This project has received funding from the European Union's Seventh Framework Programme for research, technological development and demonstration under grant agreement no 612279

Aljaž Škerlavaj, Enrico Nobile
DIA - Dipartimento di Ingegneria e Architettura
Università degli Studi di Trieste

Esercitazioni di Termofluidodinamica Computazionale

CFD Simulation of a simple centrifugal pump

May 2016

1 Introduction

This document is a detailed description how to perform a CFD (Computational Fluid Dynamics) simulation with *ANSYS CFX 16.2* (hereafter "CFX") of a simple centrifugal pump. The geometry, as well as experimental and numerical results are described in [1]. The geometry of the case is provided in three *.stp* files. The goal is to create computational grids, make a CFX setup file and run the simulation.

2 Case description

Centrifugal pumps are widely used sin engineering applications. Therefore, the current document instructs to create computational grids (meshes) for a given geometry and to prepare a setup for the CFD simulation.

The presented case is a very simple shrouded centrifugal pump impeller (Fig. 1), the (scaled) Grundfos CR4, with medium specific speed. The PhD thesis by Pedersen [1] includes experimental and CFD results for the pump. The experiments were performed for a transparent impeller (produced in perspex) by Particle Image Velocimetry (PIV) (Fig. 2) and Laser Doppler Velocimetry (LDV) techniques. The CFD analysis was performed with Large Eddy Simulation (LES).

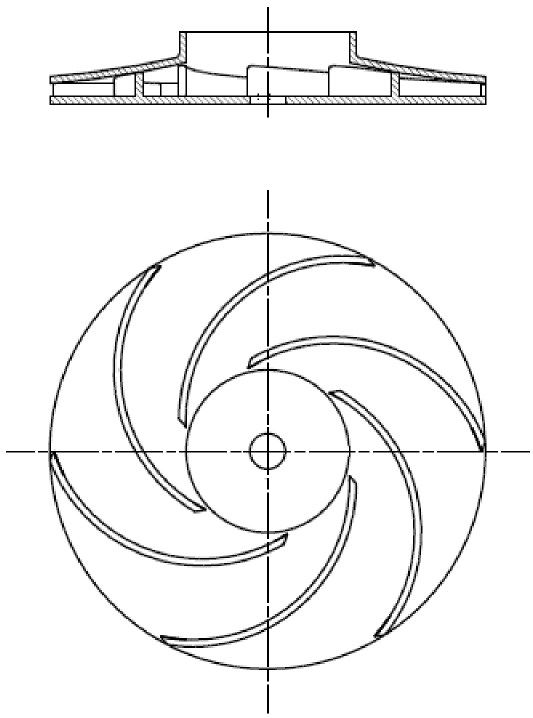

Figure 1: Geometry of the pump impeller [1]

The experimental observations at the design operating conditions ($Q = Q_d$) did not show separation in blade passages, and flow in all six channels was similar. On the other hand, at quarter-load conditions ($Q = 0.25Q_d$) a non-rotating stall was observed, consisting of

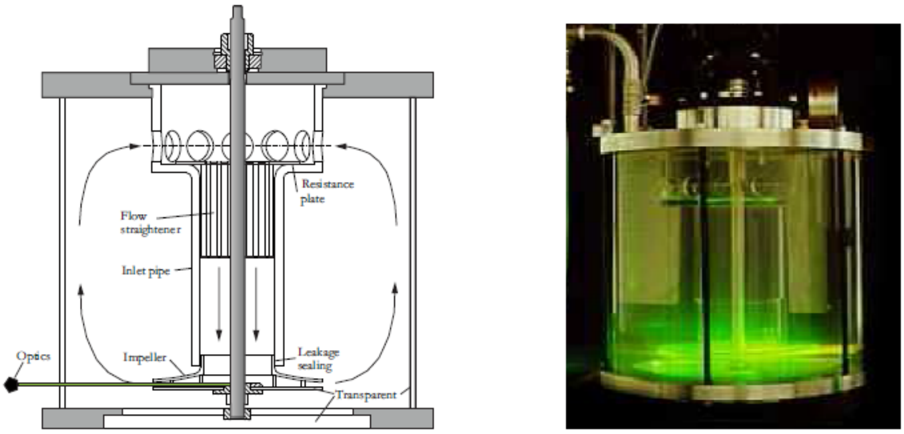

Figure 2: Details of the PIV setup [1].

alternate stalled and unstalled passages. In the latter case, flow in every second passage was similar.

3 Analysis with ANSYS CFX

3.1 Geometry

Currently, the provided impeller geometry is an approximation of the geometrical data (Table 1) provided in [1].

Dati caratteristici del problema.				
Inlet diameter	D_1	=	77	mm
Outlet diameter	D_2	=	190	mm
Inlet height	b_1	=	13.8	mm
Outlet height	b_2	=	5.8	mm
Number of blades	Z_1	=	6	-
Blade thickness	t_i	=	3	mm
Inlet blade angle	β_1	=	19.7	$^\circ$
Outlet blade angle	β_2	=	18.4	$^\circ$

Table 1: Impeller characteristics [1].

Usually, the computational domain of interest is extended with an inlet and an outlet part/domain. In our case, for the impeller in Fig. 1 an inlet extension and an outlet extension should be provided. The inlet extension provides a more realistic inlet conditions, whereas the outlet one provides more realistic conditions (velocity vectors) at the outlet from the impeller. For instance, if there is a large vortex at the outlet from the impeller, the outlet or opening boundary conditions at the impeller outlet surface might produce incorrect results.

Therefore, we will be using three computational 'domains' of our CFD simulations: inlet, impeller and outlet domain. The geometry is provided by three files. In Table 2 the relation of filenames and computational domains is provided. The whole geometry is provided for a 60° section ($1/Z_1$ of the full circle).

Filename	Part No. in Fig. 3	Computational domain
geometry_inlet3.stp	1	inlet
geometry_impeller3.agdb	2	impeller
geometry_outlet3.stp	3	outlet

Table 2: Geometry filenames and computational domains.

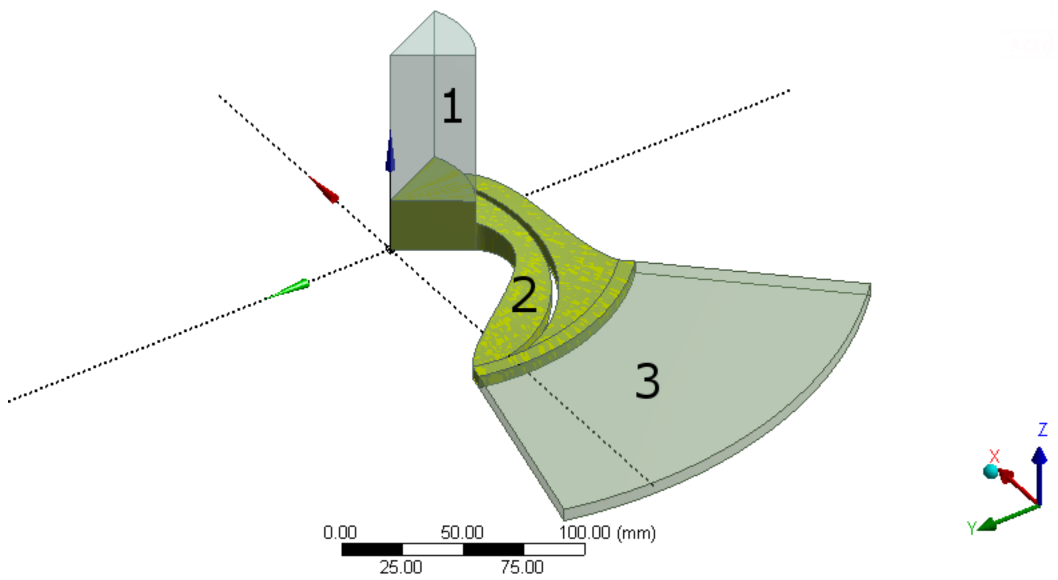

Figure 3: Geometry, specified by three geometry files

In the following section it will be described how to create computational grids for the three geometry files.

3.2 Computational grids - ANSYS ICEM

Computational grids for all three parts (provided by three geometry files) will be created in *ANSYS ICEM*. It is suggested to put each geometry file in a separate file directory.

3.2.1 Computational grid 1

Start *ANSYS ICEM* by clicking *Tutti i programmi* → *ANSYS 16.2* → *Meshing* → *ICEM CFD 16.2*. Set the working directory of the first file (geometry_inlet.stp): *File* → *Change Working Dir...*.

Open geometry file of part 1: Click *File* → *Import Geometry* → *Legacy* → *STEP/IGES*, select a file, click *Open*, then click *Apply* in the window presented in Fig. 4.

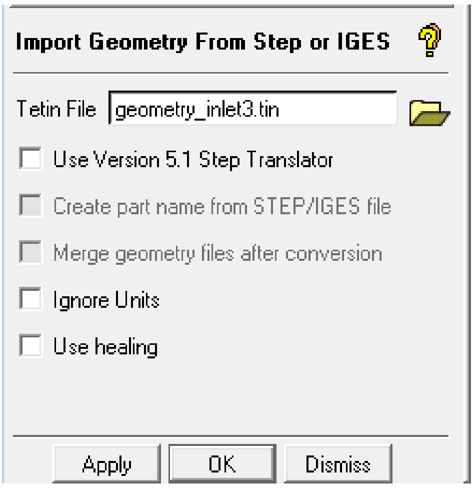

Figure 4: Importing the geometry file.

The geometry is imported. Save the project to select project's name: *File* → *Save Project As...*, write *mesh_inlet.prj* and click *Save*.

In the window that represents geometry the *left mouse button* (LMB) rotates the geometry, the *central mouse button* (CMB) moves geometry and the *right mouse button* (RMB) scales up/down the geometry. In the left region of the screen a Display Tree (Fig. 5) is shown, which can be used for control of display in the main window. It is possible to click on the + sign to expand the tree.

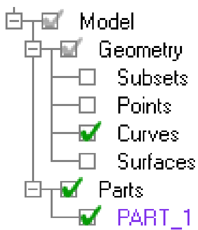

Figure 5: Display Tree.

The imported part represents an inlet part above the impeller in Fig. 2. From the figure it can be concluded that side walls of the inlet geometry do not rotate.

The first step is to set the correct units and size. Click (LMB) *Settings* → *Model/Units*. Set the units to Milimeters and click *OK*. The geometry size should be increased by 1000 times. In the Main Toolbar click *Geometry* → *Transform Geometry*: . Use the settings according to Fig. 6 and click . Enable selection of points, curves, surfaces and bodies in the floating menu (choices in the menu should be enabled). Then press "a" on the keyboard (shortcut for "select all") and click *Apply*. Click Fit Window icon () to fit the geometry in the screen window.

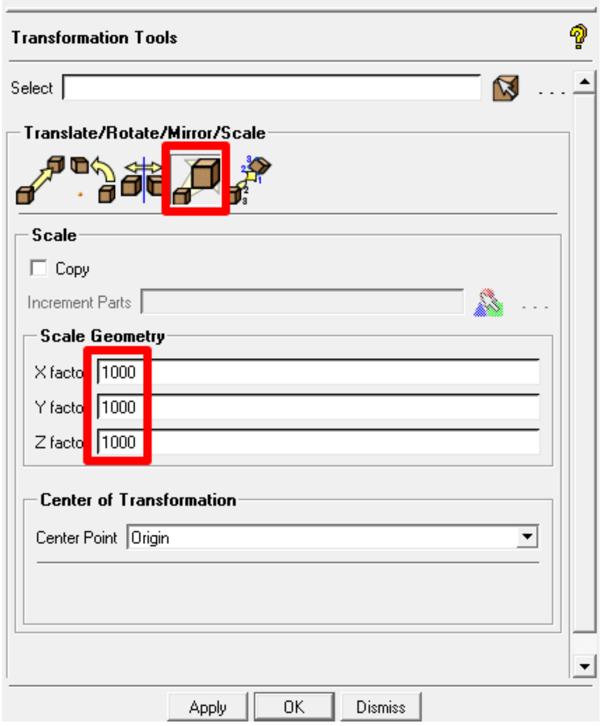

Figure 6: Scaling the geometry by a factor of 1000.

The next step is the creation of the "parts" in the Display Tree, which will later represent selectable surface parts of the meshes for which we will prescribe boundary conditions. Another advantage of creation of parts is the prescription of element size for each part (in case of unstructured grids).

Five parts will be created in the Display Tree of the mesh_inlet.prj project: geom1_in, geom1_out, geom1_walls, geom1_per1 and geom1_per2 (the 'geom1' label was used to represent the inlet part/domain of the whole case).

Rotate the geometry so that Zaxis is pointing upward, turn on surfaces in the Display Tree and click on *solid simple display* icon  in the Main Menu. The result is presented in Fig. 7.

To create a part (containing a surface) the following procedure can be used:

- Right-click on *Parts*, then left-click on *Create Part*
- Enter(write) the part's name in the top field
- Click on icon  next to *Create Part by Selection*. The most important functionality of the pop-up window is to toggle on/off selection of points, curves, surfaces and bodies by clicking the four icons. Turn on only the selection of surfaces: 
- Pick-up the desired surface(s) by left-clicking (LC) on surface(s), then finish with a middle-button-click (MC). Before finishing it is possible to undo the picking-up actions (in a reverse direction) by using a right-mouse button (RC). During the pick-up process it is possible to rotate/move the geometry by pressing *F9* button (and re-pressing to re-enter the pick-up process).

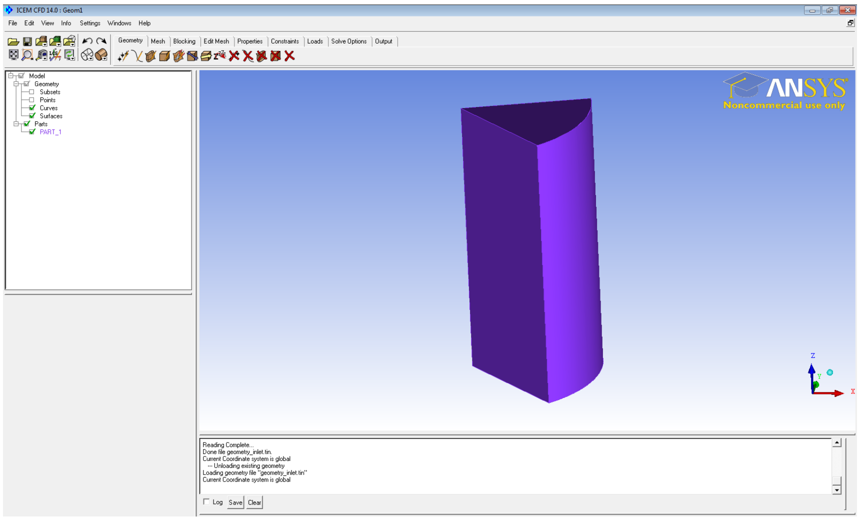

Figure 7: The geometry of the first domain (inlet domain) before creation of parts.

- MC again to finish the selection process.
- A new part appears in the Display Tree and the part's surface(s) change its(their) colour.

By using the described procedure create the five parts. The part geom1_in includes the inlet surface (with the highest Z value) - the surface at the top in Fig. 7. The part geom1_out includes the outlet surface (with the lowest Z value). The part geom1_walls includes the annular section/surface. There should be two periodic parts. Suppose that index Per1 of the periodic parts is on the left side of index 3 in Fig. 3 (for geometry 3), whereas index Per2 is on the right side of index 3. Therefore, the part geom1_per1 is the visible vertical surface in Fig. 7, whereas the part geom1_per2 is the hidden vertical surface in Fig. 7.

In the Main Toolbar click *Geometry* $\rightarrow$ *Create Body*: .

Turn on points visibility and turn off visibility of surfaces in the Display Tree. Create a body (part) named BODY_INLET by using *Centroid of two points* and picking up two points (e.g., as in Fig. 8). MC twice. Check that the centroid (body) point lies within the geometry, bounded by surfaces (rotate the geometry to check it).

In the Main Toolbar click *Mesh* $\rightarrow$ *Global Mesh Setup*: . Set the global mesh parameters according to Fig. 9 (Max element size=4.0, Prism initial height=0.1, ratio=1.2, layers=10, Rotational period. axis = 0 0 1, angle=60). Click *Apply* in each of the three windows. It should be noted that in some cases (complex ones) the creation of prism layers mail fail with the latter setting. In such cases it is better to set only two parameters for prism elements (thus leaving the total prism height 'floating'). For instance, set only the ratio with e.g. 7 layers, and later subdivide and rearrange the layers (as it will be done for the blade mesh).

Besides setting the global parameters it is possible to set local parameters for specific parts. In the Main Toolbar click *Mesh* $\rightarrow$ *Part Mesh Setup*: . A pop-up window appears. Choose parameters accurding to Fig. 10 (BODY_INLET part will have a hexa core mesh, max. size=4, prism layers will be located at part GEOM1_WALLS). Instead of putting 4 to the *max size* it would be possible to leave the setting as 0, because of the previously defined global parameters. Prism elements are needed only at walls. The hexa-core mesh converts

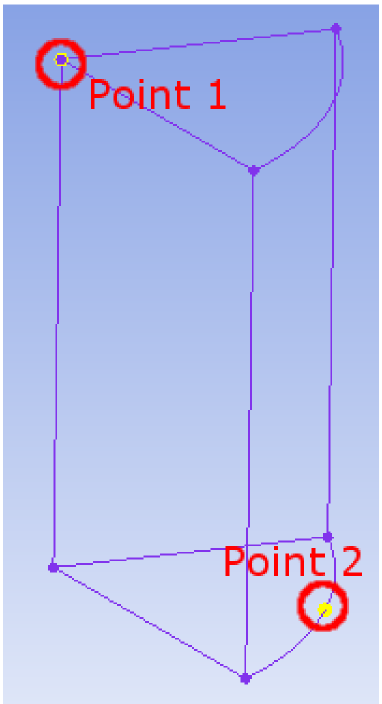

Figure 8: Create Body: selection of two points.

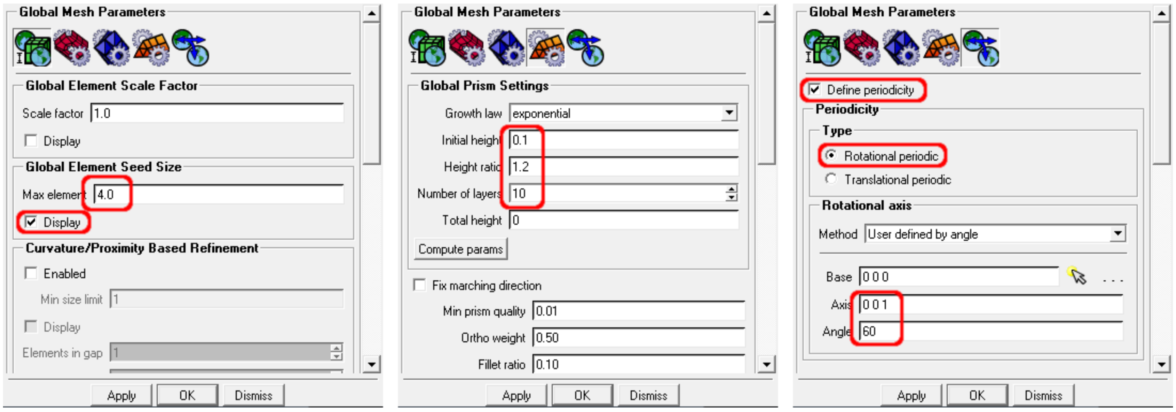

Figure 9: Global mesh setup.

tetrahedral elements to haxagonal elements, thus reducing number of elements (decreased elements vs. nodes ratio). To finish, press *Apply*, then *Dismiss*.

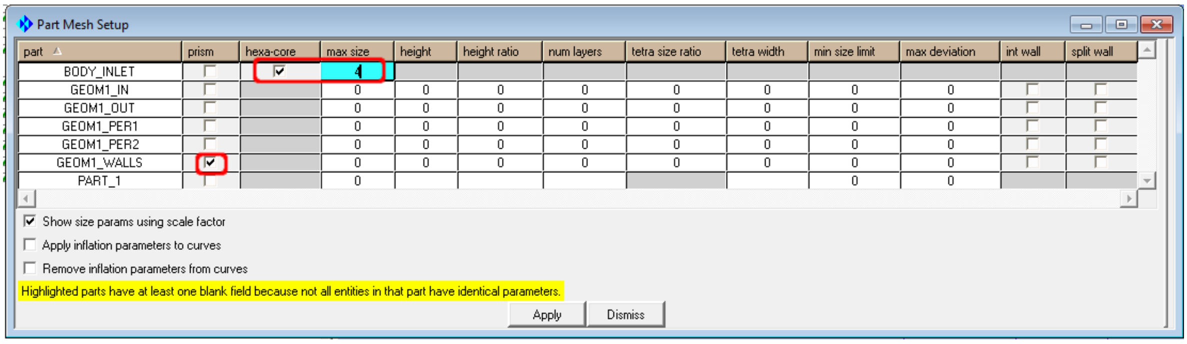

Figure 10: Part mesh setup.

After defining the mesh parameters it is time to create the mesh. In the Main Toolbar click *Mesh* → *Compute Mesh*:  . In the Compute Mesh window turn on creation of prism layers and mesh hexa-core after the tetra meshing (Fig. 11). It would be also possible to create prism layers and hexa core in a separate meshing process. This can be useful for complicated geometries, where it is better to create a high-quality tetra mesh first (quality above 0.2 or 0.3). Press *Compute*.

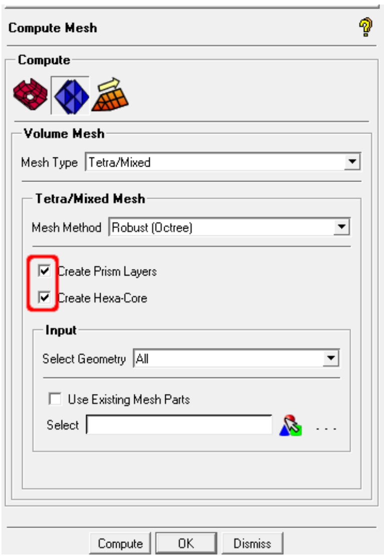

Figure 11: Compute Mesh window.

To view the mesh (if it is not visible) turn on Shells in Mesh tree of the Display Tree, as well as surface parts, left-click  in the Main Toolbar, then click *Fit Window* icon  in the Main Toolbar. The mesh is presented in Fig. 12.

Check the mesh for errors. Click *Edit Mesh* → *Check Mesh* (). Click *Apply*. A pop-up window appears. Select the two periodic parts and click *Accept*. Confirm the deletion of unconnected vertices (click *Yes*).

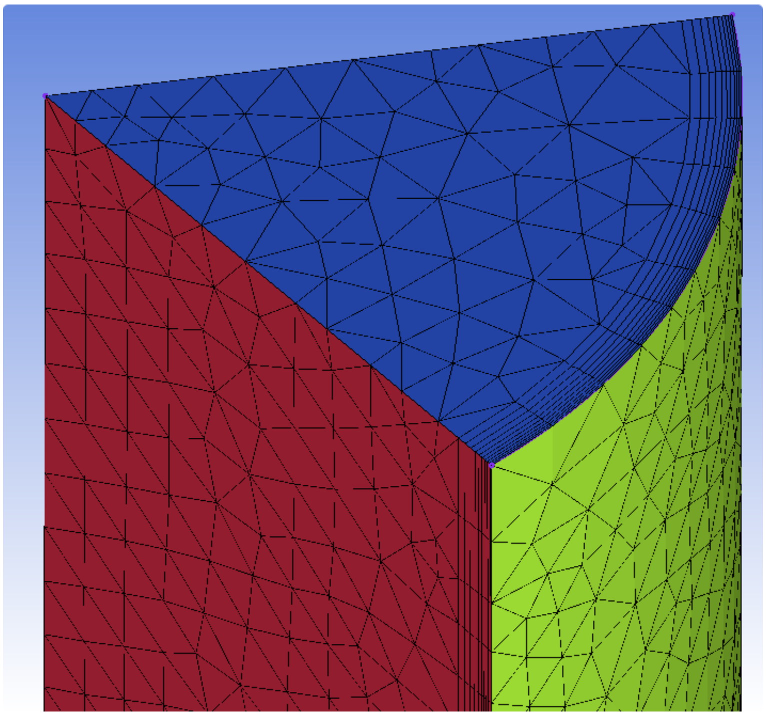

Figure 12: Mesh - inlet part.

Check the quality of the mesh. Click *Edit Mesh* → *Display Mesh Quality* ( ). The mesh quality is just above 0.2.

Improve the mesh quality. Click *Edit Mesh* → *Smooth Mesh Globally* ( ). Set the smoothing parameters according to Fig. 13. Click *Apply*. The new quality is above 0.25. If it was smaller, the smoothing would be repeated with quality limit set to 0.2 and all previously 'Frozen' types of meshes set to 'Smooth'.

Repeat the previously described Check Mesh step.

Click *File* → *Save Project*.

The mesh has to be exported to a .cfx5 file format. Click *Output* → *Select solver* ( ) and set the field *Output solver* to *ANSYS CFX*. Click *Apply*. Then click *Output* → *Write input* ( ). Confirm saving the project. Click *Done* in another pop-up window. Wait until you see 'Done with translation' message in the Message window. The mesh is exported. Now close the project (you can as well save it).

3.2.2 Computational grid 2

Start *ANSYS ICEM* by clicking *Tutti i programmi* → *ANSYS 16.2* → *Meshing* → *ICEM CFD 16.2*. Set the working directory of the second geometry file (geometry_impeller3.agdb): *File* → *Change Working Dir...*.

Open geometry file geometry_impeller3.agdb: Click *File* → *Import Model*, select a file, click *Open*, in the menu set units to Milimeter and then click *Apply*.

Save the project to select project's name: *File* → *Save Project As...*, write *mesh_impeller.prj* and click *Save*.

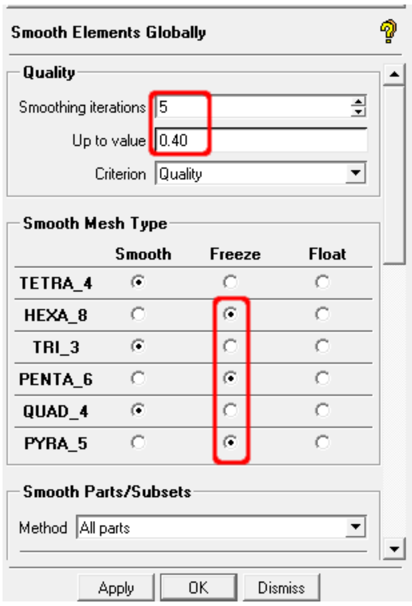

Figure 13: Smoothing the mesh.

Click *Settings* → *Model/Units* in the Main Menu. Set Topo Tolerance to 0.0008 and Triangulation Tolerance to 0.00001, then click *OK*.

Create the following surface parts (remember, the Surfaces in the Display Tree should be enabled for the selection of the surfaces):

- geom2_in,
- geom2_out,
- geom2_blade_LE,
- geom2_blade_PS,
- geom2_blade_SS,
- geom2_blade_TE,
- geom2_hub,
- geom2_shroud,
- geom2_nowalls_top,
- geom2_nowalls_bottom,
- geom2_per1,
- geom2_per2.

The surface parts are presented in Fig. 14 and Fig. 15. The inlet surface is the one at the largest value of Z. The index LE stands for leading edge of the blade (small surface), TE stands for trailing edge of the blade (small surface), the SS is suction side and the PS is pressure side. Hub is the (non-curved) surface at the impeller bottom (at the lowest value of Z), whereas shroud are the two curved surfaces opposite to the hub. The top and the bottom surfaces at the largest radius (after the TE), marked geom2_nowalls, do not represent the impeller. They are there just for numerical purposes. Create a body with name BODY_IMPELLER.

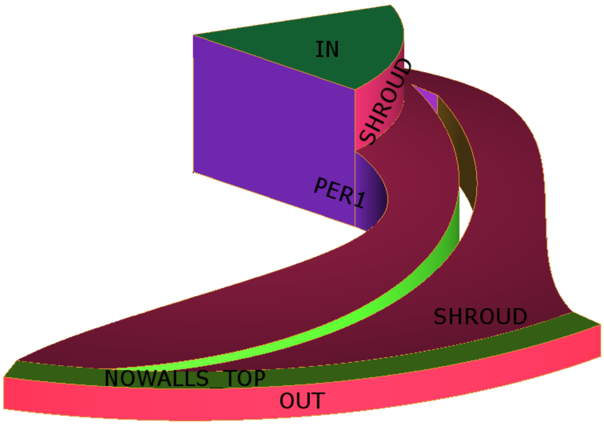

Figure 14: Surface parts of the impeller blade.

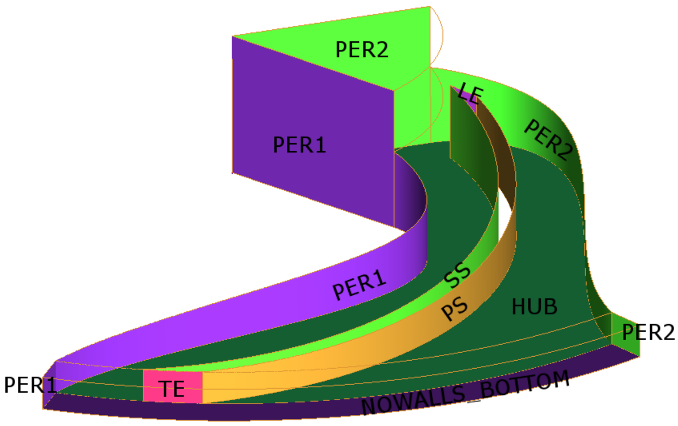

Figure 15: Surface parts of the impeller blade.

Delete two curves and six points, as presented in Fig. 16. Create part POINTS with all points in the geometry. Create part CURVES that should contain all curves in the geometry.

Set the global mesh setup: in the Main Toolbar click *Mesh* $\rightarrow$ *Global Mesh Setup*: . Set the global mesh parameters according to Fig. 17 (Max element size=4.0, Tetra edge criterion=0.05, No. of prism layers=1, Rotational period. axis = 0 0 1, angle=60). In addition, in the Global Prism Settings, set the Number of surface smoothing steps to 0. The height of the channel at the outlet from the impeller is equal to 5.8 mm, which is similar to the maximum size of the mesh element. Therefore, only one prism layer was created, which will be later

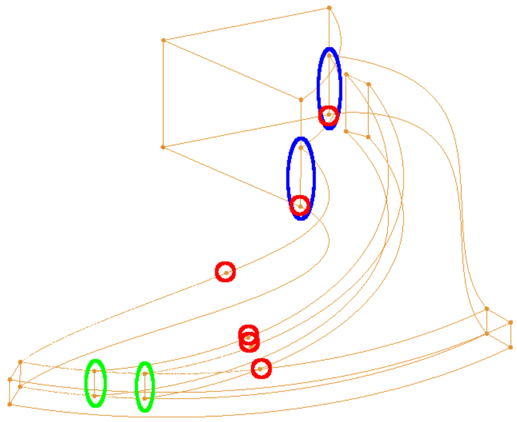

Figure 16: Curves and points. Blue: curves to be deleted. Red: points to be deleted. Green: curves to be set with Curve Mesh Setup.

subdivided into more layers. The tetra edge criterion was decreased to capture the curves of the trailing edge properly (otherwise the trailing edge might become jagged).

The next step is setting parameters of the mesh for the parts. In the Main Toolbar click *Mesh* → *Part Mesh Setup*: . A pop-up window appears. Choose parameters accurding to Fig. 18 (BODY_IMPELLER part will have a hexa core mesh, max. size=2, prism layers will be located at all walls). The size of the hub, shroud and nowalls parts is 2.5, and remains so for two layers away from the surface (tetra width). To finish, press *Apply*, then *Dismiss*.

The last step of the mesh setup will be a setup of elements sizes on two curves at the trailing edge of the impeller. In the Main Toolbar click *Mesh* → *Curve Mesh Setup*: . For the two curves indicated in Fig. 16 set the parameter 'Number of nodes' to 8.

Now the mesh can be computed. In the Main Toolbar click *Mesh* → *Compute Mesh*: . In the Compute Mesh window turn on creation of prism layers and mesh hexa-core after the tetra meshing (Fig. 11). Press *Compute*. At the trailing edge, the mesh will look like as presented in Fig. 12.

Since the prism elements consist of only one layer, we have to subdivide it. In the Main Toolbar click *Edit Mesh* → *Split Mesh*: . Set the parameters according to Fig. 20: Prism Volume Parts=BODY_IMPELLER (click on the icon), No. of layers=5. Click *Apply*. Afterwards, the mesh at the outlet from the impeller will look like as presented in Fig. 13. Note: in general, there should be 10-15 prism layers in the boundary layer due to wall function requirements.

Check the mesh for errors. Click *Edit Mesh* → *Check Mesh* (). Click *Apply*. A pop-up window appears - select the two periodic parts and click *Accept*. Confirm the deletion of unconnected vertices (click *Yes*).

Check the quality of the mesh. Click *Edit Mesh* → *Display Mesh Quality* (). The mesh quality is below 0.2.

Improve the mesh quality. Click *Edit Mesh* → *Smooth Mesh Globally* (). Set the smoothing parameters according to Fig. 13. Click *Apply*. The step can be repeated with the quality limit set to 0.1, with Prism and Pyramid elements set to 'Smooth'.

Repeat the previously described Check Mesh step and save the project (click *File* → *Save*

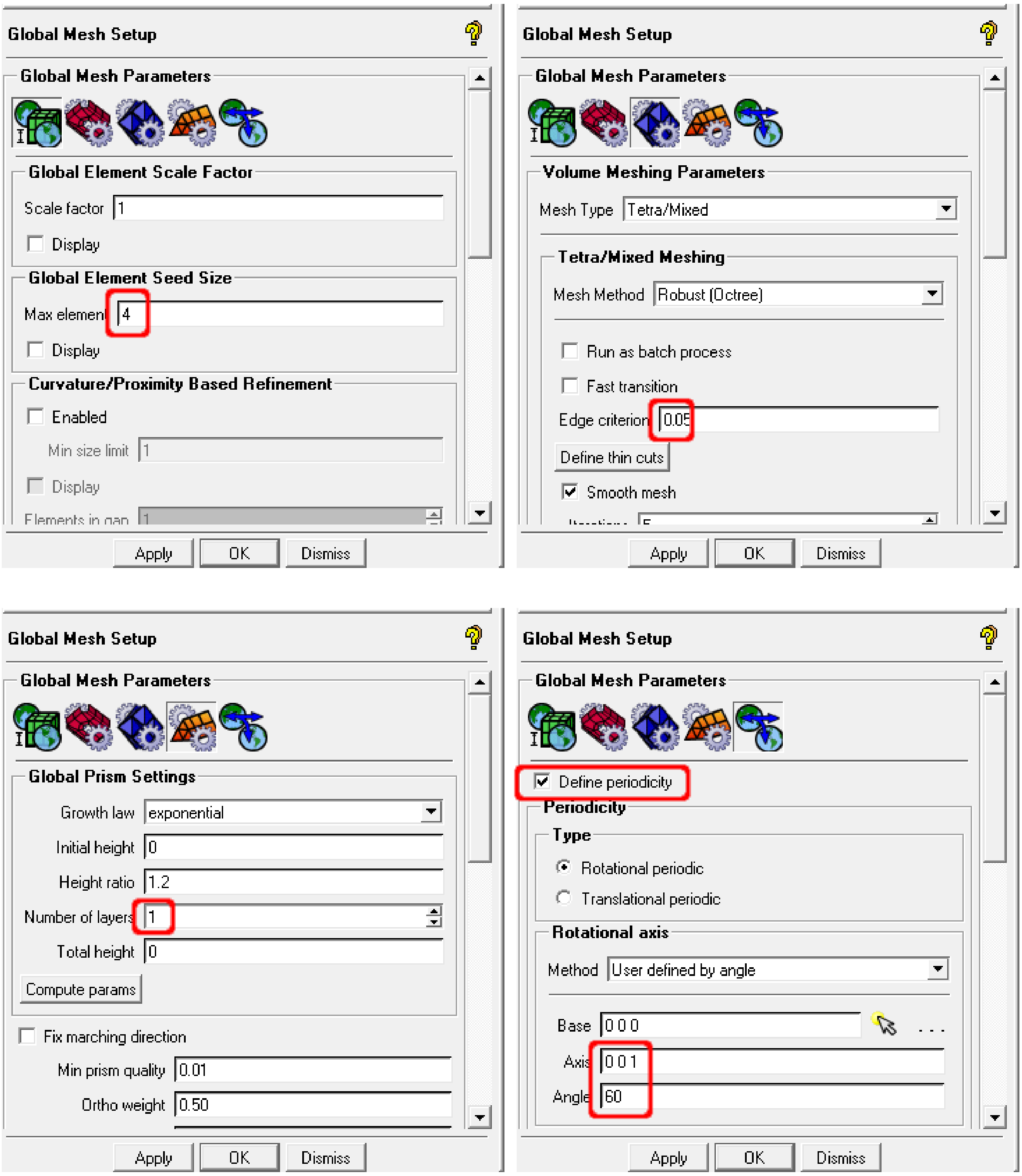

Figure 17: Global mesh setup for the impeller blade.

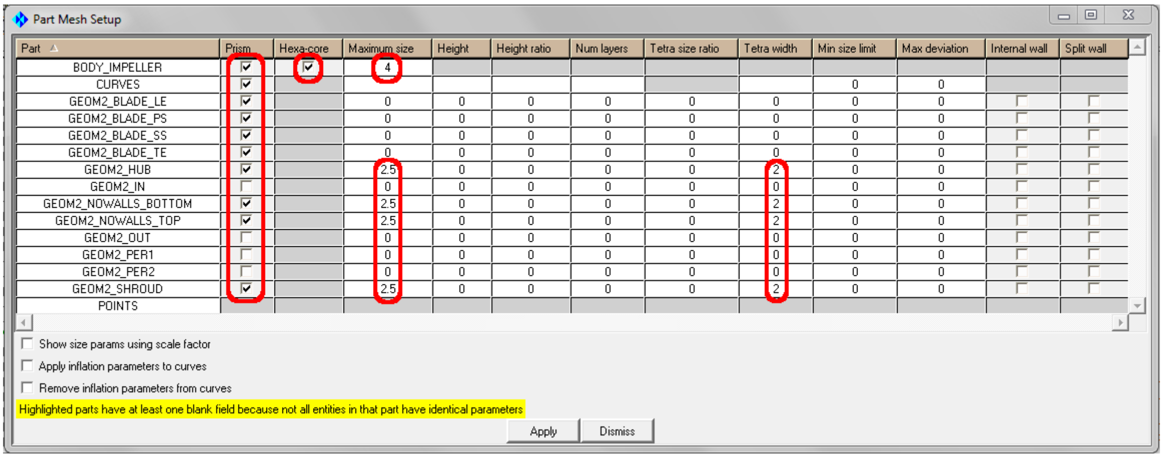

Figure 18: Part mesh setup for the impeller blade.

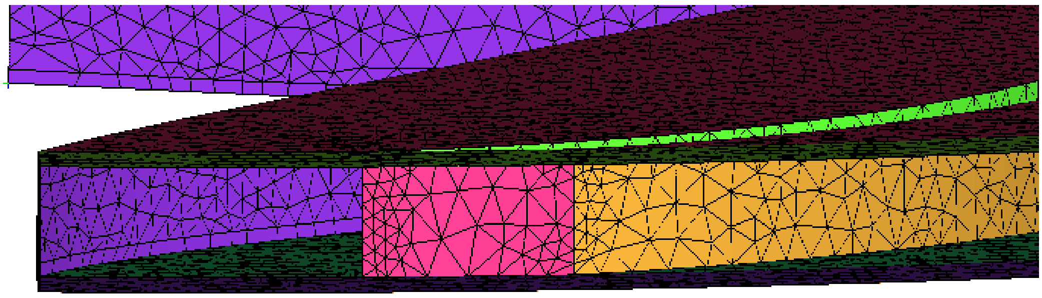

Figure 19: Single layer of prisms. Sharp trailing edge.

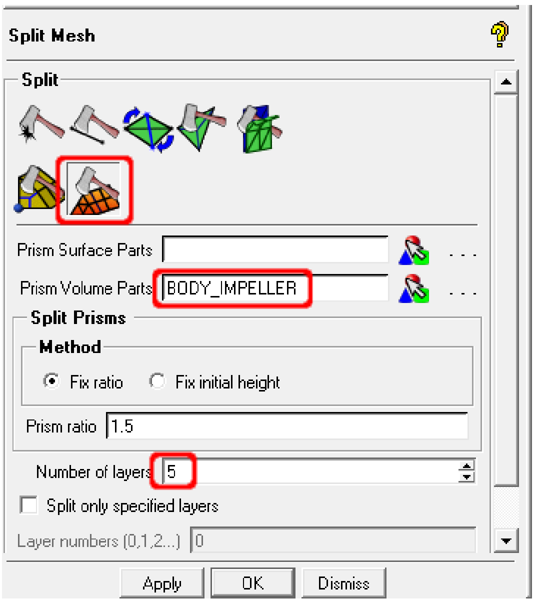

Figure 20: Split Prisms setup.

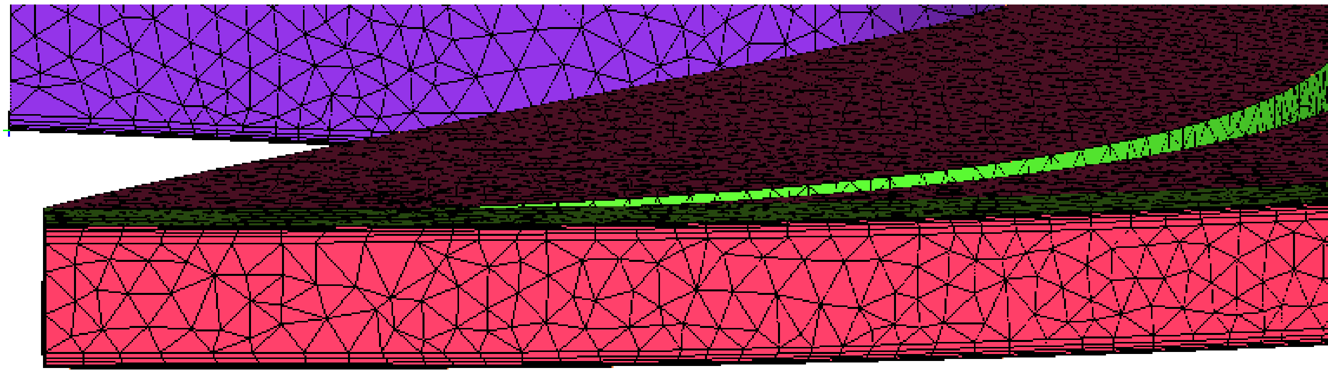

Figure 21: Prism layers at the impeller outlet after subdivision of the initial layer.

Project). As in the previous chapter, export the mesh. Close the project.

3.2.3 Computational grid 3

Start *ANSYS ICEM* by clicking *Tutti i programmi* → *ANSYS 16.2* → *Meshing* → *ICEM CFD 16.2*. Set the working directory of the third geometry file (geometry_outlet.stp): *File* → *Change Working Dir...*.

Open geometry file: Click *File* → *Import Geometry* → *Legacy* → *STEP/IGES*, select a file, click *Open*, then click *Apply* in the window presented in Fig. 4.

Save the project to select project's name: *File* → *Save Project As...*, write *mesh_outlet.prj* and click *Save*.

Click *Settings* → *Model* in the Main Menu. Set Topo Tolerance to 0.001, Triangulation Tolerance to 0.00001 and units to Millimeters. As for the inlet mesh, scale up the whole geometry by a factor of 1000 (Fig. 6), then rescale the view.

Create the following parts: geom3_in, geom3_out, geom3_top, geom3_bottom, geom3_per1 and geom3_per2. The index LE stands for leading edge, the SS is suction side and the PS is pressure side. The bottom surface is the one at the impeller bottom (at the lowest value of Z), whereas the top one is at the largest value of Z.

Create a body named BODY_OUTLET.

We will create a blocking for the block-structured mesh, with 40 evenly-distributed nodes in the circumferential direction and 25 nodes in the radial direction. For the height of the 'channel' 19 nodes will be used, with spacing 0.1 and ratio 1.3 from both sides. This will result in 19,000-node mesh. In the Main Toolbar click *Blocking* → *Create Blocks* . In the window on the left side of the screen (Create Block window) click on the icon and press *a* on the keyboard to select all the geometry. A blocking will appear in the screen (black lines in Fig. 22).

To associate blocking vertices with points click *Blocking* → *Associate* in the Main Toolbar. Associate all eight points, as indicated in Fig. 22, by clicking the in the Blocking Associations windows in the left side of the screen (use selected radio button *Point*). To select the vertex, LC in the field *Vertex*, pick a vertex (LC), then pick a point. To finish use MC.

To associate blocking edges with curves click *Blocking* → *Associate* in the Main Toolbar. Then click the in the Blocking Associations windows in the left side of the screen.

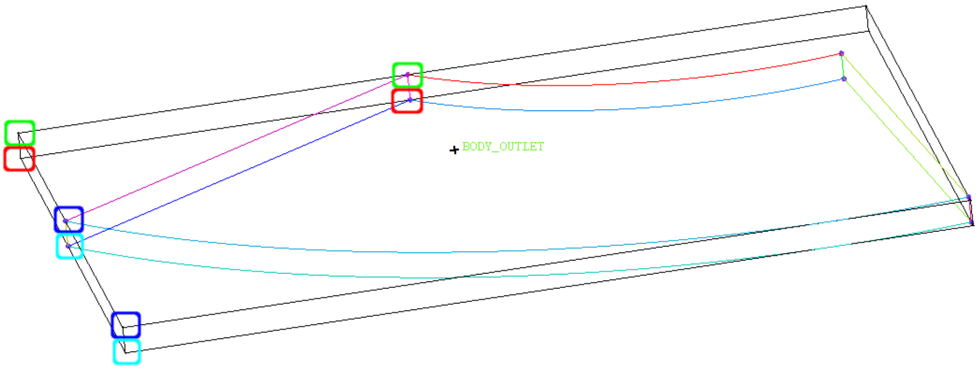

Figure 22: Blocking (black lines). Associate vertices with points (indicated with similar colours: red to be associated with red, green with green, etc.).

To select the edge, LC in the field *Edge(s)*, pick an edge (LC+MC), then pick a corresponding curve (LC+MC). To finish use MC. When an edge is associated it turns green (turn off visibility of curves to check it). To check the edge associations, in the Display Tree find *Model* → *Blocking* → *Edges* and right-click *Show Accociation*.

To define spacing of mesh elements use *Blocking* → *Pre-Mesh Params* in the Main Toolbar, then choose *Edge Params* icon () in the Pre-Mesh Params window on the left. Do not forget to turn on the tick at *Copy Parameters* (*To All Parallel Edges*).

The next step is to take care of the periodicity. Define the 60-degree periodicity, as in the right-hand-side part of the Fig. 9. In the Main Toolbar click *Blocking* → *Edit Block*: . In the Edit Block menu on the left, click Periodic Vertices: . Choose (set) the pairs of periodic vertices, as indicated in Fig. 23. The periodicity of vertices, as presented in Fig. 23, can be observed by displaying the vertices in the Display Tree Menu and choosing (RMC on Vertices) *Periodic*.

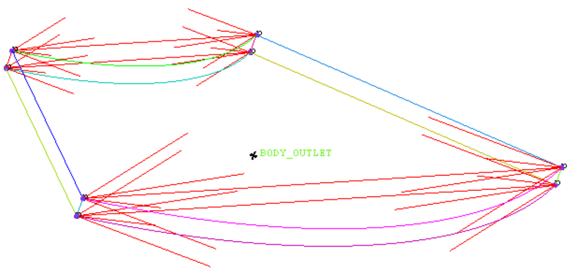

Figure 23: Periodic vertices (indicated by red arrows).

Create a pre-mesh: in the Display Tree RC on *Pre-mesh* under the *Blocking* tree, then choose *Recompute*. Turn on the visibility of pre-mesh. By using the blocking parameters described previously, the blocking should look like as in Fig. 24.

Check the pre-mesh quality: click *Blocking* → *Pre-Mesh Quality Histograms* in the Main Toolbar, choose *Quality* for the criterion. Click *Apply*. The result should be close to 1.

Convert the blocking to unstructured mesh (in the Display Tree RC on *Pre-mesh* under the *Blocking* tree, then choose *Convert to Unstruct Mesh*).

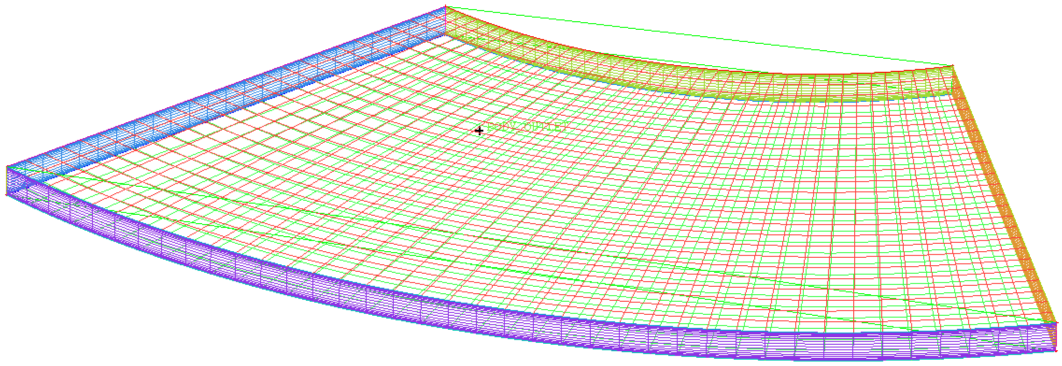

Figure 24: Pre-mesh of the outlet geometry.

Save the project (click *File* → *Save Project*). Export the mesh. Close the project.

3.3 Numerical setup - CFX-Pre

After creating the 3 meshes it is possible to create a numerical definition (.def) file. The file will contain the 3 meshes, information about the simulation type, boundary and initial conditions, the models used, etc.

Start *ANSYS CFX* from *CFX Launcher* by clicking *Tutti i programmi* → *ANSYS 16.2* → *Fluid Dynamics* → *CFX 16.2*. Set the working directory. Then click *CFX-Pre 16.2*.

Create new case (CTRL+N). If asked, choose *General* simulation type and click *OK* in the information window that follows. Save the case as 'impeller_Qd.cfx'.

To import the three meshes, use *File* → *Import* → *Mesh*. Choose the directory where mesh(es) is(are) located, set 'Files of type' to *ICEM CFD (*cfx *cfx5 *msh)* and units to mm. Then click on the mesh file to select it and click on button *Open*.

The design operating point for the pump impeller with outer diameter 190 mm [1] is defined with rotational speed ($n = 725$ rpm), flow rate at best-efficiency point ($Q_d = 3.06$ l/s) and head ($H_d = 1.75$ m).

A steady-state simulation will be performed. The default type of a simulation is a steady-state one, which can be checked by clicking the icon  in the Main Toolbar.

First of all, we have to create three domains because the inlet and outlet meshes are in non-rotating domain, whereas the impeller is in a rotating one. The domain is created by clicking the icon  in the Main Toolbar. Create the domain names as specified in Table 3, in the Domain window use the locations as specified in the table.

Domain Name	Mesh (location)
dom_inlet	BODY_INLET
dom_impeller	BODY_IMPELLER
dom_outlet	BODY_OUTLET

Table 3: Domain names and corresponding mesh(location) names.

Material selection: double-click on one of the created domains and set *Basic Settings* → *Fluid and Particle Definitions...* → *Fluid 1* → *Material* to *Water*. Click *OK*.

Heat transfer will not be used: double-click on one of the created domains and set *Fluid Models* → *Heat Transfer* → *Option* to *None*. Click *OK*.

The Shear Stress Transport (SST) turbulence model will be used: double-click on one of the created domains and set *Fluid Models* → *Turbulence* → *Option* to *Shear Stress Transport*. Click *OK*.

For the impeller domain prescribe angular velocity: double-click the domain and set *Basic Settings* → *Domain Models* → *Domain Motion* to the following values. Set 'Option' to *Rotating* and set 'Angular Velocity' to *725 [rev min^-1]* . Leave 'Rotation Axis' set to *Global Z*. Click *OK*.

To define the boundary conditions (BCs) in a specific domain, use a pull-down menu from the icon , which is located in the Main Toolbar. Mass flow rate through the inlet BC is equal to (3.06 l/s * 997 kg/m3)/6. Set the BCs according to Table 4.

Domain	BC Name	Type	Location	Boundary Details
dom_inlet	in	Inlet	GEOM1_IN	Mass Flow Rate, 0.5085 kg/s
dom_impeller	blade_LE	Wall	GEOM2_BLADE_LE	No Slip Wall
dom_impeller	blade_TE	Wall	GEOM2_BLADE_TE	No Slip Wall
dom_impeller	blade_SS	Wall	GEOM2_BLADE_SS	No Slip Wall
dom_impeller	blade_PS	Wall	GEOM2_BLADE_PS	No Slip Wall
dom_impeller	walls	Wall	GEOM2_HUB, GEOM2_SHROUD	No Slip Wall
dom_impeller	slip1	Wall	GEOM2_NOWALLS_BOTTOM, GEOM2_NOWALLS_TOP	Free Slip Wall
dom_outlet	out	Outlet	GEOM3_OUT	Aver. Stat. Pressure, 0 Pa
dom_outlet	slip2	Wall	GEOM3_TOP, GEOM3_BOTTOM	Free Slip Wall

Table 4: Boundary conditions.

To define the domain interfaces click the icon  in the Main Toolbar. Set the interfaces according to Table 5. In case the stage interface set the pitch ratio option to *None*, whereas for the frozen rotor interface specify pitch values equal to 60° at both Side 1 and 2 of the GGI.

To define the expressions, as specified in Table 6 use the icon  in the Main Toolbar. Another possibility is to import the expressions (import (append) the file expressions2016.ccl).

The impeller efficiency is calculated as a ratio between hydraulic power P_{hyd} and shaft power P_t. Hydraulic power is equal to $P_{hyd} = \Delta p_{tot} Q = \varrho g H_{imp} Q$, where Q is volume flow and Δp_{tot} a difference in total pressures between outlet and inlet from/to the impeller. The H_{imp} is the head difference expressed in [m] between the impeller outlet and impeller

Name	Region1	Region2	Interf. Model	Mixing Model
GGI_inl_imp	GEOM1_OUT	GEOM2_IN	General Conn.	Frozen Rotor
GGI_imp_out	GEOM2_OUT	GEOM3_IN	General Conn.	Stage
GGI_per1_per2	all three PER1	all three PER2	Rotational Period.	

Table 5: Domain interfaces.

Name	Expression
dens	997 [kg m^-3]
nrot	725
omega	(2*pi*nrot)/60
mFlow	abs(massFlow()@in)
TorLE	torque_z()@blade_LE
TorTE	torque_z()@blade_TE
TorPS	torque_z()@blade_PS
TorSS	torque_z()@blade_PS
TorChanWalls	torque_z()@walls
Tor	abs(TorChanWalls+TorLE+TorTE+TorPS+TorSS)
H1	massFlowAve(Total Pressure)@GGI_inl_imp Side 1
H2	massFlowAve(Total Pressure)@GGI_imp_out Side 2
H1p	areaAve(Pressure)@GGI_inl_imp Side 1
H2p	areaAve(Pressure)@GGI_imp_out Side 2
Himp	(H2-H1)/(dens*g)
HpImp	(H2p-H1p)/(dens*g)
Eff	((H2-H1)*mFlow)/(Tor*omega*dens)

Table 6: Expressions.

inlet. The shaft power (from motor to the shaft) is equal to $P = T\omega$, where T is torque and ω is angular velocity. In our case, the impeller outlet was moved slightly away (further downstream) from the true outlet.

It is possible to observe results during the simulation. To do this, click Output Control icon in the Main Toolbar. Go to 'Monitor' tab, enable Monitor Objects and click Add New Item icon at the bottom.

To monitor efficiency, enter name 'mon_eff', then use 'Option' *Expression* and in the 'Expression Value' RC(right-click) $\rightarrow$ *Expressions* $\rightarrow$ 'Eff'. Click *Apply*. Similarly, enter monitor point 'mon_Himp' for the 'Himp' expression, 'mon_HpImp' for the 'HpImp' and 'mon_Tor' for the 'Tor' expression.

The last setup that remains is the Solver Control setting . Set the parameters according to Fig. 25. In the same figure the geometry, with some BCs, is visible.

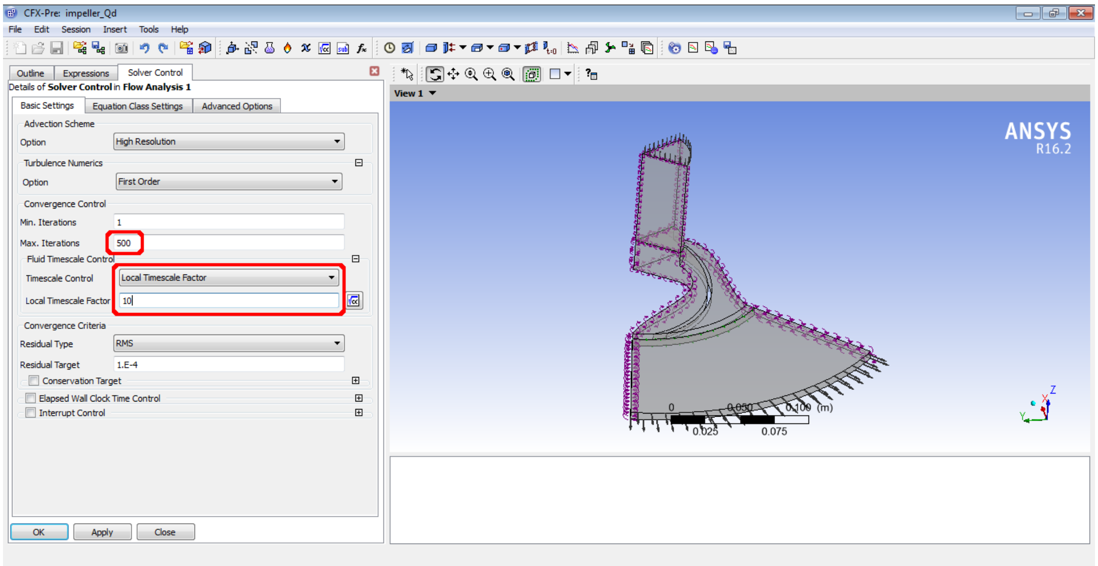

Figure 25: CFX-Pre window with Solver Control settings on the left and geometry and BCs on the right.

Save the file. Click *Define Run* icon , then *Save* to save the .def file. A *CFX-Solver* window appears. Now you can close the *CFX-Pre*.

3.4 Numerical simulation - CFX-Solver

Define the run settings, then click *Start Run*. To observe the predefined monitor points click *Workspace*$\rightarrow$*New Monitor*, define name (eg. 'eff'), then click *OK*. Finally, associate the entered name with the monitor point and choose the min. and max. values Fig. 26.

3.5 Postprocessing - CFX-Post

When the Solver is finished it is possible to check the results in CFX-Post. One option is to click the icon in *CFX-Solver*. Another option is to click *CFD-Post 16.2* in *ANSYS*

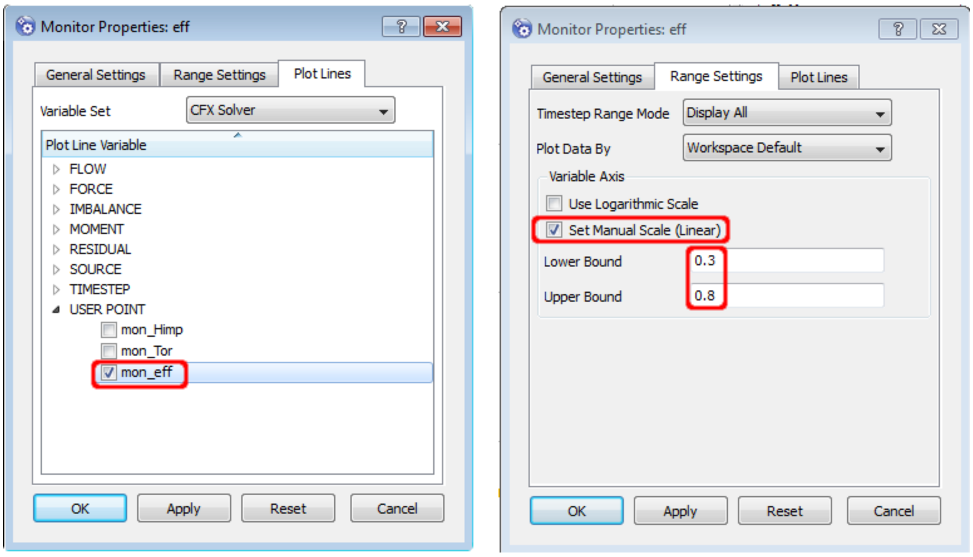

Figure 26: Define monitor points in *CFX-Solver*.

CFX-16.2 Launcher (*Tutti i programmi* $\rightarrow$ *ANSYS 16.2* $\rightarrow$ *Fluid Dynamics* $\rightarrow$ *CFX 16.2*).

The experimental results ([1]) are presented in Fig. 27. The left graph is presented for the original size of the impeller, whereas the right graph is scaled from the left one, for the scaled-up impeller by a factor of two. Our CFD results should be compared to the right graph. Comparison of design parameters of both impeller sizes is presented in Table 7.

It seems that the Head presented in Fig. 27 is actually a pressure difference, since in [1], p. 116, the author mentiones that the design total head is 2.6 m, based on velocity measurements. Total head, predicted by our cfd simulation, is approximately 2.22 m, whereas the pressure difference is approximately 1.725 m (both predicted at slightly larger outlet radius).

Experimentally obtained [1] relative speed W distribution and vectors are presented in Fig. 28 and Fig. 29. Experimental results for vorticity are presented in Fig. 30. Results for deviation angle are represented in Fig. 31.

Investigation	n $[rpm]$	R_2 $[mm]$	b_2 $[mm]$	Q_d $[l/s]$	H_d $[m]$	ϕ_I $[-]$	ϕ_{II} $[-]$	ψ_I $[-]$	Re_I $[-]$	N_s $[-]$
Present	725	95	5.8	3.06	1.75	0.122	0.015	0.33	$1.4 \cdot 10^6$	26.3
Original CR4	2900	47.5	2.9	1.5	7.0	0.122	0.015	0.33	$1.4 \cdot 10^6$	26.3

Table 7: Design parameters of original and scaled CR4 impeller [1].

CFD results for stage and frozen rotor general-grid interface at the outlet from the impeller are presented in Fig. 32 and Fig. 33.

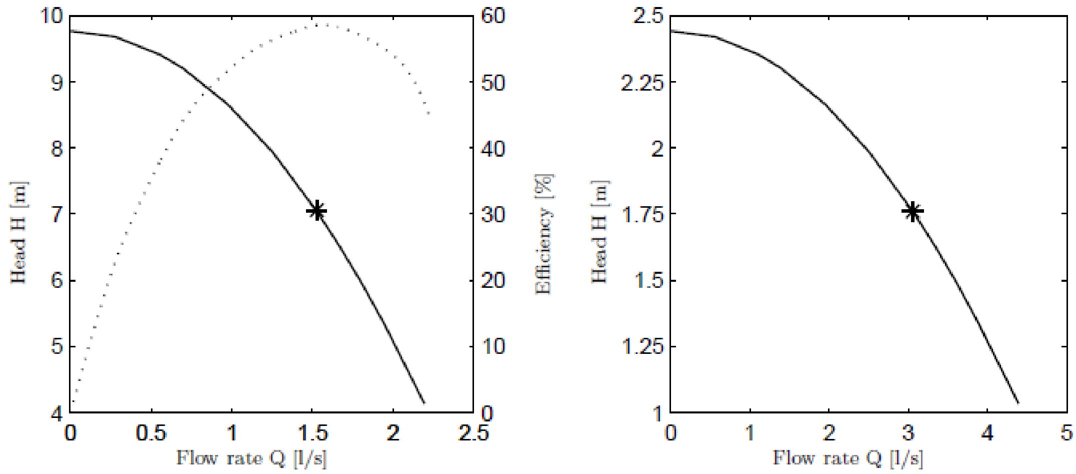

Figure 27: Experimental results [1]. Left: Performance and efficiency for original (non-scaled) CR4 impeller. Right: Performance curve for scaled test impeller as predicted by similarity laws. Stars represent design points.

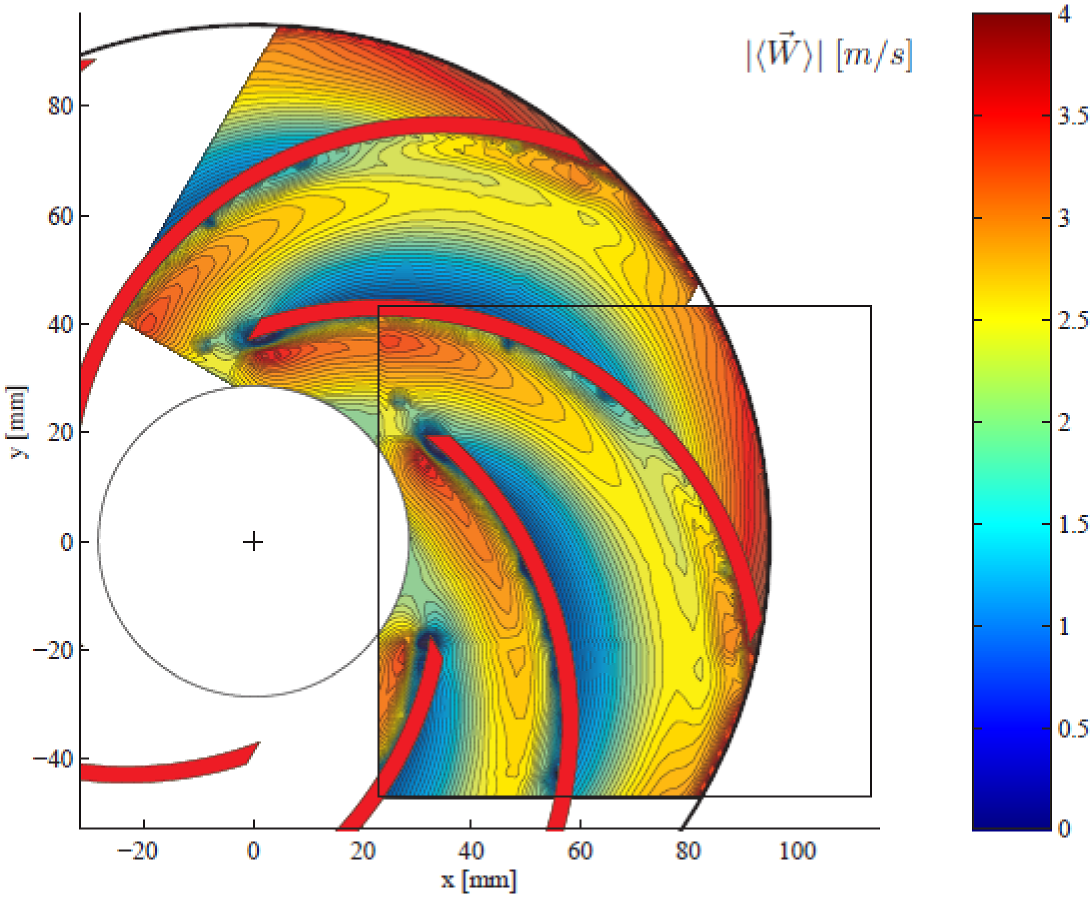

Figure 28: PIV ensemble averaged relative speed W at design operating point [1].

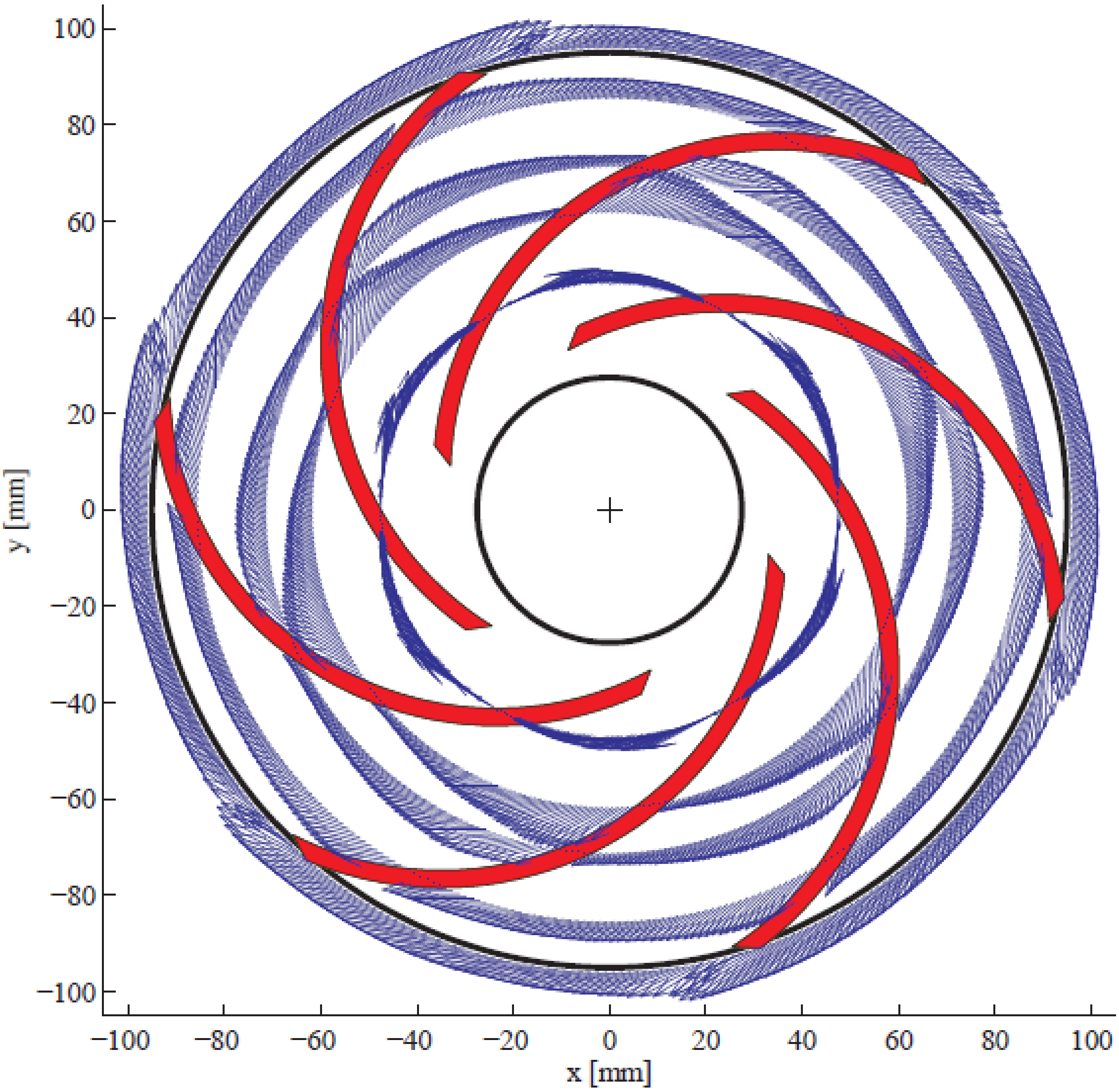

Figure 29: Vector plot of relative velocity W measured with LDV at radii $r/R_2 = \{0.5, 0.65, 0.75, 0.9, 1.01\}$ for design operating point [1].

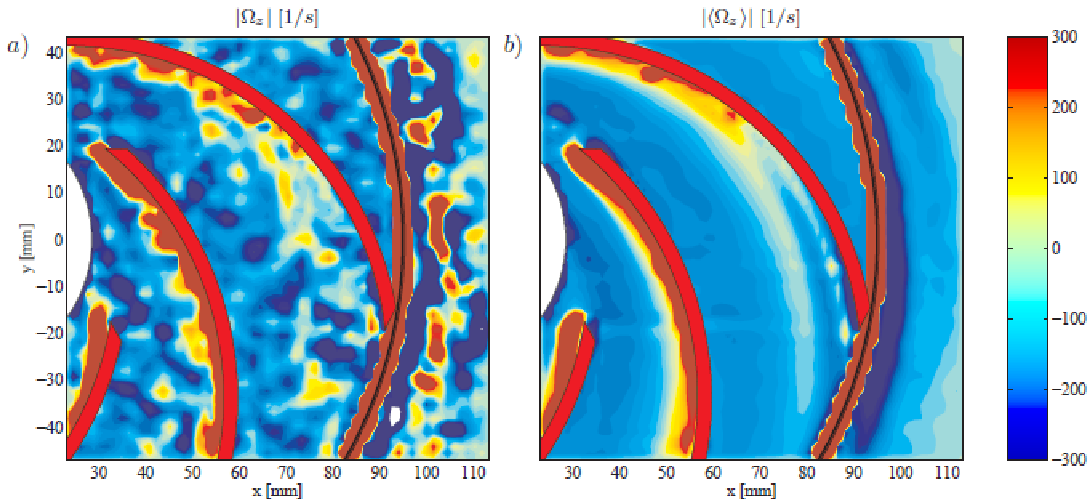

Figure 30: Vorticity in Z direction for design operating conditions [1]. a) Instantaneous sample; b) Ensemble average.

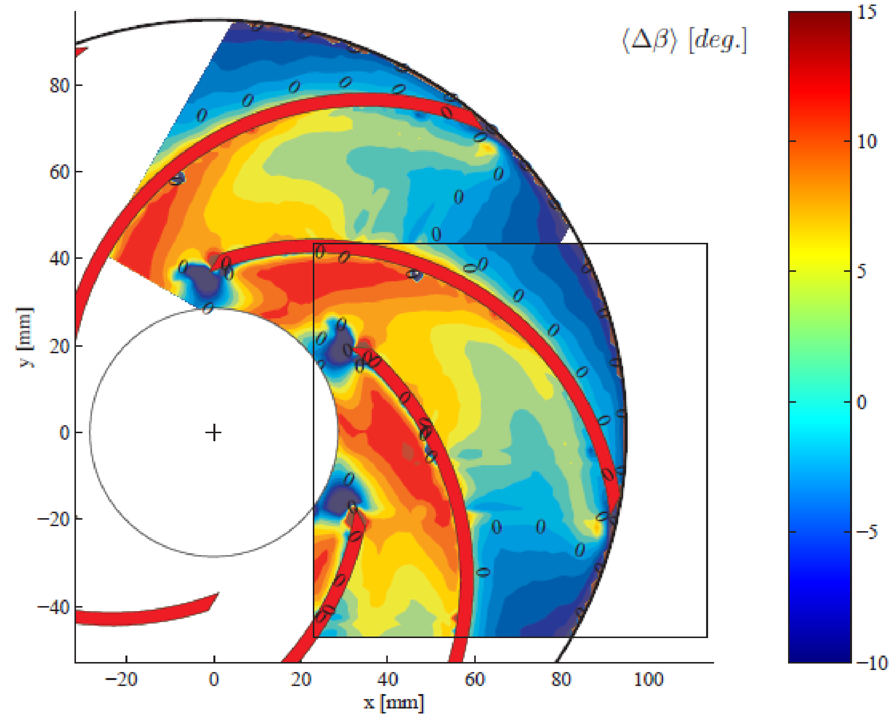

Figure 31: Ensemble averaged deviation angle $\langle\Delta\beta\rangle = \langle\beta\rangle - \beta_2$ between relative flow angle $\beta = atan(W_r/W_t)$ and the outlet blade angle $\beta_2 = 18.4°$. W_r represents relative radial velocity, W_t represents relative circumferential velocity. Experimental result for design operating conditions [1].

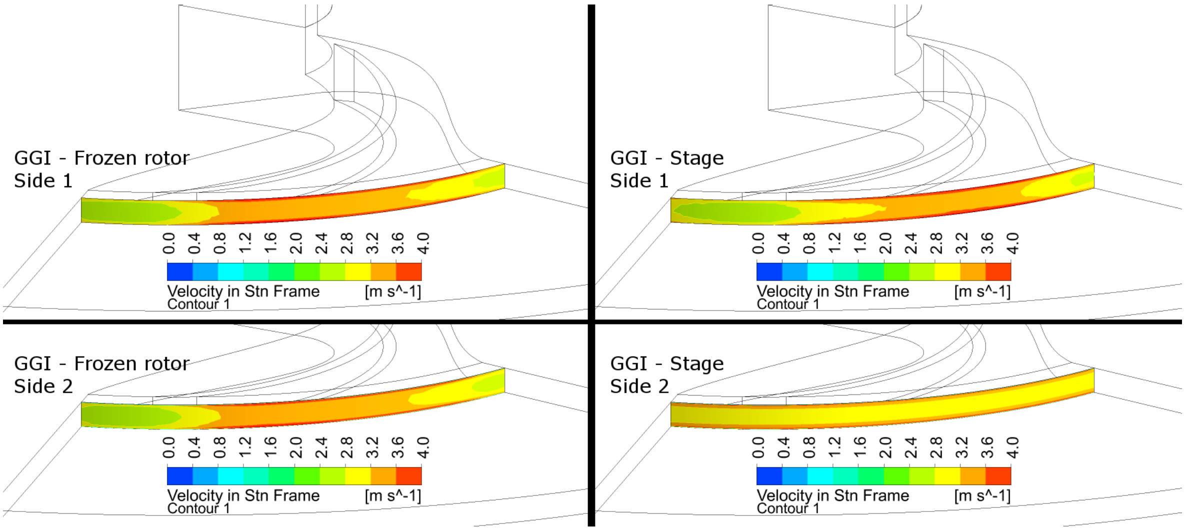

Figure 32: Comparison of Velocity in stationary frame at both sides of the GGI, for stage and frozen rotor type of the GGI.

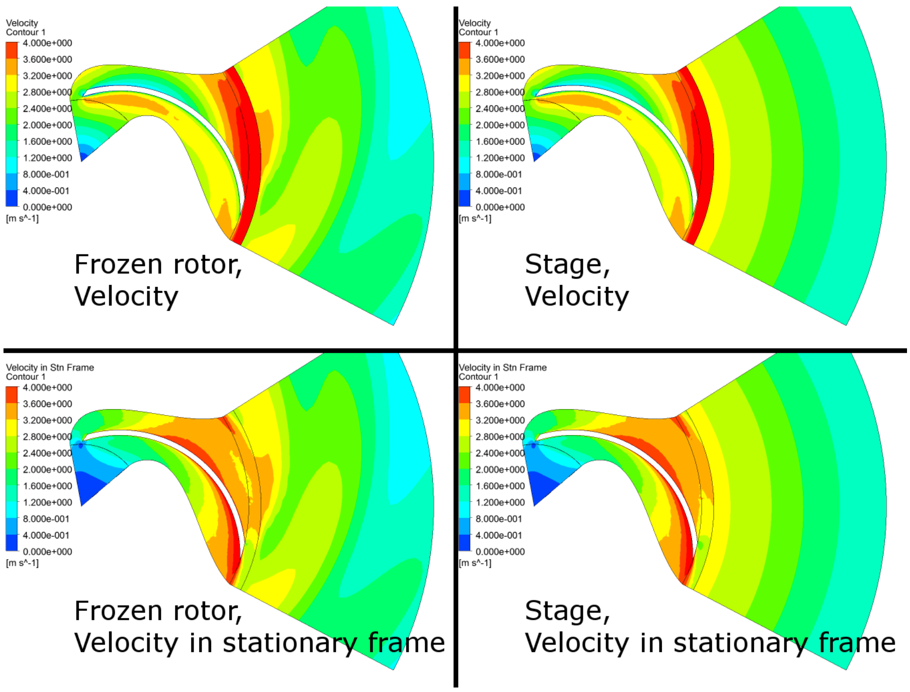

Figure 33: Comparison of Velocity and Velocity in stationary frame in a plane at Z=29 mm, for stage and frozen rotor type of the GGI.

Acknowledgement

This document was created as a result of dissemination during the ACCUSIM project. The ACCUSIM project has received funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/2007-2013/ under REA grant agreement n°612279.

References

[1] Pedersen, N. (2000) Experimental Investigation of Flow Structures in a Centrifugal Pump Impeller using Particle Image Velocimetry. *PhD Thesis*, Technical University of Denmark, Department of Energy Engineering, Lyngby, Denmark. ET-PHD 2000-05. http://orbit.dtu.dk/services/downloadRegister/5451968/Nicholas.PDF (20/05/2015).