Parametric Optimization of the Aerodynamic Shape of an Aircraft Engine Axial Entry Fan Blade

Prashant Sethi  
*Master Thesis Intern*  
*Altair Engineering Hellenic*  
*Balkan Center, Building C*  
*Thessaloniki, Greece*  
*theseth.prashant@gmail.com*

**Abbreviations:**  
Bpr - Bypass Ratio

**Keywords:** Aerodynamic, CFD, Optimization, Morphing

**Abstract**

Turbofan engines are the most common propulsive devices used in model aviation. A turbofan engine has a gas turbine driven core which drives a large diameter fan at the front of the engine. Thrust produced in any engine is directly proportional to the mass flow rate through the engine. A significant proportion of the air that enters a turbofan engine through the fan does not enter the core, and instead flows in the annulus between the core housing and the engine casing. The proportion of this volume to the volume that enters the core is called bypass ratio. Bypass ratio of aircraft engines has been on constant rise ever since their inception because of their higher specific thrust. While engine designers are still working towards achieving higher bypass ratios, it is a challenging task because of the (centrifugal) loads that accompany running such engines of large diameter at high speeds. Thus another way of maximizing thrust with a constant bypass ratio is to aerodynamically optimize the shape of the fan blade row to achieve higher mass flow through the engine. This study attempts to achieve such an optimization by integrating a CFD solver *AcuSolve* with an optimizer program *Hyperstudy*. The original baseline geometry is analyzed to begin with and the results are saved as benchmark. These results are then compared to the results from various different shapes of the fan blade with some variation from the original shape. For the purpose of creating the new shapes, instead of remeshing the modified geometry, the original mesh used for the baseline shape analysis is ‘morphed’ using certain shape variables as parameters of morphing. The shape variables used are rotation and translation of the leading edge of the blade. This saves considerable amount of time which will otherwise be spent in creating the different shapes for analysis, and meshing each shape independently. Morphing also preserves the mesh topology thus reducing any variations in result of the different cases due to mesh noise. An experiment design with both the shape variables as input parameters of the experiment is set up and the various shapes are analyzed according to this setup. The case which shows the best improvement in mass flow rate, which is the response parameter for this optimization study, is presented and discussed.

**INTRODUCTION**

*Modern turbofan engines*

Modern turbofan engines are the most popular engines for commercial civil aircraft propulsion. A turbofan engine is an air breathing jet engine, where a fan driven by a turbine pushes the pressurized air through the rear of the engine thus generating forward thrust which propels the aircraft. The diameter of the fan is larger than the diameter of the ‘engine core’. The engine core is the actual power plant of the engine which houses the compressor-turbine arrangement generating the power to drive the engine and the aircraft peripherals. Thus a certain percentage of air which enters the fan passes through the engine core while the rest of the air bypasses the core. The air passing through the core is compressed in the compressor, mixed with fuel and burnt in the combustor, and this hot mixture is then expanded in the turbine to generate the power. The ratio of the two volumes of air (bypass/core) is called the bypass ratio of the engine. Figure shows schematics of a low-bypass and a high-bypass turbofan.

The total thrust generated by a turbofan engine is the sum of two thrust components – one produced by the acceleration of the bypass air flow, and the second is the thrust generated by the engine core. The flow on the outside, or the bypass flow, is responsible for a major fraction of the thrust generated by a turbofan. The total thrust produced by the engine is directly proportional to the mass flow of the air through the engine.
Simulate to Innovate

Thus, higher thrusts can be obtained from the engine if a higher mass flow through the fan can be achieved.

*Figure 1: (a) High and low bypass turbofan, (b) Various optimization approaches and their application to design processes*

In the current optimization study it has been attempted to achieve such a result by aerodynamically optimizing the blade shape of a fan used in a turbofan engine. The objective is to arrive at a shape which is aerodynamically superior or at least at par with the original shape, and also facilitates a higher mass flow rate through the fan. Computational Fluid Dynamics (CFD) analyses are used to generate the results for comparison. Optimization is carried out using a parametric study where the shape of the blade in the given fan model (baseline shape) is modified parametrically. Two parameters are used to define the modified shape of the blade – the rotation and translation of the leading edge of the blade. For carrying out a CFD study, the geometry must be discretized into a fine grid called the mesh. Mesh morphing technique is used to morph (or adapt), the mesh created for the baseline shape to the modified shapes of the blades. Using morphing, an existing mesh can be morphed to fit a modified geometry without having to remesh the new geometry from scratch. This can save a lot of time otherwise spent in remeshing and thus the saved time can be used to test more number of setups for comparison.

**Optimization using CFD**

The CFD (and computational analysis methods in general) has reached a stage where it enjoys the confidence of engineers in the results it produces, and also the manner in which it produces these results. With high computing power available cheaply now, and with possibility of further augmenting such with parallelization of processing power across a number of dedicated computing nodes, engineers can quickly solve a large number of complex flow problems in a comparatively very short time as was possible few years ago. And with the constant advancement in both the available algorithms and computing power, the CFD techniques continue to be applied to more and more challenging cases, with complex flows inside complex geometries being one of the drivers of this research.

CFD tools can thus be used to generate a large number of parametric results quickly without needing the large number of physical prototypes. The advantages of CAD and CFD together in terms of amount of time, money and energy utilized to generate an equal number of virtual prototypes and test them is significant. These advantages offered by CFD have caught the attention of designers and researchers alike and there are a number of optimization models available now to the users. Typically these optimization models will work alongside the traditional CFD solver, generating results from various prototypes according to a preset arrangement decided by the user. Thus at the end when the optimization model has finished running, the user will have a set of results to analyze and compare and thus make an informed decision regarding the best design for the application under consideration.

In this study a partial parametric optimization approach has been used to arrive at the optimized shape for the test object. In such an approach, an existing or baseline design, is optimized to the output requirements for a new application by controlling only a few design parameters, while keeping the rest of the parameters unchanged or fixed.
Mesh Morphing

When CFD is used as a design tool, the process starts with a conceptual design which then undergoes a series of analyses and improvements before being deemed fit for use. What this means is that CFD will not be used just once in the design process, but every time a design is changed. In the traditional CFD process this means that for a component with complex geometry, for example a turbomachinery blade, the mesh will have to be generated again and again every time a change in the geometry is done.

A case in consideration here is when a design is being using one of the optimization algorithms discussed in the last topic. A parametric optimization algorithm might have parameters which define the geometry of the design being tested. This is also the case with the current study, where the optimization is being done for the shape of the fan. As different values of such parameters will be realized during the process, each case will have a different geometry thus requiring a mesh to be generated separately for itself. Since in case of such an optimization process, the total number of individual cases may be huge, the need to remesh the geometry every time can be inconvenient and a bottleneck process in the chain.

This is where the mesh morphing technique comes to the rescue, which as its name implies, is capable of ‘morphing’ an existing mesh from one geometry to another, to adapt it to the new geometry. An important characteristic of morphing techniques is that it does not create any new nodes. The existing nodes in a mesh are shifted in space (translated or rotated) as per the requirements of the new shape, preserving the mesh connectivity information. An illustration of how this may be achieved is shown in Figure. This has two important implications when morphing is being used as part of an optimization algorithm.

Since there are no connectivity or topology changes during a morph process, the new shape of the mesh can be defined as a function of independent parameters, usually called the shape parameters. These shape parameters can have different values and can be thus used to create different shapes of the existing mesh just by doing some transformations according to the morphing function with the parameters plugged in. Thus instead of having to reload the mesh for each case, only a single mesh with some additional parametric information is needed to run all the possible cases. This is a big save of time, and memory. The second implication lies in the effect of mesh on the convergence. A remesh of the complete geometry from scratch will most likely have some variation from the original mesh with regards to node connectivity. Thus when the results are analyzed, the variation seen in the results of the mesh of original design and the mesh of modified design can be attributed to two factors – a certain fraction to the change in shape, but also an undesired variation due to the difference in mesh. By preserving the topology information, morphing can minimize the undesired effect of a complete remesh.

MESHING

Test Object

The test object for this thesis project is an axial entry aircraft engine fan, being developed by General Electric Aviation. The fan is part of the GENX – n series engines. However, the exact make and model of the engine were not disclosed due to confidential reasons. The fan model was provided as a Solidworks CAD model file which can be imported in Hypermesh.

A hexahedral mesh is used to mesh the geometry because of the certain advantages such a mesh offers over the unstructured mesh.

- Less number of elements for a given number of nodes
- A hexahedral mesh arrangement can be used to optimize mesh density by reducing number of elements in streamwise direction.
- A better boundary layer resolution can be obtained with a hexahedral mesh.
- Information flow between nodes is ‘smoother’ because the arrangement is regular.

However structured meshes do not come without problems. Creating a structured mesh requires much more human effort and time. Even with a simple geometry, the process is not really straightforward, and can involve
considerably increasing user interaction as the geometries become more complex. It is often required to break a complex 3D geometry to smaller logical volumes, the sum of which makes up the whole geometry, and each of these individual volumes must be mappable volumes. In a mappable volume, a surface mesh existing on one face of the volume can be swept, or extruded, to the opposite face, along a logical direction, to obtain a regular solid map mesh.

The map volume meshing scheme can only be applied to volumes that can be meshed such that the mesh represents a logical cube. To represent a logical cube, a volume mesh must satisfy the following general requirements:

- There must exist exactly eight mesh nodes that are attached to only three mesh element faces. (These eight mesh nodes comprise the corners of the logical cube.)
- Each of the eight corner mesh nodes must be connected to three other corner mesh nodes by means of a straight chain of mesh edges—that is, a chain of mesh edges all of which belong to a single logical row of mesh nodes.

According to the criteria described above, the most basic form of a mappable volume is a rectangular brick. For such a volume, the mesh nodes located at the corner vertices of the brick constitute the corners of the mesh cube.

**Mesh Generation for the provided test object**

**Hypermesh** is the software that has been used for mesh generation for this study. It is capable of checking a volume for mappability, and efficiently generating solid regular hexahedral meshes by surface sweeping. As mentioned before, Hypermesh can also import the CAD model format in which the model was provided. The CAD model provided was a solid model of the fan. However, the region of interest for this study is the flow passages between the fan blades i.e. where the fluid flows through the fan. The flow passages are concentric and are assumed to be similar throughout the circumference of the fan. Thus, instead of modeling the flow through the whole fan, only one passage section of the fan is considered for analysis, and the results are assumed to be similar through the rest of the passages.

The solid component is deleted before proceeding, however preserving the surface information. The surfaces of the blades and the hub (or axis) surface in the chosen passage also become the bounding surfaces of the fluid passing through the passage. Inlet and outlet volumes are then added to keep the inlet and outlet far away from the actual blade passage. This helps mitigate the influence of the applied inlet and outlet boundary conditions on the result of the analysis. By taking away the perturbations caused by the boundaries further from the region of interest, this helps ensure with maximum accuracy that the results obtained in the region of interest are representative of the actual conditions.

Since it is being aimed to generate a regular hexahedral volume mesh for this analysis, all the individual volumes to be meshed should be mappable volumes. The geometry for this study was not found to be mappable as a whole hence was split into bits of smaller volumes, such that the sum of these individual bits makes up the whole volume. This process must be repeated until each small volume is mappable individually. For this model, this splitting was done along the span of the blade.
Once all the volumes have been verified to be mappable, all the preparations for the actual meshing have been done and the meshing process can be started. Each of these small volumes can now be meshed independently using the sweeping process to generate the volume mesh. This is done using the 3D solid map meshing tool in Hypermesh after surface meshes have been generated on the surfaces of the volumes under meshing. It is also important to ensure the continuous mesh connectivity between all these individual meshes. Mesh connectivity is important to ensure that the information is passed from each node to all the connected nodes properly during solving the governing equations over the grid. This is done by equivalencing the nodes on the coincident faces such that no duplicate nodes or element exist. Care is taken to introduce boundary layers wherever required. In this study the critical wall surfaces where boundary layer is needed are the surfaces of the blade.

*Figure 3: (a) Modeled Flow Domain (b) Domain split in mappable volumes (c) Meshed domain*

**Mesh Statistics**

The following are the details of the mesh count and quality statistics.

- Total element count – 838200
- Total node count – 807576
- Total surface elements – 61950
- Total solid elements – 776250
- Minimum Jacobian – 0.44
- Maximum skew – 78 degrees
- Maximum interior angle of quad faces – 168 degrees
- Minimum interior angle of quad faces – 12 degrees
- Maximum aspect ratio – 2755
- Maximum aspect ratio in the passage elements – 38
- Minimum Jacobian in 2D elements – 0.46
- Maximum skew – 78 degrees
BASELINE CFD SETUP

The baseline configuration in this study is the blade shape of the provided model of the fan. In this section the setup used for CFD analysis applied to the baseline configuration has been specified. Acusolve is used for the setting up and running the analysis. A number of different trials were done with varying setup parameters before finalizing this setup to be used for further analyses. In the subsequent analyses, the CFD setup used remains the same. Only the shape of the model is changed for each step of optimization, so that the changes can be attributed only to the change in shape, and not any change in setup. The setup is also illustrated in the Fig.

- Type of analysis – Steady State
- Convergence level for residuals – 1e-4
- Under relaxation factor – 0.5
- Temperature equation – Off
- Turbulence model – One equation Spalart-Allmaras model
- Material model – Constant density air, standard atmospheric air density – 1.225 kg/m^3
- Rotational reference frame, rotation speed – 1080 min⁻¹
- Upstream and downstream volumes – stationary volumes.
- Passage volume – rotating volume with the reference frame applied.
- Inlet – atmospheric stagnation pressure inflow with no swirl
- Outlet – standard atmospheric pressure outflow
- Blade surfaces, hub – no slip walls, rotating
- Periodicity applied on periodic surfaces which are coincident with other passages

Results and Discussions – Baseline Shape

With the setup defined in the previous section, the baseline model of the fan solved with residual ratios reaching convergence level in 148 iterations. The solution was completed in a little under eight hours while running on a high performance node with twenty processing cores and 64 GB RAM. The run time for a model is important to be optimized because a large number of runs will be performed on a range of different shapes for selecting the optimized shape among them. As can be seen in Fig, which is the plot of the residual ratios
with respect to the iteration steps, the convergence is smooth with residual ratios falling smoothly until below the desired convergence value.

*Figure 5: (a) Residual ratios plot for the baseline shape analysis (b) Plot showing the total pressure rise*

The pressure and velocity values were evaluated at different sections in the flow region. Some of the contours obtained from post-processing the results are shown below. The mean total pressure rise in the air as the flow passes through the fan is observed to be **747.5 pascal**. This total pressure rise in the flow is later expanded in the exhaust nozzle to atmospheric pressure, and hence is a critical parameter in the thrust generation. The pressure difference between the two sides of the blade, namely the pressure side and the suction side was found to be **454.4 pascal**. This is an important parameter as it is directly related to the loading on the blade. And finally, the mass flow through the fan was observed to be **0.8837 kg/s**. Mass flow is the parameter of interest in this study as the fan blade is being optimized for maximizing the mass flow.

*Figure 6: (a) Total and (b) static pressure contours before and after the blade row (flow is from left to right), (c) Blade pressure side and (d) Blade suction side pressure contours*

**OPTIMIZATION SETUP**

Once the baseline configuration setup is fixed and the results obtained, optimization can be set up. For optimization, shape variables are created and which are then used to morph the existing mesh to the new shape. The shape variables are created with the Hypermorph tool in Hypermesh, using the freehand morphing option. The domains for morphing are defined such that only the leading edge of the blade is modified. Since the outlet angles of the fan blade row are constrained by the downstream components in the engine, such as the compressor rows, the trailing edge of the blade is set to be unaffected by the morphing. To define the new shapes for this study, the upstream elements will be shifted in space using two shape variables — rotation and translation of the nodes. The downstream elements are kept fixed because of the constraints explained above. The in-between passage elements will however be affected due to the shifting of the elements in the upstream domain, thus yielding a new shape for the fan blade. The translation of nodes is along the direction of the blade chord i.e. x direction. The rotation of the nodes is along the direction of rotation of the fan i.e. about the x axis. Fig illustrates the morphed shape of the domain when a certain rotation and translation is applied to the original shape.
The shape parameters along with the existing mesh, are then imported in the optimization tool. Altair Hyperstudy is the optimization tool used to setup the optimization study in this project. The shape variables, the rotation and translation of the nodes in the upstream morphing domain, are the optimization parameters. When one or more variables are present, as in this case, each variable traverses through the specified levels, and all the possible combinations of the different levels of two variables make up the complete repertoire of cases that will be analyzed. The experiment design is similar to the full factorial design, enabling the user to study the effect of all possible input parameters on the response parameter, and also the interaction between the parameters. For example, if $R$ is the response parameter and $X$ and $Y$ are input parameters, variation can be studied in terms of how $R$ varies when $X$ and $Y$ are varied simultaneously.

The number of levels for both these variables are specified as 5. Full factorial design will be used for the design of experiment (DOE), thus yielding a total of 25 cases that can be evaluated according to the design variable matrix. The actual bounds specified in the morphing tool while creating the shapes are 50 mm for the translation, and 10 degrees for the rotation of the moved nodes. Thus a level of -1 for translation refers to a translation of nodes by -50 mm, or 50 mm in negative $x$ direction. Similarly a level of -1 for rotation refers to the rotation of nodes by -10 degrees about $x$ axis.

Once the design variables are specified and the experiment design with the number of levels is input to the optimization algorithm, the algorithm works automatically by reading the original baseline mesh, applying the morphing parameters to generate the new shape for the case being evaluated and sending the modified shape mesh along with the setup to the CFD solver. The optimizer software interacts with the morphing tool and the CFD solver via scripts. As soon as the CFD solver completes running a case the optimizer repeats the process for the next case in the queue until the experiment has finished. The results for all the cases are saved, and the desired data can be obtained from the post processing of these results, just like the baseline case.

**Results and Discussion – Optimization Study**

*The DOE Case Matrix*

The DOE matrix which was presented in the last section can now be filled with results from the cases run for optimization. Since the response parameter for this study is mass flow, Table 1 shows the experiment matrix as discussed in the previous section, with the mass flow values for all the different cases for which the results were obtained. There were a few cases for which the results could not be obtained because the solution failed due to problems with the morphed mesh. Those cases have been marked with NA in the matrix.
Table 1: Optimization experiment design matrix populated with the study results

<table>
<thead>
<tr>
<th>Variables</th>
<th>Translation (mm)</th>
<th>Rotation (deg)</th>
<th>-10</th>
<th>-5</th>
<th>0</th>
<th>+5</th>
<th>+10</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-1  50</td>
<td></td>
<td>0.6919</td>
<td>0.9013</td>
<td>0.8754</td>
<td>NA</td>
<td>0.8960</td>
</tr>
<tr>
<td></td>
<td>-0.5 -25</td>
<td></td>
<td>0.9094</td>
<td>0.8725</td>
<td>0.8719</td>
<td>0.9091</td>
<td>0.8820</td>
</tr>
<tr>
<td></td>
<td>0  0</td>
<td></td>
<td>0.8929</td>
<td>0.8991</td>
<td><strong>0.8837</strong></td>
<td>0.8980</td>
<td>0.9161</td>
</tr>
<tr>
<td></td>
<td>0.5 +25</td>
<td></td>
<td>0.8927</td>
<td>0.8729</td>
<td>0.8685</td>
<td>0.8964</td>
<td>0.8818</td>
</tr>
<tr>
<td></td>
<td>1   +50</td>
<td></td>
<td>0.8662</td>
<td>0.8997</td>
<td>NA</td>
<td>NA</td>
<td>NA</td>
</tr>
</tbody>
</table>

From the above calculated mass flow values for different runs, it is seen that the maximum mass flow is observed in the case with the normalized variable levels as (0, +1) for translation and rotation of the upstream nodes respectively. As actual values of the variables, this translates to no translation of the nodes but their rotation by +10 degrees about x axis. The corresponding shape of the blade row with this shape applied is shown in Figure 8.

**Figure 8: The shape corresponding to the highest mass flow rate through the fan (a) original shape (b) optimized shape**

(a) (b)

Flow Parameters

Table 2 shows the variation between the important flow parameters between the baseline case discussed earlier and the selected case from the optimization runs.

<table>
<thead>
<tr>
<th>Flow Parameter</th>
<th>Optimized Case</th>
<th>Baseline Case</th>
<th>Percentage Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Pressure Rise through the fan</td>
<td>731.5 Pa</td>
<td>747.5 Pa</td>
<td><strong>− 2 %</strong></td>
</tr>
<tr>
<td>Differential pressure between blade surfaces</td>
<td>453.0 Pa</td>
<td>454.4 Pa</td>
<td><strong>0.3 %</strong></td>
</tr>
<tr>
<td>Mass Flux</td>
<td>0.9161 kg/s</td>
<td>0.8837 kg/s</td>
<td><strong>+ 4 %</strong></td>
</tr>
</tbody>
</table>

A look at the table shows that although the mass flow in the chosen case shows maximum improvement over the baseline case, the total pressure rise shows a small decline. A possible reason behind this can be that the new blade shape has a longer blade-wise chord length. The pressure difference between the two sides of the blades is almost similar, thus indicating no significant change in the loading on the blade due to the change in shape.
The objective of the study was to determine the optimum aerodynamic shape for the shape of a given axial fan. This was achieved using a partial parametric optimization approach with the optimization parameters being the shape variables for the shape of the fan blade. The optimization was done to maximize the mass flow through the fan, because a higher mass flow through the fan means a higher total thrust for the engine.

In the previous sections we have discussed the methodology and procedure followed for the study. The baseline mesh is first generated from the given fan geometry and a CFD analysis done on it to record the benchmark results which are to be improved upon by changing the shape of the blade passage. For the purpose of changing the shape, the leading edge of the blade is translated and rotated by a certain magnitude in space by means of a morphing algorithm. Each of these shapes is then solved for results through a CFD analysis similar to the baseline shape and the results compared thereafter. The primary parameter compared is the response parameter for optimization, or mass flow through the fan.

The best case i.e. the case with maximum mass flow rate among the tested cases, was observed to be case no translation of the nodes but their rotation by +10 degrees about x axis. The improvement in mass flow was observed to be ~4% over the baseline mass flow. However, in the chosen case, the total pressure rise through the fan decreased by ~2%. It was also attempted to determine a correlation between the shape variables and the response variable, but a good correlation was not found to exist. This can be, in part, because of uncertainty in CFD results. Since the test object was not available, and neither were any experimental results, or another set of results from an independent study, available (even for the baseline case), it was difficult to determine the accuracy of the CFD results. However, since the study was an optimization study and the emphasis was on comparing the results among themselves, it was taken care to follow the best CFD practices and guidelines regarding the processes followed during meshing, preprocessing and setting up the CFD solution.

The optimization was achieved much quickly than what would have been possible if the morphing and optimization tools had not been used. It would have taken much more time, computational and human resources to accumulate the same repertoire of results corresponding to many different shapes of the fan. Also, the method allowed a very logical sequence of results generation which can further be used to develop a model for the output based on the input parameters.

There still might be, however, scope for further increasing the reliability of this study. One of the ways this can be done is by further refinement of the mesh, especially within the blade passage zone. While this may lead to an increase in computation time and morphing time, a better resolved mesh almost always leads to a more accurate solution in CFD. Another option that can be looked after in a future study is to increase the number of levels that will be tested for each variable for the optimization study. A more number of levels will yield more solution points in the matrix for comparison of the shapes. Though again, this leads to an increase in number of total computation time. However, using morphing, the time is significantly less than the equal number of test cases carried out independently of each other. Secondly, as the total number of cases increase, more experiment designs can be looked into, some of which do not require all the possible cases to be run but select a few cases from the large matrix to produce the results which can be extrapolated to fill the remaining empty cells.

REFERENCES


Simulate to Innovate


[7] Stefan Harries; Practical Shape Optimization using CFD; Whitepaper, November 2014 (Version 1.0)


[17] Altair Acusolve Command Reference