



Case Studies about Combination of CAESES and ANSYS CFX for Pumps

Shigeyuki TOMIMATSU
(DMW Corporation, JAPAN)

富松 重行
(日本电业社机械制作所)



Introduction of DMW Corporation

Open up the future. ～新しい風が未来を切り開く～



DMW CORPORATION

Domestic Deployment

MISHIMA PLANT
Shizuoka Pref.



HEAD OFFICE Tokyo



and 14 Regional Offices in Japan



Mt. Fuji

Manufacturing and sale of

- Pumps
- Fans, Blowers, Turbo-compressors
- Valves
- Environment Equipment
- Electric Control / Monitoring System
- Others



International Network

Subsidiaries:

DMW India Pvt. Ltd., Mumbai

Offices:

Abu Dhabi

Amsterdam

Dalian, China

Houston, U.S.A.

Singapore

Maintenance Facilities (Local Partners):

FEC: Mumbai, India

WESCO: Abu Dhabi, U.A.E.

WAMCO: Damman, Saudi Arabia

APSCO: Iran





Case Study about Mixed Flow Pump Impeller



Introduction

In DMW, ANSYS Blade Modeler has been used for CFD modeling because ANSYS CFX has been mainly used as a CFD software.



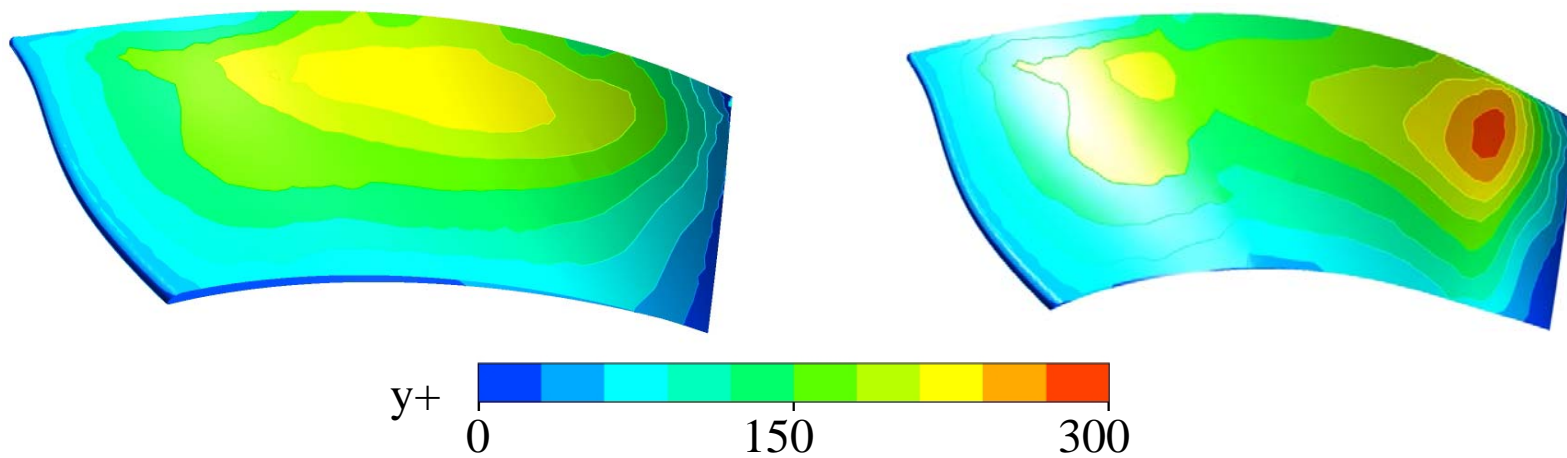
However, it is difficult to optimize a fluid machinery using the model made by ANSYS Blade Modeler.



We considered to make a CFD model of a mixed flow pump impeller using CAESES.

Mesh Quality by Mesh Morphing

The design optimization technique is often used at the pump development.



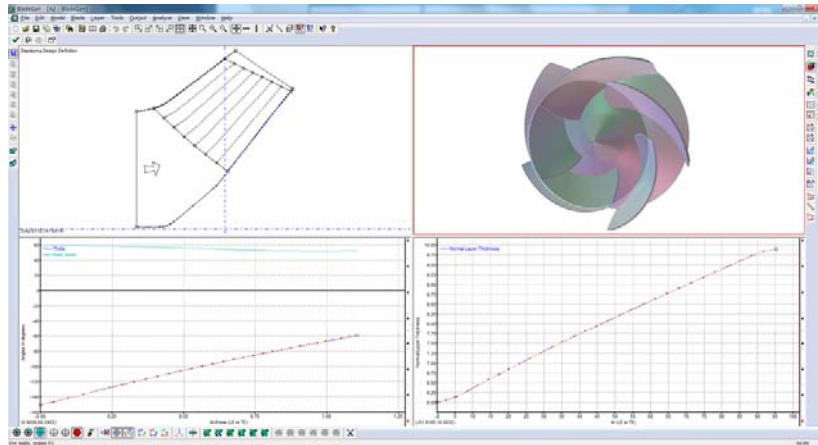
Before morphing

After morphing

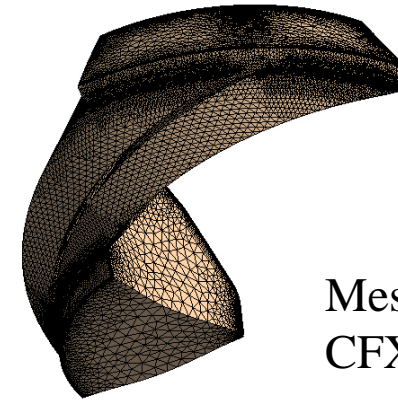
y^+ on Surface of impeller blade (Meshed by CFX-Mesh)

The value of y^+ is drastically changed by the mesh morphing.

Conventional Approach



Modeled by
CFX Blade
Modeler



Meshed by
CFX-Mesh

When CFD is conducted, a hexa-mesh model is better than a tetra-mesh model in order to keep analysis quality.



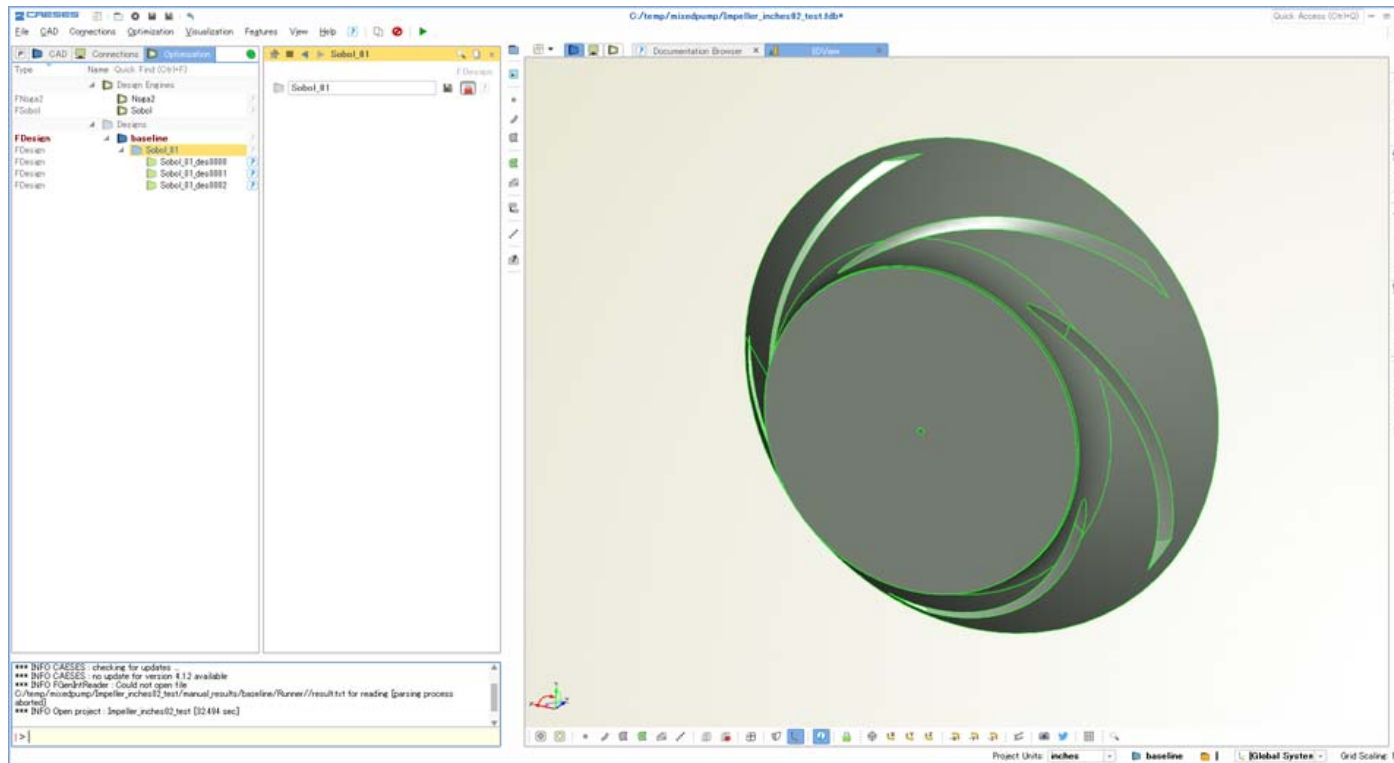
Conflict!

When morphing is conducted, a tetra-mesh model is better than a hexa-mesh model in order to keep mesh quality.

Improved Approach



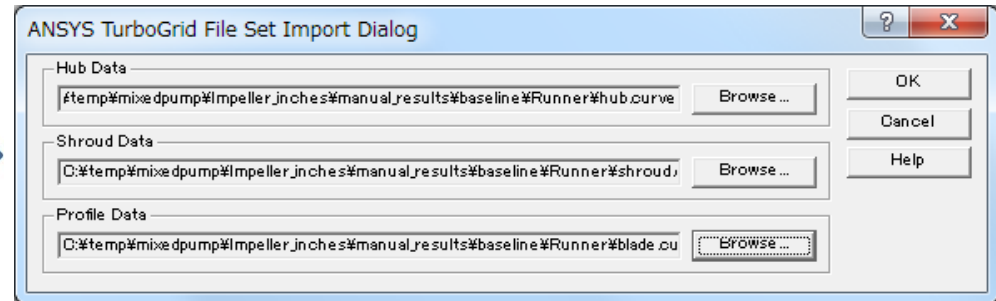
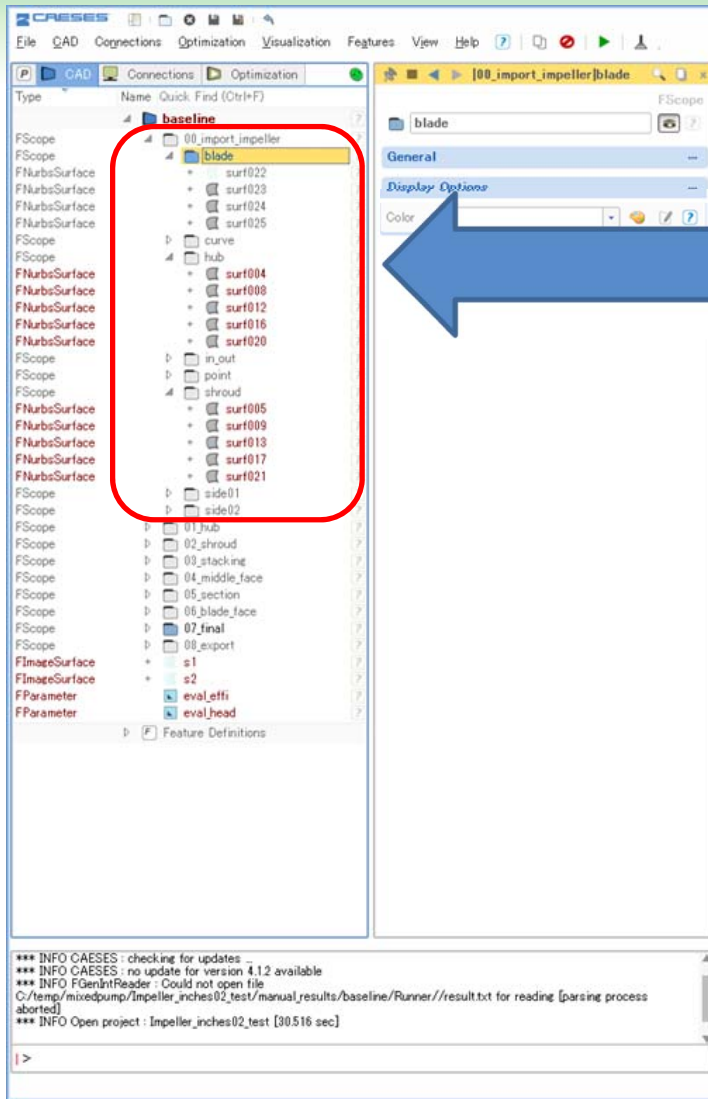
Shape morphing will be better than mesh morphing.



Fluid path modeled by CAESSES



Import Coordinate Data



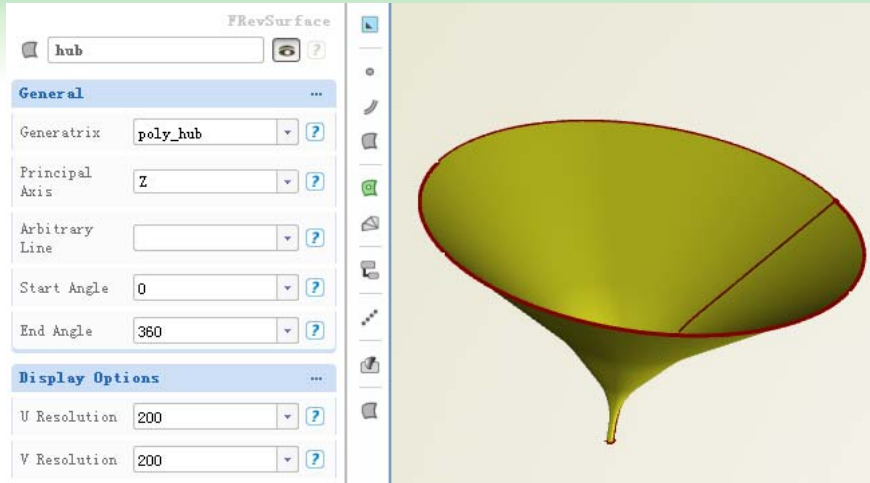
Import Dialog of ANSYS Blade Modeler

Coordinate data files for a blade, a hub, a shroud used by ANSYS Blade Modeler don't need to be changed for CAESES.

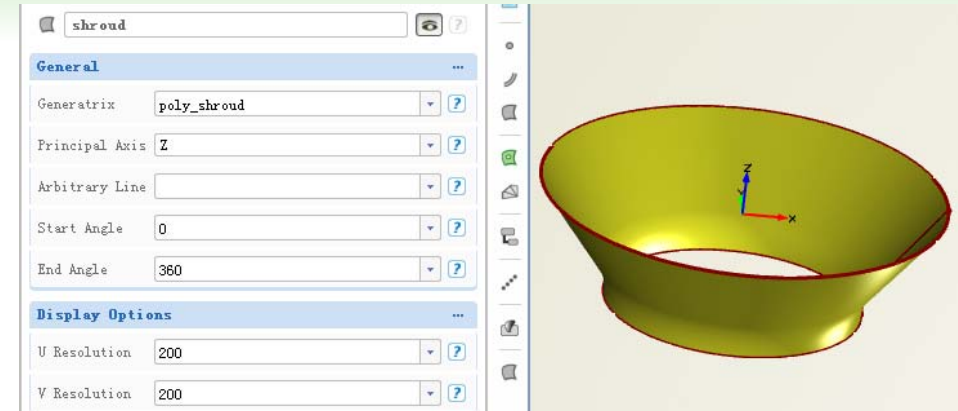


We can make efficient use of our technological asset.

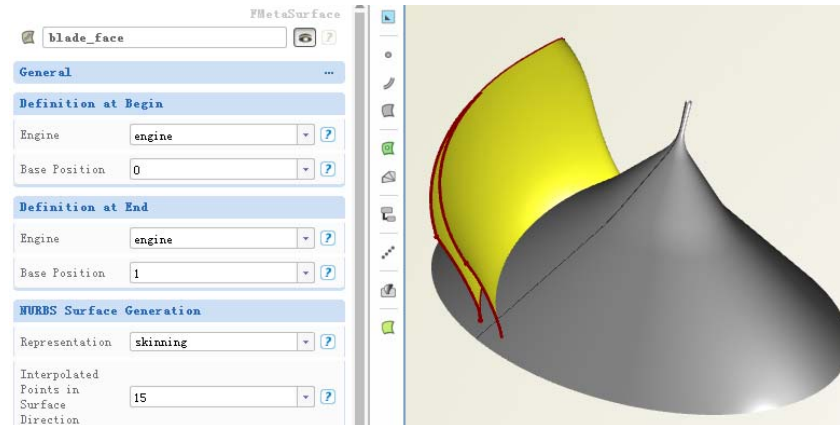
Modeling



Hub

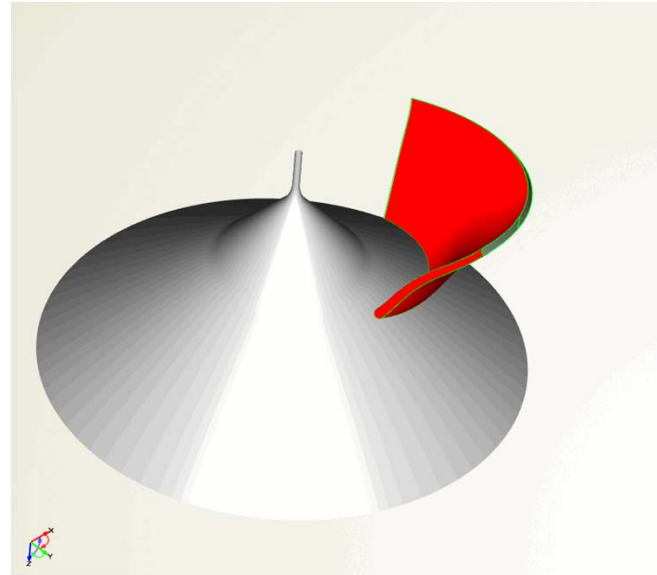
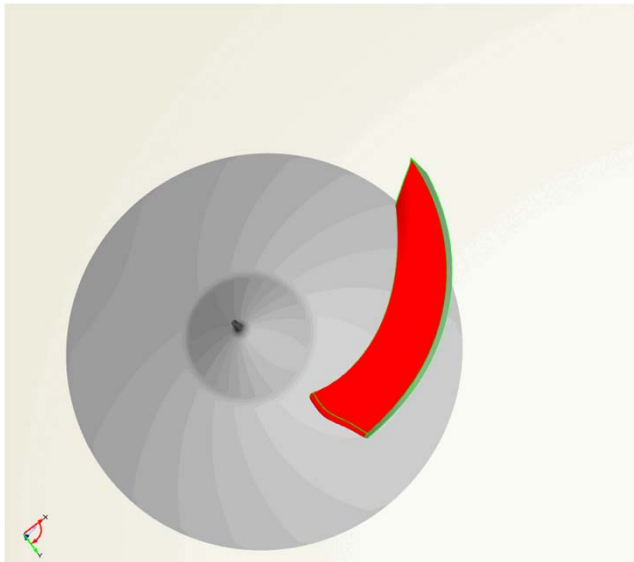
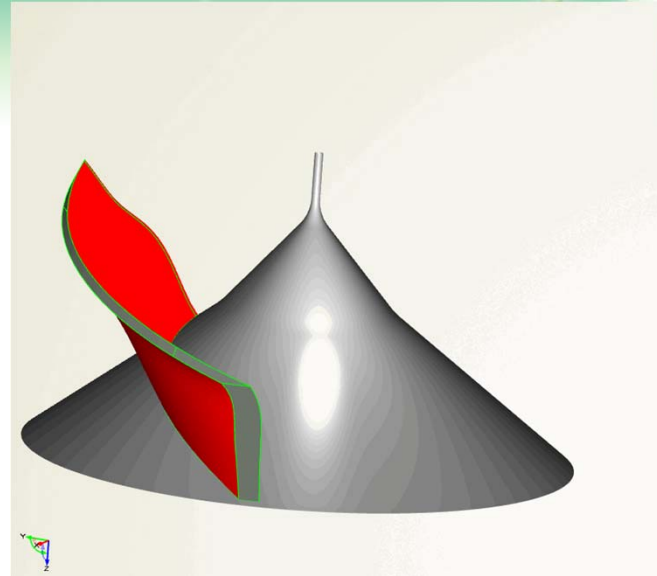
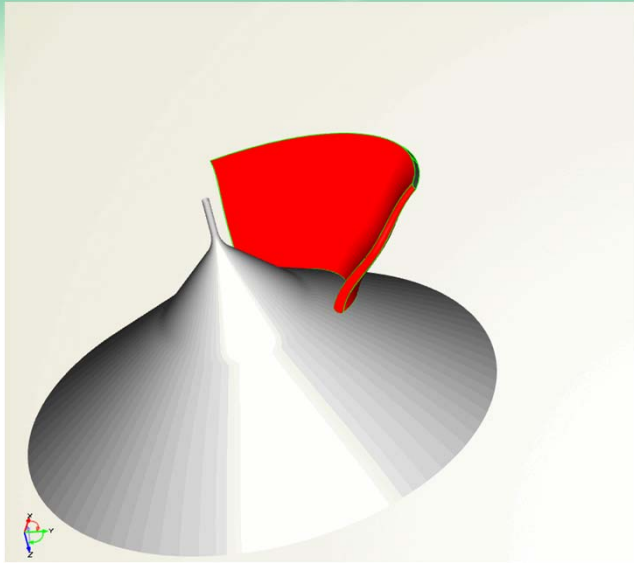


Shroud



Blade

Shape Morphing



Open up the future. ~新しい風が未来を切り開く~



DMW CORPORATION

Meshing and CFD

The screenshot displays the CAESAS software interface. The 'Runner' window is active, showing 'Execution Settings' and 'LocalApplication' sections. The 'Executable' field is highlighted with a red circle and contains the path: `$PROJECTDIR/run.cfd.bat`. A callout box points to this field with the text: "ANSYS TurboGrid and CFX can be run automatically." Below the Runner window, a separate callout box shows the contents of the batch file:

```
@echo Start Meshing....↓
"C:\Program Files\ANSYS Inc\v170\TurboGrid\bin\cfx5.exe" -batch impellernew.tse↓
@echo off↓
@echo CFX PRE...↓
"C:\Program Files\ANSYS Inc\v170\CFX\bin\cfx5pre.exe" -batch impeller.pre↓
@echo CFX SOLVE...↓
"C:\Program Files\ANSYS Inc\v170\CFX\bin\cfx5solve.exe" -batch -parallel -partition 4 -par-local -def "impeller.def" -fullname impeller_001↓
@echo CFX POST...↓
"C:\Program Files\ANSYS Inc\v170\CFX\bin\cfx5post.exe" -batch impeller.cse↓
```

A 'batch file' callout box is positioned to the right of the batch file content.

Optimization

The screenshot shows the CRESSES software interface. On the left, the 'Optimization' menu is open, and 'Sobol' is selected. The central panel displays a table of design variables for 'Sobol_01_des0000':

Name	Lower Bound	Value
[04_middle_face]end_angle	45	60
[05_section]end_thick	0.8	1.15
[03_stacking]middle02_Distance	3	5.5
[04_middle_face]start_angle	30	45
[05_section]start_thick	0.8	1

The right-hand panel shows a detailed view of a design variable and its evaluation table:

Name	Lower Bound	Value	Upper Bound	
[04_middle_face]end_angle		45	60	75
[05_section]end_thick		0.8	1.15	1.5
[03_stacking]middle02_Distance		3	5.5	8
[04_middle_face]start_angle		30	45	60
[05_section]start_thick		0.8	1	1.2

The evaluation table below it shows:

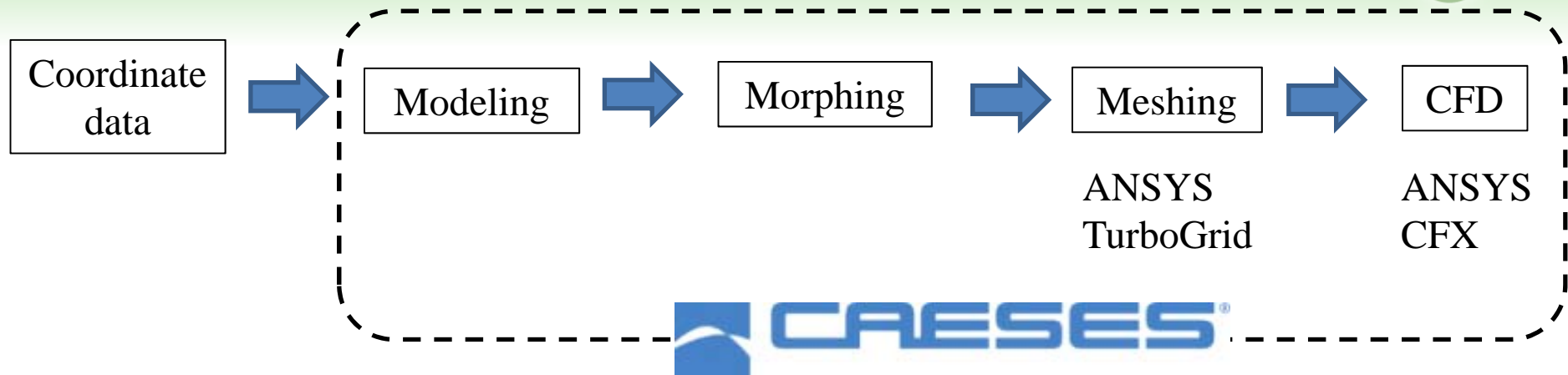
Name	Value	Is Objective
leva_effi		No
leva_head		No

Annotations include:

- A red box around 'Sobol' in the optimization menu with the text: "Sobol is selected as an optimization technique."
- A blue circle around 'leva_effi' and 'leva_head' in the evaluation table with a blue arrow pointing to the text: "Pump efficiency and head are used as objective functions."
- Two grey boxes with the word "Secret" are placed over the 'leva_effi' and 'leva_head' rows in the evaluation table.



Conclusions



These phases can be controlled by CAESES.



It will be able to be improved operational efficiency.

Future Work

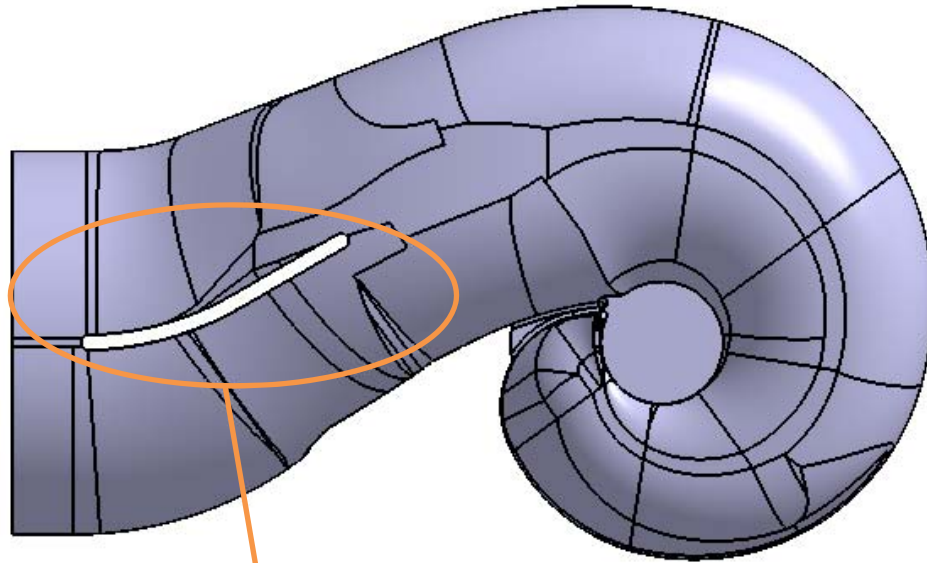
Optimization for a merged model of a rotor and a stator of a mixed flow pump will be conducted.



Case Study about Suction Part of Double Suction Pump

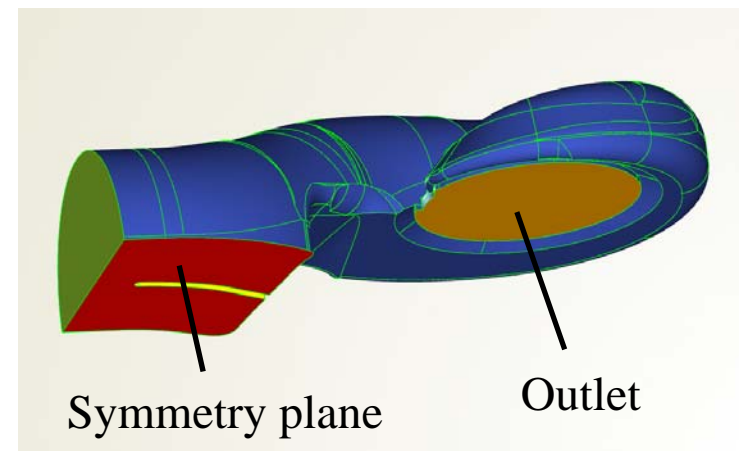
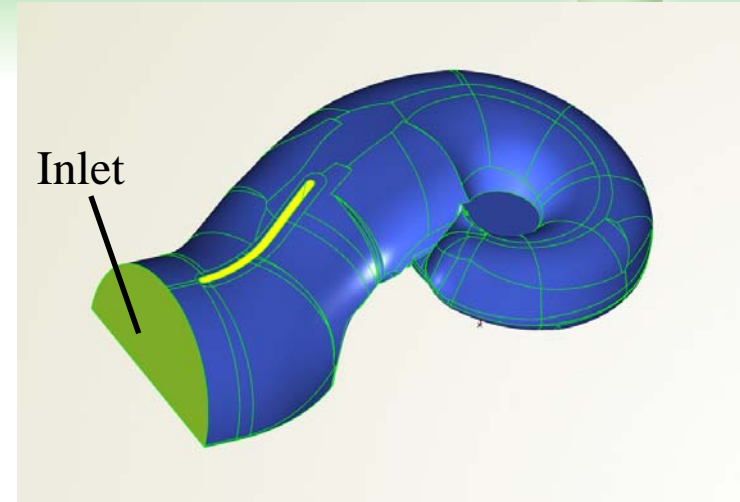


Suction Part of Double Suction Pump

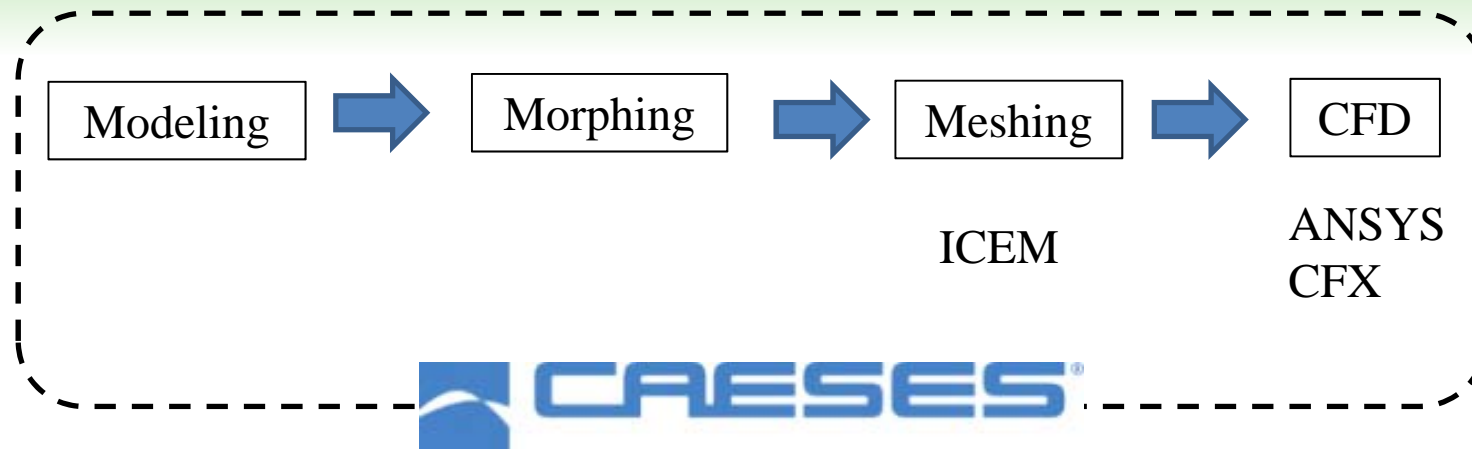


Fluid path modeled by CATIA

The optimization technique is used in order to find the shape of the guide vane having the lowest pressure drop.



Approach



Each phase is controlled by CAESES.

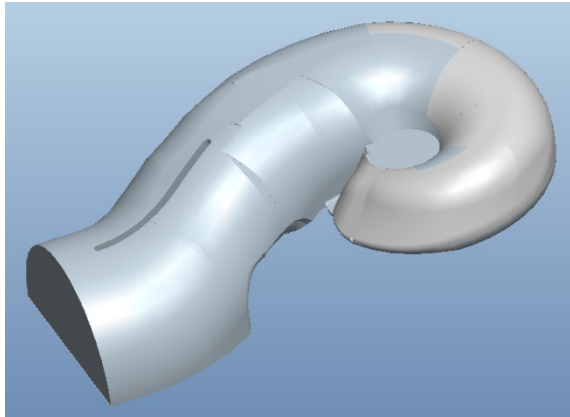


Challenge

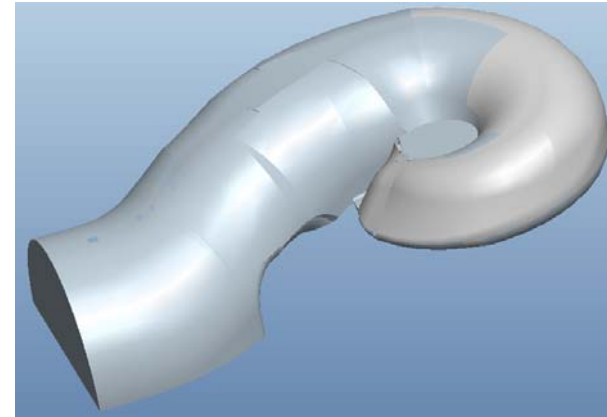
- ✓ It is reused the CATIA model as much as possible.
- ✓ ICEM is automatically controlled by CAESES.

Modeling

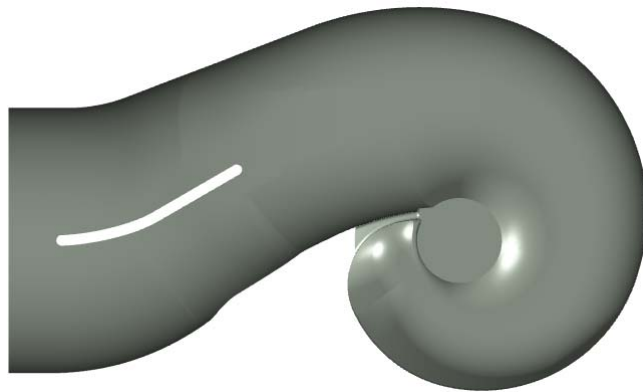
1. The CATIA model is read as a solid model.



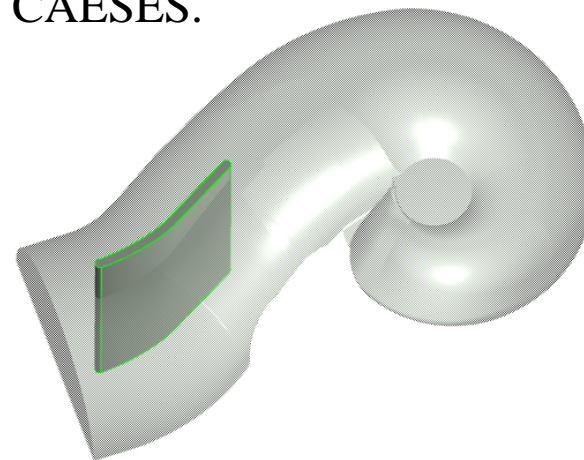
2. The guide vane is removed.



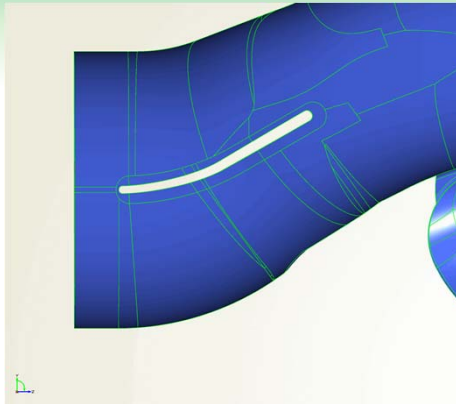
4. The guide vane is subtracted from the solid model.



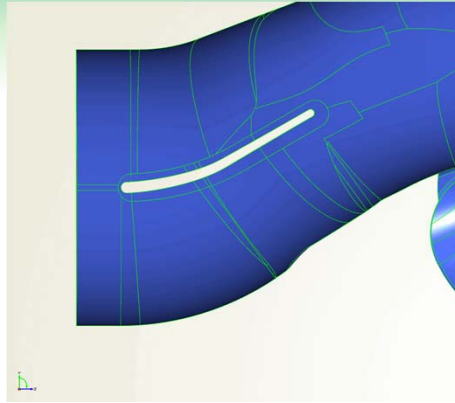
3. The guide vane is remodeled by CAESSES.



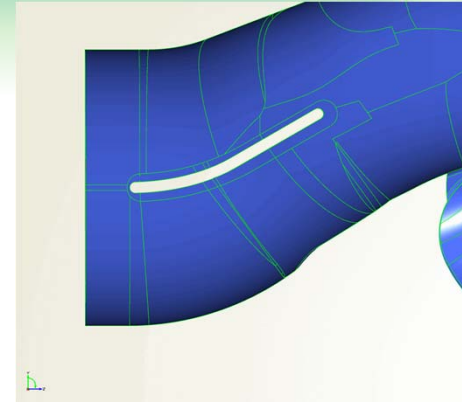
Morphing



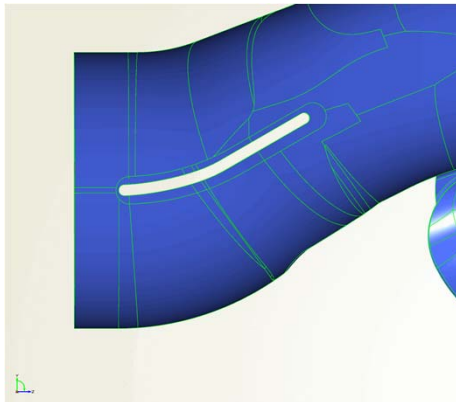
Thickness around
leading edge



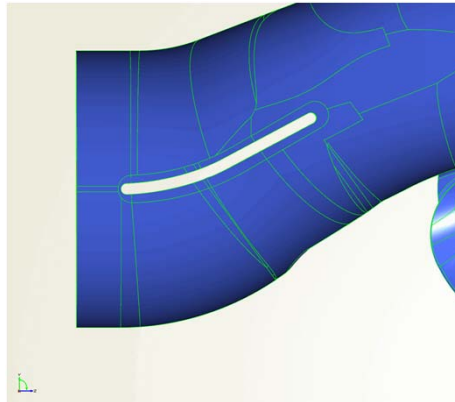
Thickness around
trailing edge



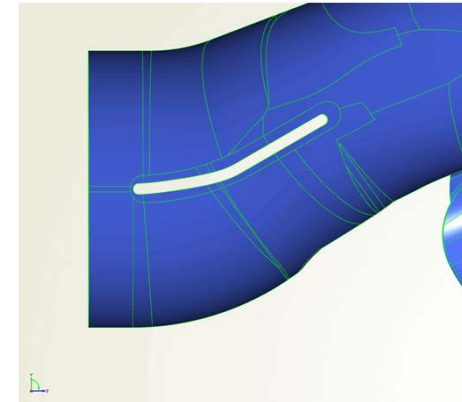
Geometry definition
parameter 1



Length of guide vane



Exit angle of guide
vane



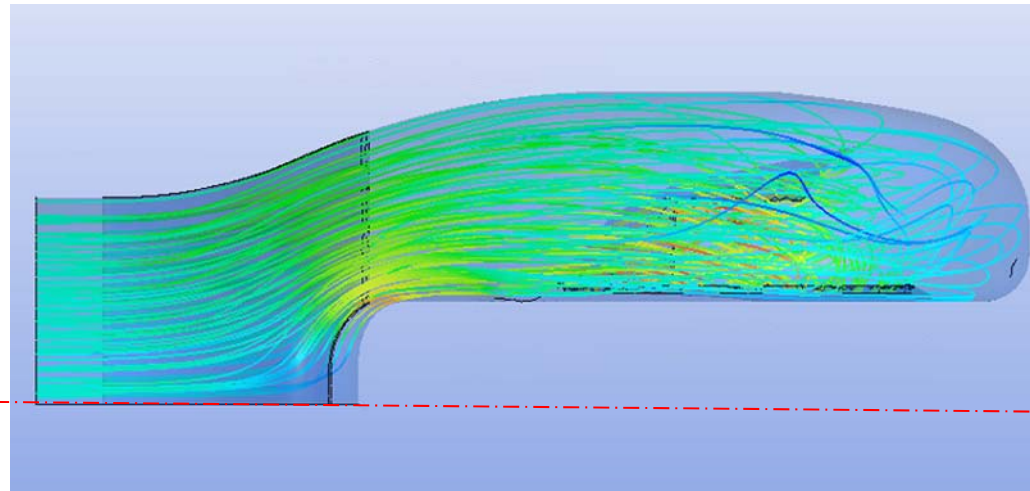
Geometry definition
parameter 2

Meshing and CFX

```
@echo Start Meshing.....↓
↓
call "<entry>ANSYS_DIR</entry>%icemcfd%win64_amd%bin%icemcfd.bat" -batch -script ICEM.rpl↓
↓
@echo CFX PRE...↓
↓
"<entry>ANSYS_DIR</entry>%CFX%bin%cfx5pre.exe" -batch DF.pre↓
↓
@echo CFX SOLVE...↓
"<entry>ANSYS_DIR</entry>%CFX%bin%cfx5solve.exe" -def "DF.def" -start-method "MPI Local Parallel" -part 4 -double -fullname DF_001↓
↓
@echo CFX POST...↓
"<entry>ANSYS_DIR</entry>%CFX%bin%cfx5post.exe" -batch DF.cse←
```

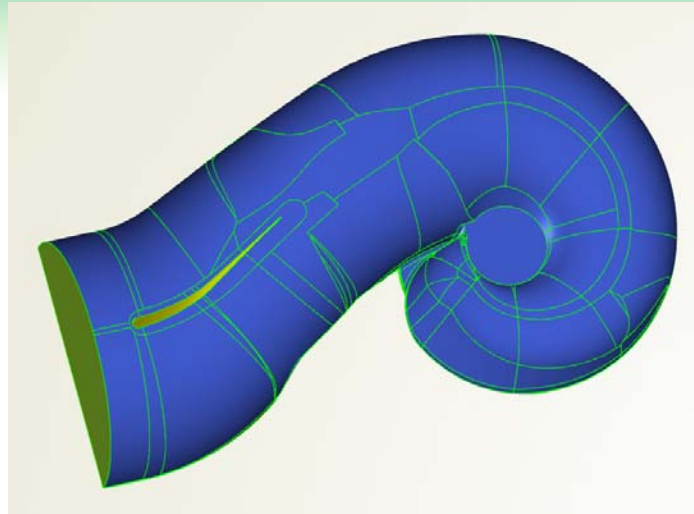
ICEM is automatically run by a batch file.

Symmetry plane

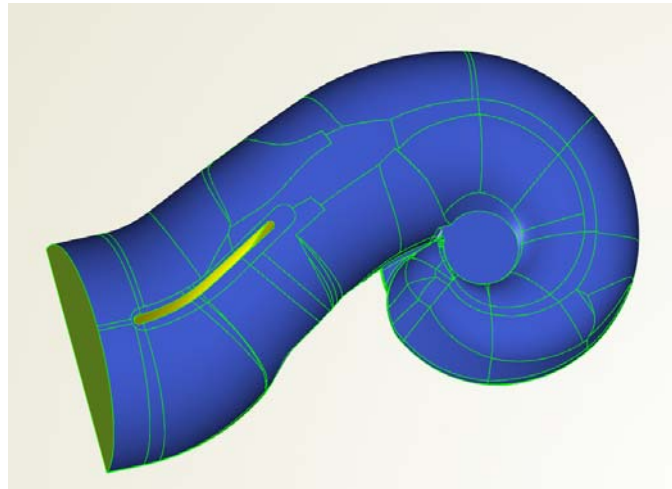


By considering the symmetry of the suction part of the double suction pump, the symmetry condition is applied to the CFD model.

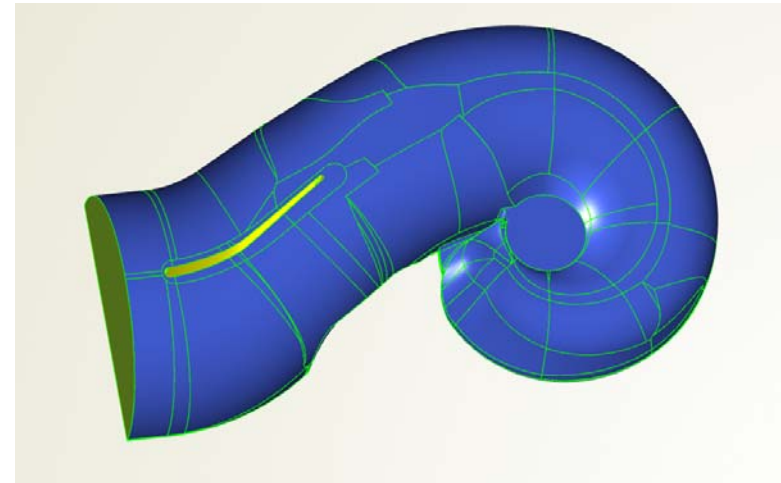
Optimization



Minimum pressure drop model

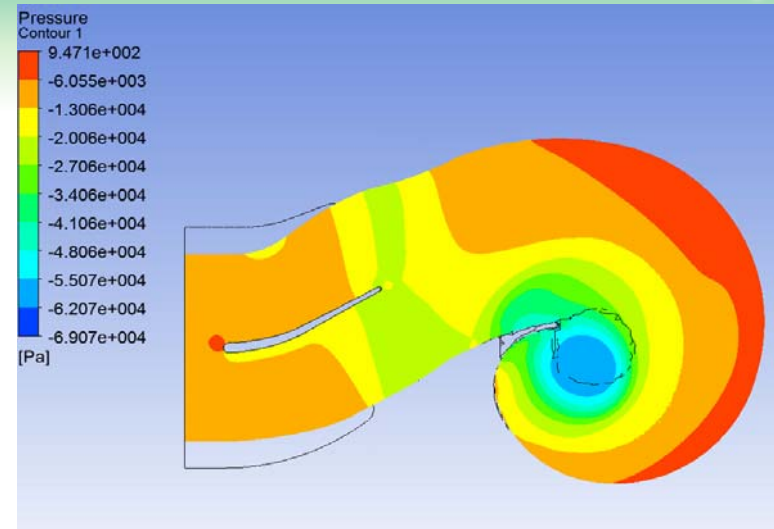


Original model

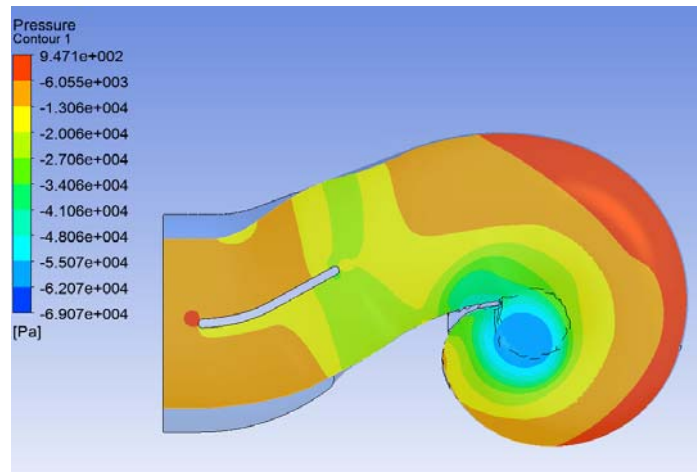


Thicker trailing edge model

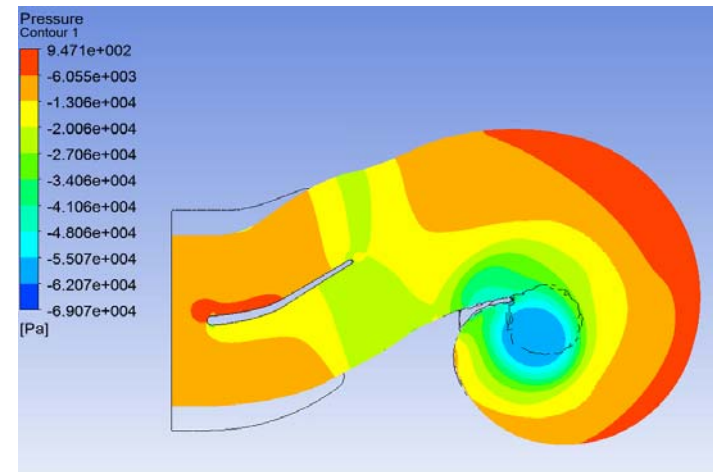
Pressure Distribution



Minimum pressure drop model

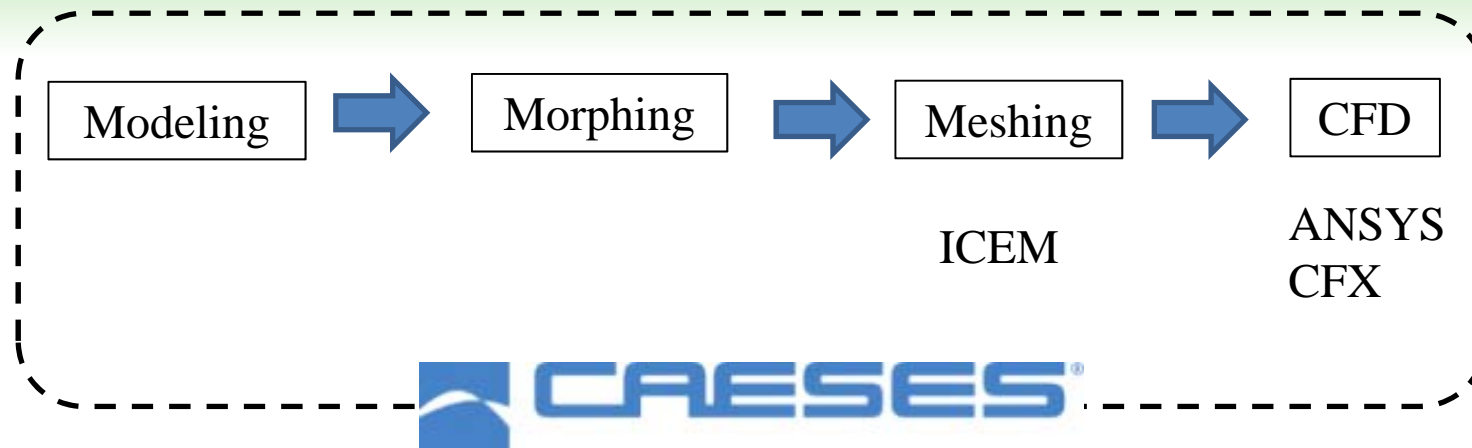


Original model



Thicker trailing edge model

Conclusions



Each phase is possible to be controlled by CAESES. ICEM and ANSYS CFX are automatically run.

The optimized geometry of the guide vane in the suction part of the double suction pump is found.

Future Work

Optimization for a whole double suction pump model will be conducted.