

# Abstract

During the design of the bi-directional fan product, Solidworks Flow Simulation (a CFD commercial software) is heavily used to influence design decisions in the project team. To be specific, CFD is used to validate and optimize 1. Fan-fan distance, 2. Tail-cone geometry, and 3. Nosecone design. The project team also present a proposed CFD method to influence design in the bi-directional fan project.

## Table of Contents

<b>Abstract.....</b>	<b>1</b>
<b>Introduction .....</b>	<b>2</b>
<b>Methodology.....</b>	<b>2</b>
<b>Overall.....</b>	<b>2</b>
<b>Flow Generating Method.....</b>	<b>2</b>
<b>Verification and Validation .....</b>	<b>3</b>
<b>CFD Results .....</b>	<b>4</b>
<b>Motor-fan Distance Study .....</b>	<b>4</b>
Setup 1: .....	4
Result:.....	5
Setup 2: .....	6
Result:.....	6
Analysis:.....	7
<b>Tail-cone Length Optimization.....</b>	<b>7</b>
Setup:.....	8
Result:.....	8
<b>Nosecone removal effect analysis.....</b>	<b>8</b>
Setup:.....	8
Result:.....	9
Analysis:.....	9
<b>Possible Design changes moving forward.....</b>	<b>10</b>
<b>Conclusion.....</b>	<b>10</b>

# Introduction

Computational Fluid Dynamics, or CFD is important in the design of performance-oriented flow systems. In the Bi-directional fan project, thrust-to-weight ratio is one of the KPP in the final deliverable. Such KPP requirement set high efficiency expectation to the design team. The use of CFD during the design phase is highly desirable to validate and inspire the bi-directional fan geometry design. The project team thus utilize CFD in many design decision makings.

## Methodology

### Overall

Research-oriented CFD usually targets achieving accurate flow simulation, the CFD results are usually validated using experimental results. Even for relatively simple geometry, an accurate CFD simulation can be very demanding in computing resources, which is usually done in a supercomputer cluster.

But in an industry setting, this type of CFD is usually not practical, because of the geometry complexity of products and the limit in computing resources. This inevitably limits the accuracy of flow that can be solved by using CFD. But in an industry setting, a noticeable flow difference between two settings is more important than the absolute accuracy of the CFD result, because of its ability to compare the performance of two settings and inform design decisions. As a result, CFD will primarily be used to inform design decisions in the Bi-directional Fan project, rather than provide absolute performance values such as efficiency.

### Flow Generating Method

To evaluate the geometry efficiency of a ducted fan, the approaches are primarily separated by the flow generating method. Here are the four methods I've been considering.

1. Pressure Boundary Condition

Replace blade fans to a donut shaped pressure boundary surface. Set total pressure boundary condition lower than the environment pressure at the disk front side, which allows air drawn to the duct inlet. Set a total pressure boundary condition higher than the environment pressure at the disk back side, which generates air flow towards fan tail.

2. Actuator Disk Model:

Replace blade fans to donut shaped surface. Set the surface to be an actuator disk, which imitates pressure difference created by rotating fan blades, thus create air flow. I found this method on a research paper on electric ducted fans. Based on conversation with CU professors, this method is not commonly used in research. But ANSYS Fluent have this implementation inside it's CFD solver.

### 3. Time-averaged flow from fan geometry (RANS):

Generate the air flow from the real fan geometry, by rotating the fan blades and apply wall boundary conditions on the fan blade surfaces. The “time-averaged” aspect of this method create a steady state flow trajectory, which represent the averaged flow properties (velocity, pressure...). Use 3d scanning technique I was able to reverse-engineered a close geometry of the JP Hobby fan blades. I primary use this method because its balance between realness and computational cost.

### 4. Fan Blade transient flow

Generate a time-dependent CFD result by rotating the fan blades. This is the most accurate CFD method in this method set, but requires extremely high computational cost. It is likely to cost months to generate a single CFD result in the team’s computation resources.

	Flow generation method	Accuracy	Computational Cost	Have Access?	Feature
1	<u>Pressure boundary condition</u>	Low	10min-1hr	Yes	Shows the flow if the blade is perfectly designed (the target flow situation)
2	<u>Actuator Disc Model</u>	Medium	<3hr	No	Use flow vector representation to mimik the flow centrifugal effect
3	<u>Time-averaged flow from fan model (RANS)</u>	Medium	1hr - 30hr	Yes	Generate a steady state flow profile
4	<u>Fan blade transient flow</u>	High	>24hr	Yes	Generate transient flow profile

*Table 1: Flow generation methods for bi-directional fan project*

The time-averaged RANS CFD model is primary used because it’s ability to imitating real flow generating, and its relatively acceptable computational cost. The pressure boundary flow generating method (#1) is also used in case studies which needs to isolate variable of fan blade geometries.

## Verification and Validation

In the use of computational fluid dynamics (CFD), any input, no matter how it’s deviated from the correct setup, will generate a flow trajectory as result, which is commonly described as “garbage in, garbage out”. So it is crucial make sure the result is usable to analyze, and this is called verification and validation in a CFD study.

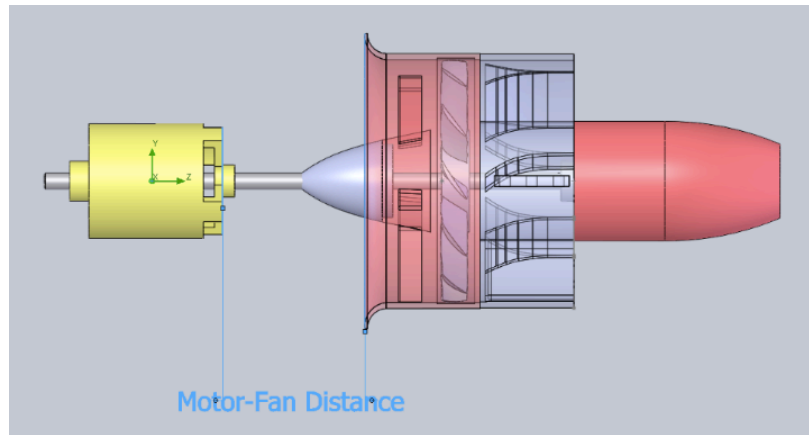
Verification is to verify that the CFD is properly set up to generate the right result of the desired CFD method. For example, incorrect mesh setup will change the fluid flow behavior such as dissipation. A widely used verification method is called “mesh convergence study”, which is to verify that the mesh is fine enough to have no effect on the CFD result. Mesh convergence study is conducted on the bi-directional fan project models. Unfortunately, the study shows current computational resource cannot obtain converged CFD result in a reasonable computation time. But fortunately, most CFD study show clear trends on performance when adjust the design such as motor-fan distance. So that CFD is still used to

inform good design practice. The limitation appears on the absolute value of CFD result, which means the result is not expected to match absolute experimental data.

Validation is to validate that the CFD method represents the correct physical effect of the fluid flow. Usually the validation step uses measured data from experiments. Since the bi-directional fan is still on its initial design phase, the experimental measurement is not feasible. As an alternate method, visualized experimental data from a research paper on electric ducted fan is used to validate that the simulation is similar to the real flow.

## CFD Results

### Motor-fan Distance Study



*Figure 1: Motor-fan distance*

According to the project KPP, the thrust to weight ratio is crucial to the bi-directional fan design. A CFD analysis to study influence of motor in front of the fan housing is conducted. If the fan is proven to provide reasonable performance with the setup, the design setup will proven to be valid. In addition, if there is a sweet spot for motor-fan distance that produce maximum thrust for this system, that distance is also beneficial to be pursued. Two setups for the motor-fan distance study was conducted, here are the results.

#### Setup 1:

1. Flow generating method: time-averaged flow from fan geometry
2. Fan RPM: 38000
3. Mesh size: ~1M
4. Computational time: ~2h
5. Motor-fan distance: 15.2mm – 91.4mm

To give detailed analyzable thrust result, the thrust is separated to “inner core”, “outer shell”, “motor bracket”. The sum of thrust force on all of those surfaces is the total thrust of the system.

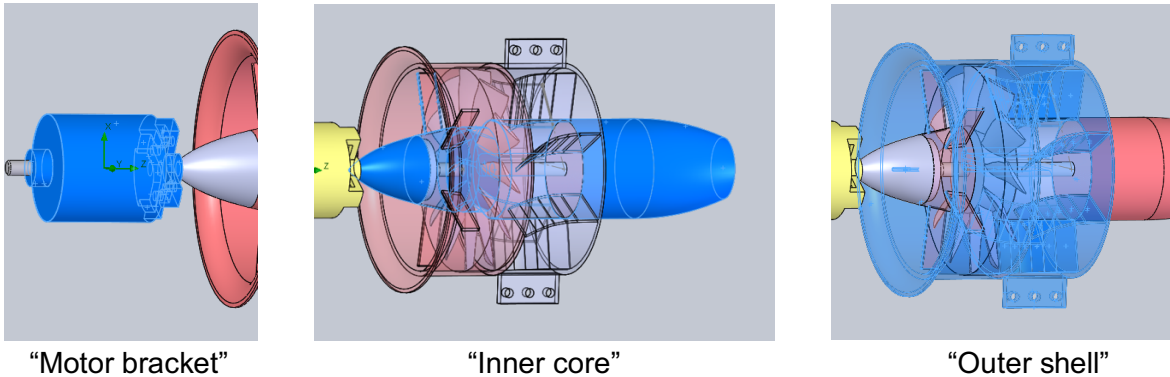


Figure 2: Blue high-lighted areas are: Motor bracket, Inner core, and Outer shell

## Result:

<b>motor-fan distance</b>	<b>[mm]</b>	<b>91.4</b>	<b>78.7</b>	<b>66</b>	<b>53.3</b>	<b>40.6</b>	<b>27.9</b>	<b>15.2</b>
<b>equivalent fan-fan distance</b>	<b>[mm]</b>	<b>232.6</b>	<b>207.2</b>	<b>181.8</b>	<b>156.4</b>	<b>131.0</b>	<b>105.6</b>	<b>80.2</b>
Blade Torque	[N*m]	0.491	0.484	0.483	0.486	0.493	0.487	0.499
Blade force	[N]	17.61	17.27	17.31	17.30	17.55	17.50	18.00
exit Velocity	[m/s]	61.78	61.52	59.70	60.25	61.11	60.81	61.00
motor bracket force	[N]	0.014	0.023	0.044	0.084	0.178	0.362	0.746
thrust from inner cone	[N]	-0.061	0.007	0.113	0.007	0.065	-0.316	0.009
thrust from outer shell	[N]	9.02	8.87	8.16	8.20	8.61	8.70	9.12
exit Mass Flow Rate	[kg/s]	0.335	0.338	0.323	0.331	0.331	0.334	0.329
exit total Enthalpy Rate	[W]	101825.25	102891.50	98266.13	100535.14	100574.20	101474.51	100231.16
Total thrust	[N]	26.55	26.13	25.54	25.42	26.04	25.53	26.39
Input power	[W]	2024.7	2045.6	2008.4	2039.4	2001.2	2063.6	2051.4
Output kinetic poewr	[W]	639.1	640.5	575.6	599.9	617.7	616.9	613.0
efficiency	[ ]	31.56%	31.31%	28.66%	29.42%	30.87%	29.89%	29.88%
thrust to power ratio	[N/kW]	<b>13.11</b>	<b>12.77</b>	<b>12.72</b>	<b>12.46</b>	<b>13.01</b>	<b>12.37</b>	<b>12.86</b>

Table 2: Motor-fan distance CFD setup 1 result

From the previous result, the mesh size is likely not sufficient that the simulation doesn't show clear trend on thrust corelated to motor-fan distance. As a result, the setup 2 with finer mesh and an additional “ring thrust” analysis is conducted. I expect a better representation of effect from the motor-fan distance.

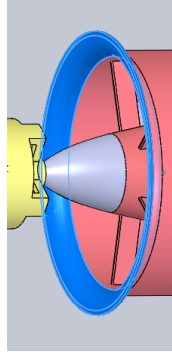


Figure 3: Blue high-lighted area is the Ring thrust

## Setup 2:

1. Flow generating method: time-averaged flow from fan geometry
2. Fan RPM: 38000
3. Mesh size: ~2.2M
4. Computational time: ~8h
5. Motor-fan distance: -3.8mm – 40.6mm

## Result:

motor-fan distance	[mm]	91.4	78.7	66	53.3	40.6	27.9	15.2
equivalent fan-fan distance	[mm]	232.6	207.2	181.8	156.4	131.0	105.6	80.2
Blade Torque	[N*m]	0.491	0.484	0.483	0.486	0.493	0.487	0.499
Blade force	[N]	17.61	17.27	17.31	17.30	17.55	17.50	18.00
exit Velocity	[m/s]	61.78	61.52	59.70	60.25	61.11	60.81	61.00
motor bracket force	[N]	0.014	0.023	0.044	0.084	0.178	0.362	0.746
thrust from inner cone	[N]	-0.061	0.007	0.113	0.007	0.065	-0.316	0.009
thrust from outer shell	[N]	9.02	8.87	8.16	8.20	8.61	8.70	9.12
exit Mass Flow Rate	[kg/s]	0.335	0.338	0.323	0.331	0.331	0.334	0.329
exit total Enthalpy Rate	[W]	101825.25	102891.50	98266.13	100535.14	100574.20	101474.51	100231.16
Total thrust	[N]	26.55	26.13	25.54	25.42	26.04	25.53	26.39
Input power	[W]	2024.7	2045.6	2008.4	2039.4	2001.2	2063.6	2051.4
Output kinetic poewr	[W]	639.1	640.5	575.6	599.9	617.7	616.9	613.0
efficiency	[ ]	31.56%	31.31%	28.66%	29.42%	30.87%	29.89%	29.88%
thrust to power ratio	[N/kW]	13.11	12.77	12.72	12.46	13.01	12.37	12.86

Table 3: Motor-fan distance CFD setup 2 result

## Analysis:

Both simulation setups show relatively high thrust-to-power ratio when the equivalent fan-fan distance is around 131mm. The distance is close to the shortest distance that components can physically designed. So the design team decides to pursue this distance.

Interestingly, the CFD shows better thrust-to-power ratio of the bi-directional setup than the original EDF's geometry. Furthermore, this bizarre result is very consistent through all the CFD setup we conducted, including the CFD using the pressure-boundary-flow-generating method. We suspect that the presence of motor creates a narrower channel for the inlet air flow, which increases air velocity and reduces static pressure nearby. At a specific range of motor-fan distance, the thrust force increased on the ring exceeds the drag created by the presence of motor, which cause a better thrust-to-power ratio on the bi-directional fan setup.

As expected, in setup 2, the motor bracket force shows increase drag when motor-fan distance decrease, while the thrust on the "outer shell" and "ring force" both shows clear trend of thrust increase as motor-fan distance decreases. Such result shows the possible correctness of the assumption.

Since the thrust to power ratio can be increased by such modification, why there is no EDF manufacture uses such design anywhere? I believe the reason is that most EDFs are designed to operate in high environment air velocity, for which drag coefficient is crucial, while such design will dramatically increase drag of the whole system. But the bi-directional fan is designed to be use as stationary as possible—to stabilize hung cargos. There is no need to optimize system overall aerodynamic performance.

After realize such difference between the original EDF and the bi-directional fan unit, I believe there are many more modifications can be done to improve system thrust-to-power ratio performance.

## Tail-cone Length Optimization

The tail-cone is originally designed to cover the inside BLDC motor for a better aerodynamic performance. Since the bi-directional fan is designed to have the motor in front of the fan, the tail cone can be shortened to reduce system overall length. We expect that shorten the tail will also increase the system overall efficiency because less surface area on the tail-cone.

6 CFD analyses are conducted, including 5 different shaped tail cones and a setup without a tail cone. The result shows very clear trend that as the tail cone shortens, the overall efficiency of the system will increase. While in the simulation setup without a tail cone, the efficiency is not as good as the ones with a tail cone. As a result, the design team decide to use the shortest tail cone in the bi-directional fan product. The round tip tail is finally chosen because it is easier to manufacture compared to the point tip tail cone.

## Setup:

1. Flow generating method: time-averaged flow from fan geometry
2. Fan RPM: 38000
3. Mesh size: ~1.4M
4. Computational time: ~3h

## Result:

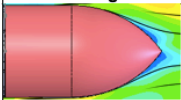
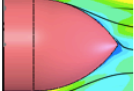
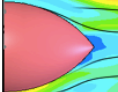


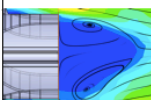
Tail cone length	Thrust (N)	Power input (W)	Power output (W)	Thrust to Power ratio (N/kW)	(Geometry + Fan) Efficiency
	25.77	2044.90	618.71	12.60	30.26%
	26.13	2053.36	643.10	12.72	31.32%
	26.12	2061.23	673.25	12.67	32.66%
	26.55	2090.34	719.05	12.70	34.40%
	26.65	2090.95	719.70	<b>12.75</b>	34.42%
	25.48	2053.64	646.45	12.40	31.48%

Table 4: Tail-cone length optimization result

## Nosecone removal effect analysis

The design team proposed to remove the nosecone for the ease of assembly. Stand from the similar argument that the bi-directional fan is used while stationary, the nosecone may not be crucial. CFD simulations are conducted to evaluate effect of removing the nosecone. Due to the limited computational resource, CFD experiment with and without a nosecone is only conducted on a setup similar to the *motor-fan distance study*.

## Setup:

1. Flow generating method: time-averaged flow from fan geometry
2. Fan RPM: 31700, 38000
3. Mesh size: ~2.8M
4. Computational time: ~9h



5. Motor-fan distance: 27.9

Result:

	with nosecone	Without nosecone	
rpm	38000	38000	31700
Input energy	2090.92	2548.34	1573.49
output kinetic energy	752.17	1075.29	686.77
total thrust	25.84	33.50	24.67
kinetic energy efficiency	35.97%	42.20%	43.65%
thrust to power ratio (N/kW)	12.36	13.15	15.67

Table 5: Nosecone efficiency CFD result

Analysis:

Both the setups were using 38000 rpm at first. But the two results are dramatically different from each other, both in flow trajectories and calculated power inputs. To match the power input to the system, another CFD with reduced rpm is conducted on the *without nosecone* setup. Although the result still doesn't show similar input energy, it produced similar total thrust to the setup with a nosecone.

Both the thrust to power ratio and efficiency shows very dramatic improvement when removing the nosecone, which is in my opinion too good to be true. The flow trajectory seems have a game changing difference between two simulation. Both CFD have a pair of vortexes downstream of the air flow, while the vortex is much closer to the fan in the nosecone CFD result than the CFD without a nosecone, which I believe is the primary reason for such effect. The vortex appears commonly in every fan-geometry-generated-air-flow CFDs, but the reason for the vortexes is still not clear to the design team. The current guess for the answer is the inaccuracy in EDF's geometry modeling.

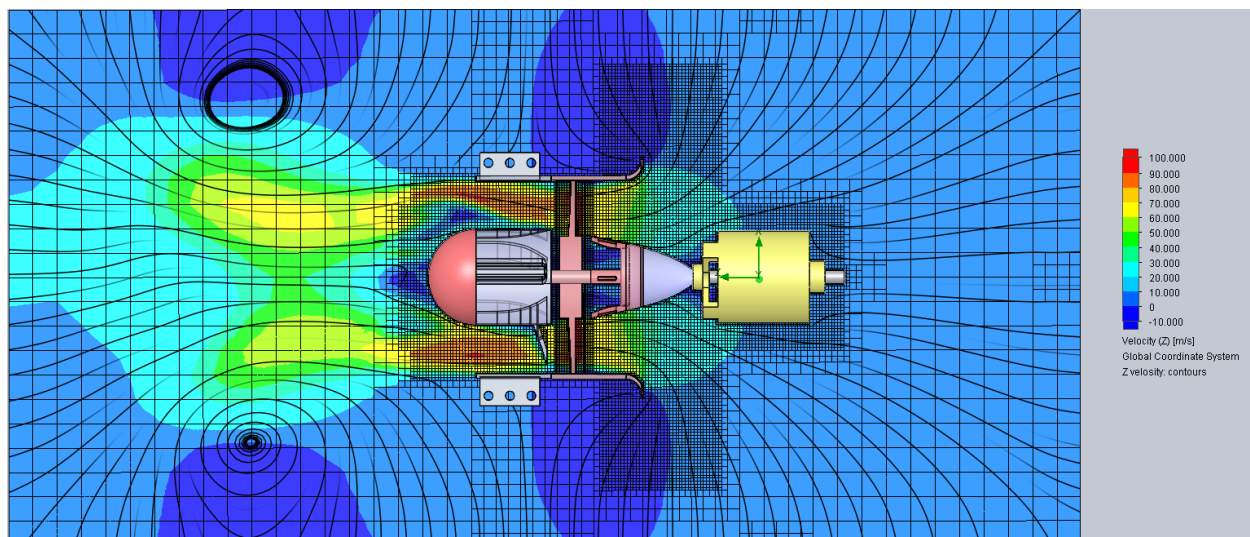


Figure 4: Flow trajectory with nosecone

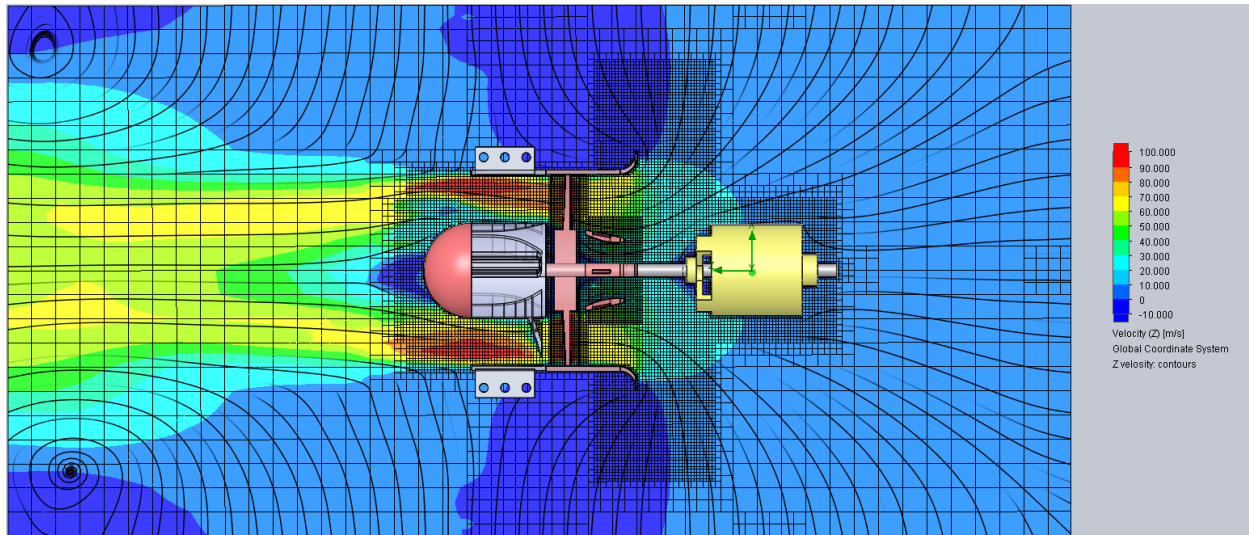


Figure 5: Flow trajectory without nosecone

As a result, the project team decide to move forward without a nosecone. However, we do not expect big efficiency or thrust to power ratio change due to such geometry change.

## Possible Design changes moving forward

There are many possible geometry changes to increase the bi-directional fan efficiency in the future, for which are most likely out of the scope and budget in the graduate design project. It is the following reasons that those changes are valid in the bi-directional fan rather than in the original EDF fan: 1. The target usage of the bi-directional fan is aiming to use while it is stationary to air velocity, 2. The motor is no longer positioned inside the fan housing.

Currently, there is only one proposed potential design change, which is to enlarging the ring on the fan housing. The ring is designed originally to generate thrust from the low-pressure air flow, enlarging it will further increase the thrust from the ring geometry. This design change is enabled because of reason #1.

## Conclusion

The project team utilize CFD to optimize design changes for overall thrust-to-weight ratio in the bi-directional project design phase. It concludes an optimized motor-fan distance—82mm, an optimized tail-cone length and shape, and a design change to remove the nose cone. During the analysis, an interesting effect that a motor in front of the fan will increase thrust is also discovered, which inspires further design modifications. The project team will continue to use CFD as a tool to meet KPPs of the bi-directional fan project in the future.