Article no. 5

THE CIVIL ENGINEERING JOURNAL 1-2015

NUMERICAL MODEL APPLICATION IN ROWING SIMULATOR DESIGN

Petr Chmátal[1], Jaroslav Pollert[2] and Ondřej Švanda[3]

1. *Czech Technical University in Prague, Faculty of Civil Engineering, Department of Sanitary and Ecological Engineering, petr.chmatal@fsv.cvut.cz*
2. *Czech Technical University in Prague, Faculty of Civil Engineering, Department of Sanitary and Ecological Engineering, pollertj@fsv.cvut.cz*
3. *Czech Technical University in Prague, Faculty of Civil Engineering, Department of Sanitary and Ecological Engineering, ondrej.svanda@fsv.cvut.cz*

ABSTRACT

The aim of the research was to carry out a hydraulic design of rowing/sculling and paddling simulator. Nowadays there are two main approaches in the simulator design. The first one includes a static water with no artificial movement and counts on specially cut oars to provide the same resistance in the water. The second approach, on the other hand uses pumps or similar devices to force the water to circulate but both of the designs share many problems. Such problems are affecting already built facilities and can be summarized as unrealistic feeling, unwanted turbulent flow and bad velocity profile. Therefore, the goal was to design a new rowing simulator that would provide nature-like conditions for the racers and provide an unmatched experience. In order to accomplish this challenge, it was decided to use in-depth numerical modeling to solve the hydraulic problems. The general measures for the design were taken in accordance with space availability of the simulator´s housing. The entire research was coordinated with other stages of the construction using BIM. The detailed geometry was designed using a numerical model in Ansys Fluent and parametric auto-optimization tools which led to minimum negative hydraulic phenomena and decreased investment and operational costs due to the decreased hydraulic losses in the system.

KEYWORDS

Rowing simulator, sculling simulator, paddle simulator, hydraulic research, numerical modeling, building information model, BIM.

INTRODUCTION

In today's sports world there is a huge competition in every subject no matter if it is athletics, team sports or individual disciplines. Multi-phase everyday practices with carefully chosen nutritious diet and foreign sport camps are the daily routine for the most of the athletes. So it is no surprise that all the teams and coaches are constantly seeking for new ways of improving the athletes' training. As a slightly different method or training device can provide a little advantage that makes the difference in the race.

For rowers and paddlers one of the ways how to increase the technique and to stay in shape during the winter months is to build an indoor rowing/paddling simulator. That allows them to continue the practices even when the weather would not allow it. As a part of a reconstruction project of Labe Arena it was decided, that such a facility should be developed.

Nowadays there are two main approaches in the rowing simulator design. The first uses static water with no artificial movement and counts on specially cut oars to provide the same resistance in the water. This solution lacks the stability when rowing, feels unrealistic, causes undesirable turbulence from the oars and provides bad velocity profile. The second approach, on the other hand integrates pumps or similar devices to force the water to circulate but all the facilities world-wide share the same drawbacks such as slow water velocity, insufficient water depth and also bad velocity profile. Therefore, the goal was to design a new rowing simulator that would provide nature-like conditions for the racers and avoid all the faults made in the past designs of such facilities.

The contractor for this project was chosen to be the company di5 architekti inženýři s.r.o. For the rowing and paddling simulator it was then decided that it should be designed by CTU in Prague, Faculty of Civil Engineering, Department of Sanitary and Ecological Engineering thanks to a lot of experience with hydraulic research. The aim was to design a facility with circularly pumped water to allow for the rowers and paddlers to experience training conditions that would resemble those in a natural river or channel where the practices usually take place. Some facility parameters are shown in the following Table.

Tab. 1: Facility parameters

	Rowing simulator	Paddling simulator
Construction size	19 x 12.5 m	14.5 x 4 m
Max stream velocity	4 m/s	2.4 m/s
Water depth for sport	0.45 m	0.65 m
Max flow per channel	3.85 m^3/s	1.33 m^3/s
Max power req.	150 kW	50 kW

The measurements of the facility were chosen due to the limitations of the housing object and were settled at 19 x 12.5 m of floor plan for both rowing and paddling simulator. In addition, the investor wanted the water velocity to reach up to 4 m/s which significantly overcomes values of any other simulator that has been in operation. The last but not least was the effort to manage and minimize both investment and operational costs of the entire project. The basic shape for the rowing simulator were two rectangular tanks horizontally separated for the water circulation. That was managed with a use of submersible pumps. The area between the tanks was allocated for the seats, rowing equipment and racers themselves. In the original design the paddling simulator was supposed to be a part of the bigger rowing simulator but after first calculations and negotiations with the investor this idea was abandoned and it was decided that the paddling simulator shall be a standalone facility placed next to the rowing one. As a structure material was chosen concrete with smoothed surface and extra epoxide coating to minimal hydraulic losses. Inlet elbow, diffuser, fin and tray were designed to be made of stainless steel. The schematic of the design is shown in Fig. 1.

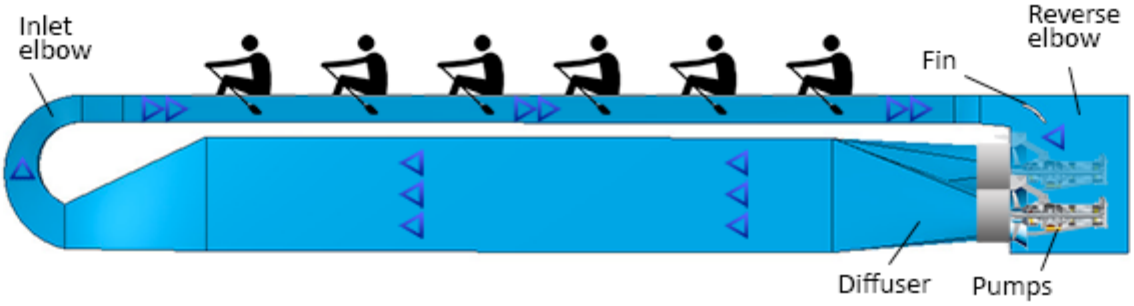

Fig. 1: Rowing simulator design schematic

DOI 10.14311/CEJ.2016.01.0005

Given the fact that there is no such a complex facility with similar parameters, it was necessary to come up with an entirely new design. This was achieved using technical calculations and simulations of water flow modeled in both 2D and 3D in Ansys Fluent. The optimization of the geometry was done within parametric tool Direct Optimizer. Some basic model parameters are listed in the following table.

Tab. 1: Basic numerical model parameters

Viscosity model	κ-ε
Multiphase	Volume of Fluid
Nodes	Ca. 1 000 000

ROWING SIMULATOR

In the rowing simulator design, it was necessary to calculate the bottom slope first. This value is important in order to secure horizontal water level. Its calculation is based on Bernoulli equation and is calculated from two profiles (inlet and outlet of the rowing platform). The difference between energy in those profiles must match hydraulic losses. The losses were calculated using Chezy equation with Manning velocity coefficient. Result of the iterative process was i_0 = 0.0023. In terms of absolute height, it means 35 mm of height difference on 15 m of length.

In the next step different flow velocities were evaluated in view of creating waves which is an unwanted phenomenon in the rowing area. The evaluation was done by using Froude number. The magnitude of the Froude number determines the flow regime. The transition from supercritical to subcritical flow is accompanied by hydraulic jump which can cause waves and negative hydraulic phenomena. This transition occurs when the Froude number moves around 1. It is clear from the calculations that the ideal conditions for running the simulator are with the flow from 0 to 2.3 m^3/s and then from 3 to 4 m^3/s. Accordingly from 0 to 1.9 m/s and from 2.3 to 3.1 m/s.

Pumps

Both left and right circuit of the rowing platform are equipped with submersible pumps whose are to provide sufficient flow and velocity. The requirements for these pumps were quite extraordinary, therefore the selection was very narrow. The output power should have been approximately 4 m^3/s with 1 m of discharge head. Also, the type must be a propeller pump in order to provide uniform velocity field at the outlet which leads to a steady water level in the rowing area. The only pumps to meet these requirements were from Amaline series, specifically 3x S 8032-465/304URG (25 kW) were used for each circuit. The combined power for each circuit was then 75 kW adding to a total of 150 kW for the entire rowing simulator.

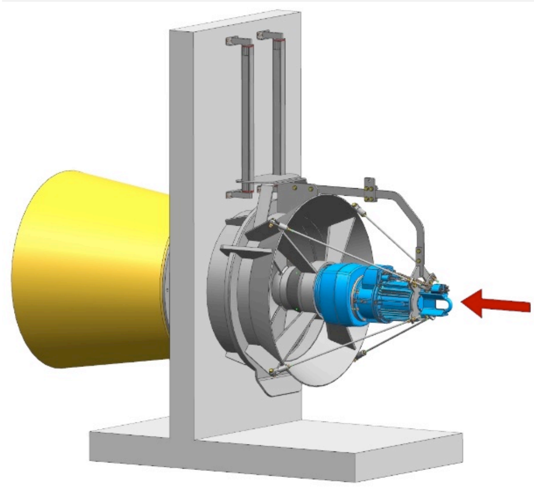

Fig. 2: Proposed propeller pump AMALINE (25 kW)

Diffuser

Important part of the design is a diffuser. It is located at the pumps discharge which are directly connected to the steel pipe flange with a diameter DN 800 and a length of 0.5 m. The main purpose of the diffuser is to provide as good diffusion of the pumped water as possible so the created velocity profile stays uniform. Besides that, the geometry of the diffuser must also cause low hydraulic losses because of the potential underpressure leading to cavity and possibly damaging the diffuser. The geometry was designed to best diffuse the water from all of the 3 pumps. The length was chosen with low manufacture costs having in mind. The shape and basic measures are shown in the following figure.

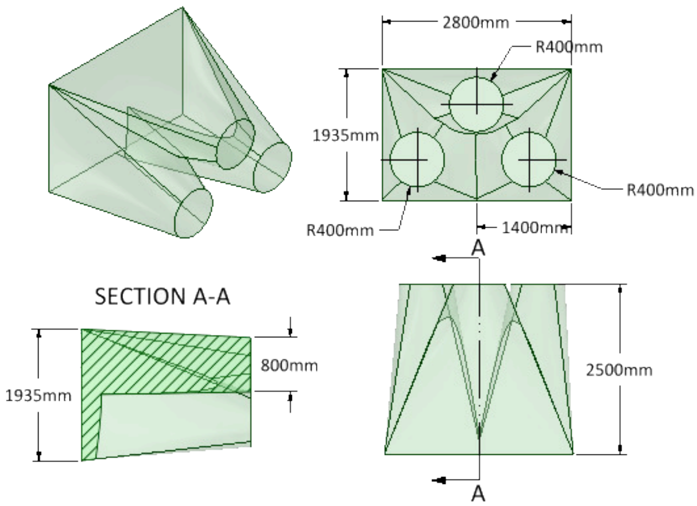

Fig. 3: Geometry of the diffuser

DOI 10.14311/CEJ.2016.01.0005

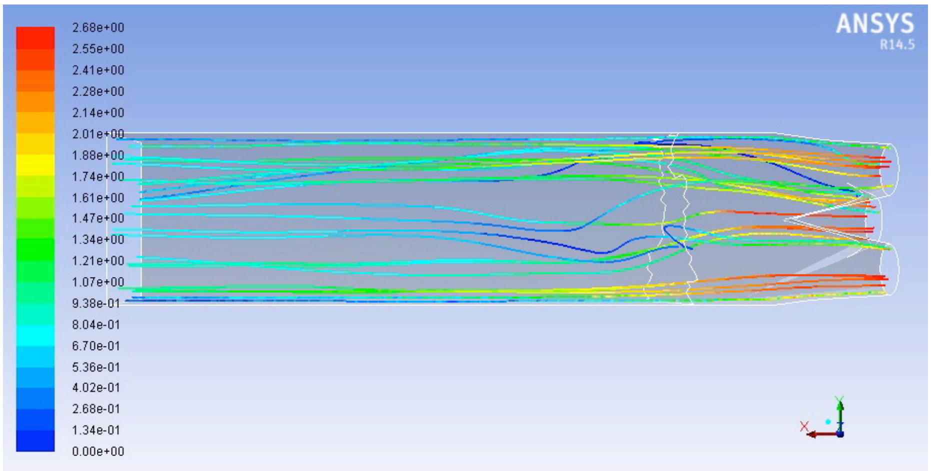

Fig. 4: Pathlines colored by velocity

Inlet elbow

More than 350 different geometries were simulated in the process of designing the inlet elbow. It was achieved using a 2D model in Direct Optimizer tool. The criteria for the calculation were set to minimum hydraulic losses and uniform outlet velocity field. The optimization started with a choosing of 200 random geometries within a certain range that were subsequently reduced using Kiring method in order to find those where matching the criteria were expected. After choosing 30 potential geometries, the simulation was carried out and the final 5 best geometries were found. The outcome of this process and the optimal geometry for the inlet elbow is shown in the next figure. Appropriate velocity profile can be seen at the end of the inlet elbow (center top).

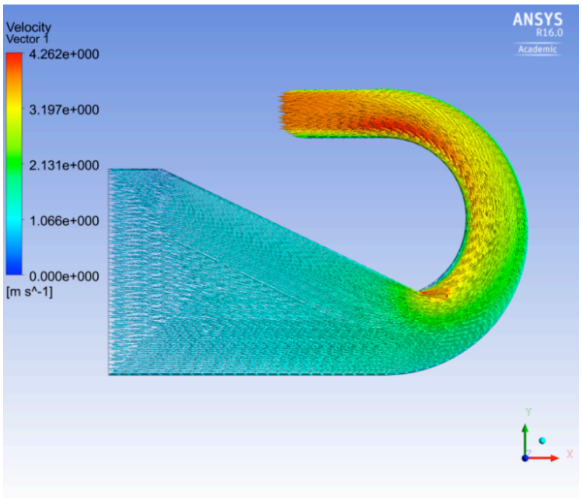

Fig. 5: Velocity vectors shown in the inlet elbow geometry

Obtained velocities and losses were slightly distorted due to the fact that the simulation was run using 2D instead of 3D, therefore the real geometric shape of the construction was not applied. Nevertheless, in this case 2D represents the outcome more than sufficiently and can be used for 3D design. The main reason for choosing 2D was to significantly lower the processing time which led to the ability to compare and evaluate more geometries.

In the next step, the transition from the elbow (pipe flow) to rowing area (open channel flow) was verified for potential wave appearance which would negatively affect the rowing experience. Outcome of the simulation proved the correctness of the design. The water flows into the rowing area with uniform velocities and does not tend to create waves of other unwanted phenomena. In the following picture you can see the phase boundary between water and air in the open channel.

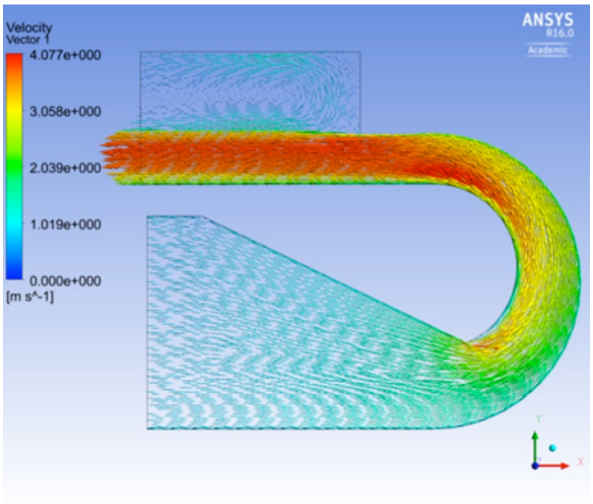

Fig. 6: Velocity vectors at the phase boundary

Reverse elbow

The geometry of reverse elbow is determined by the space availability due to the pumps' specifications and installation requirements. The inlet parameters were improved by implementing a fin. The optimal design of the fin was found out by testing 10 different geometries with choosing the best one in the end. It turned out that the best solution would be a 1-fin variant because of its low losses. By adding more fins in the design there was no improvement achieved in means of flow parameters and uniformity of the velocity field. In addition, the design of the fin prevents air intake by the pumps. It is designed to fulfill its function the best when the flow reaches the maximum at 3.85 m^3/s. It must be taken in mind that it may not function optimally with lower flows.

The fin is designed in a way that it can be detached in order to allow the installation of the pumps and for their better maintenance. In the proximity of the reverse elbow there is a whitewater area. A special attention was paid to this area to make sure that the whitewater does not occur in the rowing area but remains close to the reverse elbow and the pumps. The outcome of this evaluation is a design modification where the trapezoid stainless steel sheet is extended to the reverse elbow area and is ended with a sharp edge. In the next figure you can see the geometry of the reverse elbow including the fin.

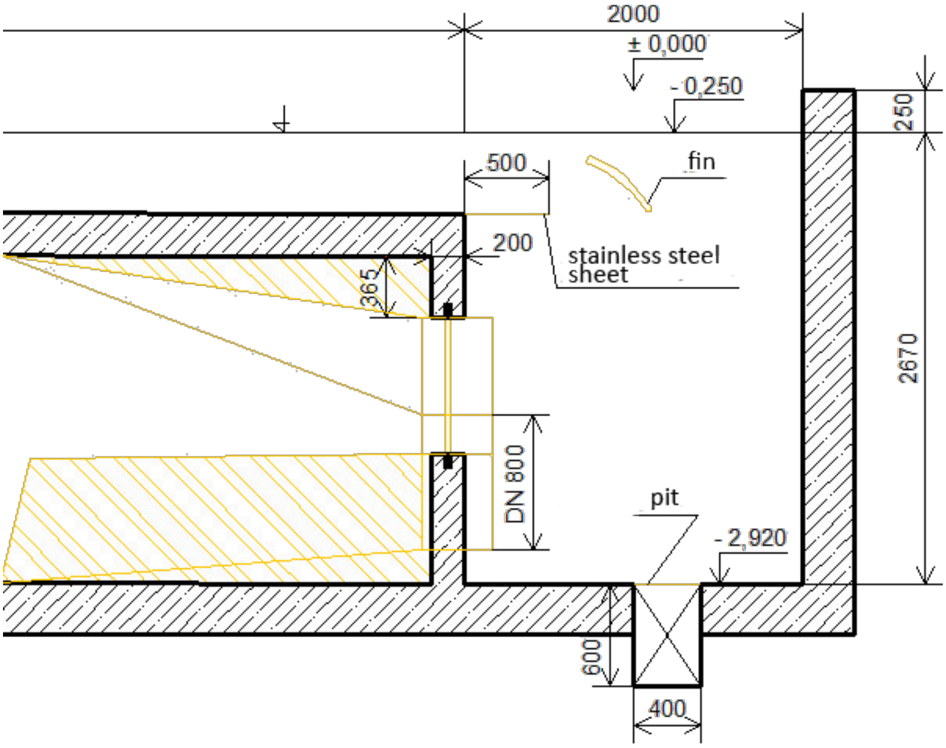

Fig. 7: Geometry of the reverse elbow with the fin

Flow simulation

Simulation of the flow in the whole rowing simulator was carried out in Ansys Fluent software using 3D model which describes the flow in the simulator the best. During the simulation the velocity of the flow was gradually increased in order to simulate the start of the pumps. It turned out that the design causes waves in the rowing area. After a geometry modification in the rowing area the following simulation proved to provide a steady flow at 3.85 m^3/s in each channel. At this setting the water level is steady with no waves and uniform velocity profile. The turbulence occurs behind the rowing area in the reverse elbow which is perfectly acceptable. The flow simulation with highlighted water level can be seen in the following figure.

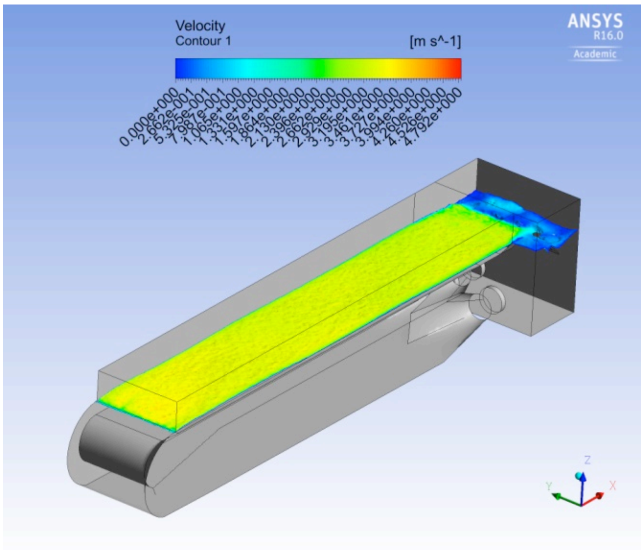

Fig. 8: Flow simulation with highlighted water level

PADDLING SIMULATOR

The design of the smaller paddling simulator was carried out in the same way as the bigger rowing one. The bottom slope was calculated including the magnitude of the horizontal force caused by the dynamics of the flow. Also different flow velocities were simulated for the wave occurring. Subsequently all the optimal geometries of the diffuser and both elbows were found using numerical model and Ansys Fluent program with the help of Direct Optimizer tool. The aim of the design were low hydraulic losses, steady flow and water level. Pumps were chosen to be the same AMALINE as in the rowing simulator. In this case only 1 pump is needed for each channel, totalling in 2 pumps of a 50 kW outcome power. Finally, the entire paddling simulator was evaluated in 3D to make sure all the designed parts make a perfect whole and no unwanted hydraulic phenomena occur in the design. The final design of the paddling simulator is shown in Fig. 9.

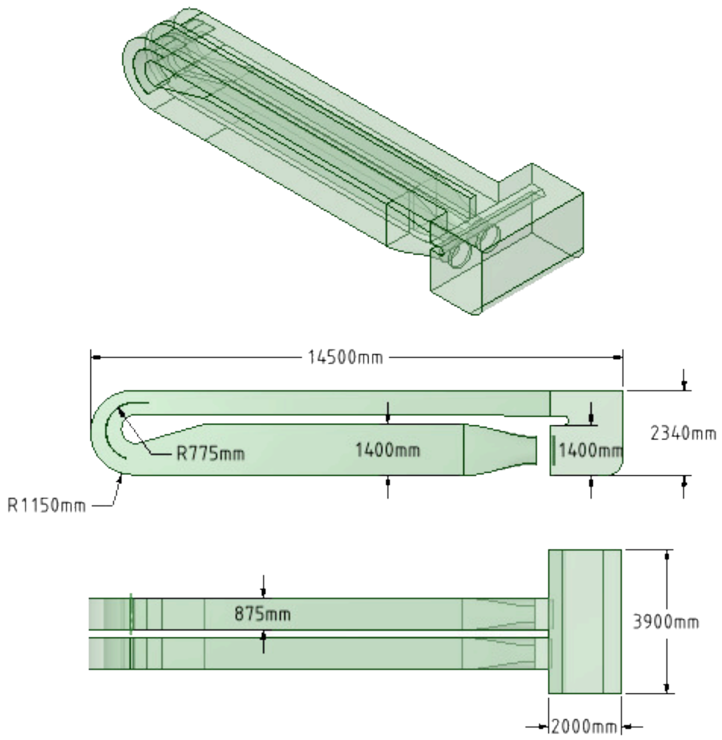

Fig. 9: Final design of the paddling simulator

CONCLUSION

A new design in the area of the rowing and paddling simulators was created with suitable geometry and hydraulic conditions. The new approach lies in the use of 2D and 3D numerical model and its evaluation in the Ansys Fluent software. In addition, a parametric tool Direct Optimizer was used to find the best suitable geometry for chosen conditions. The aim was to design a facility that would provide steady water level with high velocities, low hydraulic losses, uniform velocity field in the rowing and paddling area and low investment and operation costs. Also the dynamic forces of the construction were calculated with a respect to the structure of the housing.

The initial goal of creating a new generation of rowing and paddling simulator with significantly better training conditions and low operational costs was met. After the last simulation it is clear that despite the high velocity in the rowing simulator, the water level stays steady with no waves occurring in the rowing area. With additional funds from the investor, it is plausible that the development of the simulator could be improved in a way of adding a device that would follow the rowing movement dynamics.

ACKNOWLEDGEMENTS

This work was supported by Technological agency of the Czech Republic: TAČR TE02000077 Smart Regions – Buildings and Settlements Information Modelling, Technology and Infrastructure for Sustainable Development and by the Student Grant Foundation of CTU: SGS16/205/OHK1/3T/11.

Access to computing and storage facilities owned by parties and projects contributing to the

National Grid Infrastructure MetaCentrum, provided under the program "Projects of Large Infrastructure for Research, Development, and Innovations" (LM2010005), is greatly appreciated.

REFERENCES

1. Streeter, V.L., Wylie, E.B. and Bedford, K.W. *Fluid Dynamics. Ninth Edition.* s.l. : Mc Graw-Hill Book Company, Inc, 1998. ISBN 0 07 062537 9.
2. Streeter, V.L., Wylie, E.B. and Bedford, K.W.. *Handbook of Fluid Dynamics. First edition.* s.l. : Mc Graw-Hill Book Company, Inc., 1961. ISBN 978-0070621787.