# HEAD LOSSES ANALYSIS IN SYMMETRICAL TRIFURCATIONS OF PENSTOCKS - HIGH PRESSURE PIPELINE SYSTEMS CFD

**C. A. AGUIRRE[1], R. G. RAMIREZ CAMACHO,[2]**

[12]Instituto de Engenharia Mecânica, Universidade federal de Itajubá.
Caixa Postal: 50 - CEP: 37500 903 - Itajubá – MG Brasil.
Av. BPS 1303, Bairro Pinheirinho.

ramirez@unifei.edu.br, caguirre@unifei.edu.br

## ABSTRACT.

Systems using trifurcations allows flow of water to provide several turbines operating at the same time. This arrangement presents smaller assembly costs in comparison of independent pipeline systems. However this installation can generate high losses in the system. This study focuses the quantified losses as a function of the volumetric flow rate, using computational fluid dynamics (CFD). To determine the coefficient of losses were analyzed three mesh settings: hexahedral, tetrahedral and hybrid, considering steady state flow. Based on the literature, the k-ω turbulence model, with refinement near wall elements, quantified the y plus. Results of loss coefficients for different discretizations are presented in this paper.

***Key words***: trifurcation, numerical simulation, SST, head loss, mesh.

## INTRODUCTION

The trifurcations are part of the architectural complex that forms the hydroelectric plant, which together with others, parts and equipment has the purpose to produce electricity using the hydraulic potential existing in a damming or a river. Whereas the optimal operating point of the pipeline systems, the losses must be reduced to obtain the best operating condition, with fields of stable flow. These conditions can be defined from tests in preliminary models to obtain appropriate geometries, with controlled load losses and variations of flow supplying the turbines.

The analysis of head loss can be done in the laboratory or with the use of tools of numerical simulation with the advantage of analysis of the local flow with the real dimensions, allowing easy generation and adaptation of geometries. Considering its application, both approaches are complementary, meanings that the numerical validation must necessarily represent qualitatively or quantitatively, the experimental results.

A lot of researches have been accomplished, in order to quantify the head losses in the pipeline systems of hydroelectric plants, focusing the best possible performance.

Wanng Hua (1967) made an experimental analysis, with several wyes configurations and manifolds (Figure 1). The effects of roughness on the wall were not considered, once the pipe surfaces were polished. The head losses in the dimensionless form were quantified with relation to the average flow velocity in the main pipe. Based on the one-dimensional energy equation results were obtained using data acquisition systems such as; dynamic pressure that is representative of the flow in a particular section of pipe, the pressure reading using a catheter was inserted at a position and height where the flow is irrotational and permanent.

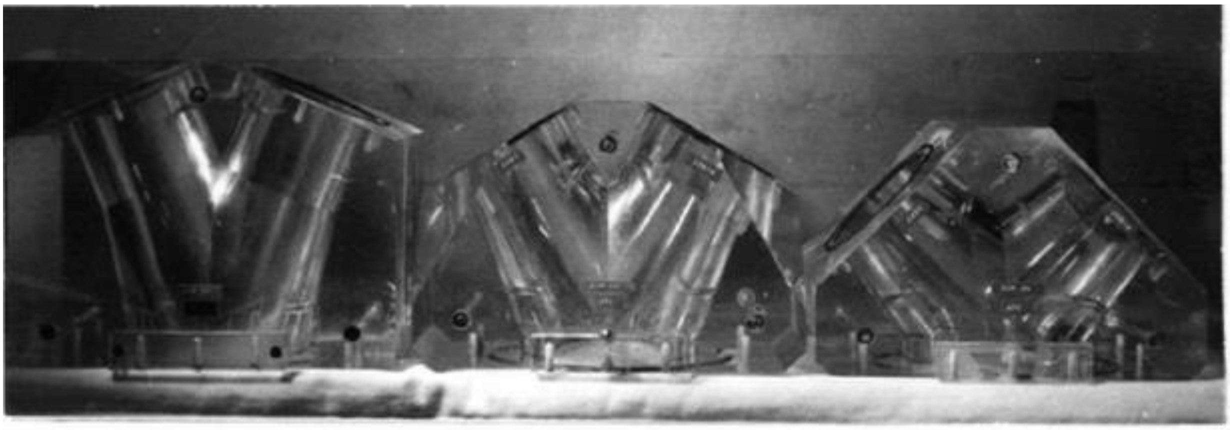

Figure 1 - Component of the critical section of wyes and manifolds, top view.

Rk Malik and Paras Paudel (2009) did an analysis for a small hydroelectric power plant of 3.2 MW, located in Kaski (Nepal). The constraints due the available space and the position of the turbines were considered for the design of the adduction system of the trifurcation, several tests were made focusing the optimal profile of trifurcation so the head losses are as low as possible.

The calculations of pressure losses were done using the energy equation between the entrance and three exits simultaneously. The turbulent and laminar regimes were analyzed using ANSYS CFD-FLOTRAN. Besides a tetrahedral mesh was generate, as shown in Figure 2. The boundary conditions were defined considering at the entrance, the gauge inlet pressure of 177 mmH2O, and the speed between 3 and 4 m/s and the static pressure at the outlet is equal to the local atmospheric pressure.

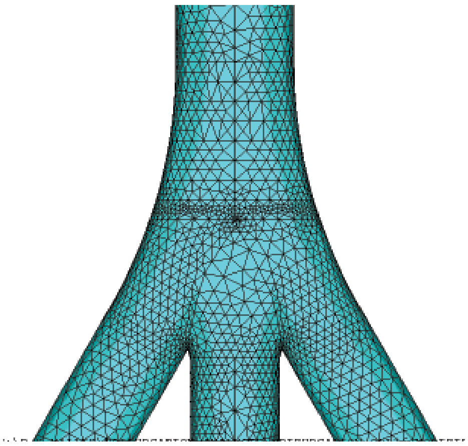

Figure 2 - Tetrahedral mesh of the trifurcation in the section of the flow separation.

Changes in the geometry of the trifurcation were made to get to a head loss of 0.42%. Hence twenty different configurations of the trifurcations were tested, including mechanical stresses analyses.

Sirajuddin Ahmed (1965) obtained results of the head loss in laboratory using three conventional configurations of the bifurcations in which was changed the angle between the branches from 60° to 90°, and the angle of taper for both 60°. Besides, the evaluations for two spherical bifurcations with angle between the branches of 90° and with different sphere diameters were checked.

During the tests, the field of turbulent flow with Reynolds number between $5x10^5$ and $3.75x10^5$ and a maximum flow rate of 0.92 cfs (0.03 $m^3/s$) were defined. The head loss coefficients for spherical bifurcations were higher than the bifurcations taper, the values of the first is 0.44 related to the bifurcation with the greater diameter sphere and 0.30 to the bifurcation with the smaller diameter sphere. The loss coefficients for the taper bifurcations are 0.16 for the 90° angle between the branches and 0.08 to 0.088 for angles of 60°. These results are for a symmetrical flow at the entrance of the bifurcation.

Buntić Ivana, Helmrich Thomas and Ruprecht Albert (2005) presented a model of Very Large Eddy Simulations (VLES). This model has an adaptive filter technique that separate the part of the fluid resolved numerically and the modeled part (Figure 3). The modeled parts use k-ε extended model of Chen and Kim. This model VLES is applied to simulate flows with unstable vortices in geometries where the turbulent flow cannot be performed with the classical models of turbulence.

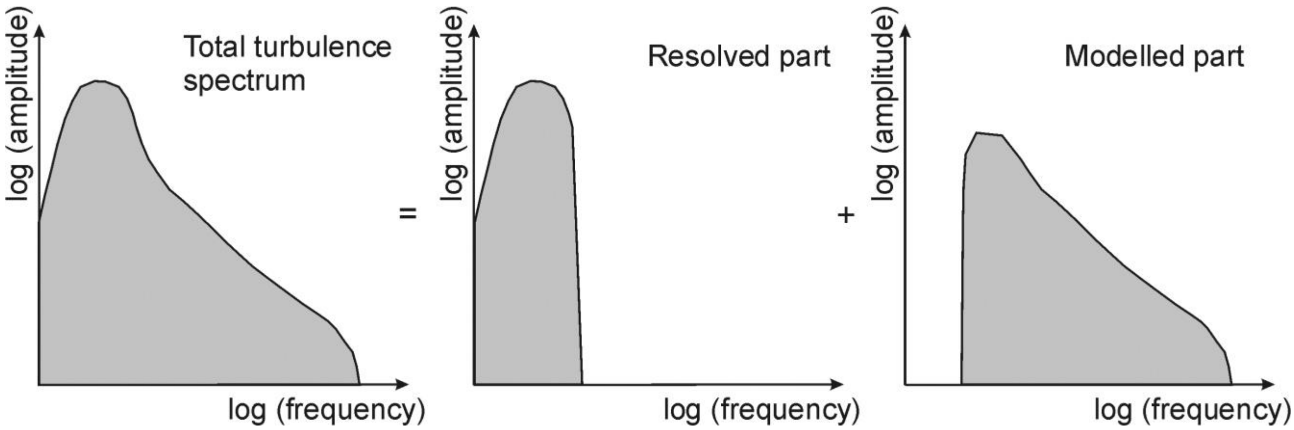


Figure 3 - Model Approach VLES

Moreover, this model tries to maintain the computational efficiency of the Reynolds-Average Navier-Stokes (RANS) and the potential for solving large turbulence structures of the Large Eddy Simulation (LES). Although the model can be applied in coarse meshes the simulation depends heavily on the modeling.

Additionally, Buntić Ivana, Helmrich Thomas and Ruprecht Albert (2005) had performed the simulation of a spherical trifurcation, Figure 4, which makes the distribution of water from the adduction system of water until the turbines. The outer branches present oscillations given by the vortices found in the flow. The variations are not periodic of a branch to another generating a high head loss.

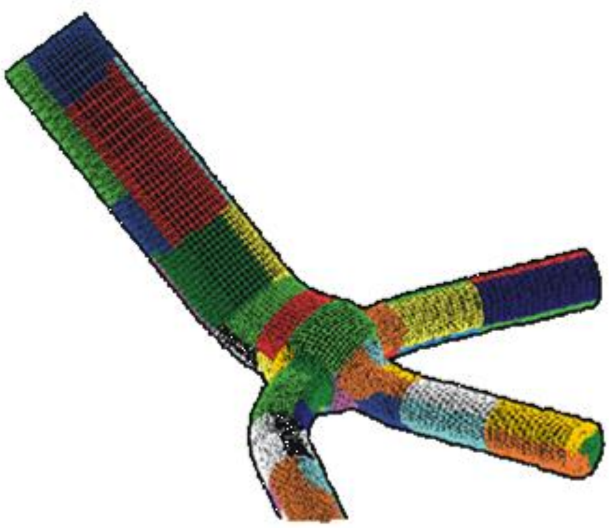

Figure 4 – Trifurcation - computational mesh.

# 1. MATHEMATICAL MODEL

Turbulent flows are characterized by transport of the large quantities of mass and momentum scalar that floating in the time and the space, not steady. The flow velocity and fluid properties have random variations in different spectrum ranges.

## 1.1 Equations for turbulent flow

The ANSYS-CFX software uses the equations of Reynolds (Reynolds Averaged Navier-Stokes RANS) to solve the problems of turbulent flow. In this model all dependent variables, scalars and vector are decomposed into a temporal average and a fluctuating part, when these variables are introduced in the conservation equation for not inertial systems results, as shown following equations.

Equation of conservation of mass

$$\frac{\partial}{\partial x_i}(\bar{v}_i) = 0 \tag{1}$$

The equation of conservation of momentum, considering the steady flow and inertial system.

$$\rho \bar{v}_j \frac{\partial \bar{v}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \mu \frac{\partial^2 \bar{v}_i}{\partial x_j^2} - \rho \frac{\partial}{\partial x_j} \overline{v_i' v_j'} + \rho \vec{g} \tag{2}$$

Generally the term of the turbulence and the viscous tensor are grouped. Thus the overall or general tensor is represented by.

$$\tau_{g_{ij}} = -\rho \overline{v_i' v_j'} + \mu \left( \frac{\partial \overline{v_i}}{\partial x_j} + \frac{\partial \overline{v_j}}{\partial x_i} \right) \tag{3}$$

The Reynolds tensor $\tau_t$ can be modeled appropriately using the Boussinesq hypothesis presented in terms of turbulent viscosity $\mu_t$ .

$$-\rho \overline{v'_i v'_j} = \mu_t \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial v_k}{\partial x_k} \right) \delta_{ij} \tag{4}$$

Where *k* is the kinetic energy and $\delta_{ij}$ is the Kronecker delta operator.

In this paper the turbulent viscosity is obtained using the SST turbulence model that uses the hypothesis of Boussinesq.

# 2. METHODOLOGY

The geometry of the trifurcation used in the research was provided by ALSTOM Figure 5. This model has 25 m wide, 7 m high and 39 m long. The pipe diameter into the fluid inlet (water 20° C) is 4.5 m and on all outputs 3 m, and in the trifurcation the approximate angle of the side branches are 60 degrees.

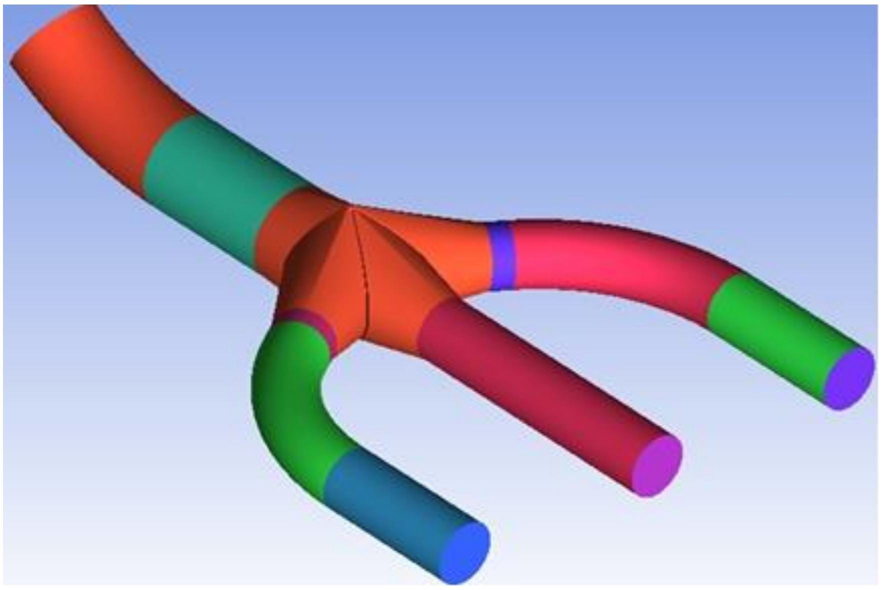

Figure 5 - General geometry of trifurcation.

Flow rates are measured within the range from 20 $m^3/s$ to 70 $m^3/s$, which still shows a permanent flow. Considering the dimensions and flow rates, the Reynolds number is approximately $2.014x10^{-7}$

As (Casartelli et al., 2010) and (Galarça et al., 2004), show that with a high Reynolds number and a complex geometry, the *SST* turbulence model can be applied, since this model can solve the problems of the models *k-ω* and *k-ε*. Thus, in regions with bends and nearby the wall is used the *k-ω* model, and regions farther from the wall the *k-ε* model. The *SST* model based on the *k-ω* considers the transport of turbulent shear stresses and provides accurate flow predictions for cases with adverse pressure gradients involving separation.

Moreover, the mesh generation requires the definition of the value of refinement of elements near the walls which can be done using an appropriated wall function "*y"*, Ariff (2009) shows how can obtain this value associated with the minimum *y+* that can be applied to the problem and the turbulence model.

Casartelli (2010) defines the *y+* range for adduction pipeline of a turbine, working with the turbulence model *k-ω SST* are between 200 and 500 because the model applies equations in the boundary layer. Thereby, this case using the *y+* of 300 defines a minimum distance for the initial layer of the mesh equal to $1.95x10^{-3}$ m.

The present work adopts ICEM-CFD® for preprocessing and generation of geometry and mesh. The geometry uses three composite meshes of different geometric elements inside it and near the surface. The main characteristics of the meshes showed in Table 1 and in Figure 7. The first mesh is hexahedral originated of approximately 400 blocs (Figure 6) with hexahedral refinement and exponential growth near the walls. The second mesh is composed of tetrahedrons and pyramids at the core and with layers of prisms with linear growth on the walls. The third mesh is composed of hexahedral and pyramids at the core and in the walls prism with linear growth.

Table 1. General characteristics of meshes.

| Mesh | Number of elements | Mesh type |
|---|---|---|
| Hexahedral | 7006388 | Structured |
| Tetrahedral | 4154711 | Unstructured |
| Hexahedral core | 2272218 | Unstructured |

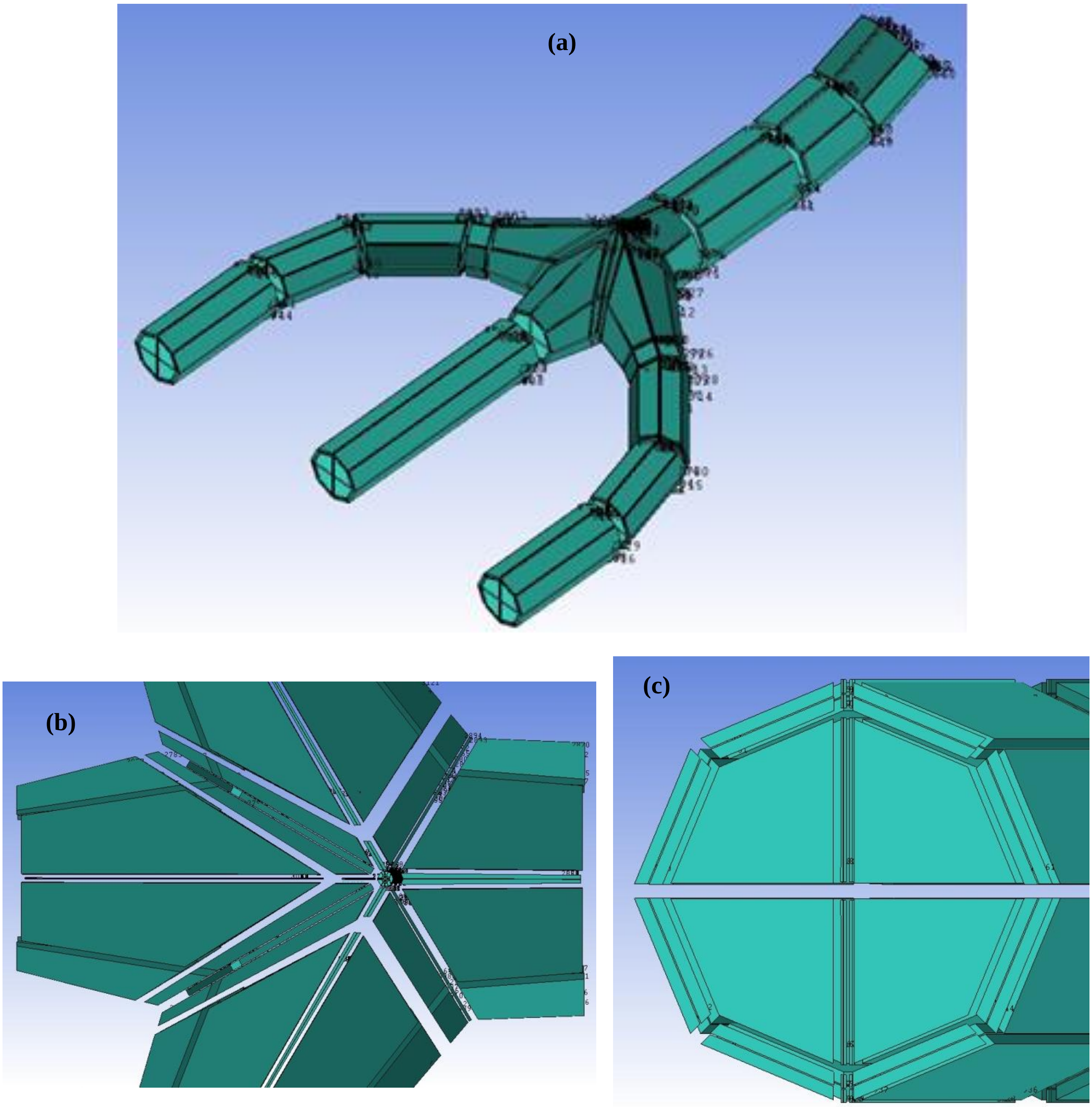


Figure 6 - Construction of blocs of hexahedral mesh isometric view (a), union of the four pipes (b) and views of the blocs that make up the cross section of the pipeline (c).

In the Figure 7 (a) shows the influence of mesh refinement near the wall with the number of elements because the refinement extends to the inside of the mesh where is not very useful, while the unstructured grids (b) and (c) present refinement only in the layers nearest to the surface reducing the number of mesh elements.

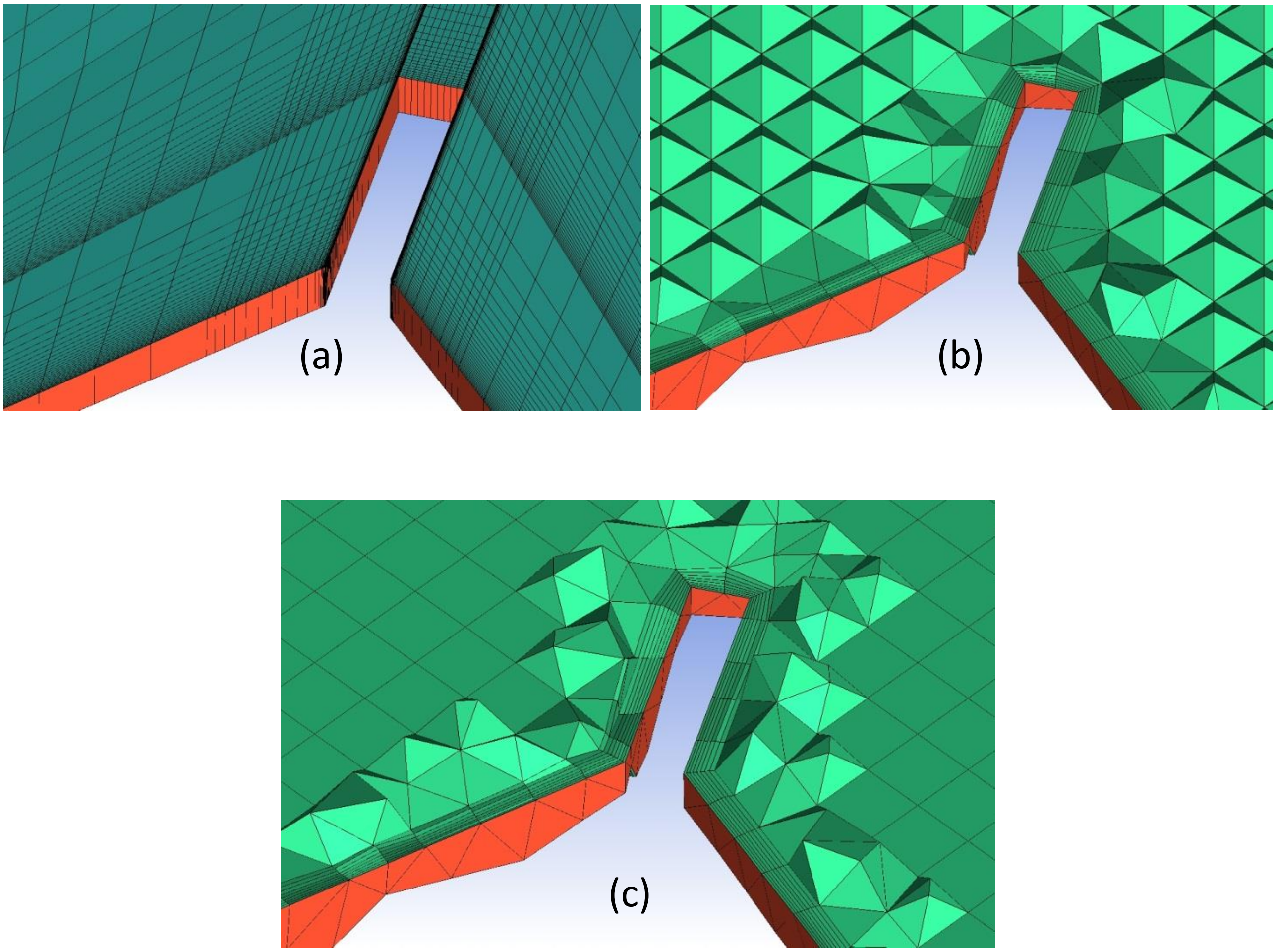


Figure 7 - Cutting Plane and behavior of the surface layers of mesh refinement for (a) hexahedral, (b) tetrahedral and (c) hybridize with hexahedral core.

With the range of mass flow rates, SST turbulence model and mesh generated, the *"solver"* software ANSYS-CFX ® is chosen for the numerical solution of the problem. The value of convergence RMS (root mean square) is fixed at $1x10^{-4}$ according to the values given by ANSYS CFX Solver theory guide (2012) for engineering researches and the ten points to be evaluated inside the range of volumetric flows are shown in Table 2. The boundary conditions for the entrance and exit are respectively mass flow and static pressure.

## 3. RESULTS

The velocity and pressure data obtained with the ANSYS-CFX program are used to calculate the head loss of each branch of the trifurcation, as shown by (Wang et al., 1967) who employs the equation 5, based on the dynamic pressure of the main pipeline for the calculation of the coefficient of head loss *k*.

$$k = \frac{(p_{T(r,c,l)} - p_{T\,Inlet})}{\frac{1}{2} \cdot \rho \cdot v_{Inlet}^2} \tag{5}$$

Where $p_{T(r,c,l)}$, corresponds to the values of total pressure in the branches, right, center and left, $v_{inlet}$, is the reference flow velocity at the entrance of the pipe.

Table 2. Coefficient of head losses of trifurcation given by the numerical approach, considering meshes with hexahedral and tetrahedral elements and hybrid mesh with hexahedral core.

| **Volumetric flow rate Q [m$^3$/s]** | **Coefficient of head losses *k*** | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| | **Left branch** | | | **Center branch** | | | **Right branch** | | |
| | **Mesh** | | | **Mesh** | | | **Mesh** | | |
| | **Hexa** | **Tetra** | **Core** | **Hexa** | **Tetra** | **Core** | **Hexa** | **Tetra** | **Core** |
| 20 | 0.513 | 0.442 | 0.444 | 0.329 | 0.268 | 0.265 | 0.515 | 0.429 | 0.431 |
| 25 | 0.456 | 0.424 | 0.423 | 0.279 | 0.252 | 0.252 | 0.457 | 0.415 | 0.412 |
| 30 | 0.448 | 0.415 | 0.409 | 0.258 | 0.238 | 0.237 | 0.443 | 0.403 | 0.403 |
| 35 | 0.446 | 0.406 | 0.404 | 0.252 | 0.228 | 0.228 | 0.442 | 0.397 | 0.399 |
| 40 | 0.426 | 0.400 | 0.405 | 0.245 | 0.220 | 0.214 | 0.423 | 0.386 | 0.389 |
| 45 | 0.430 | 0.396 | 0.397 | 0.242 | 0.213 | 0.215 | 0.442 | 0.377 | 0.391 |
| 50 | 0.424 | 0.389 | 0.400 | 0.234 | 0.208 | 0.206 | 0.446 | 0.374 | 0.373 |
| 55 | 0.423 | 0.394 | 0.403 | 0.231 | 0.201 | 0.204 | 0.429 | 0.373 | 0.373 |
| 60 | 0.426 | 0.383 | 0.397 | 0.223 | 0.198 | 0.200 | 0.402 | 0.362 | 0.372 |
| 65 | 0.435 | 0.388 | 0.392 | 0.220 | 0.194 | 0.198 | 0.433 | 0.368 | 0.372 |

In Figure 8 a, b, c are represented the losses coefficients of the three branches of the trifurcation as a function of volumetric flow rate. Three mesh configurations were analyzed: hexahedral (black line), tetrahedral with elements prismatic in the wall (blue line) and hexahedral core (red line). In all Figures 8 a, b, c, the analysis shows that the hexahedral mesh has higher values when is compared to the unstructured meshes. More specifically, in the Figures 8 a, b the hexahedral mesh has greater instability of the loss coefficient, compared to unstructured meshes. However, it shows that the unstructured meshes have similar behaviors, especially in the relation to head loss on the central branch.

These figures show that the smaller loss values are close to the nominal flow rate, 90 m$^3$/s. However, the analysis around this value requires an approach using transient models type URANS or LES. In this range, considering the phenomenon in the steady state, the desired value of convergence can be reached with *SST* (RANS) model.

The central branch presents head loss coefficients smaller, because only have change in the area of pipes due to the greater effect of energy dissipation is associated with the viscous friction at the wall, whereas the side branches have variation in cross-sectional area and a strong change in the direction of flow (secondary flow).

The trifurcation at the nominal condition, generally operates with flow rates above 60 m$^3$/s in the transient regimen where the coefficients for the central and lateral branches are around 0.2 and 0.4 respectively (Figures 8 a, b, c). Mays et al. (1997) recommends for symmetric trifurcations the value of 0.3 in the loss coefficient, for the three branches.

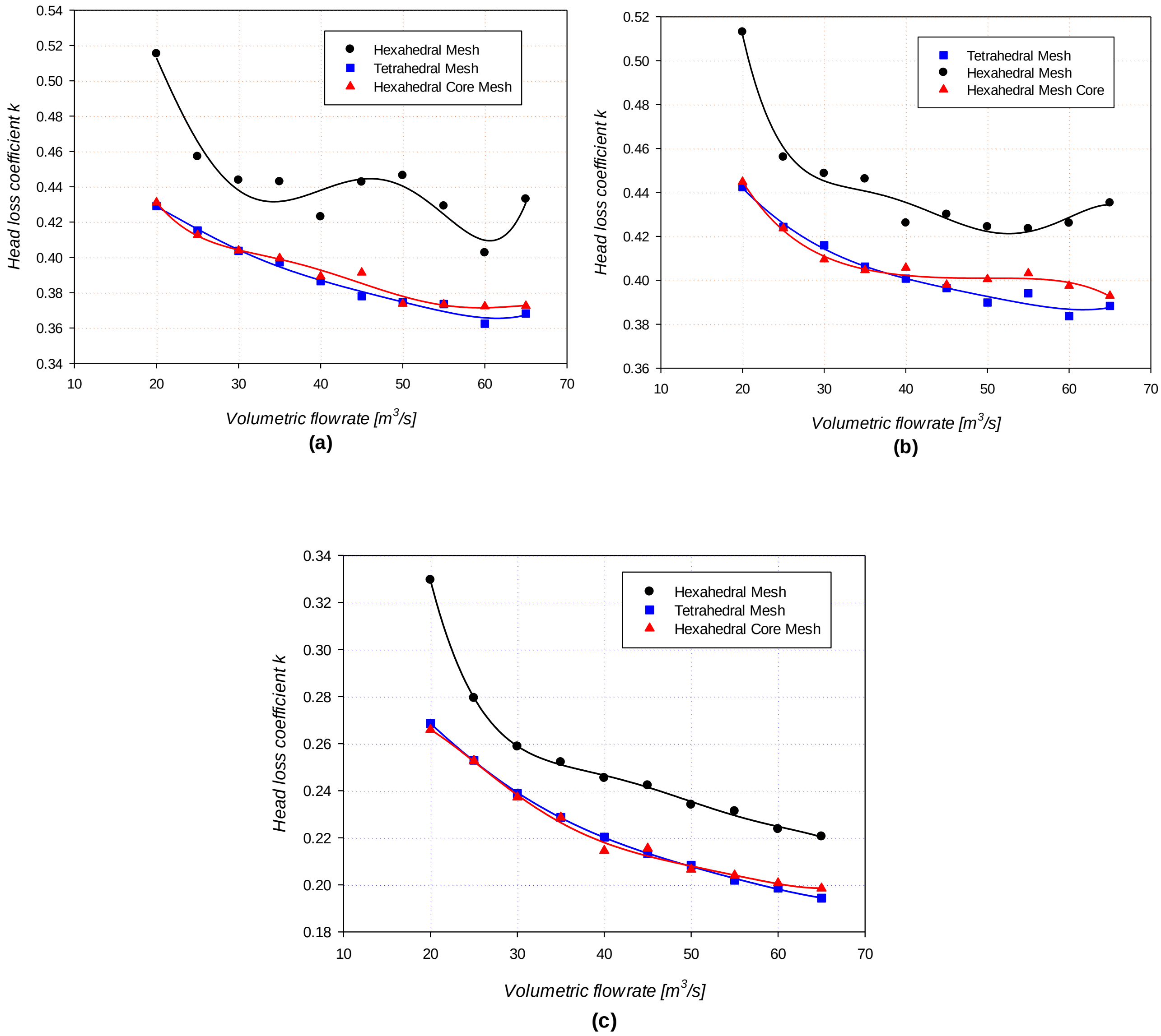


Figure 8. Head loss coefficients of the three meshes and left (a) right (b) and central (c) branches.

The behavior of the streamlines given by the velocity field show clear differences between structured and unstructured mesh as show in Figure 9, where structured meshes capture a formation and propagation of vortexes in the side branches larger than structured mesh and the velocities along the streamlines and the separation of the boundary layer are higher for the hexahedral mesh.

The hexahedral mesh in all flow rates were studied always reached the value of converge with fewer iterations than the unstructured grids. The differences in the number of iterations are between 50% and 70% less for the hexahedral mesh. Besides, comparing these meshes in relation to the number of iterations, the hexahedral mesh requires a minimal convergence value, but the tetrahedral mesh converges faster.

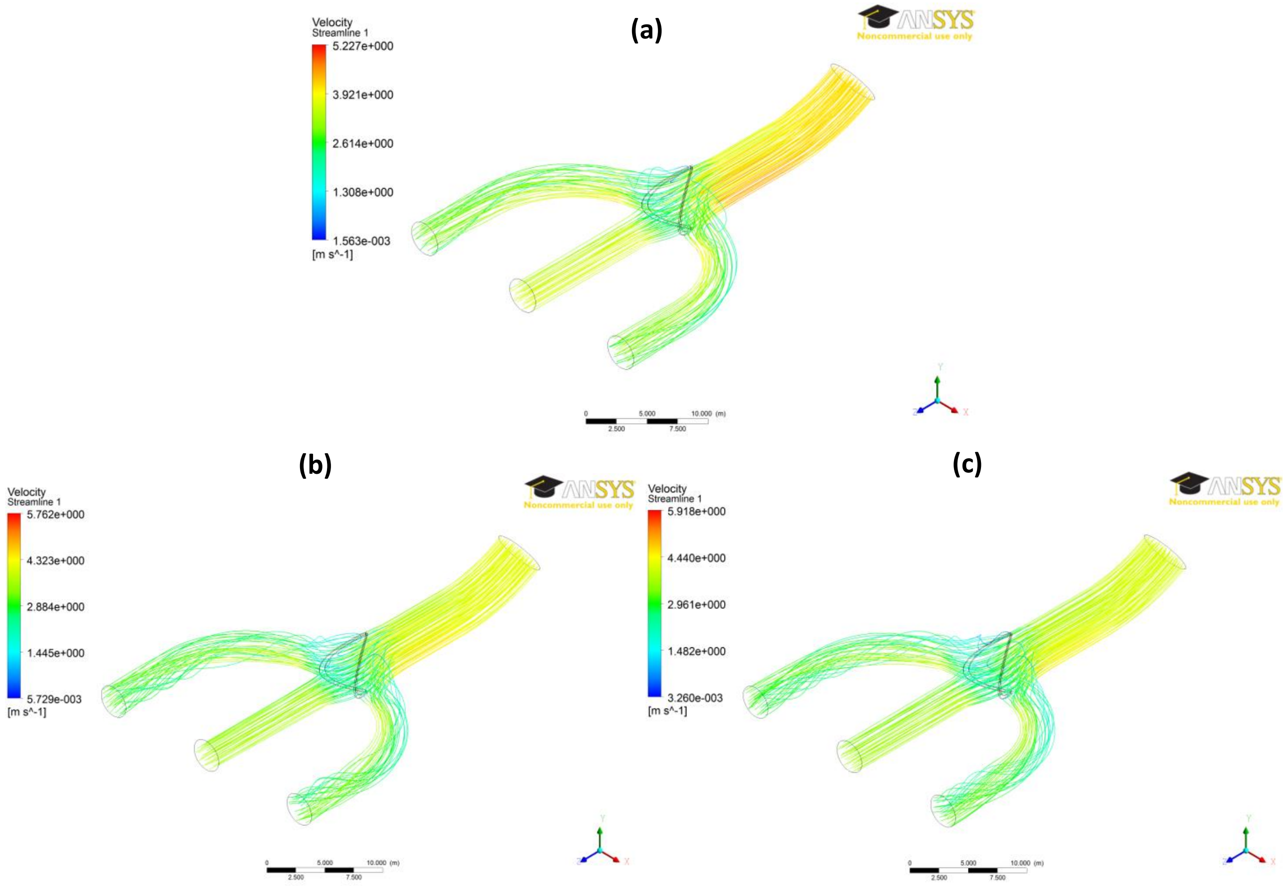


Figure 9 - Streamlines along the trifurcation of the hexahedral mesh (a), tetrahedral mesh (b) and hybridizes with hexahedral core mesh (c).

## 4. CONCLUSIONS

An analysis using Computational Fluid Dynamics CFD was presented to determine the losses coefficients in adduction systems of type *"symmetric trifurcation"*. The geometry was divided into structured and unstructured volumetric elements. Additionally, other analysis was done in relation to the velocity field, the trajectories of the streamlines checking variations when using different discretizations. Apparently the hexahedral mesh is more sensitive to quantify the head losses meanwhile the unstructured meshes show similar behavior between them and qualitatively with the hexahedral mesh. Therefore, it is necessary that the results are validated comparing its results with reduced model tests in specialized laboratories.


## ACKNOWLEDGMENT

The author acknowledges to ALSTOM Brasil Energia Transporte for financial and technical support


## BIBLIOGRAPHY


1. ANSYS, Inc. Southpointe, 2012, ANSYS CFX-Solver Theory Guide, Canonsburg, PA, USA.
2. Ariff M., Salim S. M., CHEAH S. C., 2009, Wall y+ approach for dealing with turbulent flow over a surface mounted cube: part 1 – low Reynolds number, Seventh International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia.

3. Buntić I., Helmrich T., Ruprecht A., 2005, Very large eddy simulation for swirling flows with application in hydraulic machinery, Scientific Bulletin of the Politehnica University of Timisoara Transactions on Mechanics Special issue, Timisoara, Romania.

4. Casartelli E., Ledergerber N., 2010, Aspects of the numerical simulation for the flow in penstocks, IGHEM-2010, Roorkee. India.

5. Galarça, M. M., 2004, Análise numérica para modelos de turbulência *κ-ω* e *SST/κ-ω* para o escoamento de ar no interior de uma lareira de pequeno porte, Programa de pós-graduação em Engenharia Mecânica – PROMEC, Universidade Federal do Rio Grande do Sul – UFRGS.

6. Mays L. W., 1997, Hydraulic design handbook, Editorial McGraw-Hill Education, New York, USA.

7. RK, M., Paras, P., 2009, Flow modeling of the first trifurcation made in Nepal, Hydro Nepal, Kathmandu, Nepal.

8. Sirajuddin A., 1965, Head loss in symmetrical bifurcations, The University of British Columbia, Vancouver, Canada

9. Wang H., 1967, Head losses resulting from flow through wyes and manifolds, The University of British Columbia, Vancouver, Canada.