Defence Science Journal, Vol. 70, No. 1, January 2020, pp. 10-17, DOI : 10.14429/dsj.70.14437

Estimation of an Object Trajectory in an Intake Duct using Numerical Simulation

L. Aravinth[#], N. Vidhyashankar[@], Reza Abbas[@], and N.M. Sudharsan[#,*]

[#]*Department of Mechanical Engineering, Rajalakshmi Engineering College, Chennai - 602 105, India*
[@]*DRDO-Gas Turbine Research Establishment, Bengaluru - 560 093, India*
[*]*E-mail: sudharsann@asme.org*

ABSTRACT

This research aims to study the trajectory of an object inside a serpentine duct of a gas turbine engine using computational fluid dynamics. The coupled implicit solver with 6-degree of freedom (6-DOF) and chimera mesh (Overset mesh) is used to track the object's trajectory. Various object orientation and aircraft angle of attack (AoA) at a speed of Mach 0.3 is studied. This provides an understanding of the bird's movement inside the duct that might cause damage to the engine components during takeoff and landing. It was observed that the combination of AoA and object orientation decide the length of the trajectory before impact. The object is found to travel the farther when the AoA is at -20^o with object oriented at 0^o and 45^o.The object tends pitch and yield to the flow irrespective of its initial orientation and hence the aircraft angle of attack is a more predominant factor. The effect of pressure recovery due to AoA and object orientation is also presented. The recovery is found to be at its best for AoA of 0^o irrespective of object orientation. This approach could be utilised for designing an intake duct that can limit the damage to engine components due to bird ingestion and simultaneously maintain good pressure recovery.

Keywords: Gas turbine engine, Trajectory; Angle of attack; Overset mesh

NOMENCLATURE

CG	Center of gravity
IGV	Inlet guide vane
I_{xx}	Roll moment of inertia
I_{yy}	Pitch moment of inertia
I_{zz}	Yaw moment of inertia
IO	Initial orientation
K	Kelvin
m	Mass
M	Mach number
Pa	Pascal
P_f	Engine face static pressure
P_{0f}	Engine face total pressure
$P_{0\infty}$	Free-stream total pressure
P_{sf}	Pound force per square feet
R	Rankine
T	Time
SST	Shear stress transport
X,Y,Z	Cartesian coordinates

1. INTRODUCTION

The compressor of an aircraft engine is vulnerable to a bird strike. A bird of about 1 kg in weight flying at high velocity can cause physical damage to the strut / blade even resulting in mechanical failure in the event of a direct impact. This may lead to blade-off situation or aerodynamic phenomena like stall or surge and may lead to flame out. This can be catastrophic to the engine. It is necessary to ensure the safety of forward components of aero engine such as, for example, struts, IGV and fan blade due to bird impact. Thus, it is imperative to ascertain *a priori* by simulation tools the nature and position of impact as well as structural loading. Moving mesh, particle collision, and fluid structure interaction are all areas pursued by scientists to build a robust numerical model. The limitation of handling morphed meshes, meshes with large aspect ratios and subsequent transfer of information between a static and dynamic mesh and/or fluid and structural solver are well documented. The present study aims to develop one such methodology to study bird trajectory inside an intake manifold. This work is intended to estimate the trajectory of the bird inside a generic intake duct and its position at the time of impact in the duct (either inside the duct or directly through the intake exit) using computational fluid dynamics (CFD) by modelling it as a cylindrical object.

The simulation of the bird's trajectory motion was carried out with the inlet free-stream conditions of Mach number 0.3 at takeoff. The bird fragment at the entrance of the engine depends upon location of bird strike and angle of impact with respect to intake duct. From the simulation, the bird's position and orientation were computed and plotted for various conditions such as initial orientation and angle of attack (AoA). Although the current focus is only to study the trajectory of the bird, this simulation can also help in designing a suitable duct that can limit the effect of bird impact and maintain a good pressure recovery for various inlet conditions. Lijewski[1] demonstrated a time accurate CFD approach to transonic store separation trajectory prediction. Overlapping grid approach was used coupled with an implicit Euler flow solver and a 6-degree of freedom motion solver was used to predict the trajectory. This approach is adopted here for predicting the trajectory of the bird. Ubels[2] used Arbitrary Lagrangian Eulerian (ALE) and

Received : 11 April 2019, Revised : 13 September 2019
Accepted : 02 January 2020, Online published : 10 February 2020

smooth particle hydrodynamics (SPH) method, bird model to represent a synthetic gelatin bird that was used in the physical experiment. The result obtained from numerical simulation is comparable to experiments in terms of pressure profile, Hugoniot and stagnation pressure. However, their focus was to study the structural damage due to impact, but the current research is on the trajectory.

Nizampatnam[3] described that a hemispherical projectile shape most closely matched the response of an actual bird in an experimental test. The numerical result was found to be a significant factor in predicting the correct shock pressure resulting from both normal and oblique impact. Wellborn[4] performed experimental studies for a flow in a diffusing S-duct. The results presented could help validate the current simulation. Zhang[5] conducted a numerical simulation of a flow inside an S-type duct using a two equation SST turbulence model. As this model (SST) was found to match well with experimental results in terms of pressure recovery and distortion, the same model was used in the present simulation. Kachel[6] performed the numerical flow the simulations during the design phase of highly bent intake geometry. They carried out a sensitivity study based on time step and domain volume. They also estimated a safety margin from the unsteady data for construction and testing of the wind tunnel model.

However, to bench mark the present work, recent literature that includes both experimental and CFD studies is sought for. Emphasis here is to accurately estimate the trajectory of the bird. This requires the knowledge of the fluid flow inside. The velocity inside the duct is to be estimated in order to transfer this information to the overset mesh, which in turn helps move the bird. A typical 'S' type duct is considered here. Aref[7] modelled the intake with HPCMB CREATETM – AV kestrel simulation tools. They investigated the model using a passive and an active flow control method, and found that these flow control method reduce the distortion coefficient at the engine face. The results of this work[7] have been utilised for validating the numerical results of the present work. The novelty is in incorporating the moving mesh capability along with object orientation for estimating the trajectory. The information available in open literature is limited and deals mostly with the structural effects of a bird's contact with the blades or duct walls. All other duct related literature deal with either design inside a duct or flow physics inside a duct. To the best of the authors' knowledge there is no open literature that deals with the trajectory estimation of a bird inside an intake duct.

The present work aims to address this particular phenomenon of trajectory estimation of a bird inside a serpentine intake duct under various angle of attack (AoA) and Orientations. The bird is considered to have 6-DOF with appropriate mass and moment of inertia. An overset (Chimera)[8] mesh is used to track the motion. There are many performance parameters but in our work the focus was only on developing the computational methodology for trajectory estimation and pressure recovery was just one parameter that was studied to present the effect of orientation and AoA. The main premise of this study is to predict trajectory of the object inside an intake duct and not on how the object movement influences the flow quality and engineering parameters in the engine face plane such as Pressure Recovery. Various parameters like speed, linear and angular trajectory of the object was traced and plotted till the time of impact.

2. VALIDATION

The numerical simulations performed in this research needs to be validated. From the literature review it is seen that there is no publication that deals with the estimation of the trajectory of a bird inside an S-type duct. Both orientation of object with respect to the axis of the intake duct at the entry and angle of attack for aircraft was varied and simulated. Studies have been performed on estimating the trajectory of a store separation[1] and for understanding the flow inside an S-type diffuser duct[4-7].

There is no study theoretical or experimental for comparing the results of this simulation. Therefore the validation was divided in two part. One to validate the flow inside an S-type duct and the second is to validate the trajectory. Hence the current validation is split into two part: (a) simulate the flow inside the S-type duct and compare the pressure recovery, Mach number and Static pressure ratio with published literature[7], (b) simulate the trajectory of a store separation and compare with published literature[1].

These two validation would satisfy the numerical methodology used for predicting the trajectory as well as the flow characteristics.

2.1 S-duct Diffuser Validation

The S type duct RAE M2129 is simulated and compared with the published experimental results[7]. The boundary conditions as mentioned in the experimental setup were utilised. The engine face static pressure as well as the following parameters such as engine face pressure recovery, Mach number and Static pressure ratio was obtained from the simulation and as presented in Table 1. The current simulation compares well with the results published in the literature[7].

Table 1. Validation data of the RAE-M2129 intake duct

	Pressure recovery (PR) $p_0/p_{0\infty}$	Mach number	Static pressure ratio $p_f/p_{0\infty}$
Aref[7], *et al.*	0.9744	0.4193	0.8522
Current simulation	0.9697	0.4052	0.8587
Deviation from reference	0.5	3.3	0.7

2.2 Store Separation

The trajectory estimation of a store released from an altitude for which data is available in open literature was used as a benchmark[1]. The computational validation of the coupled 6-DOF and overset mesh system was carried out using a simulation of a store separation event from underneath a delta wing under transonic conditions (Mach number 0.95) at an altitude of 7,924.8 meters for a particular weapon configuration with appropriate ejector forces. An inviscid flow was assumed to simplify the above simulations. All the domains are initialised with the free-stream values for steady and unsteady state conditions. The free-stream velocity, pressure and temperature were obtained from Lijewski & Suhs[1].

These results were obtained after performing a study for both spatial and temporal independence. The grid independence study was for both spatial & temporal. The cases studied were 4 & 8 million with same time step of 1e-^{2} sec and 8 million with 1e-^{2} & 1e-^{3} sec time step as presented in Fig. 1. A standard wall function is enabled in the solver however there were 24 layers of prismatic cells to capture wall effects at the boundary. Figure 1 presents the trajectory along Z direction of the store. It can be seen that there is a good agreement between the values of the data presented in reference[1]. The flow physics point of view is not discussed here and readers are invited to refer to Lijewski & Suhs[1] for the same.

From the above sets of validation and mesh independence study the procedure for discretisation of domain for further simulation is as follows:

(a) First prism layer close to the wall should be 0.2 mm
(b) The prism layer growth rate is 1.3
(c) A total of 24 layers to capture the wall effects, and
(d) The rest of the domain is discretised with hexahedral cells of size 8-10 mm.

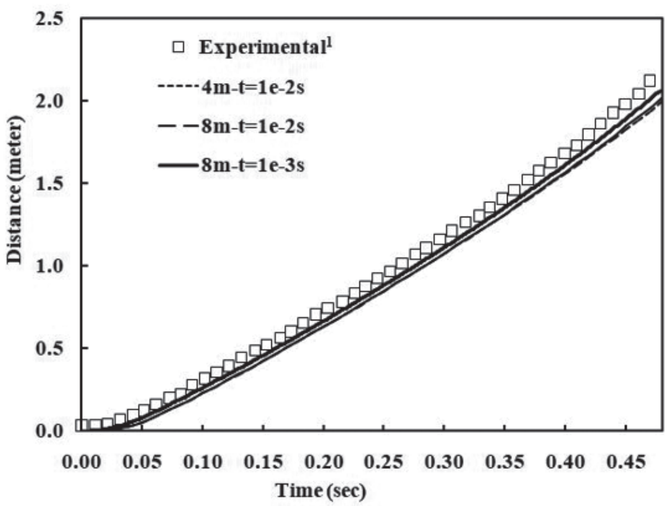


Figure 1. Trajectory along Z-direction (M=0.95).

3. TRAJECTORY ESTIMATION

The present effort is to estimate the position of bird at the time of collision with the duct and also to determine if the bird would pass through the entire stretch of the duct without colliding with the duct walls to reach the engine face directly. The duct shape was provided by airframe designer and forward portion of the airframe upstream of the intake was modelled for this simulation to account for boundary layer formation upstream of intake duct entry and study the trajectory of the object inside the duct. Based on the existing information available in open literature, simulation is proposed for 6-Degree of Freedom (6-DOF) along with chimera mesh (overset mesh) to track the trajectory of the bird. Figure 3 presents the domain to be discretised. The term ‘object’ in Fig. 2 represents the bird. It is placed exactly at the entrance of the S-type duct (Fig. 2 inset).

The closed domain is discretised with a background mesh with a free stream, zero velocity atmospheric condition. Then separate meshes are generated for the bird of given size as well as the aircraft with intake. This methodology of meshing is called overset meshing. The mesh around the bird is moved with bird through the duct during the course of simulation. There is a continuous transfer of information by linear interpolation between the overset and background mesh. This simulation stops once the bird either reached the exit or collided against the duct wall.

3.1 Computational Methodology

Once the meshing process is completed the next step is to setup the model to mimic the physics of the fluid flow. The fluid is considered to behave as an ideal gas and compressible. A standard wall function was enabled in the solver. The coupled flow solver is used to solve the governing equations using k ω-SST turbulence model[5] with 2nd order convection term. Generally, kω-SST is preferred for computations involving internal flows and as detailed in the literature[9] on the advantages of SST being robust; the same is made use of in this study. In order to ensure faster convergence without loss of accuracy the simulation is first performed as a steady state problem with 1000 iteration was used for stability. After the flow fully develops and convergence is reached, the transient

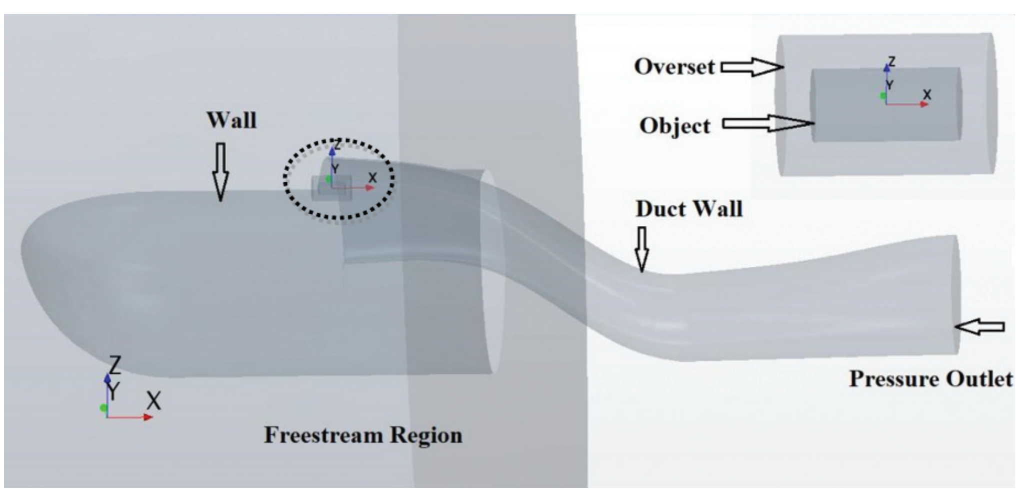


Figure 2. Air intake duct domain and overset region.

conditions are enabled to track the trajectory of the bird. The simulation was for M=0.3 and Reynolds number at exit was 4.7x 10^6. We were only tracking the trajectory of the object over time and pressure recovery at the end of impact.

The simulation was based on details provided in ref 1, 5 and 9 with respect to modelling and analysis of high speed flows. Details as to the equations and physics is already presented in this literature and hence not explained here.

The boundaries are marked in Fig. 2. Free-stream condition is applied to inlet boundary where the Mach number is defined as a boundary parameter. The outlet boundary of the fluid domain is pressure outlet, where static pressure along with static temperature is defined (Table 2).

Initial conditions such as velocity, temperature, density, are given based on the atmospheric condition and Mach number at that altitude[10]. The initial conditions are as shown in Table 3.

Table 2. Free-stream boundary values

Altitude (Km)	Mach number	Density (kg/m^3)	Pressure (Pa)	Temperature (k)	Velocity (m/s)
Sea level	0.3	1.225	101325	288.1	102.8

Table 3. Initial conditions

Weight (Bird mass)	0.907 kg
Moment of Inertia I_{xx}	8.073×10^{-4} kg – m^2
Moment of Inertia I_{yy}	2.44×10^{-3} kg – m^2
Moment of Inertia I_{zz}	2.44×10^{-3} kg – m^2
Bird initial orientation	0,45,-45,90 degrees
Aircraft Angle of attack	-20,0,20 degrees

3.1.1 Setting Solver Parameters

A linear interpolation scheme is used to interpolate between the background and overset grids. The overset mesh is assigned to a 6-DOF motion solver. As explained in the validation study, for the purpose of faster convergence without loss of accuracy, the simulation is first performed as a steady state problem. The steady state solver is enabled till all residuals reach a value of 1e^{-3} (non-dimensional) or less. The unsteady solver is then enabled till all residuals reached 1e^{-5} (non-dimensional) or less, and terminates once the bird hits the wall or leaves the duct domain. The simulation was performed on a Intel(R) core(TM) i7-6700 CPU @ 3.40GHz processor with installed memory of 64GB RAM. The time taken for running 1e^{-2} sec with 4 core license was 7 hours 26 minutes.

3.2 Initial Orientation of the Object (bird)

The orientations considered for the present set of simulation is as presented in Fig. 3.

0^o orientation: the angle between the bird's axis and intake central axis is zero degree.

45^o orientation: the angle of bird's axis with respect to intake central axis is 45 degree in clockwise direction.

45^o (negative) orientation: the angle of bird's axis with respect to intake central axis is 45 degree in counter clockwise direction.

90^o orientation: the angle of bird's axis with respect to intake central axis is 90 degree in clockwise direction.

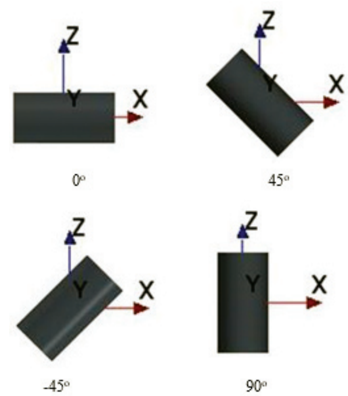


Figure 3. Initial orientation of the bird.

3.3 Angle of Attack of the Aircraft

The angle attack is the measure of angle between the fluid flow direction and intake central axis as shown in Fig. 4. It is assumed that an aircraft's angle of attack of ±20^o will approximately be the line of sight from duct inlet to exit, and at this angle the bird is expected to pass unhindered. Therefore, conditions of the angle of attack studied are 0 and ± 20 deg.

From the simulation, the bird's position, and orientation, were computed and plotted for initial conditions such as bird orientation and aircraft Angle of Attack. The length of travel and location of impact inside the duct helps in predicting the worst case scenario and vulnerability of the engine components.

4. RESULTS AND DISCUSSION

The various parameters that were varied for the present simulation is presented in Table 4. The positions of impact and the orientation of the bird in terms of Roll (X), Pitch (Y) and Yaw (Z) for the various cases studied is as tabulated and presented in Table 4.

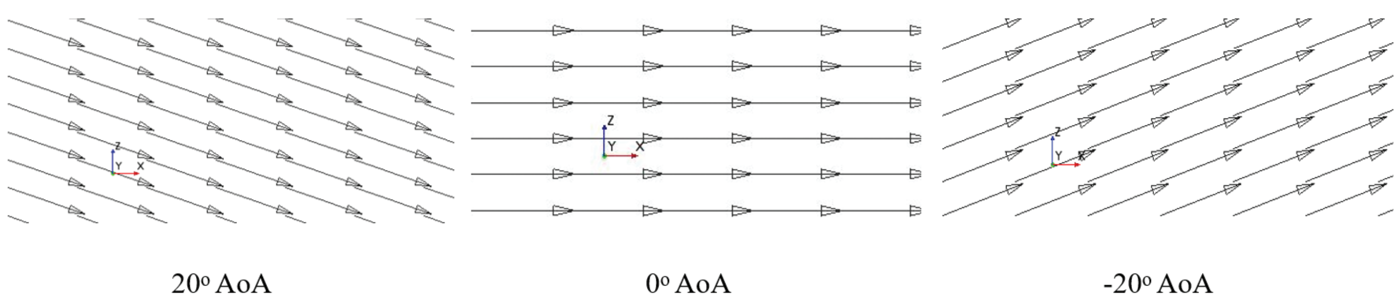


Figure 4. Angle of attack of the aircraft.

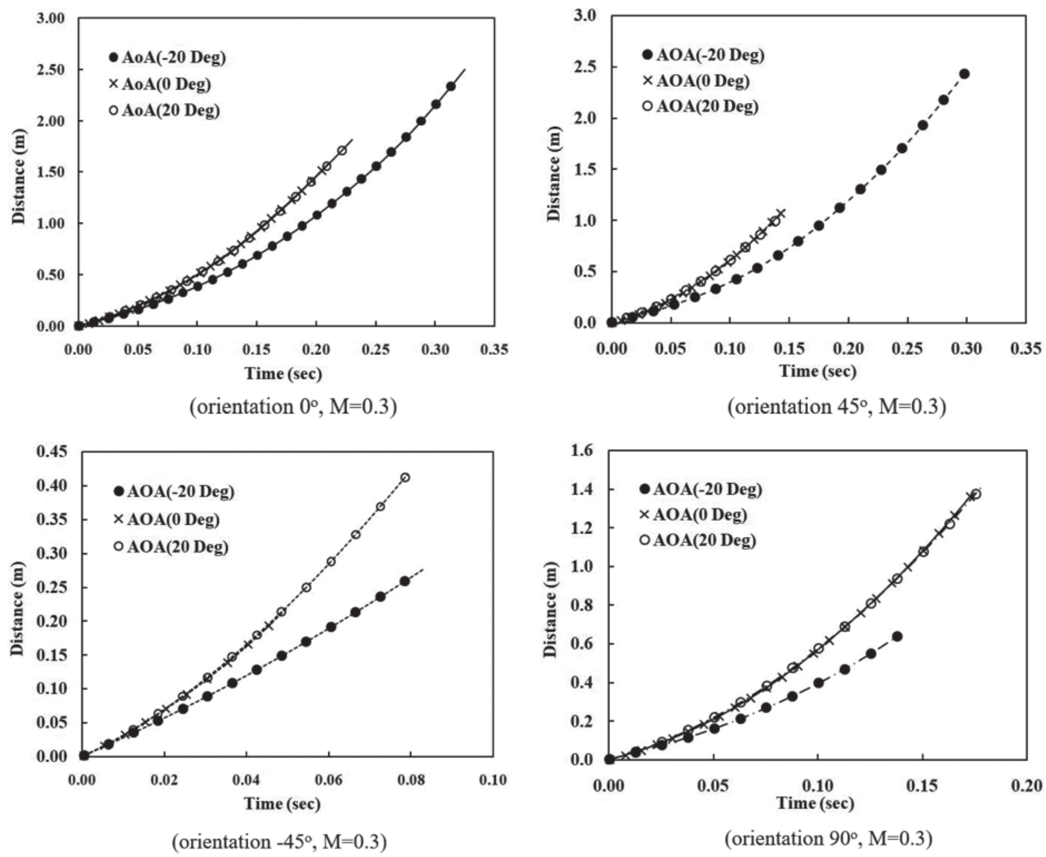
Table 4. Position and orientation of the bird at the time of impact (Mach 0.3)

Case	AoA (°)	IO (°)	Position of impact (m)				Angular orientation (deg)		
			Time (s)	(X)	(Y) 10^{-3}	(Z)	Roll (X)	Pitch (Y)	Yaw (Z)
1.1	-20	0	0.324	2.4980	15.7	-0.9392	11.20	91.76	-22.59
1.2	0	0	0.2095	1.5745	-1.3	-0.4696	-0.43	65.98	-0.33
1.3	20	0	0.230	1.8109	-1.1	-0.6123	-0.05	67.32	0.20
1.4	-20	45	0.3	2.4597	25.6	-1.0052	-34.14	61.89	-26.74
1.5	0	45	0.146	1.1051	-29.7	-0.1837	-3.48	86.31	2.02
1.6	20	45	0.1358	0.9976	-0.1	-0.1312	-7.65	83.82	-2.55
1.7	-20	-45	0.083	0.2766	-1.3	-0.0489	-0.55	6.94	0.16
1.8	0	-45	0.05	0.2207	-3.3	-0.0375	-0.51	-23.51	-0.23
1.9	20	-45	0.079	0.4158	-0.6	-0.0674	-0.34	-47.36	-0.01
1.10	-20	90	0.1385	0.6414	-18.9	-0.1134	4.04	46.95	1.11
1.11	0	90	0.1735	1.3688	2.8	-0.346	-1.86	8.20	-0.95
1.12	20	90	0.178	1.4050	-1.2	-0.3689	-1.05	5.58	1.1

4.1 Trajectory Characteristics

Table 4 presents the time taken for impact along with the distance travelled from its initial position till impact as well as its final angular orientation along X, Y, Z axis that is roll, pitch, yaw. From Table 4 it can be seen that the bird travels farthest when its initial orientation is at 0°. It can be seen from the Table 4 that the X direction trajectory and Y angular orientation has maximum variation due to the change in AoA & IO and hence only these are presented graphically. The effect of orientation and AoA on pressure recovery was affected and discussed in Fig. 8. The time mentioned here refers to the elapsed duration from start to impact. The CG of the object was traced over time to estimate the trajectory. The explanation for AoA -20° being distinct from other is explained with velocity vector plot in Fig. 7.

Figure 5 presents the X-direction distance and Fig. 6 presents the angular orientation along Y for all the cases (1.1 to 1.12). The vector plot with the object orientation at various times instants is plotted in Fig. 7 for cases 1.1 to 1.3.



(orientation 0°, M=0.3) (orientation 45°, M=0.3)

(orientation -45°, M=0.3) (orientation 90°, M=0.3)

Figure 5. Trajectory trace of bird's CG along the X-direction.

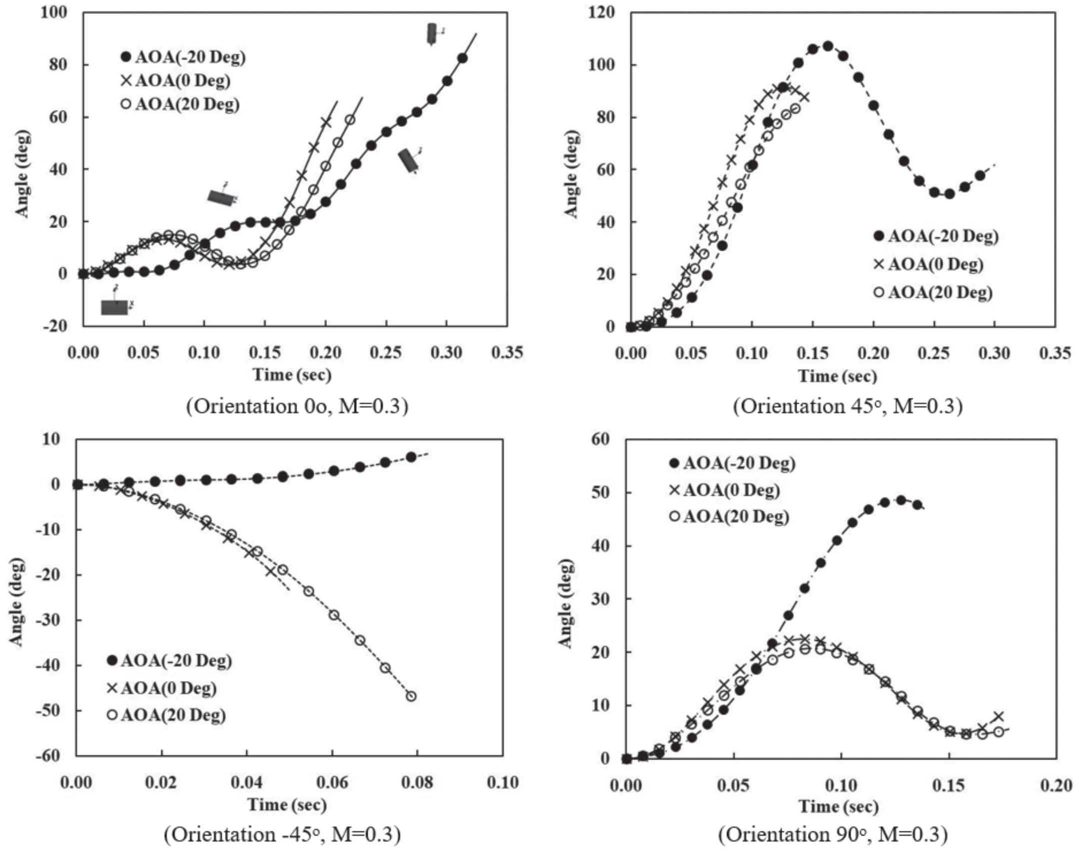


Figure 6. Trajectory trace of bird's Y-angular orientation.

Following are the observations from Fig. 5:

(a) The bird travels the farthest for AoA -20° for the IO of 0° and 45°. The distance travelled is 2.5 m and time taken is 0.32 s.

(b) It can also be seen that the X- direction trajectory for 0° and +20° have the same profile irrespective of the initial orientation of the object. AoA of -20° is seen to be distinct of all AoA's irrespective of the orientation and as discussed in Fig. 7.

(c) The object orientation of -45° is the best case scenario as it hits the duct within 0.07seconds having the shortest trajectory for all AoA's.

Following are the observations from Fig. 6:

(a) Similar to the observation made in Fig. 5, it is seen that the pitching of the object for the case AoA -20° is distinct from the other two angle of attacks namely 0 and +20° irrespective of the object orientation.

(b) The pitching for the two angle of attacks 0 and +20° have the same profile both qualitatively and almost quantitatively for all the four orientations.

(c) The pitching is gradual for the IO of -45° but exits only for a fraction of time.

(d) The IO of 0° and AoA of -20° is the least oscillatory with the object travelling the farthest.

The flow physics due to the interaction is complex in nature to quantify. However, from both Figs. 5 and 6 it can be concluded qualitatively that the effect of orientation is less predominant than angle of attack. Also the AoA -20° has a distinct profile. In order to understand this distinct feature, the vector plots at various time instants for the cases 1.1, 1.2 and 1.3 are extracted and presented in Fig. 7. The following are observed. The AoA+20° is more closely oriented to the duct profile and the flow can be seen to follow the duct profile by pushing the object slightly towards the upper portion of the duct. In the AoA 0° case, the flow sees a circular projected area of the object which in turn increases as the object yields to the flow path. This interaction causes the fluid to rotate the object and simultaneously pushes it to the duct wall. This is due to the fact that the object has not attained the flow velocity in the duct and there is a region of smaller velocity ahead of the upstream face of the object and is not in the same direction as the flow. This causes the object to deviate from the flow direction and the object invariably hits the wall. In the AoA -20° case, the flow tends to support the object keeping it in flotation as it moves thorough the given duct profile. This ensures that the object stays more or less at the center of the duct and travels longer. This is the reason for the AoA -20° being less oscillatory and travels the farthest with the pitching 'in line' with the duct profile. Thus whatever the initial orientation, the object tends to pitch and orient to the flow thus having less effect than AoA.

4.2 Variation in Engine Face Pressure Recovery Pattern for Various Orientations and Angle of Attack

Engine face pressure recovery was computed at the time of impact and presented as contour plot over the duct exit area. This was neither averaged over time nor was this tracked with time for the entire duration of simulation. As discussed earlier, the engine face pressure recovery is obtained for all the cases studied. As explained earlier, the design of the duct and the presence of the bird cause non-uniformity in the flow. This could affect the surge margin and lead to stall[7].

From Fig. 8 the pressure recovery is better for cases when the bird orientation is -45° (3rd Column) irrespective of AoA (cases 1.7, 1.8 and 1.9).The pressure recovery is affected for AoA -20° followed by 20° for all orientations as can be seen by observing rows 1 and 3 and the pressure recovery is least affected for AoA 0° (row 2). For the -20° and 20° AoA cases (rows 1 and 3) the 90° bird orientation has the worse recovery pattern with a very steep gradient.

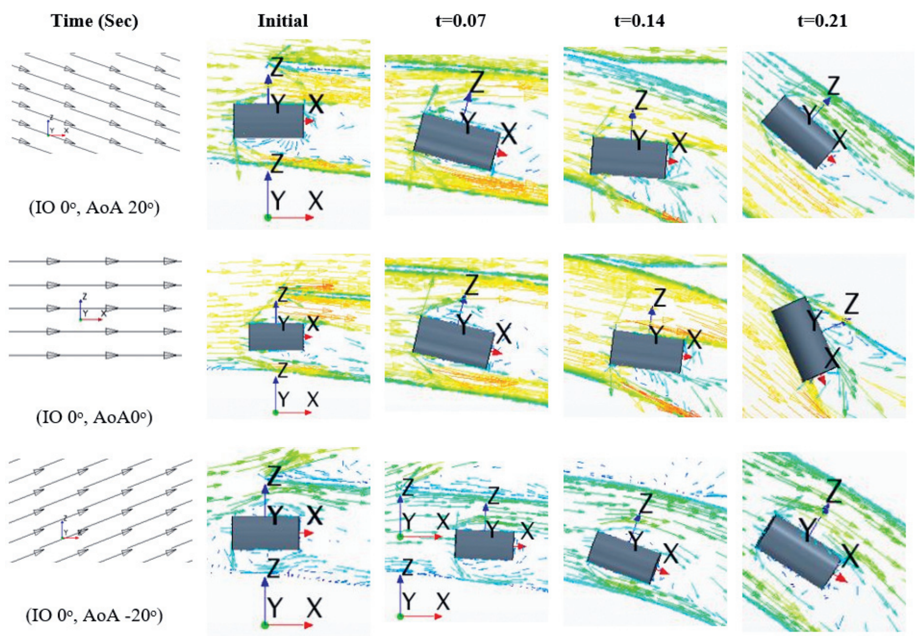


Figure 7. Vector plot at various time instants for the three AoA cases 1.1, 1.2 and 1.3.

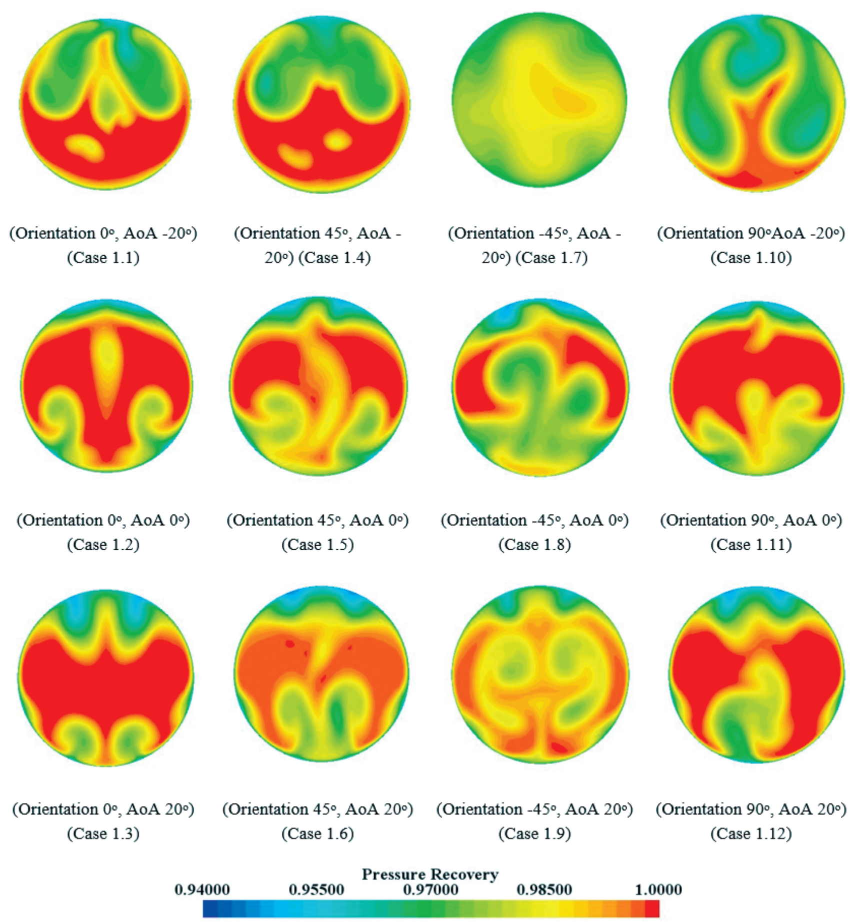


Figure 8. Engine face pressure recovery (M=0.3).

5. CONCLUSIONS

The bird's trajectory motion inside the duct was simulated using CFD and presented here for various entry conditions. For the given design of the intake duct, the possibility of the bird reaching the engine face is checked. In the present set of study it can be seen that irrespective of the object orientation, the AoA of -20^o has a distinct feature both in terms of linear trajectory as well as pitching. It was also observed that the effect of AoA is more predominant than the initial orientation. The interaction of AoA and duct shape will decide if the object will strike the engine face. Although the present work focuses primarily on the bird's trajectory, it is seen that the engine face pressure recovery is affected based on bird's orientation and AoA. This study can be further explored to understand the non-uniformity of flow due to bird ingestion in order to design a better serpentine duct that can be safe to engine components without loss in efficiency. It is expected, that due to the momentum, the bird might get fragmented before reaching the duct exit. Several possible operating conditions need to be simulated in order to optimise the duct shape for safe operation from bird hit. The effect of variation in bird mass and fragmentation due to collision also needs to be studied in future.

REFERENCES

1. Lijewski, L.E. & Suhs, N.E. Time-accurate computational fluid dynamics approach to transonic store separation trajectory prediction. *J. Aircraft*, 1994, **31**(4), 886-891.
doi: 10.2514/3.46575
2. Ubels, L.C.; Johnson, A.F.; Gallard, J.P. & Sunaric, M. Design and testing of a composite bird strike resistant leading edge. *In* 24th International SAMPE Europe conference of the Society for the Advancement of Materials and Process Engineering, Paris EXPO, Porte de Versailles, Paris, France, 1-3 April 2003, 1-14. https://reports.nlr.nl/bitstream/handle/10921/607/TP-2003-054.pdf?sequence=1&isAllowed=y [Accessed on 13 September 2019]
3. Nizampatnam, S.L. Models and methods for bird strike load prediction. Wichita State University, USA, 2007. (PhD Thesis).
4. Wellborn, R. S.; Reichert, A. B. & Okiishi, H. T. An experimental investigation of the flow in a diffusing S-duct. *In* 28th Joint Propulsion Conference and Exhibit cosponsored by the AIAA, SAE, ASME, and ASEE, July 6–8, 1992 AIAA 92–3622.
doi: 10.2514/6.1992-3622
5. Zhang, L.; Liu, Z. & Jiang, J. A numerical simulation of the flow in a diffusing S-duct inlet. *Modern Appl. Sci.*, **3**(4), 2009, 111-116.
doi: 10.5539/mas.v3n4p111
6. Kachele, T.; Schneider, T. & Niehuis, R. Steady and unsteady numerical simulation of a bent intake geometry. Springer International Publishing, AG 2018, 15-25.
doi: 10.1007/978-3-319-64519-32
7. Aref, P.; Ghoreyshi, M.; Jirasek, A. & Satchell, J.M. CFD validation and flow control of RAE-M2129 S-duct diffuser using CREATETM-AV Kestrel simulation tools. *Aerospace,* 2018, **31**(5), 1-23.
doi: 10.3390/aerospace5010031
8. Steger, J.L.; Dougherty, F.C. & Benek, J.A. A chimera grid scheme. Advances in grid generation, ASME FED **5**(1), June, 1983.
9. Menter, F.R.; Zonal two equation k-ω turbulence models for aerodynamic flows. *In* 24th Fluid Dynamics Conference*,* Orlando, Florida, July 6-9, 1993. AIAA 93-2906.
doi: 10.2514/6.1993-2906
10. Yahya, M.S. Gas tables for compressible flow calculations. 6th ed New Age International, India, 1981.

ACKNOWLEDGMENT

This work was supported by the Gas Turbine Research Establishment, Bengaluru-93. Funding reference GTRE/MMG/BMR1/1410/16/CARS/A/16.

CONTRIBUTORS

Mr L. Aravinth holds BE (Mechanical Engineering) from Anna University and currently pursuing his MS in the Department of Mechanical Engineering, REC. Currently working as a Junior Research Fellow with the Numerical Engineering and Simulation Lab, at the Rajalakshmi Engineering College, Chennai.
He was fully involved in literature review, discretisation of the domain, validation and simulation for all the parameters including analysis of results for the present study.

Mr N. Vidhyashankar is presently working as a Scientist in Compressor Group of DRDO-Gas Turbine Research Establishment, Bengaluru. His research interest include : Design of axial and centrifugal compressors, testing of compressors, design and analysis of intake ducts, aircraft-engine integration, fluid-structure interaction etc.
His contribution to the present study was to formulate the problem and establish the framework involving 6-DoF along with Overset Mesh for the prediction of trajectory prediction.

Mr Reza Abbas is presently working as a Scientist in Compressor Group of DRDO-Gas Turbine Research Establishment, Bengaluru and his research interest include design of axial and centrifugal compressors, manufacture, assembly and testing of compressors, complex CFD simulations that include various blade rows of LP and HP spools along with bypass, study of unsteady flows using CFD.
His contribution to the present study has been to validate the coupled flow solver for a standard NACA duct case and CFD studies to determine the variation of trajectory results to mesh and time step variation.

Dr Natteri M. Sudharsan PhD from Nanyang Technological University, Singapore and Post doctoral fellowship with Institute of High Performance Computing, A-Star, Singapore. Currently working as a Professor of Mechanical Engineering with Rajalakshmi Engineering College and heads the Numerical Engineering and Simulation Laboratory. His research interests are in computational heat and fluid flow, combustion modeling and medical diagnostic applications using CFD.
He provided the necessary expert guidance to the scholar in the present study.