Formation of Rotating Vortex Rope in the Francis-99 Draft Tube

To cite this article: N Sotoudeh et al 2019 IOP Conf. Ser.: Earth Environ. Sci. 240 022017

View the article online for updates and enhancements.

You may also like

- Efficient Sterilization with Reactive Oxygen Produced By Radical Vapor Reactor (RVR)
  Shoko Ishikawa, Nobuhiro Miyori and Tetsuya Haruyama

- A comparative study on the evolution of compressible vortex ring generated from a short driver section shock tube
  Santanu Dey, T Murugan and Dipankar Chatterjee

- Turbine Test Rig to Investigate Flow Instabilities in Draft Tube
  S Kumar, S Khullar, M J Cervantes et al.
Formation of Rotating Vortex Rope in the Francis-99 Draft Tube

N Sotoudeh¹, R Maddahian¹ and M J Cervantes²

¹ Faculty of Mechanical Engineering, Tarbiat Modares University, Tehran, I.R. Iran
² Division of Fluid and Experimental Mechanics, Lulea University of Technology, SE 971 87, Lulea, Sweden

maddahian@modares.ac.ir

Abstract. The aim of this research is to understand the mechanism(s) of RVR formation during the changes in operating condition from the Best Efficiency Point (BEP) to Part Load (PL). A Computational Fluid Dynamic (CFD) methodology by the means of ANSYS-CFX is applied on a reduced high head Francis turbine model. The reduced model consists of one stay vane, two guide vanes, one runner blade, one splitter and a full draft tube. Numerical simulation is first performed at BEP as well as PL to ensure the appropriate employment of turbulence models and boundary conditions. In the second step, the inlet boundary conditions are changed linearly from BEP to PL in order to achieve the transient conditions inside the draft tube. The initial condition of the second step is the converged BEP result. The transient simulation is continued until the RVR is fully developed in the draft tube at part load condition. The numerical results for BEP, PL and BEP to PL are in a good agreement with the experimental data. The effect of the RVR is considered from two aspects. The first one is the frequency, and the amplitude of the pressure pulsations induced by the RVR in the draft tube. The second one is the velocity field in the draft tube which is investigated over time during load rejection. Moreover, the flow structure is visualized using the $\lambda_2$ criterion. The mechanism(s) of RVR formation and damping is accurately investigated by the presented approach. Furthermore, the results provide a better understanding of the physics behind the RVR formation. The obtained results aim to design an effective RVR controlling approach.

1. Introduction

The subjects of global warming and reduction of greenhouse-gas emission have attracted the interest of researchers to renewable sources of energy, especially the hydraulic power plants. Receiving energy from moving water has been used since ancient times by utilizing watermills. Recently, the energy of the moving water has been used in hydraulic turbines to produce electricity. Hydraulic turbines are mostly designed to work at their Best Efficiency Point (BEP). Due to the variation of electricity demand and also deregulation of electricity market, turbines should more often run at off-design conditions. Therefore, as turbines are operating in conditions far from the designed conditions, there would be some inevitable consequences. From another point of view, renovation in a large number of turbine constructions is crucial since they are mostly old. Due to the renovation costs, the focus of the renovation is mainly on redesigning guide vanes and runner, however, ignoring redesigning spiral casing, stay vanes and draft tube can cause an unstable flow field, especially in the draft tube at off design condition. Thus, a precise study on off design operating conditions can be helpful to design/redesign turbines to operate more efficiently, with higher output power and large operational capability.
In 1940, some fluctuations in the output power of the generator at Part load (PL) was realized by Rheingans. The power fluctuations are the result of unstable flow field, especially the unsteady vortices in the entrance of the turbine's draft tube. Rheingans called this phenomenon the Rotating Vortex Rope (RVR) and found that the ratio of RVR frequency to the runner’s speed is equal to 0.278 [1]. However, due to integrated structure of runner and generator at that time, it was hard to separate the effect of different parts on the out coming power. Benjamin (1962) predicted the Vortex breakdown by investigating the characteristics of the swirling flow [2]. Cassidy and Falvey presented a non-dimensional parameter called swirl intensity, which is the ratio of angular momentum to axial momentum [3]. Nishi et al. (1982) visualized the swirling flow field in the turbine draft tube and categorized four different flow regimes during load rejection by measuring the location and the volume of the stalled region. They stated that the decrease in flow rate results in developing some stalled regions in the entrance of the draft tube. When the turbine operates in the PL condition, stalled regions grow and connect together and create a merged stalled region at the center of the draft tube. Subsequently, the stalled region rotates and moves downward gradually in the draft tube and creates an RVR [4]. RVR is the actual consequence of high shear on the interface between the recirculation central region and the main flow and causes a low frequency and high-amplitude pressure fluctuation [5-7].

The effects of the RVR rotation were investigated by Escudier. He expressed that not only the RVR existence affects the runner rotation and produces severe vibration and noise, but also the flow field inside the draft tube becomes unsteady. Both effects reduce the efficiency of the turbine, and the RVR rotation produces unfavourable low-frequency pressure pulsations [8]. The RVR existence results in a low performance of the whole turbine, reduction in the available area in the diffuser and larger axial velocity near the wall. The high-velocity gradient and non-uniform flow will increase hydraulic losses considerably. In addition, if the frequency of pressure pulsations equals to the natural frequency of the system, a resonance would occur in the hydraulic turbine and power plant constructions. On the other hand, in the edge of the RVR vortices, the velocity is high and if the pressure reaches to the vapor pressure, the cavitation phenomenon occurs. The irregular burst of the bubbles would be a new source of pressure fluctuations and causes the corrosion damage to the draft tube’s wall [9].

The primer numerical efforts to simulate the flow field inside the turbine draft tube started in 1952 by Wu. Employing some approximations, the flow was assumed to be two-dimensional and the governing equations were applied on different surfaces in the draft tube cone [10]. Several assumptions such as symmetric or inviscid flow are also considered by researchers to simplify the flow field [11-13]. Recent investigations focus on the turbine operating conditions such as BEP or PL. Xiao-xi et al. performed a numerical simulation based on experimental transient data on a Francis turbine by applying a 1D-3D coupling approach. The results showed that however the one-dimensional model can predict the characteristics of the flow properly, it is unable to predict the flow pattern in the spiral casing, draft tube and runner and thus, utilizing a three-dimensional model is vital [14]. Trivedi et al. studied six different transient operations containing load acceptance and load rejection in a high head Francis turbine experimentally. They reported that the maximums of pressure fluctuations were seen at load rejection because of the RVR existence [15]. Amiri et al. also worked experimentally on measuring unsteady pressure on the runner and the stalled regions in a Kaplan turbine during load acceptance and load rejection. The research was dedicated to the formation and vanishing of the RVR and the effects of it on the runner. The results showed that the pressure fluctuations are more significant in load rejection in comparison to load acceptance and the reason standing behind this fact is RVR formation in PL [16].

Gagnon et al. studied experimentally on the runner responses during load rejection with / without water admission. The results showed that using water admission reduces the amplitude of pressure fluctuations on the runner [17]. Goyal et al. (2016) investigated the load rejection of a Francis turbine experimentally. The results showed that the runner blades passage frequency is visible all over the turbine's domain, including the vane-less spaces and the draft tube, but the amplitude of the runner passage wave changes in the domain due to the mismatch of the outlet flow from guide vanes to the runner during load rejection. Moreover, they discussed that closing the guide vanes during transition causes remaining extra swirl in
the draft tube and RVR formation [18]. Recent numerical investigations were dedicated to the transient simulation of turbine during the load acceptance or rejection [19-20].

One important point which is not addressed precisely in the previous investigations is the mechanisms and flow physics behind the RVR formation. Therefore, the focus of this research is on capturing RVR formation in a Francis turbine during load rejection from BEP to PL. The simulations are performed for a reduced model of Francis-99 turbine. The present work is the first step of numerical investigation of the Francis-99 draft tube during the transient condition and is devoted to the unsteady simulation of flow during the load change from BEP to PL. Governing equations are presented in section 2. The physical model and the numerical method are reviewed in section 3 and 4, respectively. Section 5 is devoted to the results from the simulation and discussions.

2. Physical Model

Francis-99 is a turbine test model located in Norwegian University of Science and Technology (NTNU). The turbine geometry consists of 14 stay vanes, 28 guide vanes, 15 runner blades, 15 splitters and a full draft tube. The experimental measurement of velocity and pressure has been published by NTNU during two workshops and provides the opportunity for scientists to perform numerical analysis based on experimental data. In this research, the transient data during load rejection (i.e. time variant guide vane closing) is used. The PIV velocity measurements and pressure sensors inside the draft tube are used. A schematic of turbine and measurement points are shown in Fig. 1. Details of the data, measurements, operating parameters (i.e. discharge, head, runner angular velocity and etc.) and geometry are available in [25].

![Fig. 1 The location of pressure and velocity sensors [25]](image_url)

3. Numerical Method

3.1. Governing Equations and Mathematical Model

To simulate the turbine flow field, the time-averaged Navier-Stokes equation is employed. The flow field is assumed to be isothermal and incompressible. The obtained equations are as following [21]:

\[ \frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0 \]  

\[ \rho \left( \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \mu \frac{\partial \bar{u}_i}{\partial x_i} \right) - \frac{\partial \left( \rho \bar{u}_i u'_i \right)}{\partial x_i} \]  

Where \( \bar{u}_i \) is the time-averaged velocity, \( u' \) is the velocity fluctuation, \( p \) is the time-averaged pressure, \( \rho \) and \( \mu \) are the fluid density and viscosity, respectively. The last term in the right-hand side of equation (2) which is called Reynolds stress tensor, needs some extra equations to be solved. Therefore, in the present work, the eddy-viscosity model is utilized to determine the Reynolds stress tensor. This term is approximated as follows:
\[-\rho\left(u_i'u_j'\right) = 2\mu_i S_{ij} - \frac{2}{3}\rho k \delta_{ij}\]  

(3)

where \(\mu_i\) is the turbulent viscosity, \(k\) is the mean turbulent kinetic energy and \(S_{ij}\) is the mean strain tensor.

To define \(k\) and \(S_{ij}\), the following equations can be used:

\[k = \frac{1}{2}\left(u_i'u_i'\right)\]  

(4)

\[S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\]  

(5)

The turbulent viscosity can be defined properly employing Shear Stress Transport-Scale Adaptive Simulation (SST-SAS) model as a turbulence model in simulations [22]. The SST-SAS model solves two equations for \(k\) and \(\omega\) (specific dissipation rate) with an additional source term in \(\omega\) equation. The turbulence length scale is also defined based on the Von-Karman length scale. The successful application of the SAS modification in comparison to ordinary SST model provides the opportunity to model unsteady phenomena more accurately [23]. The formulation of SST-SAS model can be found in [22, 24].

3.2. Geometry and grid

In order to reduce the computational costs, a reduced geometry is used for the numerical simulation. In the reduced model, 1 stay vane, 2 guide vanes, 1 runner blade, 1 splitter and a full draft tube are modelled instead of full geometry (see Fig. 2). In the reduced model, a runner blade and a splitter are merged together to form a rotating domain. Similarly, one stationary domain is considered including 1 stay vane and 2 guide vanes. During the transition from BEP to PL, the angles of guide vanes are changed. Although, the dynamic mesh is applied on the whole domain of stay vane and guide vane, the geometry related to stay vane remains stationary and only the guide vanes’ mesh is changed during transition.

Fig. 2 Francis-99 reduced geometry

The flow field is turbulent and swirling in the domain, especially in the draft tube. Thus, the mesh, which is generated using ANSYS-ICEM, consists of hexahedral elements to ensure the simulation accuracy and solution convergence. The total number of cells in the reduced model is 2,439 million (1 stay vane and 2 guide vanes: 829440 cells, a runner and a splitter: 718720 cells and draft tube: 890880 cells). The total number of computational cells in the draft tube is changed to ensure the grid independent results. The grid independency validation of the axial velocity in the draft tube at BEP is shown in Fig. 3. The number of the elements for the coarse, medium and fine mesh of the draft tube are 630 654, 890 880 and 1000523, respectively. As illustrated, both the medium mesh and the fine mesh results are in an acceptable agreement with experimental results.
3.3. Boundary conditions

The inlet boundary condition is set as mass flow rate with prescribed flow angle at the inlet of stay vanes. During the transient simulation, the inlet mass flow rate is changing linearly from BEP to PL mass flow rate. Also, 5 percent turbulence intensity is considered at the inlet to make the simulation more realistic. The no-slip condition is used as wall boundary condition and the automatic wall function is employed to calculate turbulent quantities near walls. The zero static relative pressure is considered for the draft tube's outlet boundary condition. The reference pressure changes linearly from BEP to PL outlet pressure during the transient simulations. Furthermore, in order to change guide vanes angle from BEP to PL, the dynamic mesh is used. In this process, the total number of the grids does not change, and only the mesh is compressed, owing to the fact that there is only 3.12 degree change in guide vanes' angle during the transition. Thus, Smoothing method was applied on the dynamic mesh process. It took 2.5 seconds to change the guide vane's angle for 3.12 degrees relative to its coordinate system [26].

The interfaces between rotating and stationary parts are Transient Rotor-Stator and capture unsteady and transient phenomena in the runner as well as the stationary parts, including draft tube, stay vanes and guide vanes. The pitch ratio is 25.7/24 for guide vanes to runner interface and 24/360 for runner and splitter to draft tube interface.

3.4. Solution Methodology

The ANSYS-CFX commercial code is used to perform numerical simulations. The advection and diffusion terms are discretized using high resolution and central difference scheme, respectively. Different time steps are employed in the simulations. The considered time steps are 3 degree, 16 degree and 3 degree of runner rotation in the BEP, transient BEP to PL and PL simulations, respectively. The total transition time from BEP to PL is 2.5 seconds.

4. Results and Discussion

4.1. Velocity results

The velocity is obtained on 28 points on line 1 (see Fig. 1) and compared with experimental measurements in Fig. 4. The axial velocity comparison is shown in Fig. 4(a) and Fig. 4(b) for BEP and PL, respectively. It should be mentioned that the reported axial velocities for PL are averaged during 1 RVR rotation. The experimental and numerical results are in a good agreement. Near the draft tube wall, the axial velocity is affected by wall region. The axial velocity increases by moving toward the draft tube center, although the difference between numerical and experimental data increases near the draft tube's center. The difference is due to the employed eddy-viscosity turbulence model. In PL, the trend of axial velocity follows the experimental results; however, opposite to experimental results, numerical results predict a reverse flow region in the middle of the draft tube. The appearance of this region might be due to the tangential velocity and swirl intensity predictions.
In order to have a better understanding during the transition from BEP to PL, a contour of axial velocity is shown in Fig. 5. In the BEP mode, the flow is completely axial and no reversal zone is seen. During the transition from BEP to PL, the swirling flow increases inside draft tube and a flow reverse zone is appeared at the center of the draft tube. The flow reversal zone is symmetric during the transition. After the transition, the reverse zone is still spreading inside the draft tube and becomes unsteady. The rotating feature of the flow reversal zone is seen in the PL mode. The radius of the reverse region is approximately $r/r_0=0.3$ in both numerical and experimental results.

4.2. Pressure results
Pressures are monitored at two monitor points, named DT5 and DT6, on the draft tube cone’s wall. The monitor points’ locations are similar to the experimental measuring points. To make a better understanding of occurred phenomena in the draft tube, a Fast Fourier Transform (FFT) is applied on the pressure results of each simulation, which are shown in Fig. 6. The reported frequencies in Fig. 6 are normalized by runner rotation frequency.

The experimental and numerical results are in a good agreement. Both experimental and numerical results in BEP show a dominant dimensionless frequency in 30 and around 15, which are standing behind all runner blades (runner blades and splitter blades) and runner blades passage frequencies, respectively. The experimental results show a dominant blade runner passage frequency of 18 instead of 15, which its reason is unknown. In PL, although experimental and numerical results show the aforementioned frequencies, there is another dominant frequency around $f/f_0=0.3$ due to the RVR rotation. The RVR frequency is shown in the zoomed part of Fig. 6(b) and 5(d). Both numerical and experimental results report this dominant frequency.
To understand the procedure of the RVR formation, a spectrogram of numerical pressure results is shown in Fig. 7. In order to illuminate the low-frequency region with a higher resolution, the range of the reported normalized frequencies in Fig. 7 is limited from 0 to 0.5. The normalized frequencies of 30 and 15 are also captured in the reported spectrogram, but the focus is on the low-frequency part and the frequencies of runner and splitters are not shown in this figure. In BEP, the RVR rotation frequency is not visible, but during the transition from BEP to PL, a new dominant frequency is seen in the figure with a normalized frequency of $f/f_0 = 0.24$. The frequency shows the trigger of RVR rotation during the transition from BEP to PL, and it remains as the simulation continues to PL. The RVR frequency shows unsteady behavior during the transition step and also at the beginning of the PL step, but after 0.31 seconds of being in PL mode, the RVR frequency becomes stable and remains in the simulation. The figure also clearly proves that the RVR rotation frequency is a low frequency with high amplitude.

Fig. 6 FFT of pressure results, (a) experimental BEP results, (b) experimental PL results, (c) numerical BEP results, (d) numerical PL results

Fig. 7 Spectrogram of numerical pressure fluctuations, Black solid line: guide vanes angle ($\alpha^\prime$); Dashed lines: start and stop of the guide vanes movement
4.3. Flow Structure

Employing the identification of rotational core ($\lambda_2$) criterion [27], the position and geometry of the RVR are visualized in Fig. 8 and Fig. 9. Fig. 8 shows different stages of the RVR formation. At the time of entering the PL operation (Fig. 8 (a)), the RVR is not formed. But as time passes (Fig. 8 (b)), the RVR starts to form. Two seconds after entering PL operation, the first signs of complete RVR formation can be seen (Fig. 8 (c)). Finally, 3.17 seconds after entering PL, the complete RVR is visible (Fig. 8 (d)).

In Fig. 9, the velocity vectors in vertical and two horizontal surfaces admit the same location for the vortex core. Moreover, it shows that although several rotating cores are created behind the runner blades; they are transformed to one unite core as moved downstream in the draft tube.

![Fig. 8 The visualized vortex core of the RVR at PL (a) 0 seconds after entering PL, (b) 0.54 seconds after entering PL (c) 2.19 seconds after entering PL and (d) 3.73 seconds after entering PL.](image-url)
5. Conclusion

The transient numerical simulation of a reduced model shows a good agreement with experimental results in both pressure fluctuations and axial velocity. The only captured dominant pressure frequencies in BEP condition are related to runner blades' passing frequency. Changing the working condition from BEP to PL, a new dominant frequency is seen on \( f/f_0 = 0.3 \), which is the RVR rotation frequency. Moreover, the trend of axial velocity during the transition from BEP to PL shows that in both numerical and experimental results, a low axial velocity region is formed and spread inside the draft tube with a radius of \( r/r_0 = 0.3 \). As both numerical and experimental results reported, the velocity of this region is much lower in PL in comparison to BEP. To visualize the RVR, \( \lambda_2 \) criterion is employed and shows that the predicted cores are accurate. The reduced model showed reliable results and can be used to reduce the computational costs effectively.

References

[1] W.J. Rheingans, "Power swings in hydroelectric power plants," Trans. ASME, vol. 62(3), pp. 171-184, 1940.
[2] T.B. Benjamin, "Theory of the vortex breakdown phenomenon," Journal of Fluid Mechanics, vol. 14(4), pp. 593-629, 1962.
[3] J.J. Cassidy, and H.T. Falvey, "Observations of unsteady flow arising after vortex breakdown," Journal of Fluid Mechanics, vol. 41(4), pp. 727-736, 1970.
[4] M. Nishi, "Flow regimes in an elbow-type draft tube," in IOP Conference Series: Earth and Environmental Science, 1982, p. 38.
[5] U. Andersson, 2009. An experimental study of the flow in a sharp-heel Kaplan draft tube, Luleå University of Technology (Doctoral dissertation, Doctoral thesis, ISBN: 978-91-86233-68-6).
[6] Jonsson, P., 2011. Flow and pressure measurements in low-head hydraulic turbines (Doctoral dissertation, Luleå tekniska universitet).
[7] Mulu, B., 2012. An experimental and numerical investigation of a Kaplan turbine model (Doctoral dissertation, Luleå tekniska universitet).
[8] M. Escudier, "Confined vortices in flow machinery," Annual Review of Fluid Mechanics, vol. 19(1), pp. 27-52, 1987.
[9] Zobeiri, A., 2009. Investigations of Time Dependent Flow Phenomena in a Turbine and a Pump-Turbine of Francis Type: Rotor Stator Interactions and Precessing Vortex Rope. EPFL Doctoral Thesis, (4272).
[10] C.H. Wu, "A general theory of three-dimensional flow in subsonic and supersonic turbomachines of axial-, radial, and mixed-flow types," (No. NACA-TN-2604). NATIONAL AERONAUTICS AND SPACE ADMINISTRATION WASHINGTON DC. 1952.

[11] C. Hirsch and G. Warzee, "A finite-element method for through flow calculations in turbomachines," Journal of Fluids Engineering, vol. 98(3), pp. 403-414, 1976.

[12] C. Hirsch, and G. Warzee, "An integrated quasi-3D finite element calculation program for turbomachinery flows," Journal of Engineering for Power, vol. 101(1), pp. 141-148, 1979.

[13] W.D. McNally and P.M. Sockol, "Computational methods for internal flows with emphasis on turbomachinery," Journal of Fluids Engineering, vol. 107(1), pp. 6-22, 1985.

[14] X. Zhang, Y. Cheng, J. Yang, L. Xia and X. Lai, "Simulation of the load rejection transient process of a Francis turbine by using a 1-D-3-D coupling approach," Journal of Hydrodynamics, vol. 26(5), pp. 715-724, 2014.

[15] Ch, Trivedi, M.J. Cervantes, G. Bhupendrakumar and O.G. Dahlhaug, "Pressure measurements on a high-head Francis turbine during load acceptance and rejection," Journal of Hydraulic Research, vol. 52(2), pp. 283-297, 2014.

[16] K. Amiri, B. Mulu, M. Raisee and M.J. Cervantes, "Unsteady pressure measurements on the runner of a Kaplan turbine during load acceptance and load rejection," Journal of Hydraulic Research, vol. 54(1), pp. 56-73, 2016.

[17] M. Gagnon, J. Nicolle, J.F. Morissette and M. Lawrence, "A look at Francis runner blades response during transients," in IOP Conference Series: Earth and Environmental Science, 2016, p. 052005.

[18] R. Goyal, C. Bergan, M.J. Cervantes, B.K. Gandhi and O.G. Dahlhaug, "Experimental investigation on a high head model Francis turbine during load rejection," in IOP Conference Series: Earth and Environmental Science, 2016, p. 082004.

[19] P. Mössinger, R. Jester-Zürker and A. Jung, “Francis-99: Transient CFD simulation of load changes and turbine shutdown in a model sized high-head Francis turbine,” in IOP Conference Series: Journal of Physics, Vol. 782, No. 1, 2017, p. 012001.

[20] K.R.G. Jakobsen and Holst, "CFD simulations of transient load change on a high head Francis turbine," in Journal of Physics: Conference Series, 2017, p. 012002.

[21] P. Dörfler, M. Sick and A. Coutu, "Flow-induced pulsation and vibration in hydroelectric machinery: engineer’s guidebook for planning, design and troubleshooting," ed: Springer Science & Business Media, 2012.

[22] F. Menter and Y. Egorov, "A scale adaptive simulation model using two-equation models," in 43rd AIAA Aerospace Sciences Meeting and Exhibit, 2005, p. 1095.

[23] R. Maddahian, M.J. Cervantes and N. Sotoudeh, "Numerical Investigation of the Flow Structure in a Kaplan Draft Tube at Part Load," in IOP Conference Series: Earth and Environmental Science, 2016, p. 022008.

[24] Y. Egorov and F. Menter, "Development and application of SST-SAS turbulence model in the DESIDER project," in Advances in Hybrid RANS-LES Modelling, ed: Springer, 2008, pp. 261-270.

[25] NTNU Webpage. 2016. Second Workshop. [ONLINE] Available at: https://www.ntnu.edu/nvks/second-workshop. [Accessed 31 March 2018].

[26] N. Sotoudeh, "Numerical Simulation of Transient Rotating Vortex Rope (RVR) Formation in the Draft Tube of Francis-99 Turbine," Master Thesis, Mechanical Engineering, Tarbiat Modares University (TMU), 2017.

[27] J. Jeong and F. Hussain, "On the identification of a vortex," Journal of fluid mechanics, vol. 285, pp. 69-94, 1995.