of the 90-degree dish in the wind tunnel.

4. RESULTS AND DISCUSSION OF WIND ANALYSIS

In conducting the wind analysis of the dish antenna, advanced computational fluid dynamics (CFD) simulations were employed using ANSYS Fluent software. This allowed for precise calculations of drag and lift forces within a controlled virtual wind tunnel environment. The analysis encompassed a comprehensive range of air velocities, varying from 20 to 45 km/h, to accurately simulate diverse wind conditions encountered in real-world scenarios. Moreover, the angular positioning of the dish antenna within the virtual wind tunnel was systematically adjusted from 0 degrees to 90 degrees, enabling a thorough assessment of force distribution across its surface. This approach facilitated the examination of aerodynamic interactions between the dish's geometry and the airflow from different angles, providing valuable insights into its performance under varying wind conditions. The wind analysis was conducted separately for both the front and back sides of the dish antenna using ANSYS Fluent. The results revealed a notable discrepancy in the forces experienced, with the front side consistently experiencing higher wind forces compared to the back side. This observation underscores the significance of comprehensively understanding the aerodynamic behavior of the dish antenna to optimize its performance and structural integrity. To present these findings effectively, detailed tables summarizing the calculated drag and lift forces were incorporated into the report. Table 6 offers an overview of the forces observed during front-side wind analysis, while Table 7 outlines the corresponding data for the back side of the dish antenna. By leveraging ANSYS Fluent for simulation and analysis, the report provides valuable insights into the aerodynamic behavior of the dish antenna under varying wind conditions, facilitating informed decision-making for design optimizations and operational enhancements

Table-2 Displays findings from wind analysis on the dish antenna's back side airflow.

Dish position	Forces	20km/h wind speed (N)	30km/h wind speed (N)	40km/h wind speed (N)	45km/h wind speed (N)
0	Drag	508.75	1144.07	2033.26	2762.38
	Lift	0.8629	2.12	3.72	4.209
30	Drag	383.79	863.34	1533.75	1939.21
	Lift	236.85	532.75	946.48	1196.77
45	Drag	251.24	564.91	1002.87	1270.29
	Lift	279.88	628.55	1116.43	1414.96
60	Drag	141.24	317.87	565.09	712.72
	Lift	287.19	645.93	1148.64	1448.78
90	Drag	55.20	145.12	212.27	272.16
	Lift	25.73	47.84	76.82	100.93

Table-3 Displays findings from wind analysis on the dish antenna's back side airflow.

Dish position	Forces	20km/h (N)	30km/h (N)	40km/h (N)	45km/h (N)
0	Drag	427.85	831.20	1407.43	1782.95
	Lift	1.06	2.63	4.51	6.16
30	Drag	255.34	574.16	1020.58	1296.47
	Lift	116.01	260.99	464.06	587.30
45	Drag	150.12	335.75	617.39	780.88
	Lift	125.30	305.42	450.13	569.39
60	Drag	68.88	154.74	274.64	347.29
	Lift	62.25	140.04	248.84	314.56
90	Drag	55.20	145.12	212.27	272.16
	Lift	25.73	47.84	76.82	100.93

From the table above, it's evident that the maximum force is exerted when the wind flows directly forward at 0 degrees. Therefore, to determine the maximum force, we need to calculate the wind force at high velocity when the angle of incidence is 0 degrees.

Dish position	Forces	50km/h wind speed (N)	60km/h wind speed (N)	70km/h wind speed (N)	80m/h wind speed (N)
0	Drag	3407.44	4906.69	6678.44	8718.57
	Lift	1.63	2.051	2.94	6.65
Dish position	Forces	85km/h wind speed (N)	87km/h wind speed (N)	90km/h wind speed (N)	
0	Drag	9838.72	10311.84	11047.29	
	Lift	7.13	8.12	17.55	

5. CONTOUR PLOTS

In the wind analysis of the dish antenna, detailed contours were generated using ANSYS Fluent to visualize the distribution of wind forces exerted on the dish within the wind tunnel. These contours provide a comprehensive depiction of how the airflow interacts with the surface of the dish, highlighting regions of high and low pressure, as well as areas of increased drag and lift. By analyzing these contours, valuable insights were gained into the aerodynamic behavior of the dish antenna, aiding in the assessment of its structural integrity and performance under varying wind conditions. The figures below depict contour plots illustrating the variation in wind force distribution across the dish antenna at different angular positions within the wind tunnel.

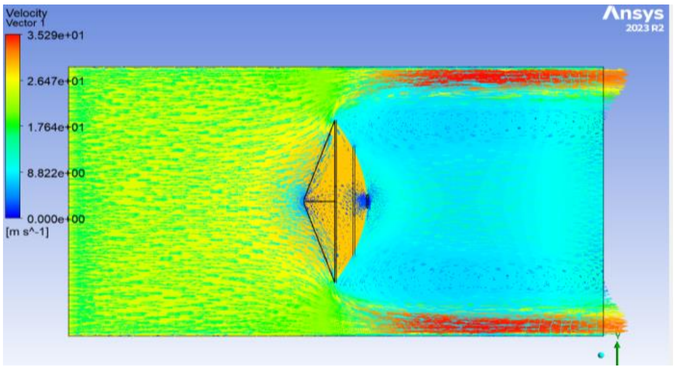


Fig-9 Front wind flow on dish at 0-degree position.

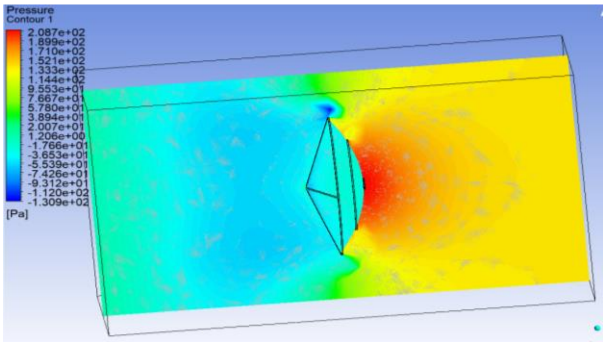


Fig-10 Contour plot of dish at 0° in wind tunnel, wind flows over back-side.

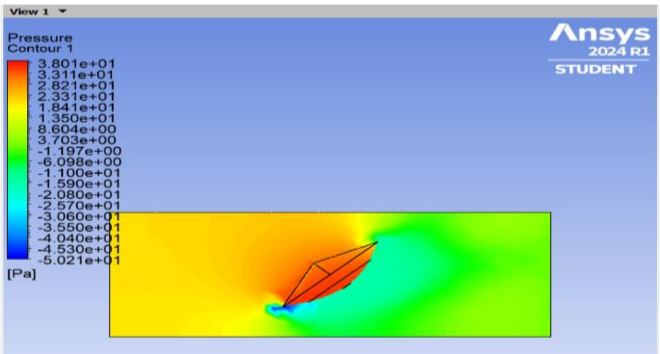


Fig-11 Contour plot of dish at 45° in wind tunnel, wind flows over front-side.

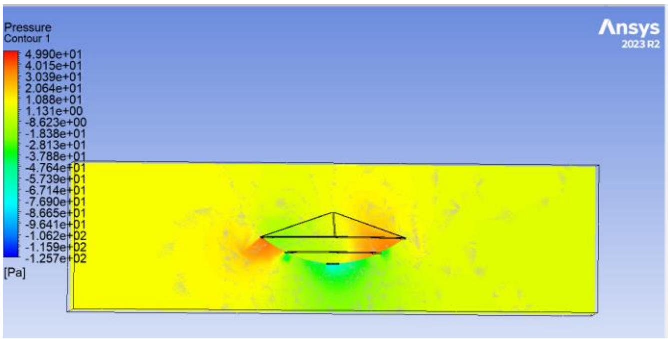


Fig-12 Contour plot of dish at 45° in wind tunnel.

6.CONCLUSIONS

The wind analysis conducted on the Three Meter Antenna dish using advanced computational fluid dynamics simulations provided valuable insights into the aerodynamic behavior of the structure. By systematically varying wind speeds and dish positioning from 0 to 90 degrees, it was observed that the maximum wind forces act on the dish when positioned at 0 degrees, while the forces are minimized at 90 degrees. This indicates that the structural integrity of the dish is most vulnerable to failure under high wind forces, especially when positioned at 0 degrees. Conversely, positioning the dish at 90 degrees results in reduced forces, enhancing its stability and resilience against wind loads.

Furthermore, the analysis revealed significant differences in wind forces experienced by the front and back sides of the dish, emphasizing the importance of understanding aerodynamic interactions for optimizing structural integrity. The contour plots generated from the analysis provided visual representations of pressure distribution on the dish, highlighting areas of maximum and minimum pressure. This comprehensive assessment of pressure distribution and force dynamics provides a thorough understanding of the dish's performance under diverse wind conditions, essential for ensuring its reliability and functionality in real-world scenarios.

Overall, the wind analysis conducted through computational fluid dynamics simulations using ANSYS Fluent software has contributed valuable insights into the aerodynamic behavior of the Three Meter Antenna dish, shedding light on critical factors influencing its structural integrity and performance in varying wind conditions.

In the second part static analysis will preform on the dish frame to check the failure point of dish. At which wind speed the dish frame will fail can be determined by comparing results of wind and static analysis.

REFERENCES

[1] https://youtu.be/W1eTcXMSfuk?si=No2dkj_g0UOeKGXs

[2] Ansys Workbench Product release note by ANSYS, Inc. Southpointe 275 technology drive Canonsburg, 245-256.

[3] ANSYS Meshing User's Guide by ANSYS, Inc. Southpointe 275 technology drive Canonsburg, 120-132

[4] Patricia Martín, Experimental study of the effects of dish antennas on the wind loading of telecommunication towers, Journal of Wind Engineering and Industrial Aerodynamics, Elsevier, pp. 41-47, 2015.

[5] Celio F. Carril Jr., Experimental study of the wind forces on rectangular latticed communication towers with antennas, Elsevier, pp. 1007-1022, 2003.

[6] J.D. Holmes a, R.W.Banks a and G. Roberts b, Drag and aerodynamic interference on microwave dish antennas and their supporting towers, pp. 263-270, 1993.