TECHNICAL MEMO

APHRES review

Resistance reduction of rotating partially submerged circular cylinders

Project No:
10175780
Memo No:
11H90437-1/ OROGN
Revision No:

Date of issue:
2020-01-22

To:		**From:**	Maritime Hydrodynamics & Stability
Carlos Arcusin		**Prepared by:**	Olav Rognebakke
Attn: Carlos Arcusin		**Verified by:**	Jens Helmers
Copied to:		**Approved by:**	Olav Rognebakke

Applicable contract(s) governing the provision of this Report:

1 CONCLUSIONS

The model tests document a significantly reduced resistance on the frame supporting one or two partially submerged circular cylinders when the cylinders are free to rotate as they are forced through water in a direction normal to the cylinder axis.

Computational fluid dynamics (CFD) analyses support the findings and provide more insights due to the ability to extract more information, like forces on the individual cylinders split into friction and drag contributions.

The model test and CFD results compare well for one non-rotating cylinder, while differences seen for one rotating cylinder may be caused by the difference in rotational speed. During the experiments the cylinders were free to move, while in CFD the rotational speed was precisely specified.

This review confirms the two Arcusin principles. These are included in the Introduction.

The CFD analyses show a dependency of non-dimensional resistance coefficients on physical scale when the geometries are modelled with sharp cylinder edges. With a slight rounding of the edge the results for a physical cylinder size 10 times larger than tested show a large and surprising change. A large sensitivity to a local geometrical feature may indicate a lack of convergence in the CFD results.

More studies are needed to produce accurate nondimensional resistance coefficients for different physical cylinder sizes (scales). Further CFD analyses should document convergence of the results with mesh size and time step.

2 INTRODUCTION

The reported work reports two physical phenomena, denoted as Arcusin principles 1 and 2 respectively:

1. A cylinder submerged to 30% of its diameter, rotating about its axis with a tangential velocity matching the towing speed of the cylinder, experiences huge reduction in resistance when towed normal to the diameter axis.
2. Two identical cylinders submerged to 30% of their diameter, positioned with parallel axes and at a minimum distance of 5% of the diameter, experience large reduction of total resistance when the cylinders rotate around their axes at the velocity where tangential surface speed matches towing speed. The foremost cylinder gets a force in the direction of motion (thrust).

DNV GL has been approached by Carlos Arcusin to perform a document review in order to possibly confirm the two principles.

3 BASIS FOR WORK

This memo is written based on the compilation report entitled "Technical Report Phase 2-n4, Summary of detailed results for 1 and 2 cylinders providing ARCUSIN principles" dated November 4th, 2019, which supplements the various documents provided as background in earlier communication. The file name is "APHRES Project summary 12 cylinders.pdf".

No independent analyses are carried out as part of this work.

An extension to this document was received later, with the same title adding a subtitle "Extra Questions required by DNV GL" dated January 13th, 2020. This document provides partial answers and feedback to a list of questions and comments from DNV GL to the first report.

4 REVIEW OF RESULTS

The first part of the submitted technical report /1/ treats the model tests carried out at Universidad Politécnica de Buenos Aires many years ago.

The results show that one or two partially submerged circular cylinders free to rotate experience a lower drag than non-rotating cylinders when towed normal to the cylinder axes. The reduction in resistance depends on the towing speed, but it is significant for most of the tested towing velocities.

It is natural to assume that the rotational speed is somewhat below the so-called synchronized speed, when the tangential velocity matches the towing speed. This assumption is founded on the need for some friction, i.e. velocity difference between water and cylinder surface, to maintain the cylinder rotation.

The report treats new results based on Computational Fluid Dynamics (CFD). OpenFOAM is used to model the flow around the advancing partially submerged cylinders. Rotation is modelled by imposing a tangential velocity matching the advance speed as boundary condition on the cylinder surface.

The supplementary report provides more information regarding numerical model, domain size, boundary conditions and mesh properties. No convergence study is documented, but the numerical model is reasonable based on DNV GL experience with similar types of flow.

The meshes show a high y+ and a wall model is used to model the boundary layer. This normally implies an assumption of turbulent flow (depends on wall model, so this is anticipated based on best-practice). In the model tests, it is likely that the flow remains laminar on at least the forward cylinder, unless a trip wire or similar was used. The difference between a laminar and turbulent boundary layer will affect the separation point and consequently the pressure resistance. However, the CFD results agree quite well with the experiments for a non-rotating cylinder, so this difference may not be significant.

4.1 One-cylinder case

The resistance for one rotating cylinder is lower in the CFD predictions than observed in the experiments. This could be due to a larger rotational speed in CFD, as friction drives free rotation in the model tests.

CFD allows for quantifying the resistance contributions from friction and pressure separately. The results illustrate how the wave profile behind the cylinder changes with rotation speed. A synchronized speed causes a dynamic pressure build-up downstream of the cylinder, which reduces the resistance.

As expected, when the tangential speed of the cylinder matches the advance speed, there is almost no frictional resistance.

Over rotation is studied, and the conclusion is that minimum resistance is obtained when the tangential speed matches the advance speed.

4.2 Two-cylinders case

The model tests show that in a two-cylinders set-up, where the two identical partially submerged cylinders are positioned with an internal distance of 5% or more of their diameter, free rotation of both cylinders significantly reduce the total towing force.

CFD is used to study a case similar to the model test No. 5, with three different cylinder distances.

The lower total resistance is obtained for a cylinder separation of 5% of the diameter, when a negative resistance (thrust) is calculated on the front cylinder.

Different rotational speeds are studied for the 5% diameter separation case, and again the synchronized rotation results in minimum resistance.

The CFD model is made with sharp cylinder edges, mesh M2, which is mentioned to result in some instabilities. Details are not given, and it is assumed that mesh M2 provides results with some uncertainties.

4.3 Mesh and geometry sensitivity

The cylinder geometry and mesh is discussed in Section 3.1 in /1/. Two different geometries and associated meshes are used. M2 models sharp cylinder edges, while MF7 includes some small rounding of the edges. The rounding is reported to stabilize the flow compared to the sharp edge. This is somewhat surprising as a sharp edge often helps in defining a separation point.

Report /2/ shows that there is quite some unsteadiness in the flow and force acting on the non-rotating cylinder. This is to be expected as the separation point changes behind the blunt moving body. The interaction between the cylinder edge and flow separation point is expected to be challenging to model accurately.

In general, a convergence study should be carried out to demonstrate that the sensitivity to changes in the mesh size and time step is reasonably small. The lack of this means that some inaccuracies and uncertainty are expected in the CFD results. This is illustrated in the next section studying scale effects while comparing different meshes.

4.4 Scale effects for two cylinders

Possible scale effects are studied by using different physical size of the cylinders in CFD calculations. The scale notation is different from normal, in that scale 20 means a cylinder that has a 20 times larger diameter than the one tested in the laboratory.

CFD analyses are done with different physical sizes, but also using both meshes M2 and MF2, with sharp and rounded cylinder edges respectively. Froude similitude is assumed when determining the advance speed.

Table 3 in /2/ shows that y+ values change, which means that the mesh is not updated to keep this similar (Reynolds scaling of local mesh). This will affect the accuracy of the results across scales.

The accuracy of the CFD calculations is not sufficiently well documented for the change in non-dimensional resistance coefficients with scale to be generally accepted.

5 REFERENCES

/1/ Technical Report Phase 2-n4, Summary of detailed results for 1 and 2 cylinders providing ARCUSIN principles" dated November 4th, 2019

/2/ Technical Report Phase 2-n4, Summary of detailed results for 1 and 2 cylinders providing ARCUSIN principles -> Extra Questions required by DNV GL" dated January 13th, 2020