



Article Minor Loss Coefficient for Abrupt Section Changes in a Cylindrical Pipe Using a Numerical Approach

José González^{1,*}, Andrés Meana-Fernández², Iván Vallejo Pérez³ and Jesús M. Fernández Oro¹

- ¹ Área de Mecánica de Fluidos (Energía), Universidad de Oviedo, Campus de Viesques, 33204 Gijón, Spain; jesusfo@uniovi.es
- ² Área de Máquinas y Motores Térmicos (Energía), Universidad de Oviedo, Campus de Viesques, 33204 Gijón, Spain; andresmf@uniovi.es
- ³ Volkswagen-Navarra, 31170 Pamplona, Spain; ivanvallejoperez14@gmail.com
- Correspondence: aviados@uniovi.es

Abstract: Abrupt section changes are a classic problem in the study of flow in cylindrical ducts or pipes. For its analysis, there are a wide set of exiting data from previous studies, among which some authors stand out and will be mentioned. Those previous works have been used to obtain reliable results for the resolution of section changes along a pipe, either due to cross area increases or reductions on a 1D basis. It is also known that a numerical 2D axisymmetric simulation (CFD) could find a consistent result compared to experimental data in almost all fluid flow fields. The main novelty of the present study is the development of a simple numerical approach used to solve the minor loss calculation. Firstly, a theoretical analysis is developed, and then the results of the numerical simulations carried out on the behavior that affects the water and air flow rate in an abrupt section change, for both contraction and expansion problems, are presented. In both cases, the results are analyzed with different meshes (discretizations) and turbulence models. Finally, the obtained numerical results are compared with those in the technical literature. Also, a theoretical approach is shown in order to show a whole frame of the discussion. The core results are the loss coefficient evolution as a function of the section change both for the sudden contraction and the expansion of a pipe flow. As the results follow the existing experimental values, it is concluded that the developed model provides a feasible and quick design tool to analyze possible geometrical changes without the need for further experiments.

Keywords: pipe flow; numerical flow simulations; CFD analysis; turbulent models; abrupt section change; area ratio; pipe flow patterns

1. Introduction

Due to different elements used in pipe flows to correctly deliver or handle a given fluid, not only the pipe's linear (or major) losses but also the minor ones have been extensively studied and constitute a classic problem in Fluid Mechanics. Among the minor loss calculations, the sudden change in cross-section at a given pipe constitutes a good example of an extensively studied problem. Nevertheless, not many numerical approaches have tackled it and, to the authors knowledge, some innovative approaches in the loss coefficient calculations are still to be developed. Obviously, using a fully 3D solution would be the best option, but our interest was to develop a quicker and more engineering-like valid solution.

The section changes in a cylindrical pipe can be branched out in different cases depending on the how such section change is defined [1]. Within those classifications, the following can be considered, among others: abrupt contraction, rounded contraction, conical contraction, and beveled (flanged) contraction. In this article, the analysis will be focused on the abrupt contraction, whose model is shown in Figure 1.



Citation: González, J.; Meana-Fernández, A.; Pérez, I.V.; Oro, J.M.F. Minor Loss Coefficient for Abrupt Section Changes in a Cylindrical Pipe Using a Numerical Approach. *Fluids* **2024**, *9*, 152. https://doi.org/10.3390/ fluids9070152

Academic Editors: Kamil Urbanowicz and Markus Klein

Received: 23 April 2024 Revised: 24 May 2024 Accepted: 21 June 2024 Published: 26 June 2024



Copyright: © 2024 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (https:// creativecommons.org/licenses/by/ 4.0/).



Figure 1. Streamline sketch through an abrupt contraction in the pipe (sketch adapted from [2]).

The main fluid feature behind the solution for the abrupt contraction flow, is the fact that this section change promotes the sum of two effects: a first contraction of the flow that produces an acceleration, followed by an expansion that produces a deceleration [1–4]. The point where the change from acceleration to deceleration occurs is the area where the flow reaches its smallest section, known as the contracted vein or "vena contracta", see [1].

In this phenomenon, shown in Figure 1, the total head loss is the sum of the losses due to the contraction between the last point of adherence (Section 1) and the "vena contracta" (Section 2), plus the losses of the expansion from the area of contracted vein (Section 2) and the exit of the flow or reattachment area (Section 3).

The sudden expansion process can be defined, parallelly to the contraction geometry, on the basis of the section exchange rate [1]. The following geometrical options are available in this classification: abrupt expansion, direct expansion with conical diffuser, multi-stage expansion with conical diffuser, expansion with curved wall diffuser, and pipe reducer, among others. In the present article, the analysis will be based on the geometry of an abrupt expansion, whose model can be observed in Figure 2.



Figure 2. Streamline sketch through an abrupt pipe expansion (sketch adapted from [2]).

The main flow feature for the abrupt expansion is the separation of particles from the imposed geometry change at the separation point which is located just at the corner defined by the expansion zone (abrupt section change, as shown in Figure 2). That separation induces the eddy generation in a recirculation region, also called the free mixing zone. Downstream of separating point and the so-defined recirculation zone, the flow returns to

the imposed geometrical condition at the reattachment point (right-hand side in Figure 2). The critical issue there is to correctly describe the reattachment point.

As depicted before, flows through section changes have been widely studied by numerous authors in previous works. In such works, difficulties came from the accurate flow description when the case of abrupt section changes were considered. The correct location of the separation and reattachment positions are always the key issue in many studies. A first part with initial approaches (from the early XX century) is developed here, and more recent values are then discussed in the final part of the article. In order to show an evolution of this subject over the years, the contributions of some of the most important authors in this field are reviewed chronologically.

An interesting review in the study of the flows along a cylindrical duct, calculating the so-called minor losses, as those generated due to contractions and expansions, can be found in [3].

Previously, the BSc thesis in [4] develops an experiment about abrupt section changes in the water flow through a pipe. From those results, it is concluded that the losses due to expansion are approximately three times those produced during contraction. The study relies on previous studies by Gibson (in 1910 and 1912) and Weisbach (in 1855), as stated and reordered in [3]. In addition, [4] determines two exponential formulas for the estimation of the load losses due to the sudden section change, following:

$$\begin{cases} \text{Expansion}: & h_{loss} = 0.00932 \ R^{0.413} V^{1.885} \\ \text{Contraction}: & h_{loss} = 0.00187 \ R^{0.88} V^{1.84} \end{cases}$$
(1)

In [5], an equation is provided for the calculation of energy losses due to abrupt expansion. This relationship has the particularity of depending on the speed of flow in both sections:

$$h_{\rm loss} = \frac{(V_2 - V_1)^{1.919}}{2g} \tag{2}$$

Later on, [6] provided experimental results about load losses and the influence of the Reynolds number on them. To analyze the influence of the Reynolds number, changes in the speed profiles were examined. In the textbook [7], it is determined that the coefficient due to abrupt expansion is determined by the following equation:

$$K_{\rm cont} = C_{\rm a} \left(\frac{A_1}{A_2} - 1\right)^2 \tag{3}$$

where C_a is an experimental factor (dimensionless) that is determined by an experimental graph [7].

On the other hand, [8] started to use the non-dimensional equation to determine the loss coefficient, that is:

$$h_{\rm loss} = \frac{V_2^2}{2g} \tag{4}$$

The main goal in [2] was to define a coefficient to determine the losses during the contraction. Thus, the following relation is obtained from data obtained through experimentation, using water and air as fluids, and their extrapolation is:

$$h_{\rm exp} = 0.57806 + 0.39543\beta - 4.53854\beta^2 + 14.24265\beta^3 - 19.22214\beta^4 + 8.54038\beta^5$$
(5)

Then, in 1969, Idelcik published his well-known work, "Memento des pertes de charge" (in English: "Handbook of hydraulic resistance" [9]), in which one chapter deals with the flow in pipes and channels. In the chapter, Idelcik proposes a value for the coefficient of head losses in various types of contraction, including the case of an abrupt contraction. This same equation is later proposed by Reza (1985) and takes the following formulation ($\text{Re} > 10^4$):

$$h_{\rm loss} = \frac{\gamma V_{D_2}^2}{2g} \left(\frac{1}{2} \left(1 - \frac{A_2}{A_1} \right) \right) \tag{6}$$

In [10], a work is developed from the results and data provided by [6], and based on his conclusions, two different approaches to the two considered problems are proposed, both for expansion and contraction, including the consideration of the "vena contracta".

As stated in [3], Weisbach (in 1972) provided a summary of the previous work on this subject, publishing a book containing the results of all the previous authors. He also contributed new experiments carried out by his team, which were adjusted from [1].

In [11], the authors studied the differences in the definition of the loss coefficient from a one-dimensional analysis of incompressible flow, highlighting some difficulties with the geometry of the contraction and flow for real pipes and fluids. Those authors have published many articles on the topic. In particular, in [12], they used the equation by Bullen et al. (back in 1983) to obtain, in an experimental and theoretical way, coefficients using different area ratios and an Re range that oscillates between 4×10^4 and 2×10^5 . The definition selected for the pressure loss coefficient in the contraction is the following:

$$K_{\rm cont} = \frac{2gA_2^2h_{\rm loss}}{Q^2} - \left[1 - \left(\frac{A_2}{A_1}\right)^2\right]$$
(7)

The study concludes that the values of pressure loss coefficients from the static wall pressure distribution are directly proportional to the Re (in a study range of $4 \times 10^4 < \text{Re} < 2 \times 10^5$). It also determines that the pressure loss is affected by how abrupt the contraction is. It also ensures that the results are valid with an accuracy of $\pm 5\%$ to $\pm 10\%$ at high Reynolds numbers. Using low Reynolds numbers, this hypothesis should be used with caution.

In more recent years, numerous studies and articles have been presented with the aim of making new contributions in this field, as well as compiling information from previous work, some of which are briefly mentioned in what follows.

As explained in [13], other contributions used the equation from the work of [5] to calculate the energy losses caused by abrupt expansion. In [14] and in [15], it is established that the loss coefficient due to abrupt expansion might be determined by means of a graph, based on the area ratio between the two conduits. For instance, [16] compiles the previous values and provide as new data a value of the pressure loss coefficient for the inlet from a pipe to a large tank, where the following condition is imposed: $D_2 >> D_1$ (so $D_1/D_2 \approx 0$). In [17], on the basis of previous documents, the following equation is proposed for the loss coefficient in an abrupt contraction, namely:

$$K_{\rm cont}\left(\frac{V_2^2}{2g}\right) \approx 0.42 \left[1 - \left(\frac{D_2}{D_1}\right)^2\right] \left(\frac{V_1^2}{2g}\right)$$
 (8)

Also, in [1], some previously developed studies were considered to finally obtain the "shrinkage loss coefficient" based on a document dated from "The Crane Company" (1957). In [1], there is a study that adjusts the data obtained in [3]. From that study, a large number of equations and tables were developed to obtain the total losses in different types of contraction and expansion (varying according to the model of section change). The following equation is finally proposed for calculating the losses in an abrupt contraction:

$$h_{\rm loss} = K_{\rm acc} \left(1 - \beta^5\right) \lambda^2 + (\lambda - 1)^2 \tag{9}$$

where K_{acc} represents the coefficient of losses due to the acceleration of flow and is taken as a value of 0.0696. In addition, $\beta = D_2/D_1$, and the flow speed ratio λ is found as:

$$\lambda = 1 + 0.622 \left(1 - 0.215\beta^2 - 0.785\beta^5 \right) \tag{10}$$

In [1], a value is also provided for pressure losses in an abrupt expansion based on the Borda–Carnot Equation [18]:

$$h_{\rm loss} = \frac{(V_1 - V_2)^2}{2g} \tag{11}$$

Some of the previously mentioned results are plotted in Figure 3, in the same kind of graph and variables as were chosen in [12], namely the loss coefficient as a function of the area ratio. Although a complete list would be almost impossible to compile, and more than the ones mentioned have been considered, it could be worth mentioning here the results in [1,2,6,9]. The results of [2] come from the plot of Equation (5). Even though the equation is a continuous curve, only four discrete values have been plotted for clarity (not to be confused with the values from [1]).



Figure 3. Comparison of the experimental loss coefficients in an abrupt contraction [1,2,6,9].

Recent numerical approaches for similar problems have been proposed by different authors: for a numerical strategy, see [19], or for experimental data in convergent–divergent flows, see the work by [20], which considers a light slope instead of a sudden section change. A basic approach intended for CFD teaching was also proposed by [21], and a solution similar to the one presented here but, once again, for a progressive section change, not for an abrupt one and in the frame of a porous media kind of application, was published by [22].

From the previously mentioned bibliographical study, a theoretical 1D model is developed in the following text (Part 2) to then show a 2D numerical axisymmetric model (Part 3) and both analyze its results and compare it with the exiting data in the bibliography (Part 4). The 2D numerical model developed in the present article offers a practical and efficient method to analyze pressure losses for contraction or expansions in pipe flow. Since the numerical results follow the existing experimental data, it can be used as a quick loss predictor for geometrical changes in conduits without the need for further experimentation. A model verification with existing bibliographic values is shown as a conclusion.

2. Theoretical Model

The two different geometries are now considered on a theoretical 1D basis in an effort to have a clear insight into the involved variables. What becomes straightforward when reading the different available bibliography is that the abrupt expansion is a much easier problem than the sudden contraction. The main reason for this is the actual complexity that the contraction adds to the problem in terms of the description of the "vena contracta" [23]. Therefore, even though both have been considered in theoretical description, only the equations for the sudden contraction will be shown and described.

Both problems will be handled using the energy equation with the losses of the element and the control volume momentum conservation equation. A classical solution that goes back to the Borda–Carnot Equation for a given element in an incompressible flow (*V* being the flow average velocity in the element), namely:

$$h_{\rm loss} = \frac{(V - V_2)^2}{2g}$$
(12)

The calculation for the losses starts with the control volume definition. In this part of the present study, a control volume is chosen to handle the geometry on the basis that the friction losses are negligible, and that then, the only effects to be considered are the boundary conditions (at flow inlet and outlet) and the pressure in the surface after the sudden contraction (see Figure 4).



Figure 4. Control volume for the sudden contraction geometry (1D model).

The first approach is a basic calculation, considering that $V_2 > V_1$, and the other boundary conditions are shown in Figure 4. And then, the forces equilibrium is derived as:

$$P_1A_1 - P_2A_2 - P_0(A_1 - A_2) = \rho Q(V_2 - V_1)$$
(13)

If the linear losses after the expansion are to be neglected, $(P_0 \approx P_2)$, and then:

$$P_1A_1 - P_2A_2 - P_2(A_1 - A_2) = (P_1 - P_2)A_1 = \rho Q(V_2 - V_1) = \rho A_1 V_1(V_2 - V_1)$$
(14)

And using the energy equation from (1) to (2) (Figure 4), the following can be obtained:

$$h_{\rm loss} = \left(\frac{f_1 L_1}{D_1}\right) \frac{1}{2} \frac{V_1^2}{g} + \left(\frac{f_2 L_2}{D_2} + \xi_{\rm cont}\right) \frac{1}{2} \frac{V_2^2}{g}$$
(15)

Now, the value of (15) is substituted in (14) considering the boundary conditions. Those include the inherent potential energy or height variations equal to zero. The linear (or major) losses that are non-relevant for the given geometry are also considered. Then, the losses in the expansion are found as:

$$\frac{V_1(V_2 - V_1)}{2g} = \frac{V_2^2 - V_1^2}{2g} + \xi_{\text{cont}} \frac{1}{2} \frac{V_2^2}{g}$$
(16)

And the final expression for the coefficient is then obtained as:

$$\xi_{\rm cont}\left(\frac{1}{2}\frac{V_2^2}{g}\right) = \frac{V_2^2}{2g}\left[1 - \frac{A_2}{A_1}\right]^2, \qquad \xi_{\rm cont} = \left[1 - \frac{A_2}{A_1}\right]^2$$
 (17)

Obviously, this solution is quite symmetrical to the case of the expansion or enlargement, but it does not look very realistic as the flow overcomes many more different features, such as the stagnation of the flow and subsequent "vena contracta", which seems again to be in the root of such behavior. The question lays then on the value chosen for the velocity on the initial section (see Figure 4). Accordingly, a table can be built by choosing different plausible values. The results are shown in Table 1, which include as a final row the value of (17), as developed in the present model, and will furtherly be referred to as "simple 1D theory", in the final figures of the article.

Table 1. Different losses according to the bibliography and 1D developed model (Equation (17)).

Value of V_0 , (m/s)	Losses, h_{Loss} (m)	Cont ()	
$V_0 = \frac{V_1 + V_2}{2}$	$\frac{1}{2}\frac{V_2^2}{g}\left[\frac{1}{2}-\frac{1}{2}\left(\frac{A_2}{A_1}\right)\right]^2$	$\left[rac{1}{2}-rac{1}{2}\left(rac{A_2}{A_1} ight) ight]^2$	Equation (6) by [9]
$V_0 = V_1 + \frac{V_2}{2}$	$\frac{1}{2}\frac{V_2^2}{g}\left[\frac{1}{2}-\left(\frac{A_2}{A_1}\right)\right]^2$	$\left[rac{1}{2}-\left(rac{A_2}{A_1} ight) ight]^2$	
$V_0 = \frac{V_1(A_1 - A_2) + V_2 A_2}{A_1}$	$\frac{1}{2} \frac{V_2^2}{g} \left[1 - \left(\frac{A_2}{A_1} \right) \right]^4$	$\left[1-\left(\frac{A_2}{A_1}\right) ight]^4$	Equation (3) by [7]
$V_0 = V_2 \frac{A_2}{A_1}$	$\frac{1}{2} \frac{V_2^2}{g} \left[1 - \left(\frac{A_2}{A_1} \right) \right]^2$	$\left[1-\left(rac{A_2}{A_1} ight) ight]^2$	1D model Equation (17)

3. Numerical Model

3.1. Geometry

The simulations and the computational solutions have been developed from the basic geometries shown in Figure 5, which is a simple sketch (not scaled). As the geometrical values are a critical issue for the final results, and considering a 2D axisymmetric model, the cylindrical pipe with the sudden cross-section change (expansion/contraction) is then numerically treated, and details are given in what follows. For the contraction geometry (right), the velocity profiles will be shown in x = 1.0 m, x = 1.7 m and x = 3.5 m (end of the domain), while the data reduction for the loss coefficient is done in x = 1.7 m and x = 1.0 m, x = 1.7 m and x = 3.5 m (end of the domain), while the data reduction for the loss coefficient is done in x = 1.7 m and x = 1.0 m, x = 1.7 m and x = 3.5 m (end of the domain), while the data reduction for the loss coefficient is done in x = 1.7 m and x = 1.5 m and x = 1.95 m.



Figure 5. Data recording positions in the two geometries: contraction (left), expansion (right).

The geometries have been generated using the software application Gambit 2.4.6 (2007), and the contraction (Figure 6, left) consists of two pipes: a first one with diameter $(D_1 = 0.025 \text{ m})$ and a second one, whose diameter (D_2) varies depending on the case of the simulation, as shown in Table 2. A global arrangement and mesh detail is shown in Figure 6. The total length of the computing domain is 3.5 m, ensuring the axial length necessary to achieve a fully developed flow at the outlet, and placing the section change close to the midpoint of the total length of the duct. More details will be given when boundary conditions are discussed later on.



Figure 6. Global mesh and detail for a given case (contraction), in the present model.

Studied Cases	Mesh Type	Number of Cells
Geometry with $D_2 = 0.9D_1$	Single	322,050
Geometry with $D_2 = 0.7D_1$	Single	410,250
Geometry with $D_2 = 0.5D_1$	Coarse Standard Refined	245,850 420,679 694,028
Geometry with $D_2 = 0.3D_1$	Single	395,000
Geometry with $D_2 = 0.1D_1$	Single	356,250
No step geometry	Coarse Standard Refined	70,000 115,000 210,000

Table 2. Summary of the generated meshes.

For the cases of no section change or only a linear loss geometry ($D_2 = D_1$) and contraction pipes with an area ratio of 0.5 ($D_2 = 0.5D_1$), three different meshes have been generated, increasing the number of cells in each one and using these cases to ensure that the results obtained are independent of the mesh.

The program used for the simulations was Ansys-Fluent 6.3.26 (2005). The 2D axisymmetric and stationary model has been selected to analyze the flow in the contraction and expansion study. The size of the meshes in the area near the wall is smaller, in order to adapt it to the thin layer with higher gradients expected there, as explained in Figure 6, mainly at the critical part of the domain, where the section change is produced. The cell size of wall-adjacent meshes in all solid boundaries has been fixed to preserve y+ values below unity.

3.2. Mesh Independence Study

In order to study the independence of the mesh, pressure gradients and pressure losses will be analyzed throughout the analysis domain of the pipe, both for the phenomenon of contraction and for that of expansion, using different meshes in the case of $0.5D_1$ (coarse, medium, and fine meshes) and always considering the turbulence model of κ - ϵ .

The study of the independence of the mesh has been carried out through the three meshes for the case of $0.5D_1$, simulated from the turbulence model of κ - ϵ . The results of both the pressure drop and subsequent pressure loss have been analyzed using the data obtained from the simulations with the different meshes, and it will be proved that the results are independent of the mesh used for the simulations.

As can be seen in Table 3, the values of pressure and head differences between the inlet and outlet are similar in all the meshes used for each phenomenon. Therefore, it can

be concluded that the simulations are correct and that the results obtained with the refined model are representative of the solution to the studied problem.

Geometry	Mesh Type	Pressure Losses (Pa)		
	Coarse	36,395.11		
Sudden contraction	Standard	36,422.75		
	Refined	36,286.85		
Sudden expansion	Coarse	20,304.47		
	Standard	20,295.86		
	Refined	20,501.45		

Table 3. Mesh independence tests for the case $D_2 = 0.5D_1$.

3.3. Turbulence Model

The physical characteristics that allow the identification of a turbulent flow have been defined by many scholars (see, for instance, [24]) and are the following: irregularity, high diffusivity, high Reynolds number values, high dissipation, and continuity. The turbulent model is the term often used in numerical flow simulation to handle a proper approach to the turbulent shear stress tensor, that is, with the tensor as:

$$\overline{\tau_{i,j}} = -\rho \overline{u'_i u'_j} \tag{18}$$

Such a tensor is often referred to in the bibliography as the Reynolds stress tensor, and it shows up along the averaging process of the Navier–Stokes Equations. That averaging process is quite convenient to solve the incompressible turbulent flow by splitting a given unsteady variable into an average term plus an oscillating term.

On the present article, the three turbulent models that are compared are the Spalart– Allmaras model (S-A) and two different two-equation models (κ - ϵ and the κ - ω). Therefore, for that two-equation models, the presented simulations will lie in the RANS (integral zone in the graph in Figure 7). The two equation models are used independently, not as in the Menter's SST model [25].



Figure 7. Different levels in the approach of the turbulence effects (adapted from [25]).

As a final complement and to check the influence of the possible two-equation closure conditions [26], a calculation with a Reynolds stress model (RSM) is performed. Usually, the RSM is better indicated for 3D geometries, but in the case of 2D axisymmetric ones, it seems that accurate results can also be obtained. Moreover, this RSM looks to be best suited to predict recirculation and reattachment. Therefore, a comparison of the results for the studied geometry will be shown.

The RSM calculations were implemented in the Ansys Fluent 5-Equation model, [27]. It uses a linear pressure-strain approach, with wall boundary conditions from the κ -Equation and wall reflection effects, as developed by [28]. The near wall treatment was chosen to be the enhanced one.

3.4. Fluid Properties

Regarding the conditions of the fluid, data have been taken from the program for liquid water (water-liquid), which are, among others, the following values, summarized in Table 4.

Symbol	Value (unit)
ρ	998.2 kg/m ³
Cp	4182 (J/(kg K))
μ	$1.0003 \times 10^{-3} \text{ kg/(m s)}$
М	18.052 (g/mol)
	Symbol ρ C _p μ Μ

3.5. Boundary Conditions and Geometry

The general convergence criterion used when carrying out numerical flow simulation (CFD) analysis is variable throughout the literature in this area, as previous works use different methods to either verify or validate their results. In this work, numerical computations are considered to have converged when the residues of the different variables are less than six orders of magnitude in all cases (sum of residuals in the used domain under 10^{-6}), reaching down to eleven orders of magnitude in some variables and simulations (10^{-11}) . To carry out the simulations, it is necessary to establish the boundary conditions. The following ones were considered:

- 1. Domain outlet: Atmospheric pressure conditions are stablished.
- 2. Domain inlet: It is necessary to discern the cases of contraction and expansion. The details of the conditions are explained in the strands (3) and (4), which follow.
- 3. Inlet velocity for the sudden contraction: before the simulations results are considered valid, the speed profile at the exit of the finest straight pipe mesh is obtained. This speed profile is imposed as the input speed of the contraction simulation. In such a way, the velocity profiles at the inlet are fully developed ones.
- 4. Inlet velocity for the sudden expansion: before the recorded/final simulations, the speed profile at the exit of the finest pipe mesh in the contraction is obtained. This speed profile is imposed as the input speed of the expansion simulation. Therefore, the velocity profile at the inlet are again fully developed ones.
- 5. Wall condition, for the pipe surface, in the different domain limits that correspond to the solid wall condition.
- 6. Axial symmetry of the whole domain, as global condition for the flow behavior.

The numerical model firstly managed to obtain a series of main parameters to be fixed, namely the number of cells and global lay-out of the mesh to be ready for the calculations. After that, following the model's detailed setup as already explained in the previous sections, a double-step protocol was started up: firstly, to stablish the best option for the turbulence model, and secondly, to run the model for the different geometrical arrangements. For the last one, two different kinds of results will be shown both for the contraction and the expansion geometries, namely the global results and the study of flow fields, basically the velocity and pressure fields. The studied geometries are considered in Table 5.

Contraction (water)	D_2/D_1	0.1	0.3	0.5	0.7	0.9	1.0
Contraction (air)	D_2/D_1		0.32	0.55	0.71	0.84	0.95
Expansion	D_{1}/D_{2}	0.1	0.3	0.5	0.7	0.9	1.0

Table 5. Considered geometries in the present study.

The total length of the model is fixed to 3.5 m, leaving the quotient of the length over diameter at least 60 times higher from the contraction/expansion to the beginning/end of the domain (according to Figure 5, for the expansion geometries, the value was $\frac{L_2}{D_1} = 68$, and for the contraction geometries, the values becomes $\frac{L_1}{D_1} = 64$). In such a way, the effect of the boundary conditions it is considered to be far enough from the core region, where the change in the cross-section is produced.

4. Numerical Model Results

The analysis of the results begins by carefully studying the speed and pressure profiles for each of the turbulence models. After a global frame is stablished, the local velocity profiles are obtained and conclusions on the turbulent model are drawn. Then, a deep look at the velocity fields and streamlines is proposed. Finally, pressure losses are obtained, and core results of the article lead to final conclusions. All results are shown for the finest mesh.

4.1. Pressure and Velocity Fields

From the numerical results, a wide range of data became available, and the postprocessing is advanced towards a comprehensive data generalization. As a starting point, the pressure and velocity fields are observed, and a global frame is stablished. For instance, Figure 8 shows the pressure field and velocity values for a contraction with $D_2 = 0.5D_1$. The first ones are plotted in the lower part of the figure, within a range of $[0, 1.13 \times 10^5 \text{ Pa}]$. The expected pressure drop is numerically obtained.



Figure 8. Pressure (lower part) and velocity (upper part) distributions for $D_2 = 0.5D_1$.

Also in Figure 8, in the upper part of the figure, the velocity values on top of the streamlines are shown for a particular contraction case. The velocity ranges for this particular case were (according to initial numerical trials) within the limits [0, 13.5 m/s]. Those ranges fitted for most of the studied cases and a proper analysis was carried out to determine the validity of the results considering the global expected trends for the velocity and pressure distributions. Based on a deeper analysis of the obtained results, velocity profiles at three different sections will be shown in what follows. The previous split on the different available numerical values will be kept, showing first the results for the con-

traction and then for the pipe expansion. A non-dimensional representation is chosen in the local velocity analysis to better analyze the obtained results. To account for such idea, non-dimensional velocity is plotted vs. a normalized position from the pipe wall.

Regarding the contraction, the comparison between the different velocity profiles at some points of the pipe for the models used in the simulation is shown in Figure 9, retrieving the expected turbulent-like profiles for the case $D_2 = 0.5D_1$. A fairly good agreement with the theoretical profiles was observed, and in the detailed comparison, almost equal values for the three different models was obtained. Almost identical behavior was observed for the x = 1 m location (before contraction), while more relative differences were found for the x = 1.7 m and x = 3.5 m (domain exit). For those two locations, the κ - ω model is the one with higher losses, with higher velocity gradient in the near-wall region. However, the three models provide similar values in all locations.



Figure 9. Sudden contraction ($D_2 = 0.5D_1$) velocity profiles for the three turbulence models at different locations, namely x = 1 m, x = 1.7 m, and x = 3.5 m.

From the values in Figure 9 and similar ones for the other studied contractions, the κ - ϵ model was found to be a good approach and was chosen for the final core results comparison. On the other hand, as the differences encountered were not so big, the final model selection was not so critical. It was then used for the subsequent data reduction, together with the finest grid, as explained in the previous part.

A plot with the velocity fields is then proposed in Figure 10. The field is made nondimensional using the inlet velocity, and then five different contractions are studied (D/D_1) . Similarly to what was proposed in Figure 8, the lower part of each figure is used to plot this non-dimensional field, while the upper part of each figure is used to plot the streamlines.

As expected, the velocity increase is higher for higher contractions, reaching its maximum at $D_2 = 0.5D_1$. Both patterns, streamlines and velocities, are also in agreement with the trends typically expected in the defined problem. That is, from the literature review and the flow behavior in the different cases, it can be concluded that (at least) the obtained results would be considered as feasible.

In Figure 10, for the cases with higher cross-section change, there is an upstream effect with high velocities even for x positions before the section change is produced. For the lower cross-section changes (cases with $D_2 \le 0.5D_1$), the geometry effect on the flow velocities resembles a practically symmetric potential source with a monopole vortex (cases with $D_2 \ge 0.5D_1$). On the other hand, the low-speed corner, just before the contraction, becomes lower with decreasing contraction degree, being minimum for the value at $D_2 = 0.9D_1$. The minor losses of the element are then a straightforward effect of the shown velocity changes and, as a matter of fact, the higher the decrease in the cross-section is, the higher the velocity gradients retrieved are, and therefore the higher the losses will be.



Figure 10. Streamlines and velocity values for the contraction (κ - ε model).

In what refers to expansion, a comparison between the different velocity profiles at three points of the pipe for the used turbulence modeling in the simulation are provided; the results, again on a non-dimensional basis, are shown in Figure 11. Parallelly to the studied contraction, the analysis of the results of the expansion process begins by reviewing the speed and pressure profiles for each of the turbulence models.



Figure 11. Sudden contraction ($D_2 = 0.5D_1$) velocity profiles for the three turbulence models at different locations, namely x = 1 m, x = 1.7 m and x = 3.5 m.

In Figure 11, the κ - ε model provides the lowest friction losses in the near-wall region for the $D_2 = 1.5D_1$ at x = 1 m. Then, at x = 1.7 (just after the expansion), both κ - ε and κ - ω provide almost the same velocity profile, which includes some negative values as the recirculation bubble is placed in that section. In section x = 1.7, only the κ - ω and the Spalart–Allmaras (S-A) models are shown as the κ - ε model provides very similar values to the κ - ω one. Finally, for the x = 3.5, all three models practically collapse into only one curve. After a full study of the different cases for the two possible changes in section (namely Figures 9 and 11 and equivalent ones for other diameter ratios), the κ - ε model was considered for further analysis.

A non-dimensional plot of the velocities is presented in Figure 12, using as a reference the outlet velocity for the different expansions. Again, as in Figure 10, the upper part of the plots is used to plot the streamlines and the lower part shows the non-dimensional velocity distributions, namely the values V/V_{exit} .



Figure 12. Streamlines and velocity values for the expansion (κ - ε model).

Almost all cases (except the one with lower section change, that is, the one with $D_2 = 1.1D_1$) show a big recirculation bubble, this being the main part of a pair of contrarotating vortices. Also, the flow field is divided into two main zones: the core or jet region and the low-speed corner, with the recirculation bubble. For the lower cross-section changes (cases with $D_2 \le 1.5D_1$), there is low interaction between the two zones, while for cases with $D_2 \ge 1.5D_1$, the interaction is higher. Again, as expected, the higher D_2/D_1 , the higher the losses.

The re-attachment position, not shown in Figure 12, follows the expected trend [26]. It is not shown for the sake of uniformity in the representation, to set the focus on the expansion zone.

4.2. Pressure Losses for the Abrupt Section Change

From the already discussed derivations, the best parameter to compare the different values would be the minor loss coefficient, defined as follows both for the pipe contraction and expansion:

$$\xi = \frac{h_{\text{loss}}}{\left(\frac{1}{2}\frac{V_2^2}{g}\right)} \tag{19}$$

And then, from the developed numerical models for the contraction and the expansion geometries, a possible comparison of the numerically obtained results with the existing bibliography becomes available. The only considered issue is that both conditions differ in the value of the higher diameter: in the contraction, the reference value is the initial one and, for the expansion, the higher one is the final diameter. Bearing that in mind, and using Equation (19), two different figures are plotted: Figure 13 for the pipe contraction and Figure 14 for the pipe expansion. The minor loss coefficient is plotted versus the change in diameter. Such a representation is considered to be the most relevant output from the performed study and resembles the values previously considered in the bibliographical review (see Figure 3, where the variable used was the cross-section ratio, namely the parameter $R = \beta^2$).



Figure 13. Comparison of the minor loss coefficient vs. $\beta = D_2/D_1$, for a sudden contraction [2,9,17].



Figure 14. Comparison of the minor loss coefficient vs. D_1/D_2 , for a sudden expansion [8,29,30].

Figure 13 plots, on one hand, some of the existing published results from [2,9] and [17]. Then, the values developed in Equation (17) with the simple 1d theory are plotted. From the numerical calculation, the deeper analyzed values were the ones using the κ - ε modeling, and two different curves are shown, i.e., one for water as fluid and another one for air, in order to check possible density influence on the results. Finally, a curve using the RSM is also plotted. Those last three curves are named as the "Present model", with the different particular conditions.

Although several other values are available in the literature for such problems, three have been selected for comparison, namely the values found in [2] and the derived and proposed values by [9] and the more recent [17]. The results from the developed model are plotted considering water and air as fluids. The water model seems to follow the expected trend, but only catches the actual values in a range for the diameter ratio [0.3, 0.7]; for lower values, it overpredicts the losses and, what is more critical, for higher values, it underpredicts the losses. On the other hand, the air model follows quite closely the values in [9] and reaches quite good results at high diameter ratios (close to 1). Globally speaking, both models show a good trend in the loss prediction, and it seems that the water model better finds the experimental values for the range [0.3, 0.6], while it finds a drop in the losses that is not so coherent for diameter ratios higher than 0.7.

The simple theoretical equation derived in previous sections (Equation (17)) is also plotted, and although it follows the global trend, the error is too high to be considered as a design tool. In other words, up to a given global view, it somehow follows the expected

trend, but it does not catch the actual values at all, providing a loss coefficient almost double in comparison with any of the other previously plotted and commented values (loss overprediction, particularly at low values of β).

For high contractions ($\beta < 0.5$), the results from RSMs replicate the behavior observed previously with the κ - ϵ models. In addition, at contractions below $\beta < 0.3$, there is a significant overprediction of all the numerical models tested with respect to the experimental observations. On the other hand, when the contraction is moderate to weak ($\beta > 0.5$), RSMs estimate a higher value of the loss coefficient, very similar to the results of Benedict et al. [2]. Moreover, turbulence closure with κ - ϵ models clearly underpredict the loss coefficient if $\beta > 0.5$. In those situations, with such slight diameter reductions, it is presumable that Reynolds stress models (RSMs) can capture mild anisotropic effects better than two-equation models, as it happens in the presented results.

Similarly, in Figure 14, the comparison of the loss coefficient for different bibliographical values and the present model is performed. The coordinate system is equivalent to Figure 13, and the minor loss coefficient is plotted versus the quotient D_1/D_2 .

In Figure 14, the values provided by [8,29,30] are plotted together with the prediction by [30]. In fact, the last one is the closest to the present model's results. Nevertheless, it seems that the model overpredicts the losses for the whole range of the diameter ratio. Only for the lower values, in the range [0.1, 0.4], can it be considered that there is a kind of agreement between the model values and the ones obtained in the equation by [30].

It seems that the location of the reattachment point does affect the reliability of the numerical model results. It is likely not even the flow solution, which the authors consider to be up-to-date as regards the present technology, but the way the data reduction is performed and the right calculation of such a point that may include some numerical additional uncertainties.

Globally speaking, the comparisons in Figures 13 and 14 show how the 2D model can capture the physics behind the flow behaviors in a sudden pipe contraction, while the same model for a sudden expansion finds more difficulties and fails to accurately describe the losses in the whole D_1/D_2 range, clearly overpredicting the losses.

5. Conclusions

The numerical modeling of a sudden or abrupt change in a pipe section presents a widespread application for pipe flow. However, a quick but accurate numerical procedure for the calculation of such losses for different practical situations is not so common in the bibliography. In the present article, minor losses are considered for the sudden change in a section of a pipe, both for the contraction and for the expansion geometries. An initial 1D model is developed and its limits are clearly shown.

The developed numerical 2D axisymmetric model yields a very accurate description of both practical situations, which can be very useful to understand the physical mechanisms governing the losses in such classical Fluid Mechanics problems. Those minor losses referred to have been widely studied on an experimental and theoretical basis, but not so much using a numerical approach. Within the proposed methodology, a numerical calculation of the losses for the two possible pipe section changes has been shown.

The numerical model has proven its robustness in what refers to mesh independency on the final results. For the obtained sensitivity of the developed model, even a lower number of cell meshes could be enough to reach the same values. A wide study has been carried out on such mesh independency, with several tests to prove the conclusion.

A wide set of both velocity and pressure maps have been obtained for the two practical situations, revealing the influence of the turbulence model being used. Three different models have been tested, namely the Spalart–Allmaras, κ - ε , and κ - ω (SST) models, which were used alongside the calculations. Although slight changes in the near-wall velocity profiles were found, when calculating the head losses and loss coefficient, almost equal values were retrieved for the three turbulent models, with less than 0.5% variations. A final check with an RSM was also conducted for the contraction flow condition, and the results

show the consistency of the developed models, with an improvement in the prediction for the RSM in the range $\beta = [0.5, 1]$, for which the two-equation models underpredict the losses.

The most relevant finding in this study is the evolution of the loss coefficient versus the diameter quotient for the different geometries. Regarding the pipe contraction geometry, a double simulation was performed using air and water as fluids. The results for both show quite good agreement with the existing data. A third approach in terms of a theoretical development was introduced, although that final value does not fit the experimental and numerical results.

For the pipe expansion, the question regarding the reattachment point has been a cornerstone throughout the different studies and becomes a critical issue in any possible approach to the problem. The comparison in this problem was not so promising as that for the previous one. As can be observed, quite different values and trends were obtained and reported in the existing values. However, the presented 2D model somehow overpredicts the losses.

The global picture of the head losses for a pipe section change has been numerically studied, and particular values for loss coefficient predictions are available. A comparison with the existing bibliographical experimental values has also been tackled. The actual comparison shows good agreement for the contraction and is fairly different to experimental values for the expansion.

Author Contributions: Conceptualization, J.G. and J.M.F.O.; methodology, J.M.F.O.; software, A.M.-F. and I.V.P.; validation, I.V.P. and A.M.-F.; formal analysis, J.M.F.O.; resources, J.G.; data curation, I.V.P. and A.M.-F.; writing—original draft preparation, J.G. and A.M.-F.; writing—review and editing, J.M.F.O. and I.V.P.; visualization, I.V.P.; supervision, J.G.; project administration, J.G. and J.M.F.O.; funding acquisition, J.M.F.O. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: Data will be shared under requirement.

Acknowledgments: A grateful acknowledgement should be given to the financial support from Project ENE2017-89965-P provided by the Spanish Ministry of Economy, Industry and Competitiveness. It has also important contribution the economic support of the "Agencia Estatal de Investigación" (AEI) of the Spanish Ministry of Science and Innovation, in the context of the State Program to Promote Scientific-Technical Research and its Transfer, through the Project "Optimization through flow control techniques of a vertical axis wind turbine for urban environments" (ref. TED2021-131307B-I00), included in the NextGenerationEU funds of the European Community. Some additional contributions of the University Institute for Industrial Technology of Asturias (IUTA), through projects SV-19-GIJON-1-15, SV-20-GIJON-1-17 and SV-22-GIJON-1-02, have allowed the development of this study and are also appreciated.

Conflicts of Interest: The authors declare no conflicts of interest.

Nomenclature

A_1	Inlet section contraction/outlet expansion, (m ²).
A_2	Outlet section contraction/inlet expansion, (m ²).
Ca	Experimental factor, (), from [7].
Cp	Pressure coefficient, ().
Ċ.V.	Control volume, as often used in Fluid Mechanics.
D_1	Inlet diameter, (m).
D_2	Outlet diameter, (m).
f	Friction coefficient, ().
h _{exp}	Experimental head loss, (m).
$h_{\rm loss}$	Head loss when changing section, (m).
K _{cont}	Head loss coefficient, ().

Kacc	Head loss coefficient due to flow acceleration, ().
L_1	Length inlet contraction/outlet expansion, (m).
L ₂	Length output contraction/inlet expansion, (m).
Μ	Molecular weight, (g/mol).
Р	Pressure, (Pa).
P_0	Pressure in the contraction section, (Pa).
P_{1}, P_{2}	Pressure at the inlet/outlet sections, (Pa).
Q	Flow rate, (m^3/s) .
R	Cross-section quotient, $R = A_2/A_1 = \beta^2$, ().
Re	Reynolds number, ().
RSM	Reynolds stress model.
/	Fluctuating velocity field, (m/s).
S-A	Spalart–Allmaras model.
V_0, V_{exit}	Inlet velocity inlet (contraction), exit velocity (expansion), both in (m/s).
V_1	Inlet velocity, contraction/outlet velocity, expansion (m/s).
V_2	Outlet velocity, contraction/inlet velocity, expansion (m/s).
μ	Fluid viscosity, (kg/(m s)).
х, у	Coordinates, axial and radial one, (m).
y+	Wall distance, ().
t	Time, (s)
g	Gravity acceleration constant, (m/s^2) .
β	Diameter ratio, D_2/D_1 , ().
ε	Specific dissipation rate, $(1/s)$.
ξ	Pressure loss coefficient, ().
ξ _{cont}	Pressure loss coefficient for a contraction, ().
к	Turbulent kinetic energy, $(J/kg = m^2/s^2)$.
λ	Flow speed ratio, Equation (10), ().
ρ	Fluid density, (kg/m ³).
τ _{i,j}	Stress tensor elements, (Pa).
ω	Specific viscous dissipation rate or specific vorticity fluctuations, (s/m^2) .

References

- 1. Rennels, D.C.; Hudson, H.M. *Pipe Flow: A Practical and Comprehensive Guide*; John Wiley & Sons: New York, NY, USA, 2012; Chapter 10: Contractions.
- Benedict, R.P.; Carlucci, N.A.; Swetz, S.D. Flow losses in abrupt enlargements and contractions. *Trans. ASME J. Eng. For. Power* 1966, *88*, 73–81. [CrossRef]
- 3. Weisbach, J. Mechanics of Engineering; (Trans. Coxe, E.B.); van Nostrand Book Co.: New York, NY, USA, 1992.
- 4. Polkowski, H. Effect of Sudden Expansion or Contraction in a Pipe upon the Flow of Water. Bachelor's Thesis, University of Illinois, Champaign, IL, USA, 1912.
- 5. Archer, W.H. Loss of head due to enlargements in pipes. *Trans. ASCE* **1913**, *76*, 999–1026.
- 6. Kays, W.M. Loss coefficient for abrupt changes in flow cross section cross section with low Reynolds number flow in single and multiple tube systems. *Trans. ASME* **1950**, *72*, 1067–1074. [CrossRef]
- 7. Vennard, J.K.; Street, R.L. Elementary Fluid Mechanics, 4th ed.; John Wiley & Sons: New York, NY, USA, 1961.
- 8. Brater, E.F.; King, H.W.; Lindell, J.E.; Wei, C.Y. Handbook of Hydraulics; McGraw-Hill: New York, NY, USA, 1996.
- 9. Idelcik, I.E. Memento des Pertes de Charge; Meury, M., Translator; Eyrolles: Paris, France, 1969. (In French)
- 10. Martin, J.J. Expansion and contraction losses in fluid flow: An example of engineering analysis. Chem. Eng. Educ. 1972, 8, 138–140.
- 11. Bullen, P.R.; Cheeseman, D.J.; Hussain, L.A. A study of turbulent flow in pipe contractions. *Proc. IMEchE Part E J. Process Mech. Eng.* **1996**, *210*, 171–180. [CrossRef]
- 12. Bullen, P.R.; Cheeseman, D.J.; Hussain, L.A.; Ruffell, A.E. The determination of pipe contraction pressure loss coefficients for incompressible turbulent flow. *Int. J. Heat Fluid Flow* **1987**, *8*, 111–118. [CrossRef]
- 13. Schmidt, J.; Friedel, L. Two-phase pressure drop across sudden contractions in duct areas. *Int. J. Multiph. Flow* **1997**, 23, 283–299. [CrossRef]
- 14. Fox, R.W.; Pritchard, P.J.; McDonald, A.T. Introduction to Fluid Mechanics; John Wiley & Sons: Hoboken, NJ, USA, 2006.
- 15. Munson, B.R.; Young, D.F.; Okiishi, T.H. Fundamentals of Fluid Mechanics; John Wiley & Sons: New York, NJ, USA, 1998.
- 16. Barrero, A.; Pérez-Saborid, M. *Fundamentos y Aplicaciones de la Mecánica de Fluidos;* Mc. Graw-Hill Iberoamericana: Madrid, Spain, 2005. (In Spanish)
- 17. White, F.M. Fluid Mechanics; Mc Graw-Hill: New York, NJ, USA, 2006.

- 18. Wikipedia. Article on the Borda-Carnot Equation. Available online: https://en.wikipedia.org/wiki/Borda%E2%80%93Carnot_equation (accessed on 20 December 2020).
- Darihaki, F.; Zhang, J.; Shirazi, S.A. Application of Scale-Resolving Simulations and Hybrid Models for Contraction-Expansion Pipe Flows. In Proceedings of the ASME 2021 Fluids Engineering Division Summer Meeting (FEDSM2021), Virtual, 10–12 August 2021; Volume 1, p. V001T02A026.
- 20. Ye, J.; Cheng, Y.; Xie, J.; Huang, X.; Zhang, Y.; Hu, S.; Salem, S.; Wu, J. Effects of divergent angel on the flow behaviors in low-speed wind accelerating ducts. *Renew. Energy* **2020**, *152*, 1292–1301. [CrossRef]
- 21. Kaaushik, V.V.R.; Ghosh, S.; Das, G.; Das, P.K. CFD modelling of water flow through sudden contraction and expansion in a horizontal pipe. *Chem. Eng. Educ.* **2011**, *45*, 30–36.
- 22. Rashid, F.L.; Alhabeeb, B.A.; Alhwayzee, M.; Mohammed, A.A. Study the sudden expansion and contraction in the pipeline on the distribution of pressure at the presence of a porous media by using CFD simulation. *J. Eng. Sci. Technol.* **2021**, *16*, 3960–3973.
- 23. Wikipedia. Article on the "Vena Contracta" (Basic Idea on the Phenomena and Implications). Available online: https://en. wikipedia.org/wiki/Vena_contracta (accessed on 3 March 2024).
- 24. Leschziner, M.A. *Statistical Turbulence Modelling for the Computation of Physically Complex Flows;* Lecture Series; von Karman Institute: Sint-Genesius-Rode, Belgium, 2001.
- Davidson, L. Fluid Mechanics, Turbulent Flow and Turbulence Modelling. Master's Thesis, Chalmers University, Göteborg, Sweden, 2017; pp. 1–363.
- 26. Chieng, C.C.; Launder, B.E. On the calculation of turbulent transport downstream from an abrupt pipe expansion. *Num. Heat Transf.* **1980**, *3*, 189–207.
- Menter, F.R.; Lechner, R.; Matyushenco, A. Best Practice: RANS Turbulence Modeling in Ansys CFD (Version 1.0). ANSYS-Fluent Tutorial. Available online: https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node78.htm (accessed on 3 March 2024).
- Gibson, M.M.; Launder, B.E. Ground effects on Pressure fluctuations in the atmospheric boundary layer. J. Fluid Mech. 1978, 86, 491–511. [CrossRef]
- 29. Miller, D.S. Internal Flow Systems; BHRA Information Services: Bedford, UK, 1978.
- 30. Cengel, Y.A.; Cimbala, J.M. Fluid Mechanics: Fundamentals and Applications; Mc. Graw-Hill: New York, NJ, USA, 2004.

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.