A brief review study of flow phenomena over a backward-facing step and its optimization

Elham Montazer, Hooman Yarmand, Erfan Salami, Mohd Ridha Mohamad, S.N. Kazi, A. Badarudin

1. Introduction

Liquid flow via expansion geometries is additionally related to some practical engineering applications such as processing of food stuff, extrusion processes, mold filling, ceramics and pharmaceutical matters are among others. For an extensive period, research on the flow of channels with reversals have transformed from symmetry to asymmetry which has received great enthusiasm. To aid further numerical and experimental research on such flows, the flow through an abrupt enlargement has been regarded as a benchmark since it comprises a layout possessing among the most primary geometries. However, this simplicity of its geometry is not necessarily indicating that the flow concept is simple as well. Furthermore, reattachment, flow separation and multiple re-circulating zones of fluid flow are those rich devices in which the majority of scientists are interested in. Use of sudden expansion configurations is quickly expanding. The volume of existing literature which has received great enthusiasm is to provide a summary of the recent research progress on the flow specifications of fluid flowing via abrupt enlargement configurations in order to suggest several potential explanations about implementation of these configurations in industrial applications.

2. Plain sudden expansion

2.1. Turbulence

The main numerical work within this area of plain abrupt enlargement was conducted by Abbot and Kline [1], who experimentally studied the subsonic turbulent flow over both single and double backward-facing steps. Fig. 1 presents a diagram of the double-step expansion that they investigated. An investigation was performed on separated regions of subsonic and turbulent fluid flow at the downstream of 2D systems, backward-facing steps for the Re ranges from a value of $2 \times 10^4$ to $5 \times 10^4$.

It was concluded that three zones of flow exist within the turbulent separation flow regions: (1) A 3D zone exists directly downstream of step face, and possesses possibly multiple vortexes that rotate around an axis normal to the direction of through flow; (2) a 2D zone downstream...
of the previous zone, which comprises classic stall patterns of flow moving upstream across the wall, and downstream adjacent to the through flow; and (3) a time-dependent tail zone that momentarily varies in size. They discovered three varying flow zones in the turbulent divided region. The first zone is 3D, directly downstream of the step face, and is generally characterized multiple vortices that rotate around an axis that is normal to the direction of flow.

The second flow zone is 2D, downstream of the previous zone, while the third and final flow zone is time-dependent, and changes its size momentarily. Additionally, they mentioned that the three zones have varying lengths for ratios of area over 1.5 for double-step configuration that shows the existence of flow asymmetry. However, for expansion ratios lower than 1.5, the double step approach is used for single step configurations with symmetrical stall areas. According to Fig. 2, the Reynolds number and the turbulence intensity values ($\frac{u''}{\mu}$) had a significant impact on flow patterns or reattachment lengths.

Khezzar et al. [2] experimentally carried out quantitative analyses of combusting and isothermal flows at the downstream of plain abrupt enlargement. They considered an expansion ratio of 2.86, and a Re of 20000. The evaluated equivalent ratios for the rough and smooth combustion values were 0.92 and 0.72 respectively. The final result demonstrated that the degree of asymmetry position of the isothermal flows was decreased by coupling the pressure among the two areas of recirculation.

Mizushima and Shiotani [3] in their work on structural instabilities of bifurcation graph discovered that flows in a symmetrical channel with a sudden expansion have a transition from a symmetrical flow, to an asymmetrical flow, because of a symmetry-breaking pitchfork bifurcation on raising the Re in the case the system was precisely symmetrical. An inconsiderable imperfection in the model can cause the pitchfork bifurcation to become imperfect. The research developed non-linear stability analyses on a weekly basis to study the structural instability in the bifurcation of these flows. The researchers came up with an amplitude equation for a disturbance by considering the impact of the imperfection of the models. They stressed that this considered weekly non-linear stability theory demonstrated the important and skeleton dynamics of the bifurcation concept, which made the examination of the associated physics of these instabilities possible. In this work, the amplitude equation was used to evaluate the equilibrium amplitude value of the disturbance and then it was compared with the two experimental outcomes yielded in the earlier cases, as well as with the computational solution of the entire non-linear flow equations in mildly asymmetric channels. Comparing the result of the weekly non-linear stability analyses process, to the numerical and experimental results, made it possible to suggest that the parameter range by which the weekly non-linear stability theory provides an accurate approximation limited to the vicinity of the critical point.

Pressure drop and heat transfer for turbulent airflow within an abrupt enlargement pipe which is equipped with spiral spring item or a propeller swirl generator with multiple pitch ratios were studied by Zohir et al. [4]. They conducted the tests at the Re ranges of 7500–18500, and at uniform heat flux conditions. The experimental tests were carried out in three independent pitch ratios for the spiral spring (P/D = 10, 15, 20) and in three zones for a propeller fan (n = 15 blades; blade angle = 650°). The effect of the utilization of the freely rotating propeller was discussed, as the input spiral spring on heat transfer improvement and pressure drop. In these trials, the spiral spring and swirl generator were utilized to produce a swirl within the tube flow. They considered both mean Nusselt numbers and relative mean pressure drop and compared them with those acquired from various analogous scenarios. The outcomes from experimentation suggest that the tubes with the propeller at the entrance have shown a great development of heat transfer ratio compared to the plain tube, about 1.69 times for X/H = 5. Regarding the tubes with spiral springs, the heat transfer rate compared to the plain tubes was about 1.37 times for the case of P/d = 20. The higher in pressure with utilization of the propeller was shown to be 3 times, and for the case of the spiral spring it was 1.5 times compared to the plain tube. The researchers had developed correlations for the average Nusselt number, spiral spring pitch and the fan location.

Zohir and Gomaa [5] considered heat transferring and pressure drop properties for turbulent air-flow within an abrupt expanded channel (d/D = 0.72) that had a propeller swirl generator with varying blade angles. The diagram of the considered test section is presented in Fig. 3. The researchers had examined the effect of Re, in the range of 10000–40000 at uniform heat flux conditions. In these tests three different propeller fans with five blades and swirl angles of $\theta = 15^\circ, 30^\circ, 45^\circ$ for the upstream flow; and one propeller fan with swirl angle of $45^\circ$ for the downstream flow were added separately to the test section. The swirl propeller fan was positioned at various locations inside the test pipe, S = 10 H, 20 H and 40 H for the upstream values of the tube were maintained for the higher rate of heat transfer (up to 190%) for all of the swirl angles with the highest values were at $\theta = 45^\circ$. They concluded that addition of the propeller at the downstream of the tube allows the further enhancement in the rate of heat transfer, and putting the propeller at the upstream of the tube also allows further improvement. Moreover, in the case of the propeller at the upstream of the tube at a swirl angle of $\theta = 45^\circ$, the rate of heat transfer was increased to 225%. The optimal value of the enhancement factor for the downstream swirl was approximately 326%, while this value was approximately 213% for its upstream counterpart. Fig. 4 displays correlations for the relative mean Nusselt number and the enhancement factors for the various fan positions, swirl angles and the Re.

2.2. Laminar flow

Macagno and Hung [6] examined flow embodiment within a sudden expansion experimental setup for the axisymmetric flows at the Re ranges of 36–4500, using computational simulation. The aspect ratio
considered was $2$; and $\text{Re}$ reached a peak of $200$. They determined that, for the laminar flow, the primary task of the eddy is to shape the flow with a quite low energy exchange.

Durst et al. [7] studied the Newtonian fluid flow within a $1:3$ planar symmetrical enlargement by using laser Doppler anemometry (LDA). After several trials, two symmetrical vortices across the walls of enlargement were recorded at $\text{Re} = 56$. In the case of $\text{Re} = 114$, flow bifurcation was directly seen with vortices of disparate sizes in both of the prominent corners.

The empirical estimations of Chedron et al. [8] depend on LDA as well. But they were far more generalized, and they also recorded flow patterns and irregularities in channels with the symmetrical enlargements. The impact of the aspect ratio of the tested geometry was studied as well. Their investigation for conduits with moderate enlargement values showed that, in the case of lower $\text{Re}$ values, the channel flow remained steady, $2D$ and symmetric with two separation locations in close proximity of the enlargement corners, the overall size of which is improved with the increase in $\text{Re}$. On the other hand, for greater estimation of $\text{Re}$, the overall flow generally remains steady, and $2D$, nevertheless is also turns into asymmetric, with two different separation areas of varying lengths that can be linked to the lower or upper walls of the corresponding conduit. At a greater $\text{Re}$, further recirculation areas are seen across the walls of the channel.

Armaly et al. [9] used LDA to search the reattachment lengths and the velocity distribution at the downstream of a single backward-facing step within a $2D$ conduit. In their work laminar, the transitional and the turbulent air flow at the $\text{Re}$ range of $70–8000$ were taken into account. They recorded that the several flow phenomena were controlled by common variations of the separation lengths with $\text{Re}$. The results suggest that regardless of the manner in which the inlet flow was controlled by common variations of the separation lengths with $\text{Re}$. The results suggest that regardless of the manner in which the inlet flow was controlled by common variations of the separation lengths with $\text{Re}$. The results suggest that regardless of the manner in which the inlet flow was.

Fig. 4. Local Nusselt number variation with dimensionless tube length values for a propeller type swirl generator located at (a) $X/H=1$, (b) $X/H=5$, (c) $X/H=10$ in the plane pipe at various Reynolds numbers [5].

Fig. 3. The test section detail [5], 1- Teflon Piston, 2- Test Section, 3- Propeller Fan, 4- Electric Heater, 5- Spring, 6- Insulation, 7- Flange.
recirculation eddy strength, however, was nonlinear, and dependent on Re. In addition, they mentioned that the recirculation eddy strength becomes weak with the increase in the Reynolds value.

Hawa and Ruzak [14] conducted bifurcation analyses, linear stability examination and direct numerical simulations with the dynamics of a 2D, incompressible laminar flow in a symmetric long conduit with an abrupt enlargement configuration. The bifurcation analyses of solutions of a set of steady Navier Stokes equations focused on the equilibrium conditions around the critical Re, in which the asymmetric states were simultaneously prevailing. The stability analyses performed were based on the underlying liberalized motion equations for the development of irrelevant 2D disorders enacted on both the steady asymmetric and symmetric conditions. The simulations suggested the relation among the linear stability result and the asymptotic behaviour time of the flow, as mentioned by the asymptotic steady-state solutions. Fig. 6 shows that the symmetric flows with Re < Re_c are linearly stable in 2D disturbances, while the symmetric conditions with Re > Re_c remain unstable. The dynamics of large and small domain disturbances within the flow are defined, as well as the transformation from symmetric to asymmetric flow is clearly presented. Therefore, the researcher mentions that the critical state is an exchange point of stability for the asymmetric and the symmetric conditions. The overall image of the present research is; in the case of sufficiently lower Re value, the flow is unable to successfully sustain the disturbance or any primary disturbance which diminishes via viscous dissipation. With an increase in the Re value, the viscous dissipation was decreased, and the symmetric flow was altered to stability.

In numerous real-world scenarios, the fluid flowing via flow devices are non-Newtonian, and thus it displayed behaviour as per the complex rheological character. More precisely, they show shear-thinning viscosity, based on the kind of fluid, and it was thus related to examine the non-Newtonian fluid flow in planar enlargements, starting with simplified rheological systems, to individually examine the effect of particular rheological particularities on the flow properties. The non-Newtonian solutions are not very concentrated, and the flows seem to possess a high Re, which leads to turbulent flow. The backward-facing step is a well-defined configuration for work on laminar flow instabilities at high Re. In the past few years, this has attracted scientists in the area of non-Newtonian fluid mechanics who desire to investigate the complicated interaction among these bifurcations and fluid rheology, more specifically, viscoelasticity. The non-Newtonian power – law model is the most simplified one for an entirely viscous fluid that is able to reflect the behaviour of Newtonian fluids and shear-thickening by changing the model parameter, n, which is the power law index.

2.2.1. Flow through planner sudden expansion

Kadja et al. [15] numerically carried out an examination of bifurcation phenomena that occurs in flows via planar abrupt enlargements. A novel convection arrangement; the Variable-Order Non-Oscillatory Scheme (or VONOS), with a multiple grid algorithm, was employed for a detailed examination of bifurcation phenomena occurring in flows via planar abrupt expansion. This new arrangement attributes to a great rise of convergence rate and accuracy. These outlooks allow for a suitable qualitative behaviour of the parameters of flow bifurcation. Fig. 7 presents the Reynolds numbers under Re_c (Re_c = 200); where the flow generally remains symmetric in the entire development period. During this time, the recirculation lengths steadily increases in the direction of their stationary values. However, at the Reynolds numbers over Re_c, the flow was transformed from a symmetric layout to an asymmetric...
layout in the steady state. One of the primary recirculation area grow into longer compared to the other, and a third area was seen as the short primary recirculation area on the same side.

Mishra and Jayaraman [16] experimentally studied the asymmetric steady flow pattern of the shear thinning fluid via planar abrupt expansion with a large area ratio (namely, ER = 16). Manica and De Bortoli [17] also investigated the power-law fluids flow within a 1:3 planar abrupt enlargement, with the different values of: \( n = 0.5, 1 \) and 1.5. They listed the observed vortex properties for the values of Re and \( n \) was maintained in the bounds of 30 and 125. They concluded that the bifurcation flow for the shear thinning fluids takes place at the critical Re greater than the values for the Newtonian fluids, whereas the shear-thickening fluid shows the least critical Re.

Neofytou [18] considered the Casson and power law models and assessed the transformation of symmetric to an asymmetric flow of power-law fluids such as pure viscous fluids by using power-law indices within the bounds of 0.3 and 3 in 1:2 planar abrupt enlargement. Moreover, the impact of the Re on flow patterns was investigated.

Ternik et al. [19] investigated the flow via 1:3 planar symmetric expansions of non-Newtonian fluid with a shear-thickening function with the use of power-law and quadratic viscosity models. They provided a comparison of the outcomes of the two models, with the models of Newtonian fluids, and mentioned that the flow asymmetry was significantly affected by the shear thickening behaviour. Afterwards, Ternik calculated the shear-thinning fluids flow with power-law indices (\( n \)) of 0.6 and 0.8 within a 1:3 planar abrupt enlargement. With the further increase of the general Reynolds number there appears a subsequent flow bifurcation following the initial bifurcation of an asymmetric flow where the shear-thinning causes a delay of the onset of the subsequent bifurcation.

Ternik [20] also recently once again considered the general Newtonian flow within a 2D, 1:3 abrupt enlargement, configuration by using the FOAM CFD open source software application. The fluid was demonstrated once again by the popular power-law model with a power law index within the bounds of 0.6 and 1.4. A set of simulations were carried out for general Reynolds numbers within the bounds of \( 10^{-4} \) and 10, and with emphasizing the analyses of low Reynolds number flow, under the critical condition for the pitchfork bifurcation onset. A low recirculation, common of creeping flow was seen for all of the fluids with shear-thinning behaviour where decreasing the overall strength and size of the secondary flow. Several simulations of power-law fluids flow in a planar 1:3 sudden expansion with the use of open source programs were also investigated by several works. It was discovered that the convergence was frequently a great restriction during the utilization of these codes, especially in the case the non-Newtonian behaviour.

2.2.2. Flow through sudden expansion in diffuser

Chakrabarti et al. [21] extensively investigated the efficiency of sudden expansion from the diffuser perspective. They considered the general momentum and mass conservation formulas in the bounds of 20 and 100, and an aspect ratio of 1.5–4, and solved the 2D steady differential equations for the uniform velocity profile and the entirely developed velocity profile at the inlet, as well as for the varying inlet lengths. Moreover, they also investigated in detail the impact of every considered variable on the efficiency of the diffuser, as well as the stagnation pressure drop gradient.

2.3. Effect of geometrical parameters

So and Ahmed [22] studied the impact of geometrical parameters and turbulent flow on the efficiency of dump combustors. They investigated two varying expansion ratios and the impact on the flow behaviour of rotation. They discovered that when all of the inlet geometrical and flow parameters, with the exception of the step height, remain as it is then the net impact of the expansion ratio on the flow inside the combustor is generally too low.

2.4. Effect of viscous flow

Shapira et al. [23] investigated the existence and stability of several solutions for the viscous flow within a sudden expansion channel. They have also investigated the linear stability of a 2D flow via a symmetric conduit. They carried out numerical calculations of the flow field, as well as linear disturbances with the use of the time-dependent finite element technique. The considered conduit’s semi-angle was within the bounds of 100 and 900. Their work suggested steady non-symmetrical solutions for high Re compared to a particular critical value. The primary trait of their study lies in how the stability in the lowest stable mode was specified, with use of an energy based approach. In their study, there was neither prior proof of the plurality of stable and non-symmetric solutions nor any other limitation points on the non-symmetric divisions.

Foumeny et al. [24] carried out a numerical investigation on the bifurcations of incompressible Newtonian flow via planar symmetric conduit enlargements. Their investigation placed focus on the specification of the Reynolds critical value, over which the flow is asymmetric based on the conduit configuration. They investigated two models. Firstly, they involved a conduit model where the conduit walls downstream to the enlargement were constructed with no-slip conditions. The second was in the form of a cascade model, where the cyclic boundary conditions were determined. They evaluated the acquired outcome with the numerical and experimental data collected by prior investigations.

The transformation of flow patterns for the simultaneous flow of water and high viscous oil was investigated by Balakhrisna et al. [25] in a sudden expansion and contraction horizontal channel. In the following, they concluded that these abrupt variations within the cross-sectional area greatly impacted the lube oil-water flow’s downstream phase distribution. The observation indicated a rather simplified technique to produce a core flow, and a technique to prohibit pipe wall, and witnessed several key differences in flow of water and low viscous oil via similar experimental setup. Numerous kinds of core-annular flows were witnessed for the previous scenario, where a greater interfacial distribution variety characterizes systems of kerosene water. They also investigated the pressure profiles in the simultaneous flow of water and
lube oil via the abrupt expansion and contraction, and compared them with flow of water and low viscous oil. The pressure profiles were considered independent liquid viscosity, as well as the loss coefficients were witnessed as to be autonomous of flow patterns in both of the two cases.

### 2.5. Different numerical methods

Mandal et al. [26] carried out numerical analyses and efficiency simulation of an abrupt enlargement configuration with a fence observed as a diffuser. The computational domain is shown in Fig. 8. A simple algorithm was applied to address 2D steady differential equations for the momentum and mass conservation. The Reynolds number was in the bounds of 20 and 100, as well as a fence subtended angle (FSA) within the range of 10° to 30°. The fence's position from the throat varied from 0.2 to 2.6. An aspect ratio for all calculations was considered 2.

They investigated the impact of every factor on mean static pressure, overall distance of maximum static pressure gain, overall diffuser effectiveness and the average stagnation pressure. Moreover, they provided a comparison between these with regards to simple sudden expansion in absence of the fence. Figs. 9 and 10 show the effect of non-dimensional $L_f$ and FSA on mean static pressure. Computation reveals that for smaller Reynolds values the accuracy of possessing a fence was worthy, based on the location and subtended angle of the fence. At any position, the fence consistently presented the advantage at relatively higher Re for any subtended angle of the fence. They revealed that the position of the fence and the subtended angle had no considerable effect on the distance of the optimal static pressure gain from the throat, and average stagnation pressure drop at a certain Re.

Alleborn et al. [27] studied the 2D laminar flow of a generally viscous and incompressible fluid via a conduit with abrupt enlargement. They employed a continuance approach to investigate the bifurcation layout of the discretized governing formulas. An Arnoldi-based iterative approach was applied to determine the stability of various solution branches via computing the most unstable Eigen modes of liberalized equations for all the disturbance values. During this observation process more solution branches and bifurcation points were calculated. They investigated the functions of key eigenvalues within the neighbourhood of all these bifurcation points. In the present investigation the numerical outcomes for the limiting cases, for all the flow and geometrical factors, were evaluated with the existing analytical solutions. The investigation showed the existence of weak viscous eddies, referred to as “Moffat eddies”, within the expansion corners within the creeping flow limit. The numerically calculated stream procedure was compared for the corner vortices regions with an approximation equation obtained by Moffat, where a suitable agreement with previous results, was established.

Experimental and numerical investigation of a 2D backward-facing step flow was conducted by Lee and Mateescu [28]. They applied multiple element hot-film sensor arrays in order to determine the separation and reattachment lengths on the lower and upper walls at Re below 3000, as well as the expansion values of 1.17 and 2.0 respectively. The outcomes achieved from their tests demonstrate that, with the application of multiple hot-film sensors (MHFS), arrays functioned with a set of constant temperature anemometers, the positions of flow separation and reattachment points on the lower and upper walls of the 2D conduit may be computed, simultaneously and non-intrusively. They concluded that the outcomes yielded in using this approach were sufficiently precise within the outcomes estimated by the numerical analyses processes.

Poole and Ridley [29] applied the Fluent software in order to numerically compute the development-length needed to obtain fully developed laminar pipe flow for inelastic power-law fluids. The authors
were not able to obtain a converging solution for n values less than 0.4. Ternik [30] mentioned that the iterative convergence became more time-intensive with a decrease in the power-law indicator and for n values less than 0.6. There were no converged solutions yielded with use of Foam software.

Lima et al. [31] carried out several simulations of the 2D laminar flow for a backward-facing step channel by the use of two commercial computational fluid dynamics (CFD) codes. The authors assessed the three recirculation flow regions in a unilateral abrupt enlargement. The result for laminar air flow for the Re value was less than 2500. The numerical model equations (of momentum and mass conservation) were addressed with finite element (FEM), and finite volume (FVM) approaches as well as a segregated method. To enable the grid to become independent, numerous refinement works were conducted. The final result was compared with the experimental data, and obtained relatively good agreement.

Zdanski et al. [32] yielded results for incompressible, laminar, as well as non-isothermal polymer melt flow within abrupt enlargements. The energy, momentum, and mass conservation laws in the framework of a general Newtonian formulation were incorporated in the model. Two fundamental relationships were used to explain the non-Newtonian flow behaviour, more specifically, Modified Arrhenius Power-law and Cross models. The primary formulas were discretized by the finite difference approach based on central, second-order accurate equations for both the diffusive and convective terms. Velocity and pressure coupling was addressed by considering a pressure Poisson formula. They offered outcomes for two commercially available polymers, and showed the key flow parameters, including viscosity distribution and pressure drop, which were greatly impacted by heat transfer specifications. They finally determined that for the two polymers that were subject to similar boundary conditions, regardless of the flow topology resemblance, the variation in pressure calculated was changed because of the viscosity impact. In this range of issues, the viscosity was much depended to temperature. Consequently, this introduced a strongly coupled property to the issue. The parametric analyses exhibited the pressure distribution with the channel to greatly relate to the expansion ratio as well as the inlet polymer temperature. For a particular expansion ratio, the pressure coefficient was reduced in a linear fashion with the rise in inlet temperature. Regarding the considered conditions, the pressure drop along the channel was decreased by 40% with a decrease in the expansion ratio from 4.0 to 2.0.

Zhang et al. [33] showed a numerical study of the laminar forced convection of supercritical CO$_2$ flow within a horizontal channel, along with planar symmetrical abrupt enlargement, as well as its bifurcation phenomenon. The calculations were carried out using differing Reynolds values for cases of various wall heat fluxes. They presented a factor referred to as recirculation disturbance intensity, with the aim to draw conclusions on the decrease of flow stability in the case of an increase in the Re or wall heat flux. The Nusselt number’s distribution within the symmetrical flow pattern was shown; the varying peak for the Nusselt number and its relative location were investigated. The pressure coefficient distribution values and wall friction coefficient value are discussed. It was seen that, in contrast to the gas flow, the overall size of the recirculation area had enlarged with the wall heat flux increase, which is attributable to the thermally supported improvement of the reverse pressure difference. The optimal Nusselt value improved as the wall heat flux or Re was raised. With a raise in the Reynolds value, the place of the maximum Nusselt value was not consistently within the area of recirculation, and the location was shifted to the region of recirculation from downstream, and the shifts were moved upstream in the direction of the inlet. Increase in the wall heat flux, the overall movement of the location became irregular. With an increase in the temperature gradient at the wall for greater heating intensity, this irregular value can result from the steep transformation of thermo-physical characteristics close to the wall. The critical Re was reduced with an increase in the wall heat flux. The increase in the recirculation disturbance intensity makes the flow disturbance stronger, and therefore the flow become less stable; and the critical Re is then reduced.

Nassar et al. [34] applied a constitutive model that solves the problem with the elastic behaviour of any viscoelastic liquids. They applied the formula in the case of an expansion-contraction axisymmetric flow. Subsequently, a comparison of the result using experimental data in the related literature was presented. Steady, inertia less mathematical solutions were acquired for finding a solution to the balance formulas of momentum and mass through the finite element technique. Thus, for various combinations of the governing parameters they acquired the velocity and stress fields. It was seen that elasticity causes great variations in the position and shape of the yield surface, the deformation fields’ rate in the cavity and impacting the extra stress. The trends seen were in line qualitatively with the graphical outcomes existing in the related works.

Kaushik et al. [35] worked on a computational fluid dynamic simulation to study core annular flow via abrupt expansion and contraction. They modelled a core annular flow of water and lubricating oils with a VOF approach, and a good match among the numerical data and the outcomes from the experimental setup was yielded. A deep investigation was carried out to produce the profiles of velocity, pressure, and volume fraction through an extensive variety of water and oil.
velocities, for sudden contraction and expansion. Additionally, an asymmetric nature of velocity along the radial plane in the two cases was reported by them.

2.6. Sudden expansion in rectangular duct

Durst et al. [36] investigated the planar symmetrical abrupt enlargement flow for several low Re. They conducted several numerical predictions and measurements for the flow via a planar 2D channel with an asymmetric abrupt enlargement and the expansion ratio of 1:2. The two numerical and experimental predictions agreed to witness a symmetry breaking bifurcation of flow causing a short and long separation area for Re over 125, and on the basis of the altitude of the upstream channel as well as the highest flow velocity at the upstream. They witnessed a growth in the overall length of the long separation area. The short area remained stable with an increase in the Reynolds numbers to over 125. They applied a LDA to acquire the numerical predictions and empirical data that were conducted by using a finite volume method.

Mistrangelo and Buhler [37] numerically had studied MHD flows in abrupt enlargements of rectangular channels, using an asymptotic-numerical technique. The mathematical utility comprises in an extended variant of the commercial code CFX, allowing for effective outcomes and for any Hartmann numbers up to 1000. The asymptotic technique was used for creep MHD flow at magnetic fields that are very strong. The Numerical result was recorded for higher imposed magnetic fields and for a stable Reynolds number. They claimed that, for intermediate Hartmann numbers, eddies may appear underlying the enlargement. On the other hand, they were already suppressed at Ha value greater than 60. Additionally, areas with minimal velocities were recognized near the outer corners of the cross-sectional enlargement. In terms of intense magnetic fields, the internal layers were created at the enlargements that gather the flow from the upstream core and disseminate a relatively big portion to the side layers.

The extra pressure drop as a result of 3D induced currents was quantified, and so was the flow rate by the single internal layer and the side layer. Several tests on MHD flows within electrically conducting abrupt enlargements in rectangular channels were carried out by Buhler et al. [38] for high Hartmann numbers. The overall distribution of wall potential and the distribution of pressure across the Hartmann wall were estimated up to the values over Ha = 5000. The reduction in pressure data comprised an inertial distribution that may be rather considerable for mild N, but it disappears asymptotically with the increase of N up to the value of infinity. On the other hand, even for the peak result conducted in this study (N = 39,151), a relatively small inertial contribution was seen. In the case, N moves to infinity the overall measurement shows that inertia has less asymptotic predictions for the pressure distribution. On the basis of the examination of the wall potential, it is likely to detect the reduced velocity at the core centres and the inclined velocity nearer to the side walls when moving towards the enlargement. The side wall potential profiles provided more information on the distribution of the flow rates of the local side layer. This suggested a side layer jet that was strong directly underlying the enlargement. An evaluation between calculated and measured surface potentials was carried out for the symmetrical plane and at the side wall, which displayed parallel conclusions.

Aloui and Madani [39] investigated the features of flow of aqueous foam via a horizontal abrupt enlargement that is localized at the center of a horizontal channel. Using a square cross-section 21 × 21 mm² for the upstream, as well as a rectangle cross-section 21 × 42 mm² for the downstream, this uniqueness had an aspect ratio of 0.5. With the abrupt enlargement, they investigated wet foam flow dynamics near the singularity by using of PIV, regular pressure loss, static drainage, underlying liquid film thickness, as well as bubble sizes and local void fraction. Near the expansion, a singular pressure increase was seen by extrapolation with the use of the linear static pressure measurement evolutions acquired in the downstream and upstream at a remote distance from the singularity. On the other hand, the mean and fluctuating velocities with a PIV technique close to the singularity, were acquired from a set of 150 immediate snapshots for every selected quality of the low foam reorganization. They studied the impact of the existence of this singularity on the liquid film thickness values, for all the three scenarios under investigation (scenario A: U = 2 cm/s; scenario B: U = 3 cm/s; scenario C: U = 4 cm/s). Considering 1D flow and after the completion of the gravitational drainage, the liquid film thickness value does not depend on the foam velocity. In a 2D flow, this thickness is reduced in the case the foam velocity is inclined. With regards to the gravitational drainage, they had recorded that the overall volume of drained liquid was reduced with the quality of foam N, as well as with the foam velocity. Downstream of this backward-facing step, the local void fraction and the bubble size were both impacted by singularity pressure. These outcomes suggested that the existence of these singularities within the foam flow models may have a significant impact on the foam's end-of-duct layout.

Sousa et al. [40] studied the 3D flow of viscoelastic and Newtonian fluids via square-square enlargements. Graphical representations of the flow patterns were created with the use of streak photography; the flow's velocity field was estimated in detail with the use of a particle image velocity-meter. Moreover, a pressure drop measurement was recorded. They studied the Newtonian fluid flow for several expansion ratios: 1:2.4, 1:4, and 1:8, and then provided a comparison of the numerical results and the experimental results. For all of the area ratios investigated, a corner vortex was seen downstream of the backward-facing step, and an inclined in the flow inertia causes an improvement in the vortex size. They came to a consensus for a comparison of the numerical and the experimental results. The two non-Newtonian fluids flow was studied practically for several expansion ratios 1:2.4, 1:4, 1:8 and 1:12. These were compared with other numerical simulations using FENE-MCR, Oldroyd-B and sPTT constitutive formulas. In case of shear-thinning viscoelastic and the boyer fluids, a corner vortex was developed at the downstream of the enlargement, which was reduced in strength and size with the increase in the elasticity of the flow. The recirculation created downstream of the square-square enlargement for all of the fluids and expansion ratios investigated, shows a 3D layout via a helical flow. It could be noted that this was also anticipated in the numerical simulations conducted earlier.

3. Sudden expansion with some modifications

Several experimental setups were also built to investigate the flow properties and performance of sudden expansion parameters with slight changes in values (including incorporation of blowing across wall; of fence in diffuser zone; of suction across wall; of swirling devices in abrupt enlargement; and of diverging channel subsequent to abrupt enlargement).

3.1. Modification with suction and blowing method

The first empirical setup on the efficiency of vortex-controlled diffusers was recorded by Heskstad [41]. Heskstad investigated a suction slot for the convex discontinuity of a step enlargement within a circular conduit. A set of two 30° included-angle wedges were integrated to the suction gap. The authors used two expansion ratios with flow Re values ranging between 2 × 10⁴ and 20 × 10⁴. They employed uniform inlet velocity profiles that possessed thin boundary layers. They recorded that the flow turned the corners in a way that greatly reduced the recirculation length value.

The application of flow restriction by introducing backward-facing step was reported by Hahn et al. [42]. They had introduced very closely spaced slots within the near throat area to obtain distributed suction technique. The impact of a step height and Re on the transition of the boundary layer was also studied by them in the presence and absence of
suction in the flow line. They inferred that in the solely laminar separation flow, the reattachment length value progressively inclined with step height Reynolds number. They informed that the strength of the flow turbulence at the downstream of the reattachment region was greatly managed by the suction position the suction rate. They recorded that the presence of suction in the area of reattachment region introduced the prohibition of premature transition by the detached flow for a sudden expansion.

Verigin [43] experimentally studied the impact of a porous suction and injection for the pressure and drag recovery and the performance of a conical diffuser. The researcher employed a vacuum pump for the application of the suction force, where the divergence angle values and the expansion rate ratio were maintained as 12° and 5.15 respectively. The primary and secondary flows were kept at subsonic level with the Mach number of 0.34. The Re bounds for the injection and the suction were maintained as $(0.5-4) \times 10^5$ and $(1-5) \times 10^5$ respectively. The researcher witnessed a worthy property to be noted that the injection and suction had very slight impact on the downstream and upstream positions of the separation area; however, there was a great variation in the distribution of the static wall pressure. The researcher also recorded that the pressure recovery within the diffuser configuration decreased with porous injection. On the other hand, it aided the protection of the structural element from a series of thermal stresses. It was also reported that the pressure recovery was inclined throughout the full length with the porous section.

Spall [44] carried out a numerical experiment for a prototypical vortex diffuser. The diffuser's geometry considered comprises a step expansion within a pipe that had an area ratio of 2.25:1. He also addressed incompressible Reynolds-averaged Navier-Stokes formulas, using the K-ε turbulence system in the computational analyses. Bleed rates applied were within the 1–7% range. He acquired 80% enhancement in the diffuser performance. The outcomes provided useful insights on the mechanism of a vortex-controlled diffuser operation. The outcomes yielded were not in line with the theories that the inclined turbulence produced at the suction slot constrains flow separation across the wall downstream to the fence. The final outcomes drawn the claim by Heskestad [41], who mentioned that the diffuser accuracy was a cause of the flow turning around a sharp corner, therefore reducing the overall recirculation zone length.

Yang et al. [45] numerically carried out a research on fluid flow properties within the recirculation area, for a sudden expansion model with a stable normal mass bleed rate within the turbulent flow control in Fig. 11. The turbulent controlling formulas were attempted by a control-volume finite-difference technique with a power-law model. Non-uniform staggered grids were applied. A value of 1.3 was set as the conduit enlargement ratio, and air was the working medium. The extension of the attachment point at the downstream was depend on the normal injection which greatly impacts the flow field of the recirculation area underlying the step. With the inclination of the injection rate horizontal velocity in close proximity to the wall in the recirculation zone was reduced. Fig. 12 depicts the maximum reverse velocity and the reverse flow rate value with the normal injection within the recirculation zone which tended to decrease.

Janour and Jonas [46] investigated the effect of blowing and suction via an opening at the foot of the step, for a straight conduit using a rectangular cross-section for controlling the separation area. They witnessed that the smaller conduits had a directly smaller recirculation area compared to the larger ones. They concluded that Re value impacted the length only in the scenario of a small or narrow conduit. The researchers also investigated the impact of inlet turbulence on the separation area length. They concluded that the turbulence rates did not greatly impact the recirculation length, with the exception of a small suction rate. They recorded that, in the separation zone, the outer stream turbulence as well as the suction reduced dissipation losses. They also mentioned about the controlling of the vortices by using suction.

Batenko et al. [47] computationally studied the impact of blowing and suction on heat transfer as well as on friction within the separated laminar flow underlying a backward-facing step. Their investigation enclosed channel flow and unconfined scenarios. Their investigation showed that the suction and blowing were effective parameters impacting heat transfer and friction within the recirculation area. The local properties of friction and of heat transfer within the separation area were sensitive less to blowing compared to suction. They witnessed that, as strong injection was used, the recirculation bubbles directly underlying the step shrink. They also recorded that directly behind the step, areas of negative shear stresses were created due to the integrated action of injection and separation. In addition, they witnessed that, for low Re, the transverse wall flow via the permeable wall had significantly minor impact on the reattachment length value.

Uruba et al. [48] empirically studied various techniques for controlling flow separation such as blowing and suction within a small, narrow duct flow underlying a backward-facing step. The turbulent flow was studied at a Re of $5 \times 10^4$. They investigated the impact on flow control effectiveness of blowing and suction slot shapes. For this reason, they applied several slots that vary in shape (serrated or regular) and cross-section, as depicted in Fig. 13. They recorded that
blowing and suction were effective to manage flow separation in turbulent backward-facing step flow. Their investigation concluded that suction accuracy was insensitive to orifice profiles. However, blowing greatly relied on the slot profile, as since it entirely relayed on the entrainment of relatively significant momentum fluids in the recirculation area. In the following, they noted that the eddies structures near the expansion were influenced by suction.

Layek et al. [49] numerically carried out a simulation work on laminar separated flow within a symmetric abrupt enlargement conduit to investigate the impact of blowing and uniform porous suction, as well as on the symmetry breaking flow bifurcation. The researchers applied the finite difference technique to address the Navier-stokes formula in a staggered grid layout. They witnessed that there was a great drop in the wall shear stresses within the recirculation as the blowing speed increased. Moreover, the reattachment point was transformed in the upstream direction. Based on this, they concluded that blowing reduces the large recirculation area. They observed that the increase in the critical Re for flow symmetry bifurcation the application of blowing from the step walls inclined the stability to asymmetric perturbation. But the researchers realized an entirely opposing nature of flow in the case of suction initiated from the porous step wall. They noticed that with an increase in the suction velocities, there was an increase in the overall recirculation zone length. They noted asymmetry creeping in the flow with an increase in suction velocity. They also proposed that the suction can be employed as an asymmetry generator.

Sano et al. [50] investigated the control of separation by controlling of turbulent conduit flow for a backward-facing step with unceasing suction, via a small opening at the step’s bottom corner. The researchers estimated the wall static pressure and the local heat transfer coefficient value with the application of suction on the horizontal and the perpendicular direction individually. They witnessed that the Reynolds shear stress and turbulence strength value inclined with increasing suction, which caused a development in the local heat transfer coefficient in close proximity to the step. They recorded that the suction application in a turbulent channel underlying a backward-facing step and improved the pressure recovery process. This investigation showed that the suction direction had no considerable effect on the wall static pressure and the heat transfer coefficient.

The separation control for a backward-facing step flow with unceasing suction was studied numerically by Zheng et al. [51] using a turbulence model of large eddy simulation (LES). In their effort the suction flows coefficient was modified to investigate the impact of suction control on the flow fields. Their work showed that the suction was extremely effective to shorten the reattachment length value, and it also successfully reduced tangential velocity gradient and reattached flows turbulence fluctuations.

McManus and Bowman [52] empirically examined the application of flow control techniques to enhance the efficiency of air-breathing combustors with a set of abrupt enlargement parameters. They showed that the integration of flow direction vortices in the combustor inlet flow enables a corrugated shear layer, and creates a 3D flame structure. The flame layout change caused a 75% increase in volumetric energy release.

Chakrabarti et al. [53] carried out a computational investigation of the efficiency of a diffuser with a bleed slot in various positions (horizontal and vertical walls of the plain abrupt enlargement). They obtained optimal efficiency in the case the suction slot positioned at the vertical wall's top corner.

Forliti and Strykowski [54] explained the use of counter current shear for managing the non-reacting flow within a combustor with altered abrupt enlargement geometrical features. They applied a suction (about 10% of the main flow) based technique to obtain counter current shear control, through an opening at the abrupt enlargement plane. They witnessed that the impact of counter flow was dramatic on the turbulent and mean flow statistics. However, the optimal turbulence levels were seen to rise with suction.

3.2. Sudden expansion configuration with multiple numbers of steps

Mandal et al. [55] conducted a numerical model to investigate the efficiency of a abrupt enlargement structure with several steps for Re in the 20–100 range. The aspect ratio for every setting utilized was stable at 2. The researchers investigated the impact of several steps on static pressure and Re values increase, proximity to the location of maximum static pressure and diffuser accuracy increase. The researchers witnessed the advantages on stagnation pressure drop and diffuser accuracy and for several step configurations. They witnessed that the mean stagnation pressure drop was minimal in the scenario of multistep abrupt enlargement, compared to that of a plain abrupt enlargement for a certain Re. Their investigation showed that the rise in a set of steps, did not have a great influence on the effective length for a certain Re.

3.3. Other methods of modification

Jochmann et al. [56] investigated a gas turbine combustor with an altered abrupt enlargement setting. The overall practical gas turbine combustor comprises the axisymmetric combustion chamber and the swirl.

Shy [57] studied 2D axisymmetric steady flows for a modified sudden expansion setting. Based on the Re values, the properties of the corner and central recirculation areas for common flow patterns were categorized into open (and closed) annular flow, stable central flow, as well as vortex shedding.

Kibicho and Sayers [58] conducted an empirical investigation that focused on divided flows in entirely stalled wide-angled diffusers. The impact on the flow and heat transfer of the diffuser angle was investigated by Lan et al. [59] for a plane symmetric diffuser and an area ratio of 4.7. An efficiency simulation was conducted by Chakrabarti et al. [60] for a abrupt enlargement with fence at the Re in the bounds.
of 20–100, the fence's subtended 10° angle, and a fence non-dimensional distance from the throat for 0–2 for an aspect ratio of 2. They finally yielded optimal efficiency of abrupt enlargement, seen as a diffuser in the case the location of the fence was installed from the throat, at an overall distance of roughly 1 for greater Re with a subtended angle of 10°. They yielded the optimal performance for the diffuser, for the case of the fence position from the throat, at a non-dimensional distance of roughly 1 for a greater Re value in the scenario of a fence subtended angle of 10°.

4. Conclusion

Based on the literature review, it could be learned that numerous research works were performed involving both the experimental and numerical investigations on plain sudden expansions. Moreover, investigations were carried out on modified sudden expansions, including: the integration of suction across the wall; using single or double fence; the integration of the blowing across the wall; the integration of swirling devices; the induction of diverging channels after abrupt enlargement; as well as other modifications. Both the non-Newtonian and Newtonian fluids were attempted for the numerical experiments. As a conclusion, based on the above in-depth review of the existing literature, it has been witnessed that a complete study on the flow via plain abrupt expansions and sudden expansions with further modifications ceases to exist. Moreover, notable works on the computational study of non-Newtonian flow via abrupt expansions with several modified configurations are not available. The flow properties via altered abrupt enlargement as seen in combustors with Non-Newtonian fluids are very limited, and consequently, there is a large scope that requires further investigation in this area.

Acknowledgments

The authors gratefully acknowledge UMRG Grant RP012D-13AET and BK080-2016, University of Malaya, Malaysia, for support to conduct this research work.

Conflict of interest

All authors have received research grants from University of Malaya. The authors declare that they have no conflict of interest.

References


