CFD SIMULATION OF DRAG REDUCTION IN PIPE FLOW BY TURBULENCE ENERGY PROMOTERS

Hussain H. Al-Kayiem and Javed A. Khan
Mechanical Engineering Department, Universiti Teknologi Petronas, Malaysia
E-Mail: hussain_kayiem@petronas.com.my

ABSTRACT

In oil and gas industry, transportation of crude oil from terminal to terminal is costing enormous amount of money in order to restore back the pressure, which is lost due to the inner surface friction of the pipeline, through pumping operation. In this study, a new method has been proposed to reduce the drag in pipe flow by installing energy promoter at the inner wall to change the turbulence structure in the flow. CFD simulation was used to study the drag reduction at various number and various heights of energy promoters. Mesh independency study was conducted to ensure the integrity of the result. ANSYS CFX was used to simulate the flow inside the pipeline with a section of energy promoter embedded at the inner wall. The pipeline with energy promoter is modelled using Solid works and imported to ANSYS CFX Fluid Flow to undergo simulation. With water as working fluid, the results obtained were compared with the empirical correlation results to ensure the validity of the simulation procedure. Results have demonstrated drag reduction in all cases of energy promoter's height, number, and flow rates. 1.0 mm height showed better drag reduction, and the 12 energy promoters installed on the internal surface of the pipe showed larger reduction in the drag compared to 8 and 4 promoters. The maximum drag reduction efficiency of energy promoter is approximately 7% and it is possible to further push the boundary for drag reduction efficiency limit. In summary, it is feasible to reduce drag in flow through insertion of energy promoters, and it is highly recommended to investigate the technique experimentally and extended numerically cases.

Keyword: drag reduction, energy promoter, pressure drop, pipeline, CFD simulation.

INTRODUCTION

The first pipelines were introduced in United States in 1859 to facilitate the transportation of crude oil. After one and a half century of pipeline operating practice, it is proven that the pipeline is far more efficient of huge scale transportation of crude oil as well as natural gas compared to conventional means of transportation such as rail and truck that is to be moved on regular basis [1]. Rapid development and enormous population growth has increased the demand for petroleum exponentially that will also increase the demand for petroleum pipelines. Typically, the oil is transported from one place to another through the pipelines by a series of pumping stations which usually located at every 50km of pipelines. The pumping stations are needed in the transportation process as the pump is to increase the pressure back in the pipeline due to friction [2].

There are three types of fluid flow, i.e. laminar flow, transitional flow and turbulent flow. The oil flow in the pipelines is preferably turbulent where turbulent oil flow in pipelines has several merits over laminar flow. First, the rate of building up of the deposited material will be reduced as it will be scrubbed away from walls. Second, due to constraint of costs, each grade of oil will be transported in batch by pipelines which will be used repeatedly and the turbulent flow will ensure less mixing of batches of oil compared to the laminar flow. Despite the advantages, in turbulent flow, the fluid behaves as if its viscosity is increased and this results in rise of drag in the flow. However, in turbulent flow, one thing that must be taken into account is the formation of eddies in pipelines which will cause the output flow rate is relatively lower compared to the input flow rate. Moreover, drag is caused by the friction between the fluid and the pipe wall resulting in pressure drop.

Drag reduction is carried out by the addition of (a) Polymer, (b) Riblets, (c) Compliant Surface.

i. The addition of polymer additives to the flow can reduce the turbulences friction. Even when minute amount of polymer additives is applied to the flow, the drag can be reduced drastically. In laminar flow, the viscosity of the flow could be increased which is caused by the dissolved polymers and hence, the drag is increased instead of decreased [3]. Polymer additives, which is also known as drag reducing agent has been applied widely to daily life application due to its drag reducing nature, i.e. oil pipelines, oil well operations, airplane tank fuelling. Without addition of drag reducer in airplane tank fuelling operation, would take up as much as twice time that with addition of drag reducer [4]. However, polymer additives have their own shortness. In order to reduce drag in flow, it changes the physical and also chemical properties of the flow and this is totally unacceptable in some industries such as pharmaceutical and food industry which requires the properties of the fluid unchanged to prevent any undesirable side effects to human bodies and environments. Despite its remarkable performance, the polymer additives in the flow also undergo mechanical degradation. This phenomenon is due to the polymeric chain that is playing the main role in drag reduction has undergone scission process caused by turbulence flow and hence, the percentage of drag reduction will reduce. Therefore, it is necessary to reintroduce the polymer additives into the flow in order to maintain the desirable drag reduction which will incur extra costs.

(ii) Riblets are generally longitudinal microgrooves that etched onto the wall surface. The application of riblets in drag reduction is inspired by the shark's skin which is made up of streamlined V-shaped tooth like grooves. It is learnt that this tiny groove has greatly reduce the drag when this predator is moving underwater and allows it to hunt with less effort. The V-shaped grooves are varied in shape at each parts of sharks which is to optimize the drag reducing effect. There are several theories proposed to explain the mechanism of how riblets reducing the drag. It is found that the riblets induce restrictions to the span wise movement of quasi-streamwise vortices [5]. On the other hand, Park and Wallace suggested another theory that has been proposed claimed that the drag reduction by riblets is achieved through viscous interaction [6]. Several researchers claimed that riblets are able to reduce the drag from 7% up to 10% and it is relatively low compared to the drag reduction performance by polymer additives. Moreover, the cost of reconstructing the inner surface of the pipelines is considered ridiculously high and once the low performance barrier of riblets could be overcome, the cost of restructuring inner surface of pipelines would be justified as it is permanent solution to solve the drag issue [4].

(iii) Compliant surface is considered as one of the non-intrusive as well as passive control of drag reduction methods. Compliant surface is made up of elastic walls and it was first discovered by Kramer in experiment to simulate the drag reduction nature of dolphin's skin [7]. Kramer claimed that under specific condition, compliant surface is able to reduce the drag up to 60%, however, the sensitivity to the pressure gradients is very high and the results produced is not consistent. It is observed that the transition period from laminar flow to turbulent flow would be delayed with huge factor and it is possible to achieve drag reduction [7]. Despite the several claims by other researchers, it is found that by prolong the averaging interval more than 700 viscous times, the drag reduction phenomenon will be starting to fade and he claimed this as apparent drag reduction [8]. This method is rarely applied due to the complication of the experimental set up and also slightly higher drag reduction compared to what can achieve with riblets.

In their recently published paper, [3] have presented the various methods of fabricating drag reduction surfaces covering biological sharkskin morphology mainly involving direct bio-replicated, synthetic fabricating, bio/micro-rolling, enlarged solvent-swelling, drag reduction additive low-releasing, trans-scale enlarged three-dimensional fabricating, flexible printing, large-proportional shrunken bio-replicating, ultraviolet (UV) curable painting, and stretching deformed methods.

In this project, the authors have studied the effect of insertion of protrusions in the pipelines to reduce the drag by changing the turbulence structure of the flow. How the height of the EPs affect the flow structure is the research question to be answered by this study.

The objective of the present work is to realize the height effect of streamlined EPs on the drag reduction in pipe flow. CFD simulation of water flow through protrusion ring that is installed on the inner surface of oil pipelines would be conducted. The drag reduction enhancement was predicted and compared with the case of seamless pipe (without EP) at various flow rates, various EP heights and various numbers of EPs. The results have been presented in terms of DR% versus Re.

It is expected that this research might introduce to the industry a new insight on drag reducing in pipelines that can save enormous of energy wasted to overcome the drag in fluid transportation. Its advantage that it is permanent, easy to manufactured and installed, and doesn't required further process, as in the case of polymer additives.

METHODOLOGY

a) Numerical simulation

In ANSYS CFX, Navier Stokes equations, in their conservation forms, were solved by the software. Mass, momentum and energy conservation instantaneous equations are presented in their steady state assumption and it can be written in stationary frame:

1) Continuity Equation

$$\nabla \cdot (\rho U) = 0 \tag{1}$$

2) Momentum Equation

$$\nabla \cdot (\rho U x U) = -\nabla \mathrm{p} + \nabla \cdot \tau + S_M \tag{2}$$

Stress tensor, τ is correlated to the strain rate by

$$\tau = \mu \left(\nabla \mathrm{U} + (\nabla \mathrm{U})^T - \frac{2}{3} \delta \nabla \cdot U \right) \tag{3}$$

3) Total Energy Equation

$$\nabla \cdot (\rho U h_{tot}) = \nabla \cdot (\lambda \nabla \mathrm{T}) + \nabla \cdot (\mathrm{U} \cdot \tau) + \mathrm{U} \cdot S_M + S_E \tag{4}$$

where, h_{tot} is the total enthalpy, and the term $\nabla \cdot (U \cdot \tau)$ is called viscous work term and represents viscous stresses work. The term $U \cdot S_M$ is neglected as it represents work due to external momentum sources.

Following steps were taken in completion of this project:

1) Identify general issues of conventional drag reduction method.
2) Pick up 3D modelling software, AutoCAD and CFD simulation software, ANSYS CFX Fluid Flow.
3) Design, model, and generate mesh of the Pipeline with Energy Promoters embedded inside at the inlet.
4) Perform mesh independency study and validation & verification of the simulation procedure.
5) Run the simulation at various flow conditions, and design conditions to conduct parametric study

assuming: The flow is in horizontal pipe and fully developed. The fluid is taken as Newtonian fluid. The flow is incompressible, steady state, and no- slip condition at the wall.

6) Classify, analyse and compare the results.

The proposed EP has a shape similar to the computer mouse, as shown in Figure-1.

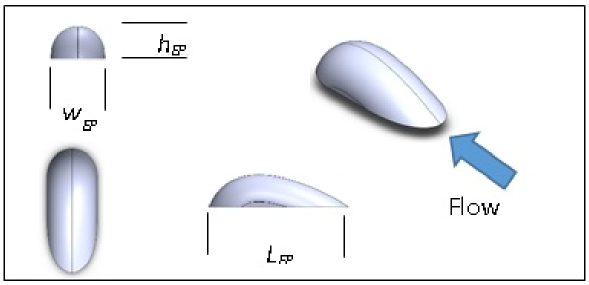


Figure-1. Proposed design of energy promoter.

The EP has geometries of width, W_{EP} = 2 mm, length, L_{EP} = 5.0 mm and two cases of height, h_{EP} = 1.0 and 2.0 mm have been simulated and discussed in this work.

The pipe segment simulated in the present work has 20.0 m long, 200.0 mm inner diameter, and 10.0 mm thickness. It was modelled using Solidworks and imported to ANSYS CFX with IGS format. Same procedure has been followed for the EPs, which were created in Solidworks and imported to ANSYS CFX. ANSYS is then generate the discretization of the flow region and generate the mesh. Figure-2 shows the arrangement of the EPs in the inlet section of the pipe.

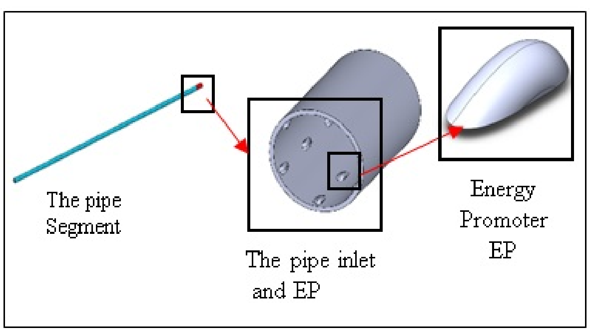


Figure-2. Conceptual drawing of pipeline with region where energy promoters are placed. To note that in this study, one raw of 4, 8 and 12 EP have been investigated.

b) Mesh independency study

To achieve mesh independence, to ensure accurate simulation results, the number of elements of the fluid in the pipe was varied. In the present investigation, change of pressure drop per length of 20 m was adopted as an indicator for mesh independent study. The number of elements was increased starting from 2,000,000 elements up to 13,000,000 elements. The selected case was with 4 EPs and the fluid velocity was 0.7 m/s. The pressure gradient results at various numbers of elements have been predicted and tabulated, as in Table-1, and plotted as in Figure-3.

Table-1. Data of mesh independency study.

Number of Elements	$\Delta P/\Delta L$ (Pa/m)	Percentage Error (%)
2,000,000	5.144	-
3,000,000	4.235	17.70
4,000,000	3.727	12.00
5,000,000	3.452	7.38
7,000,000	3.152	8.70
8,000,000	3.053	3.14
9,000,000	2.977	2.55
10,000,000	2.917	2.02
11,000,000	2.87	1.61
12,000,000	2.83	1.39
13,000,000	2.82	0.4

From Figure-3, it could be observed that the predicted results of the pressure gradient are dependent of the number of elements till around 7,000,000 elements. Further increase in the number of elements shows reduced influence on the simulated pressure gradient.

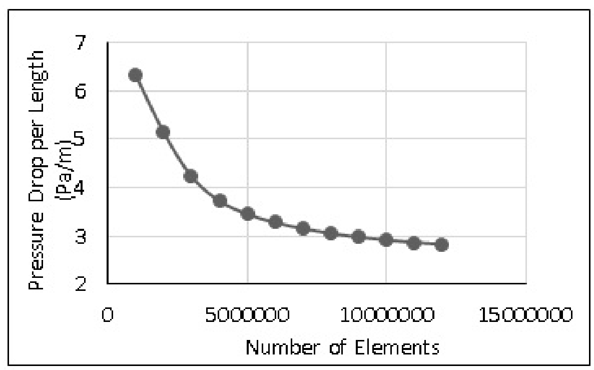


Figure-3. Pressure loss per length versus number of elements.

The predicted pressure gradient becomes less dependent on the number of elements after 7,000,000. Furthermore, the approximated computation time for number of elements around 12,000,000 is around 2.0 hours. By increasing the number of elements larger than 12,000,000, the computation time increases largely. Hence, the decision was made on 12,000,000 as the suitable setup of the simulation. In summary, by maintaining the same mesh setting, the integrity of the simulation results can be maintained. Any larger number of elements beyond that point will not have any significant effect on the accuracy and consistency of the simulation results

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

c) Verification and validation of simulation model

To verify and validate the simulation procedure, the results of the pressure gradient predicted by the simulation were compared with the pressure gradient results obtained from mathematical calculations. The pressure drop is calculated as ΔP/ΔL, where;

$$\Delta p = \rho g h_{losses} = f \frac{\Delta L}{D} \frac{v^2}{2} \rho \quad (5)$$

The friction factor, f was predicted using Colebrook equation, as:

$$\frac{1}{\sqrt{f}} = -2.0 log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \quad (6)$$

The surface roughness, ε, as adopted from the industrial pipe application, is 0.00015m. Equation 6 was solved iteratively.

d) Parametric study

There are four factors that are expected to have effect on the drag reduction efficiency of the pipeline had been identified for parametric studies through CFD analysis. The parameters included are: (a) Height of Energy Promoter; (b) Number of Energy Promoter and (d) Flow rate in oil pipelines. In this project, the drag reduction efficiency is indicated by the pressure drop per length throughout the whole pipeline.

RESULTS AND DISCUSSIONS

Verification and validation of simulation model

By varying the velocity of the model from 0.3 m/s to 1.3 m/s with interval of 0.2 m/s, the theoretical pressure drop per length and simulation pressure drop per length for different velocities are tabulated as shown in Table-2 and a linear proportional graph could be plotted with 45 degree of gradient as shown in Figure-4 which indicates the results obtained from the simulation model are in excellent agreement with the results obtained using analytical calculation.

Table-2. Validation of the simulation procedure by comparison between calculated and simulated pressure gradient.

Velocity (m/s)	Theoretical ΔP	Simulation ΔP	% of difference
0.3	5.021	5.005	0.32
0.5	13.124	13.11	0.11
0.7	24.92	24.94	0.08
0.9	40.39	40.42	0.07
1.1	59.52	59.55	0.05
1.3	82.31	82.34	0.04

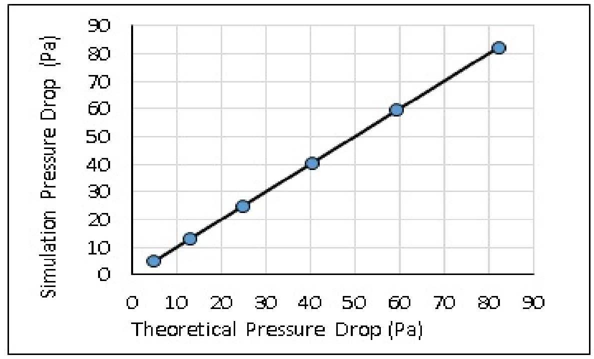


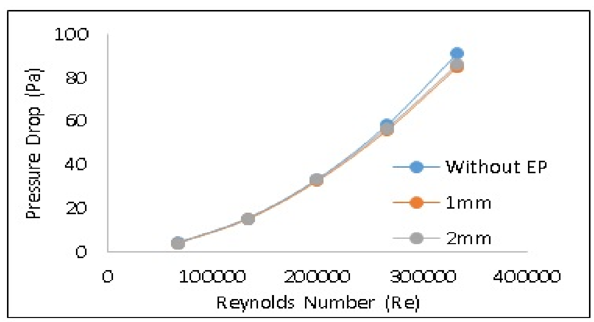
Figure-4. Theoretical pressure drop per length versus simulation pressure drop per length.

Hence, it is safe to assume that the simulation model truly represents the real system.

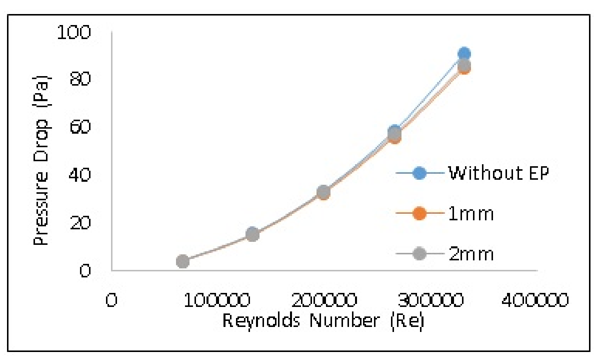
Effect of heights of energy promoters

Figure-5 displays the pressure drop across the 20 m pipe segment at various Re. Three different cases of EPs numbers, (a), (b) and (c) for 4, 8 and 12 EPs, respectively have been simulated. In each case, the flow rate was varied.

The 4, 8, and 12 EP have been arranged in the same circumferential, as one raw. The pressure drop, for each case, was estimated by predicting the pressure difference over 20 m of the pipe length. The pressure drop per length of pipeline without energy promoter has been used as the benchmark to estimate the DR reduction percentage.



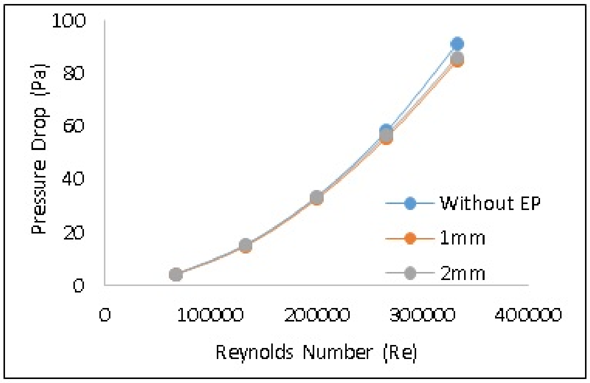
(a) 4 EPs



(b) 8 EPs

VOL. 11, NO. 24, DECEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com



(c) 12 EPs

Figure-5. Simulation results showing the effect of height of energy promoters on the pressure drop in pipeline. (a) using one raw with 4 EPs, (b) using one ring with 6 EPs, and (c) using one ring with 12 EPs.

It is observed that the predicted pressure drop, for cases of 1.0 mm and 2.0 mm EP height, is irregular. The overall drag reduction percentage for 1 mm height of energy promoter is higher than 2 mm (height) energy promoter for case 4 EP and 12 EP, on the other hand, overall drag reduction percentage for case with 8 EP with 2mm (height) energy promoter is higher than 1mm (height) energy promoter. The 4, 8, and 12 EP have been arranged in the same circumferential, as one raw.

Table-3. Effect of height of energy promoter on drag reduction efficiency in pipeline at different conditions.

Section	Height (mm)	DR (%)
(a) 4 Energy Promoter	1	4.03
	2	2.64
(b) 8 Energy Promoter	1	3.90
	2	3.95
(c) 12 Energy Promoter	1	4.25
	2	2.43

The pressure drop per length results, of each case, is summarized in Table-4. It is found that that overall drag reduction for pipeline with 1mm (height) Energy Promoter in normal direction is better than Energy Promoter in reverse direction. Meanwhile, for the pipeline with 2mm (height) Energy Promoter, the Energy Promoter in reverse direction has shown better effect in drag reduction.

Velocity of flow

As mentioned earlier, the flow rate has been varied within a range of 0.3 m/s to 1.3 m/s. The corresponding Re is varied from 66480 to 332400. The estimated drag reduction is shown in Figure-6 at various Re. Results were estimated for 1 mm and 2 mm EPs heights. It is clearly shown that in Table-6, the percentage drag reduction is decreasing gradually until Re = 190000, beyond this point, the percentage of drag reduction is increasing steadily up to Re = 350000. The results have shown that drag reduction effect of energy promoter is strong at low velocities and high velocities. The results are demonstrating that the application of energy promoter becomes more effective when the fluid flows at Reynolds number higher than 260000.

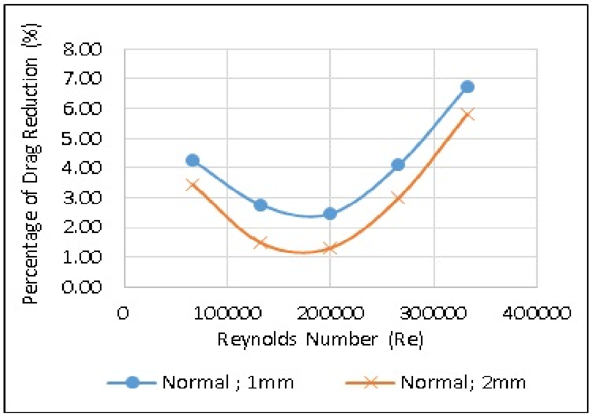


Figure-6. Correlation between reynolds number and percentage of drag reduction.

Table-4. Percentage of drag reduction under different reynolds number.

Reynolds Number	Percentage of Drag Reduction (%)	
	1.0 mm	2.0 mm
66482	4.26	3.44
132964	2.75	1.47
199446	2.45	1.30
265928	4.10	2.99
332410	6.75	5.81

CONCLUSIONS AND RECOMMENDATIONS

CFD technique has been used successfully to simulate flow in pipe with proposed energy promoters. The model has been validated by comparing the pressure drop results with theoretically estimated results. The agreement is very good.

Water, as working fluid, has been used to flow at various velocities in 0.2 m inner diameter pipe. The simulated segment of the pipe is 20.0 m using ANSYS CFX software. The followings could be drawn as conclusions from this preliminary numerical attempt.

- 1.0 mm energy promoter's height is more effective than the 2.0 mm height.
- Within the tested flow range of Re, when the fluid flows with Re higher than $2.6x10^5$, the presence of the energy promoters is more effective.

ARPN Journal of Engineering and Applied Sciences

- In terms of numbers, insertion of 12 energy promoters demonstrates larger drag reduction, up to 4.25%.

In short, by selecting the correct parameters for the dimension of the EP in the pipeline, it is possible that the drag caused by the friction between the fluid and inner wall of pipelines be reduced to the desirable level. Implementation of this technology into the pipelines that are used to transport water, natural gas and crude oil will certainly reduce the cost and energy that is used to restore the pressure.

It is highly recommended to conduct experimental investigation with various shapes and configuration to optimize the performance of the energy promoters for drag reduction.

ACKNOWLEDGEMENTS

The authors express their appreciation to the Universiti Teknologi Petronas for providing ANSYS-CFX software and research environment. Thanks are due to Teow Jing Min for his contribution during his FYP in UTP.

REFERENCES

[1] Pipeline Purpose and Reliability. (n.d.). Retrieved from Buckeye Partners, L. P.: http://www.buckeye.com/pipelineawareness/pipelinepurposeandreliability/tabid/109/default.aspx

[2] Boyun Guo; Shanhong Song; Jacob Chacko; Ali Ghalambor. (2005). Offshore Pipelines. Elsevier.

[3] Diamond, P., Harvey, J., Katz, J., Nelson, D., & Steinhardt, P. (1992). Drag Reduction by Polymer Additives. Virginia: The Mitre Corporation.

[4] Brostow, W. (2008). Drag reduction in flow: Review of applications, mechanism and prediction. Journal of Industrial and Engineering Chemistry, 409-416.

[5] Choi, S. (1985). Near-wall turbulence structure on a riblet wall. UK.

[6] Park, S., & Wallace, J. (1993). Near-Wall Turbulent Flows. B. E. Launder, 749.

[7] Kramer, M. (1960). Boundary layer stabilization by distributed damping. Am. Soc. Navig. Eng., 25-33.

[8] Xu, S., Rempfer, D., & Lumley, J. (2003). Turbulence over a compliant surface: numerical simulation and analysis. J. Fluid Mech., 11-34.

[9] Luo, Y., Xu, X., Li, D. and Song, W., (2016) Recent developments in fabricating drag reduction surfaces covering biological sharkskin morphology, Reviews in Chemical Engineering. Volume 32, Issue 1, Pages 93–113.