16th Australasian Fluid Mechanics Conference
Crown Plaza, Gold Coast, Australia
2-7 December 2007

Validated CFD simulations of vortex formation in jet engine test cells

Ho Wei Hua and Mark Jermy*

Department of Mechanical Engineering

University of Canterbury

Private Bag 4800

Christchurch

New Zealand

*Corresponding author tel. +64 3 364 2987 x. 7390 fax +64 3 364 2078 mark.jermy@canterbury.ac.nz

Abstract

Vortices can be produced and ingested in to the intake of a jet engine during its operation. This can occur when the plane is on the runway during take-off, or during engine tests in a test cell. The vortex can throw debris into the intake or stall the compressor, causing severe damage to the engine. The runway problem is solved by keeping the runway clear of debris and scheduling the throttle appropriately. However vortices can still occur in test cells. To eliminate vortices at the design stage it is necessary to be able to predict the onset of the vortex. This paper seeks to use the commercial CFD code Fluent to investigate both the runway and test cell problem. The runway problem has been investigated in previous wind tunnel studies by other authors. These studies were recreated in a CFD simulation reported in detail elsewhere. The threshold conditions for vortex formation were located and the effects of suction tube diameter, shear in the test cell inlet, ground boundary layer thickness and suction inlet Reynolds number were investigated.

With the computational techniques thus validated, the study is extended to enclosed test cell geometries. The simulations show three stages of flow regime namely regular vortex, deformed vortex and no vortex. Vortices are not formed at cell bypass ratios greater than 50-70% and stable vortices are formed at cell bypass ratios less than 20-30%.

The vortex threshold is found to be lower than the threshold for suction over ground plane simulations on the Vi/Vo against H/Di graph, i.e. vortex formation occurs over a wider range of conditions when the flow is enclosed.

Introduction

Vortices can develop in the intakes of aero engines during high power operation near solid surfaces. This may occur during take-off or during test in a ground facility. The structure of the vortex is similar to the vortex seen in a bath. One end of the vortex is anchored on the nearby solid surface while the other enters the suction intake.

In a take-off like scenario, the threshold of vortex formation depends on the thrust of the engine, height of the engine above the ground, size of the engine diameter, upstream inlet velocity and gradient, and the ground boundary layer thickness.

In a test cell, there is a flow of excess air beyond that required by the engine, driven by entrainment by the exhaust plume. This flow passes between the engine and the internal walls of the cell. It is quantified by a cell bypass ratio (CBR):

$$CBR = 100\% \left(\frac{\dot{m}_{cell} - \dot{m}_{engine}}{\dot{m}_{cell}} \right)$$

Where $\dot{m}_{cell}$ is the air mass flow rate at the cell intake and $\dot{m}_{engine}$ is the air mass flow rate through the engine including the fan and core. The CBR is distinct from the engine bypass ratio, which is the ratio of the fan to core flow rate.

A commonly used rule of thumb is that a cell must have a bypass ratio of more than 80% to avoid vortex formation. Typically cells are designed with CBRs up to, and in some cases exceeding, 200%.

CFD simulations of intake vortices have been reported by [1, 2]. However, no studies to date have used numerical methods to predict the vortex formation threshold. We have successfully reproduced the experimentally measured threshold [3]. In this paper we briefly review the results of this earlier study of a take-off like scenario and then extend the investigation to an enclosed, test cell like scenario.

Vortex Formation

The vortex type concerned in both these studies is the type which concentrates ambient vorticity leading to a vortex with a single core. This study does not consider the types which do not require ambient vorticity and which manifest vortex systems with two or more cores.

In the formation of such vortices, there exists a blow-away velocity. The blow-away velocity is the threshold velocity of upstream air above which the vortex core is convected downstream and disconnected from the inlet. Conversely if the upstream air velocity is below the blow-away velocity a vortex may be formed, subject to other conditions being favourable. The blow away condition is expressed as the ratio of the inlet velocity Vi to the freestream velocity Vo. There is a minimum value of this ratio Vi/Vo below which the vortex will not form.

Equally, if the inlet is too far away from a solid surface, there will be no stagnation point (a point with a diverging velocity profile radially) on the surface and the vortex cannot form. In other words, the capture stream-tube (enclosing the air which enters the inlet) does not intersect with any solid surface. This condition is expressed as the ratio of the perpendicular distance from the inlet to the solid surface H to the inlet internal diameter Di. There is a maximum value of this ratio H/Di above which the vortex will not form.

The threshold values of these two ratios are interdependent and can be expressed as a line representing the locus of the threshold conditions for vortex formation on a plot with axes of Vi/Vo and H/Di such as Figure 1. The locus plotted is the average of many wind tunnel studies by different workers and is taken from Nakayama [4].

Vortex Formation in Take-Off Like Scenario

Vortex formation in take-off like scenarios was investigated, using commercial CFD package ANSYS Fluent, in an earlier study [3]

These simulations have the following geometric and solver characteristics:

- Suction Inlet Diameter (Di) – 0.75 and 1.75m
- Upstream Distance – 5 x Di
- Downstream Distance – 10 x Di
- Suction Inlet Centreline Height (H)
- Cross-section of Cell is Square
- Solver Regime – Incompressible SST-kω with default settings
- Mesh Density – 100 000 to 200 000 cells

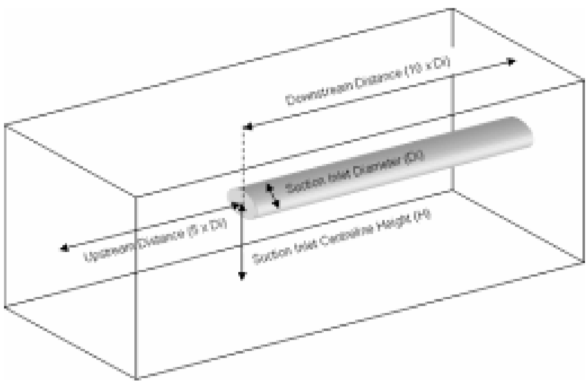

Figure 1: Geometry of Solution Space

The simulations have the following as boundary conditions in the model:

- Upstream – Velocity Inlet UDF
- Downstream – Pressure Outlet (zero gauge)
- Ground and Engine Wall– Wall
- Sides and Ceiling – Pressure Outlet (zero gauge)
- Engine Inlet – Pressure Outlet

The results of the simulations (vortex formation threshold) show a trend as shown in Figure 2 below.

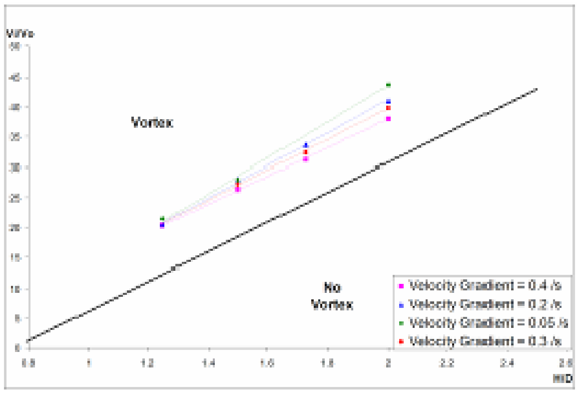

Figure 2: Generic Vortex Formation Threshold

The vortex threshold shifts with changes in the following parameters and their effects are highlighted in Table 1 and are described briefly below.

Table 1: Shifts in Vortex Formation Threshold

Parameter	Effects
Upstream Velocity Gradient (Increase)	More likely for vortex to form
Engine Inlet Diameter (Increase)	More likely for vortex to form
Ground Boundary Layer (Increase)	More likely for vortex to form

Increasing in the velocity gradient (shear) moved the formation threshold down the plot (i.e. to lower Vi/Vo, with a greater area of the plot falling in the vortex forming region). Similarly, increasing the thickness of the ground boundary layer increased the area of the plot falling in the vortex-forming region. Conversely, increasing the Reynolds number, based on the freestream velocity and the engine inlet diameter, reducing the area falling in the vortex-forming region.

The threshold derived from the CFD simulations had the same slope but a higher intercept than Nakayama's correlation. The reasons for this are under investigation and could be due to the presence of a pronounced ground boundary layer [3]. In addition the experimental results on which Nakayama's correlation is based show considerable scatter as they consist of a number of independent experiments, each with different inlet Reynolds number, boundary layer and velocity gradient.

Vortex Formation in Test Cell Like Structure

After the simulation for the take-off like scenario, walls were added to the sides and ceiling enclosing the engine in an open-end box in a test cell like structure.

The parameters that are interesting and are likely to affect the vortex formation threshold include the following:

1. The distance between the walls and the engine
2. The size of the engine inlet
3. The distance between the cell inlet and the engine inlet
4. The cell inlet velocity gradient
5. The cell inlet velocity

This paper addresses only point 4.

The vortex threshold is likely to have the same trends as that found in suction over a ground plane scenario shown in Figure 2.

Method

The simulations will be conducted, as in the take-off like scenario, using ANSYS Fluent. The geometry was meshed with Gambit 2.2.30 with tetrahedral/hybrid meshes throughout. A typical mesh is shown in Figure 3 below.

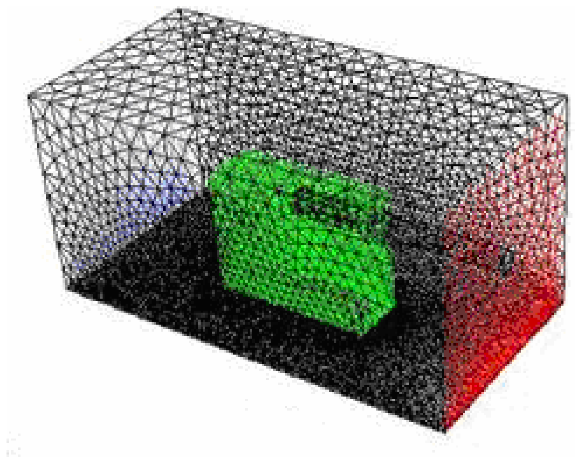

Figure 3: Typical Mesh for Suction in Box Model

The central region has a tighter mesh compared to the rest of the cell as this is where the vortex appears.

The eventual mesh has between 100 000 to 200 000 cells and was solved with an incompressible flow regime. Velocity boundaries are not recommended for compressible flow and hence control of the velocity gradient is very difficult leading to the use of the incompressible flow regime. Compressibility is only significant in the suction inlet.

The boundary conditions for the model are as follows:

Cell Inlet – Velocity Inlet UDF defining a linear velocity profile of 0.2/s

Engine Inlet - Outflow with Flow Rate Rating of 1

Cell Outlet – Outflow with Flow Rate Rating set to achieve the desired Cell Bypass Ratio

Cell and Engine Walls – No-slip walls (zero velocity on the surface)

The solution was initialised from the Cell Inlet plane.

Compressible vs. Incompressible Flow

Because of the difficulty in setting simple linear velocity gradient boundary conditions in compressible simulations, a study was conducted to investigate the difference between both flow regimes and determine if an incompressible solution was adequate. A compressible flow solution was first obtained using pressure inlet as the cell inlet boundary condition and the resultant velocity profile at the cell inlet is extracted and used as the velocity inlet boundary condition for the cell inlet in the incompressible flow solution. The results show that the position of a vortex (in a vortex producing case) is very similar. With the same inlet velocity profiles and cell bypass the results show the same phenomenon (vortex or no vortex), but a comparison of vortex threshold was not conducted because of the difficulty in controlling the resultant inlet velocity gradient in compressible solution and the inlet velocity gradient is likely to be a factor in determining the vortex threshold.

The pressure contours showing the location of the vortex is shown in Figure 4 below.

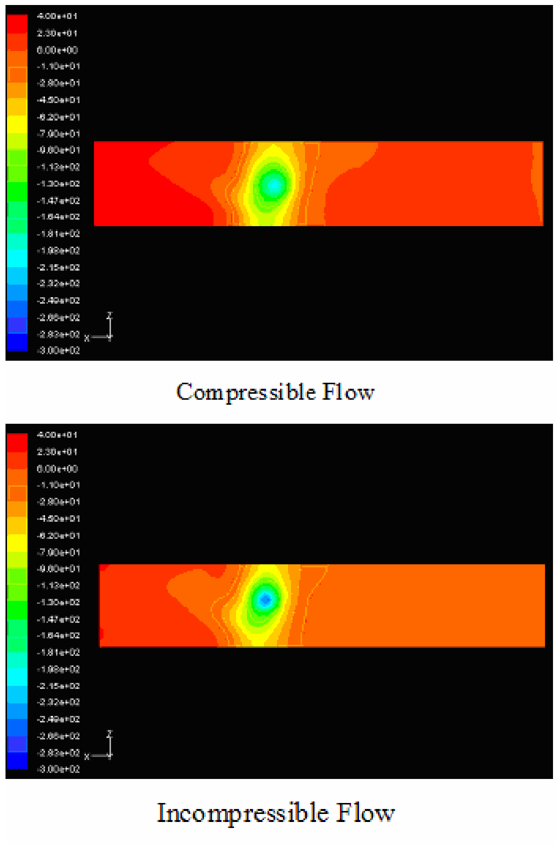

Figure 4: Compressible vs Incompressible Flow (Pressure Contours)

Turbulence Model

Turbulence was modelled with the SST k-ω scheme. This scheme was chosen as combining the best features of the k-ε scheme in free flows and the standard k-ω scheme in near wall flows, yet avoiding the computational expense of the Reynolds stress models. The difference in the two models has been investigated by Jermy and Ho [3] and the SST-kω turbulence model produced similar results to the RSM model.

Mesh Convergence

A mesh convergence test was conducted and the eventual mesh (after mesh independence was achieved) used had a mesh size as follows (the description are those as used in Gambit):

Ground (Green Zone) – 0.1m Quad

Ground (Rest of Cell) – 0.2m Tri

Cell (Green Region) – 0.35 – 0.5m Tetrahedral

Cell (Rest of Cell) – 1m Tetrahedral

Where the suction inlet diameter takes a value of 1m

Results

The solution schemes used were as follows:

- Pressure Based Solver
- First Order Discretisation scheme

The solution was initialised from cell inlet at the start of every solution to prevent a numerical equivalent of the "hysteresis effect" observed by Ridder and Samuelsson [5] in their experiments.

All other solver settings were left as default values.

For a single vortex to be formed, a slightly lower velocity region near the one of the horizontal surface (i.e. ceiling of floor) is necessary to create a preference for the vortex to be formed from one of the surface. In J or U-type test cells, which have a vertical inlet and horizontal test chamber, a thick boundary layer typically forms from the junction of these two sections, extending over the ceiling. Any vortex observed in such a test cell will be formed on the ceiling. Two single core vortices from both the ceiling and floor of the cells are shown in Figure 5 below.

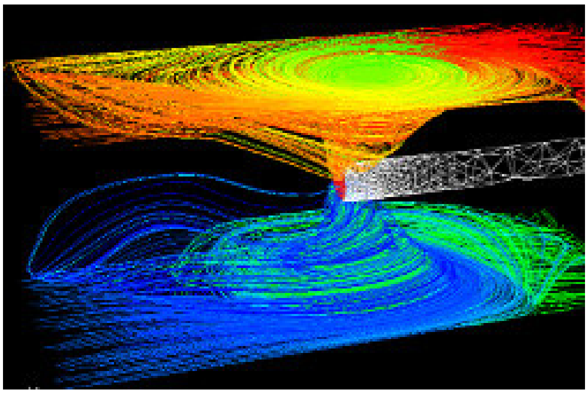

Figure 5: Two Single Core Vortex

Stages of Vortex Formation

In the suction over ground plane scenario, there are only two types of flow observed i.e. vortex formed or no vortex formed. Also the vortex always formed directly below the suction inlet, for the range of conditions tested in [3]. The size of the vortex remained the same and the shape was always circular.

However in this suction inlet in a box scenario, the presence of the walls has an effect on the vortex introducing a new regime of flow. In addition to the circular vortex and no vortex regimes, a third regime lying between the two is characterised by a deformation of the vortex from it's "perfect circular" shape and moving of the vortex away from it's "central, below suction inlet" location. In some cases this regime is characterised by the unsteadiness of the vortex.

It should be noted that stable, steady vortices have been observed in scale model experiments, and unstable, unsteady vortices have been observed in real test cells. In this case, the vortex core precesses, and the vortex disappears and reforms aperiodically.

For discussion purposes, the three stages shall be called the following:

1. Stage 1 – Regular Vortex
2. Stage 2 – Deformed Vortex
3. Stage 3 – No Vortex

An illustration of all three regimes are shown in Figure 6 and Figure 7 below

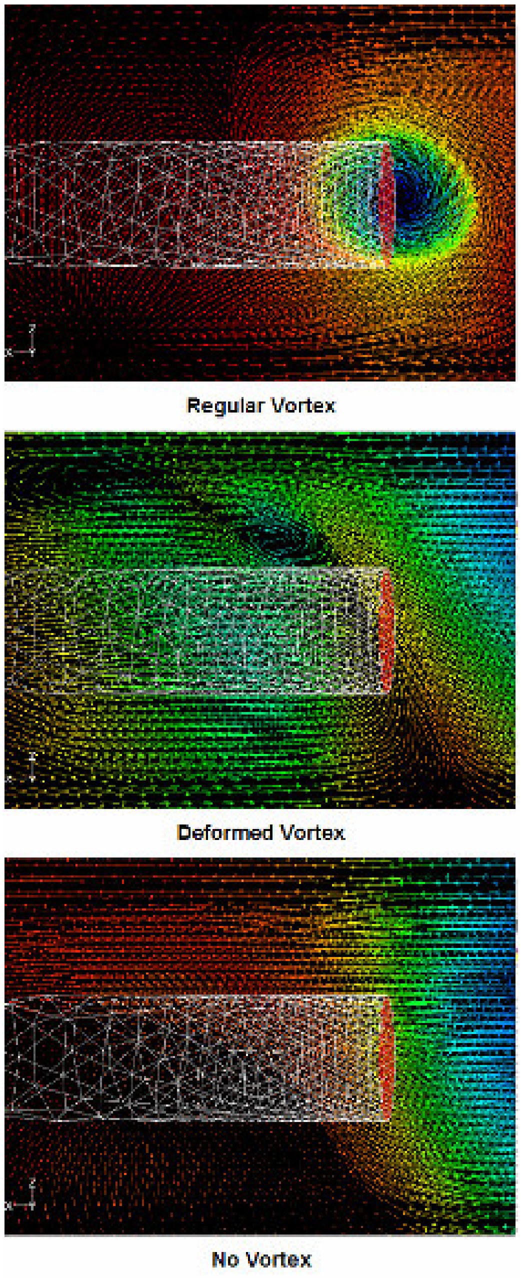

Figure 6: Vector Plot Showing Stages of Vortex

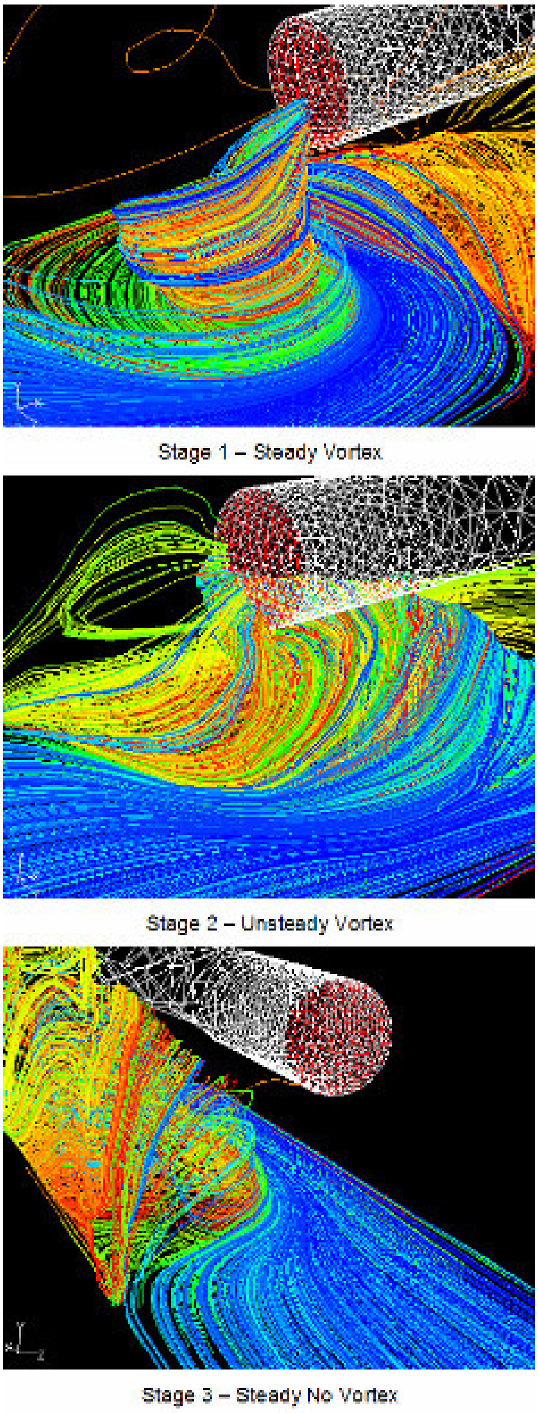

Figure 7: Pathlines Showing Stages of Vortex

Stage 1 – Regular Vortex

This stage occurs at low cell bypass ratios. The characteristic of the vortex at this stage is very similar to the one seen in suction over a ground plane simulation.

The vortex is circular in shape and occurs directly under the suction inlet. The vortex at this stage is easily identifiable in velocity vectors on the ground as well as pathlines.

The vortex in this stage is steady and the core does not move at all.

Stage 2 – Deformed Vortex

This stage occurs at higher cell bypass ratios than in stage 1. The vortex of this stage is irregular. It is elongated in shape and the core is located away from the bottom of the suction inlet, in contrast to the stage 1 vortex. An unsteady calculation of one case was performed. In this the vortex was seen to be unsteady and the core moves around the floor of the cell. The unsteady motion is shown in Figure 8 on a plane 0.5m above the floor.

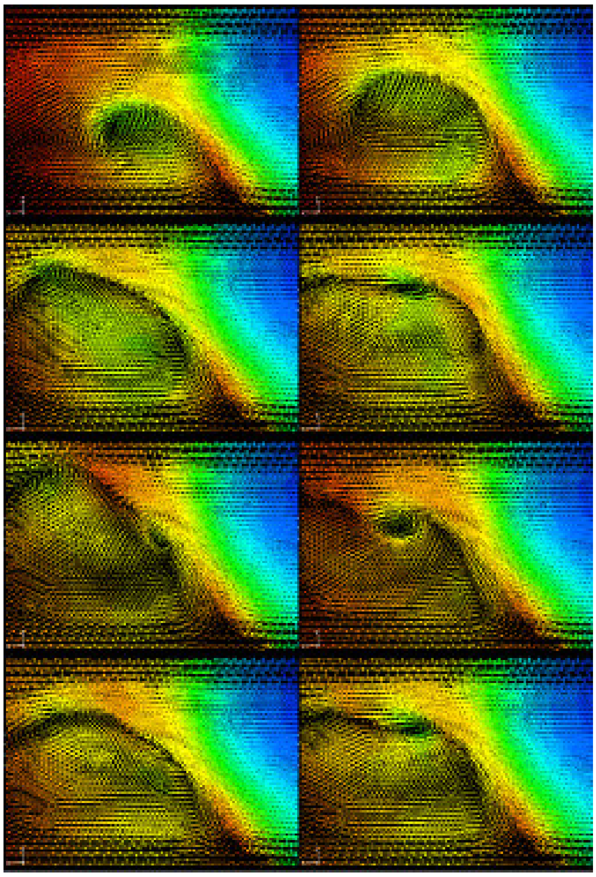

Figure 8: Unsteady Vector Plot (From Top Left to Bottom Right) – Every 2s

Stage 3 –No Vortex

This stage occurs at even higher cell bypass ratio as compared to stage 2. This stage is defined such that no pathline entering the suction inlet has rotation.

Vortex Formation Threshold

The vortex formation threshold is defined as the value of Vi/Vo (ratio of the average velocity at the engine inlet to the average velocity at the cell inlet) at which the vortex appears or disappears.

The model was solved with cell bypass ratio increasing in steps of 5% until the threshold for the three stages are found. This translates into an uncertainty in Vi/Vo of not more than ±2.6%

The vortex formation threshold for a cell inlet velocity with velocity gradient of 0.2/s is shown in Figure 9 and Figure 10. In Figure 10, the dashed line shows the threshold from the suction over ground plane simulations.

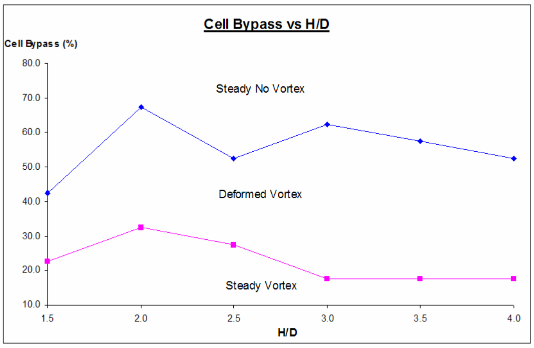

Figure 9: Vortex Formation Threshold (Cell Bypass Ratio)

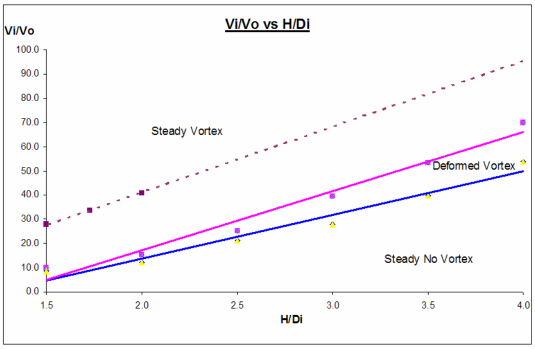

Figure 10: Vortex Formation Threshold (Vi/Vo) – Linear Fit

Discussion

Figure 9 shows that the no vortex condition requires a cell bypass ratio of more than 50-70% (the value varying with H/Di), justifying the rule of thumb used in test cell design that a cell bypass ratio of more than 80% must be used to prevent vortex formation. Below the no-vortex region, there is a wide band of CBR at which an unsteady, unstable vortex is seen, equivalent to the unsteady, inconstant vortices observed in some real test cells. At CBRs of less than 20-30% a stable vortex is seen in the calculations.

A comparison between the vortex threshold between the suction over ground plane and suction in box simulation shows that vortices are more likely to form in the latter, i.e. the vortex forming region covers a greater range of conditions.

The threshold for vortex formation predicted shows the following trends

1. Vortices form when the upstream velocity is low and are blown downstream and vanish as upstream velocity increases above the "blow away" velocity.
2. On a Vi/Vo against H/Di plot (Figure 10) the threshold for vortex formation shows a positive gradient i.e. as the height of the suction inlet increases, the blow-away velocity decreases.
3. A low velocity region near one of the horizontal surfaces is necessary to create a preference for a vortex to be formed from that surface. Otherwise, two vortices will be formed from both surfaces.

Trends 1 and 2 agrees with previous experimental data by various authors (Nakayama and Jones [4], Liu et al. [6] and Shin et al. [7]) and numerical data by Jermy and Ho [3] on a suction inlet over ground plane model.

Conclusions

The scenario of a suction inlet in a box, resembling a test cell configuration, has been studied and the threshold for vortex formation extracted.

Three cases of vortex formation are observed: no vortex, an unsteady, unstable deformed vortex, and a stable regular vortex.

In agreement with previous studies, the vortex threshold has a positive gradient when plotted on a Vi/Vo against H/Di graph. The threshold cell bypass ratio is relatively constant with H/Di,. No vortex is formed with cell bypass ratios greater than 50-70%, and stable vortices are formed at cell bypass ratios less than 20-30%. The exact value of the cell bypass ratio threshold varies with H/Di.

The vortex threshold is found to be lower than the threshold for suction over ground plane simulations on the Vi/Vo against H/Di graph indicating that vortices are formed over a greater range of conditions when the flow is enclosed.

Further investigation on the effects of cell inlet velocity gradient and other parameters has on the threshold, as detailed above, will be carried out.

Acknowledgement

We would like to thank CENCO Inc., SAFRAN Group for financial support and technical discussions, and Terry Clark and Paul Flynn for useful discussions.

---

[1]Karlsson A. Fuchs L., "Time evolution of the vortex between an air inlet and the ground", AIAA paper 2000-0990 (2000)
[2]Secareanu A., Morioanu D., Karlsson A. and Fuchs L., "Experimental and numerical study of ground vortex interaction in an air-intake", AIAA paper 2005-1206 (2005)
[3]Jermy, M. and Ho, W.H., manuscript submitted
[4]Nakayama A., Jones J.R., "Vortex formation in inlet flow near a wall", 24th Aerospace and Sciences Meeting and Exhibit, AIAA 96-0803 (1996)
[5]Ridder S.O., Samuelsson I., "An experimental study of strength and existence of domain of ground-to-air inlet vortices by ground board static pressure measurements", Stockholm Royal Institute of Technology, KTH AERO TN 62
[6]Liu W., Greitzer E.M. and Tan C.S., "Surface static pressure in an inlet vortex flow field", ASME Journal of Engineering for Gas Turbines and Power, 107, pp 387—393 (1985)
[7]Shin H.W., Cheng W.K., Greitzer E.M., Tan C.S., and Shippee C.L., "Circulation measurements and vortical structure in an inlet-vortex flow field", J. Fluid Mech. 162 pp 463-487 (1986)