MRL TURBINE SIMULATION

MODELLING TECHNIQUES FOR FLUID MACHINES

Andrea Rossi Marco Bonasegale Marco Belloli Alberto Casali

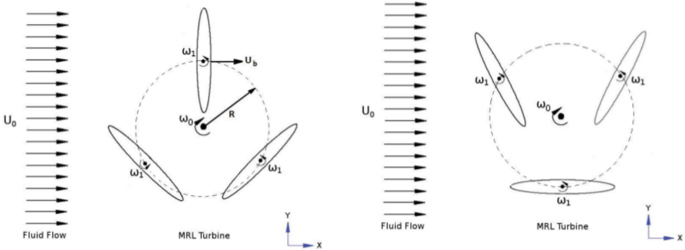
Supervisor: Gianluca Montenegro Giacomo Persico

Co-Advisor: Augusto Della Torre

Introduction

The MRL (Momentum Reversal and Lift) Turbine is a hydraulic machine. Problem data are:

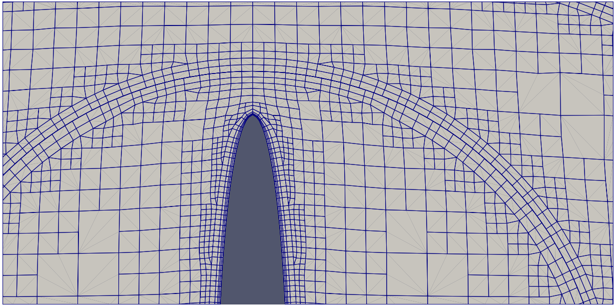
- Inlet flow velocity of $1\,\mathrm{m/s}$;
- shaft rotational speed of $100\,\mathrm{rpm}$;
- blades counter rotational speed of $50\,\mathrm{rpm}$;
- geometry of the problem.



In class work - mesh generation

The steps for the mesh generation are:

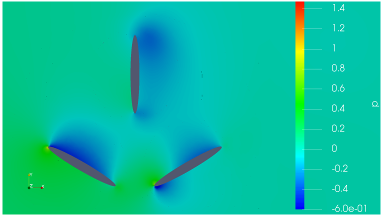
- `surfaceTransformPoints` to scale of the STL files;
- `blockMesh` and `snappyHexMesh` of each regions and blades;
- `mergeMesh` to marge the 5 previous regions;
- `extrudeMesh` to reconstruct a 2D mesh;
- `refineWallLayer` to reduce layers size near the blades.



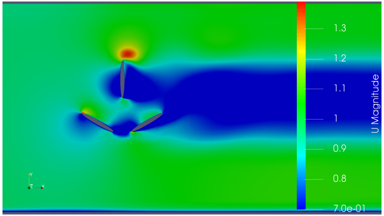
In class work - the boundaries

Inlet velocity is fixed to $1\,\mathrm{m/s}$ while all the other quatities are calculated according to the physics of the simulation.
Outlet relative pressure is fixed to $0\,\mathrm{m^2/s^2}$ (atmosferic pressure) while all the others are free to change according to upstream evolution.
Upper surface relative pressure is fixed to $0\,\mathrm{m^2/s^2}$ (atmosferic pressure) while all the others are free to change according to the flow evolution.
Bottom wall velocity is set to no slip condition to mimic adherence and so boundary layer evolution. Pressure is set to `zeroGradient`, typical condition in B.L. All the others are free.
Blades a similar no slip condition for moving walls is applied for the velocity. B.L. is isobaric without gradient of pressure. All the others depends on flow solution.

In class work - the simulation results



(a) Pressure at 2.4



(b) Velocity at 2.4

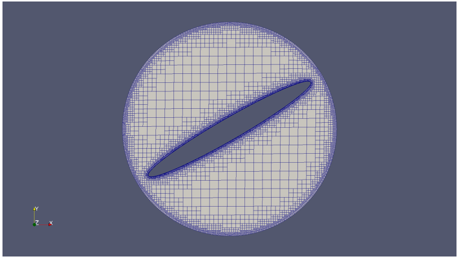
Power (Pressure) [W]	4.846
Power (Shear stress) (W)	−0.254
Power (Total) [W]	4.592

: Mean power between 1.8 and 2.4

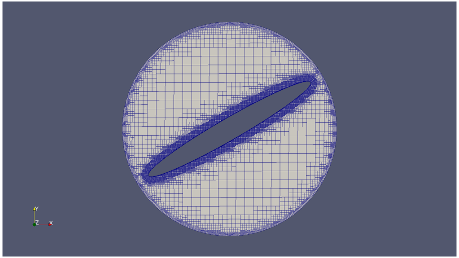
Mesh sensitivity analysis

We have performed the sensibility analysis with two different kind of grid:

- mesh without refinement region around blades;
- mesh with refinement regiorn around blades.



(c) Mesh without refinement region



(d) Mesh with refinement region

Mesh sensitivity analysis

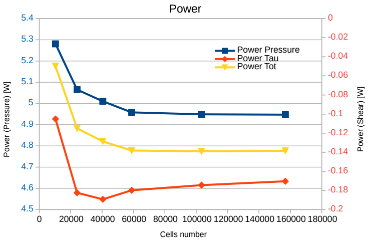
We have improved the number of cells acting on the refinement of the y axis. We have started from 20 cells and have arrived to 160 cells.

	Without region	With region
Mesh 20	10328	10328
Mesh 40	24066	25275
Mesh 60	40462	44380
Mesh 80	58877	66785
Mesh 120	103384	124366
Mesh 160	156791	196676

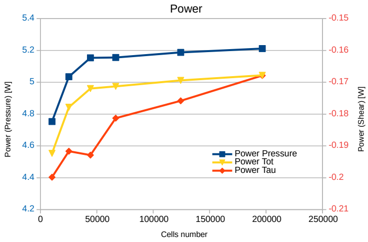
Mesh sensitivity analysis

To perform the analysis we have considered:

- total power;
- total power components (p, τ);
- total pressure drop (work and losses).



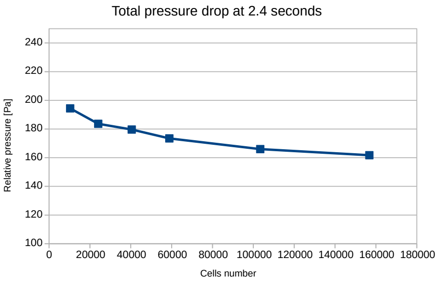
(e) Without refinement



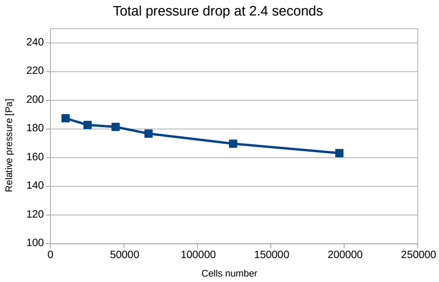
(f) With refinement

Mesh sensitivity analysis

For the total pressure drop we have considered the difference between the average at inlet and outlet at time $2.4\,$s



(g) Without refinement

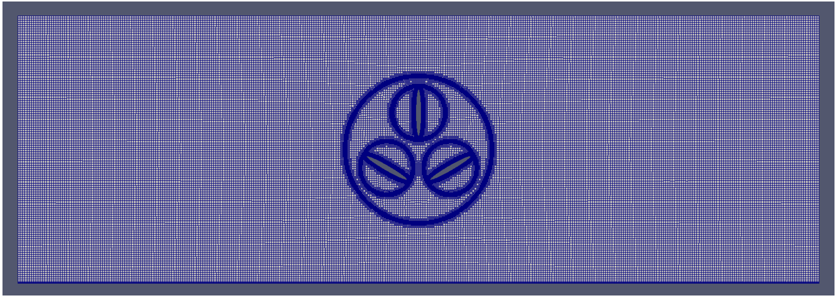


(h) With refinement

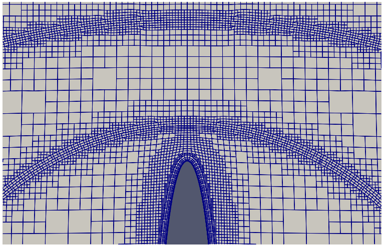
Mesh sensitivity analysis - final mesh

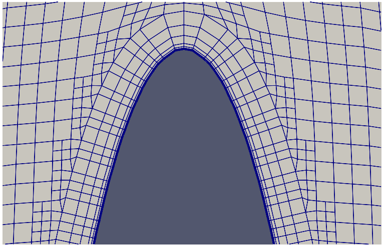
The result of the sensitivity analysis suggested up to adopt mesh 120 which is tradeoff betweeen:

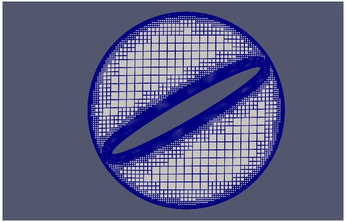
- computational cost;
- results reliability.



Mesh sensitivity analysis - final mesh



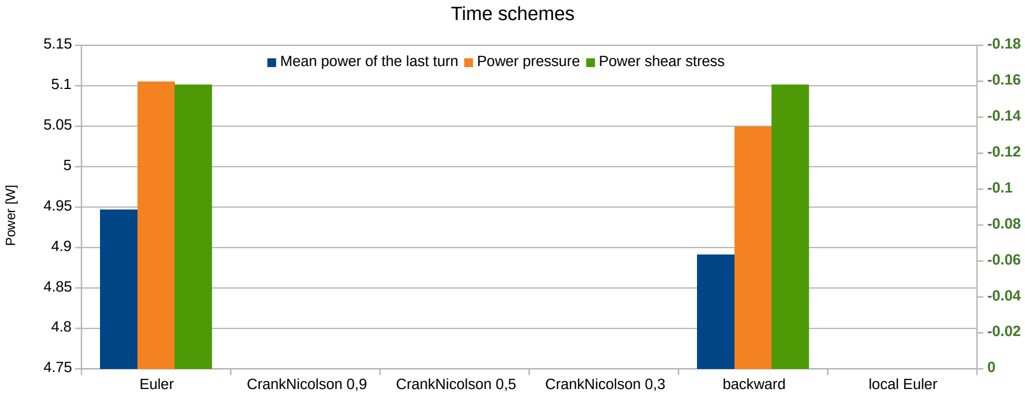




Cells	113754
Max Skewness	2.71645
Max non-orthogonality	$58.52\,^\circ$
Max aspect ratio	16.1092

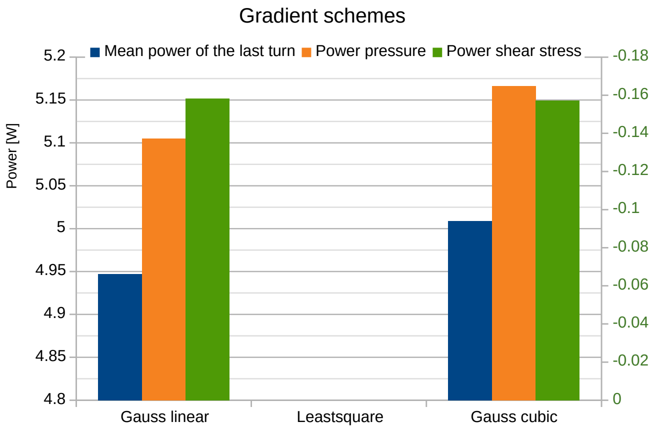
Numeric schemes - Time schemes

Time schemes define the way in which a propriety is integrated in time. Depending on the choice of the user, an old ϕ^0 or old-old ϕ^{00} solution will be required.



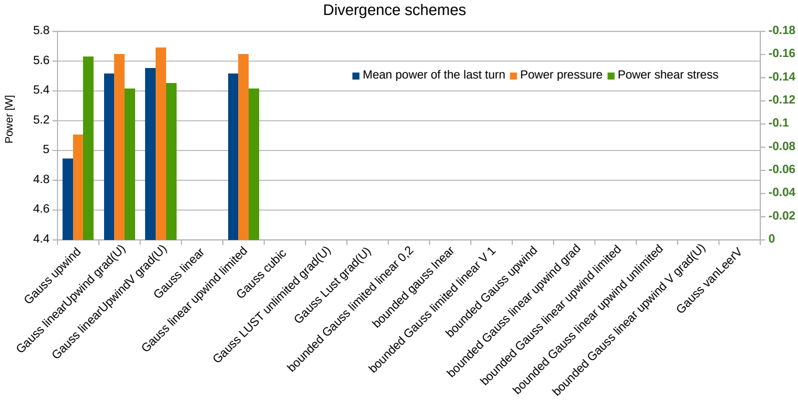
Numeric schemes - Gradient schemes

The gradient of a certain quantity ϕ rappresent the way in which that propriety is changing along a direction.



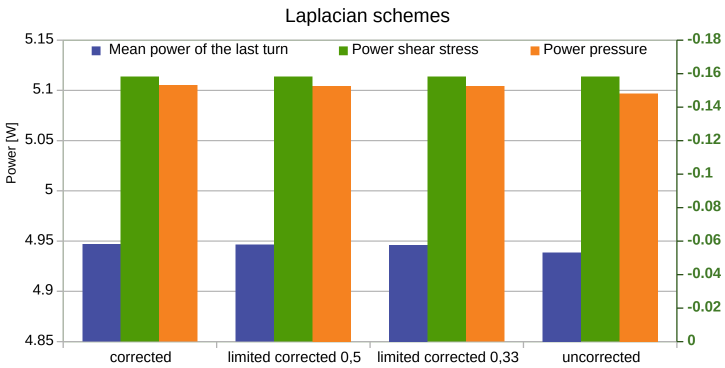
Numeric schemes - Divergence schemes

The divergence of a propriety U rappresent the rate at which that quantity is changing. We are only testing divergence schemes of the velocity since it is in our opinion the most relevant quantity.



Numeric schemes - Laplacian schemes

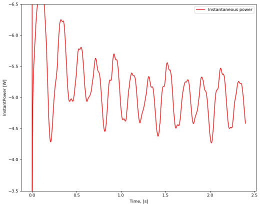
Laplacian typically it is associated to the diffusive term.
Gauss scheme is the only one available and requires the interpolation scheme used and the normal gradient scheme, defined in the proper subsection *interpolationSchemes* and *snGradSchemes*.



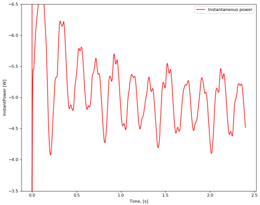
Numeric schemes - Conclusions

Finally we have chosen the following schemes:

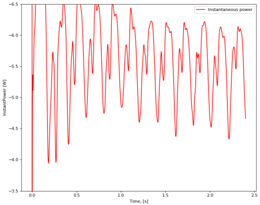
`time`	`backward`
`divergence`	`Gauss upwind`
`laplacian`	`Gauss linear corrected`
`grad`	`Gauss cubic`



(i) Reference case



(j) Backward



(k) Linear upwind

Solver parameters

Starting from in the in-class base setup, the analysis of the residuals highlights that the pressure was the most problematic quantity in terms of convergence.
To improve the results we can deal with:

- outer correctors;
- inner correctors;
- non-orthogonal correctors;
- Courant number.

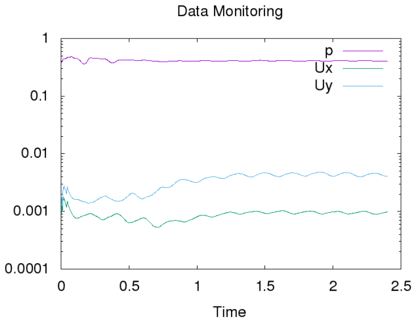
The default schemes are:

- `PIMPLE`: 50 outers, 1 inner;
- `PISO`: 1 outer, 2 inner.

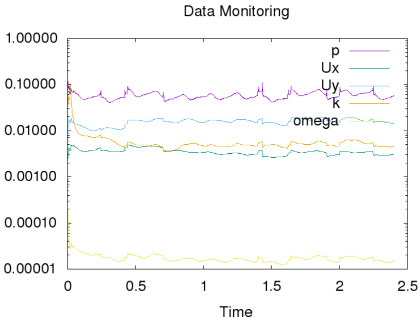
Since we want to reduce pressure residuals we have tested a different version of standard `PISO`:

`PISO`: 1 outer, 20 inner.

This modified version of the PISO allows us to increase the Courant up to 50 without comprimizing pressure residuals, but with a gain of ≈ 10 times in terms of computational cost.



(l) Pimple 50 outer 1 inner



(m) Piso 1 outer 20 inner

Turbolence

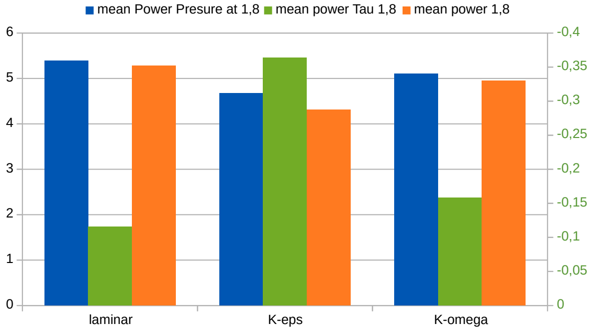
Turbolence model adopted in base case was $k-\varepsilon$.
By further investigation on mesh 120 it was highlighted that y^+ was not suitable for such grid-model combination.

	min y^+	max y^+
lower wall	0.679	4.095
blade0	0.055	0.800
blade1	0.049	0.174
blade2	0.073	0.775

Moreover we also expect to have flow detachment and recirculation because of the wide range in incidence angle.

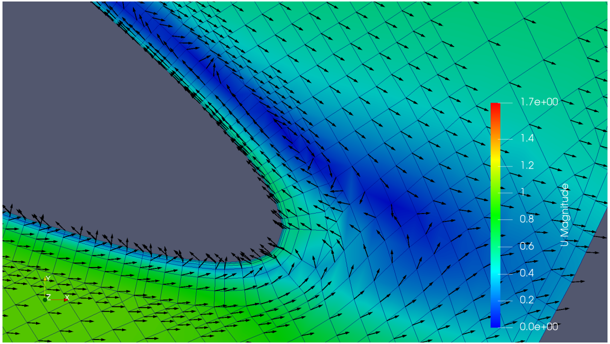
Turbolence

We will move to models which solve the boundary layer instead of applying wall functions to mimic the phenomenon in a better way ($k-\omega$ SST).
Since in turbomachinery is common the use of $k-\omega$ SST we compare it to less appropriate $k-\varepsilon$ and laminar.



Turbolence

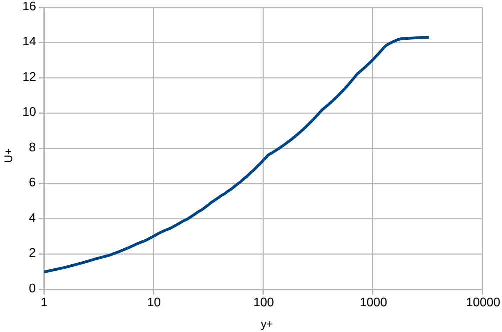
$k-\varepsilon$ model underestimate the pressure power and overestimate the shear power. $\Rightarrow$ Resulting power double penalized.
Probably the wall functions are not able to fit correctly boundary layers in all their shadings (detachment, flow recirculation, ...).



Turbolence - boundary layer

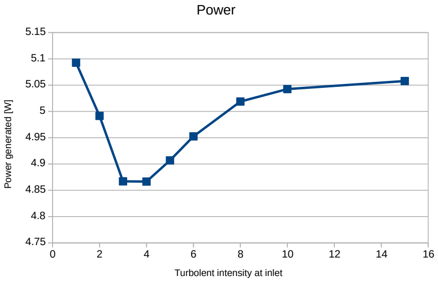
The resulting trend of the velocity U^+ inside the boundary layer with $k - \omega$ SST model reveals the typical regions of the theoretical analysis of the plane plate:

- linear for $y^+ < 10$;
- buffer for $10 < y^+ < 30$;
- logarithmic for $y^+ > 30$;

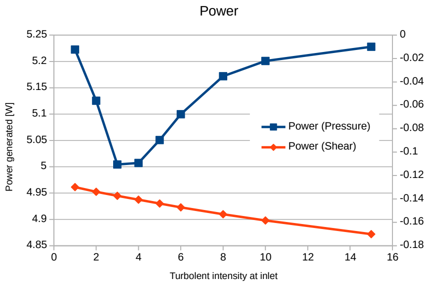


Turbolence intensity

Up to now we have performed all simulations with 5% turbolent intensity. How about changing the value?
To reply to this answer we have once more implemented a sensibility analysis based on the variation of turbolent intensity.
We range from a very moderate up to a quite intense turbolence.



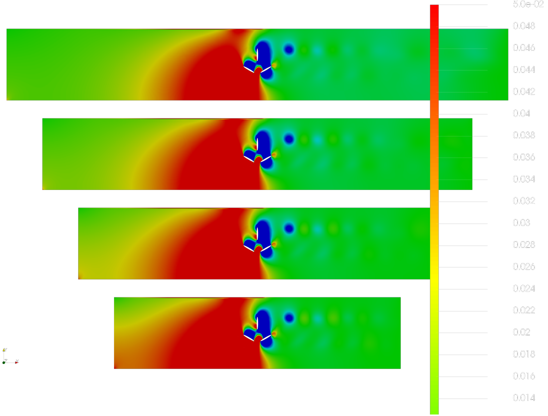
(n) Total power.



(o) Normal and tangential power.

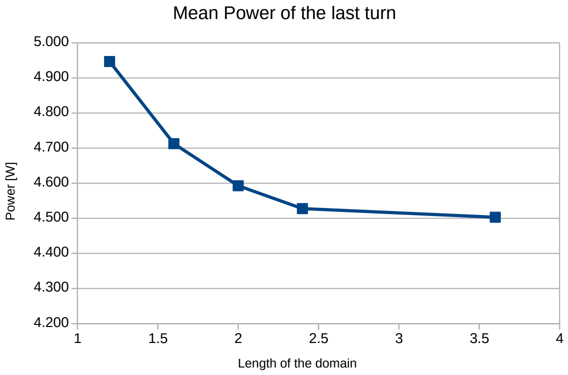
Extended domain

Looking at our symulation we can clearly see that before and after the turbine, the flow has not reach yet an unperturbed motion. And since there is the possibility that boundary will affect our solution, we run the reference case with different domain length.



Extended domain

Starting point is the case where the mesh is limited between -0.6 m to 0.6 m. Then different meshes were tested increasing dimension in symmetric way, to take into account the pressure rise effect at the bottom of the channel as well as the wake and mixing downstream.

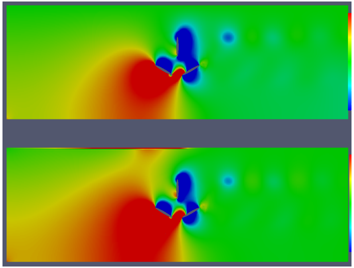


Gravity effect and enhanced boundary conditions

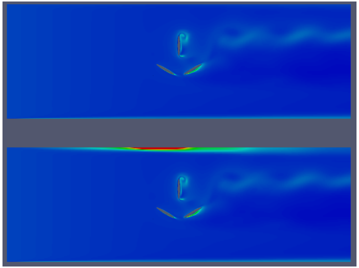
To correct rappresent the problem our idea is to consider also gravitational effect. However before doing so we need to modify boundary conditions.

- **Velocity**: change on the upper wall from `slip` to `freestreem`
- **Pressure**: change on outlet from uniformly fixed to $0\,\mathrm{m}^2/\mathrm{s}^2$ to `zeroGradient`. This is the crucial B.C. that allows us to implement gravitational effect.
- **Turbolent kinetic energy**: change on initial condition (not B.C.) to $3/2 \cdot (I\,\mathrm{U})^2 = 0.00375\,\mathrm{m}^2/\,\mathrm{s}^2$. the upper wall is set to `freestreem`.
- **Turbolent kinematic viscosity**: change only on initial condition to 0.000335 to try to speed up regime convergence.

Enhanced boundary conditions



(p) Pressure

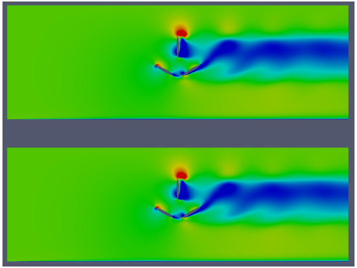


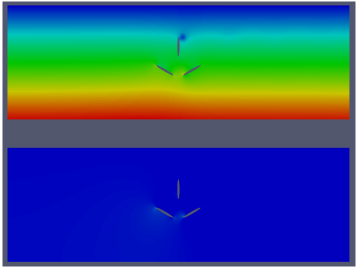
(q) Turbolent Kinetic energy

	Default boundaries	Enhanced boundaries
Power [W]	4.947	5.029
Power (pressure) [W]	5.105	5.186
Power(Shear stress) [W]	−0.158	−0.157
Total pressure inlet (2.4 s) [Pa]	650.303	632.008
Total pressure outlet (2.4 s) [Pa]	482.617	442.600
Total pressure drop (2.4 s) [Pa]	167.687	189.409

Gravitational effect

Even if gravitational force completes our model, there is no significant impact on the final results.

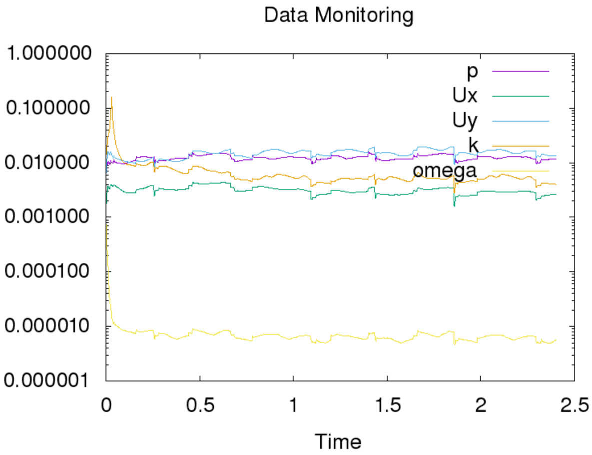




	Gravity enabled	Gravity disabled
Power [W]	5.028	5.029
Power (pressure) [W]	5.185	5.186
Power(Shear stress) [W]	−0.157	−0.157
Total pressure inlet (2.4 s) [Pa]	3059.583	632.008
Total pressure outlet (2.4 s) [Pa]	2870.286	442.600
Total pressure drop (2.4 s) [Pa]	189.297	189.409

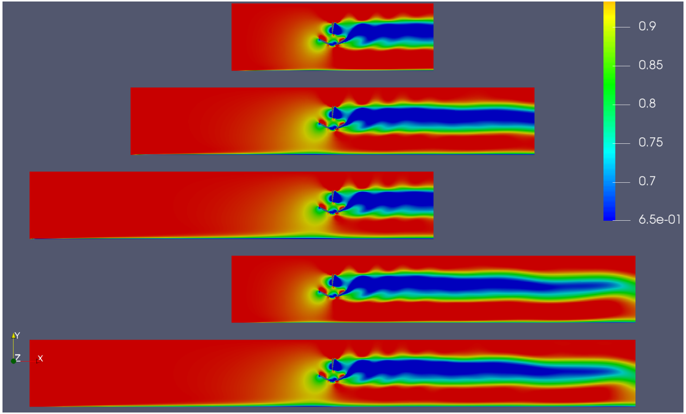
Gravitational effect - residuals

Even if the contribution of both the new B.C. and the gravity seems quite negligible, a positive effect can be noticed looking at the residuals trend.



Gravitational effect - extended domain

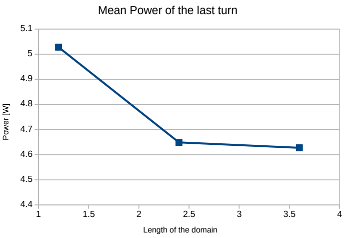
To conclude the gravitational field analysis, we have performed few simulations with extended domain. We have tried to both extends upstream, downstream and in a symmetric way the total domain. (Figure below represents the velocity field.)

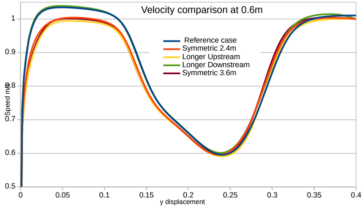


Gravitational effect - extended domain

It is interesting to notice that only upstream region has an influence on the power.

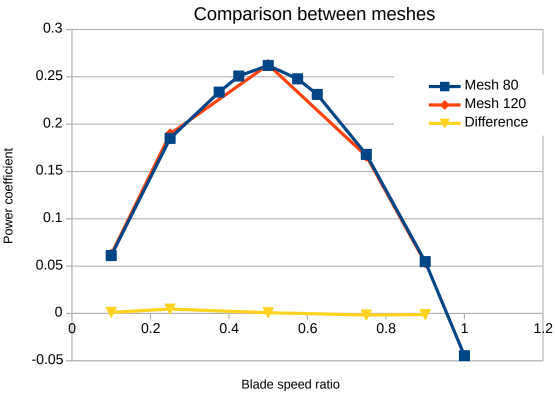
	Power [W]	Power (pressure) [W]	Power(Shear stress) [W]
Gravity enabled	5.028	5.185	−0.157
Gravity disabled	5.029	5.186	−0.157
Gravity (2.4 m symmetric)	4.649	4.803	−0.154
Gravity (2.4 m downstream)	5.033	5.190	−0.157
Gravity (2.4 m upstream)	4.616	4.768	−0.152
Gravity (3.6 m symmetric)	4.627	4.780	−0.152





Blade speed ratio

From mesh sensitivity analysis, the power is quite accurately computed even for a mesh size smaller that that we consider mesh independent. To obtain more points we have decided to run most of the simulation with a mesh 80 instead of 120. Cells number is reduced from 110K to 60K.



Blade speed ratio

For the calculation of the power coefficient we have considered as reference area:
$\mathrm{A}_{\mathrm{reference}} = 2\,\mathrm{R} \cdot t = 0.0396\,\mathrm{m}^2$ where:

$R = 0.09$ m distance between center of rotation and blade tip

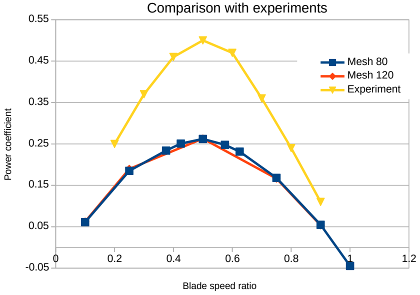
$t = 0.22$ m is blade thickness

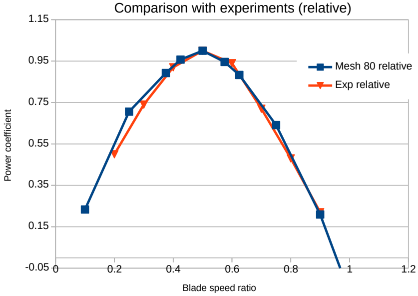
BSR	Inlet U [m/s]	$\mathrm{P}_{\mathrm{flow}}$ [W]	Power [W]	c_{P}
0.100	5.760	3783.024	231.310	0.061
0.250	2.304	242.114	44.783	0.185
0.375	1.536	71.737	16.777	0.234
0.425	1.355	49.280	12.360	0.251
0.500	1.152	30.264	7.930	0.262
0.575	1.002	19.899	4.933	0.248
0.625	0.922	15.495	3.586	0.231
0.750	0.768	8.967	1.507	0.168
0.900	0.640	5.189	0.284	0.055
1	0.576	3.783	−0.169	−0.045

: Blade speed ratio results with mesh 80.

Blade speed ratio

Even in the absolute value is almost the half respect to the experimental data, we can notice that the trend instead shows a good agreement with experimental results.





Final results

After the boundary enhanced analysis we have noticed that pressure residuals are not a preblem anymore.
We decided to reduce courant number to 5 since we can manage a single simulation which takes longer time.
To be sure to reach regime condition we have also increased simulation time from $2.4\,\mathrm{s}$ to $3.6\,\mathrm{s}$.
The domain has been expanded just upstream.

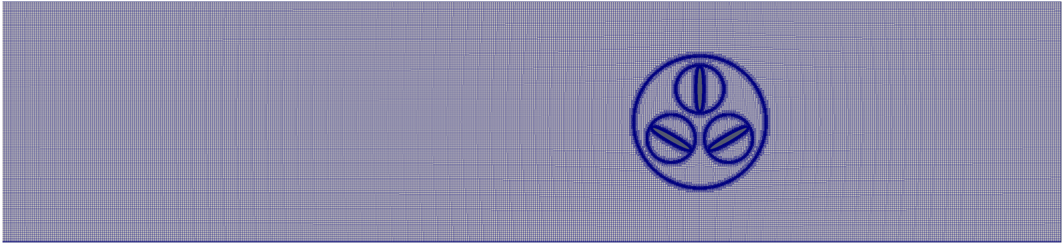
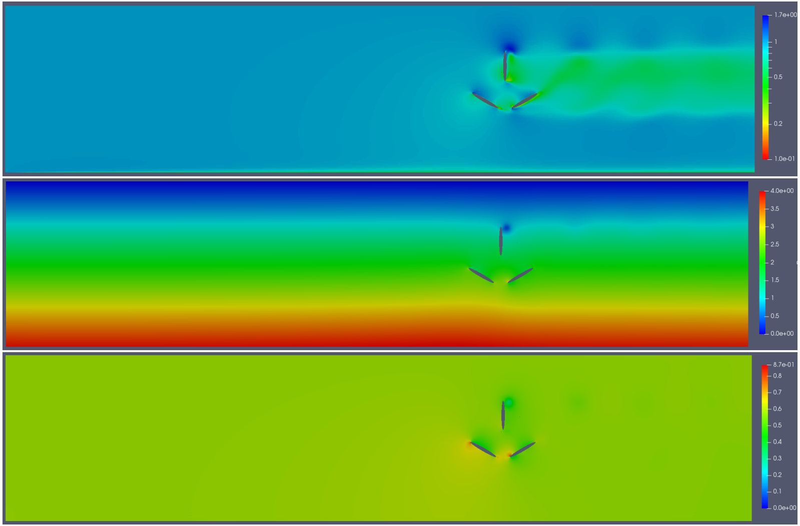


Figure: Final mesh representation.

Final results



Final results

For the final value of the power of this report we have decided to take the last turn that we have simulated.

$$\textbf{Power} = 4.493\,\text{W (Computational time} \approx 12\text{h)}$$