Higher-order modelling of one-dimensional flows

AIMETA 2017
XXIII Conference
The Italian Association of Theoretical and Applied Mechanics
Luigi Ascione, Valentino Berardi, Luciano Feo, Fernando Fraternali and Antonio Michele Tralli (eds.)
Salerno, Italy, 4–7 September 2017

HIGHER-ORDER MODELLING OF ONE-DIMENSIONAL FLOWS

E. Carrera[1], D. Guarnera[2], and A. Pagani[2]

[1] Mul2, Department of Mechanical and Aerospace Engineering, Politecnico di Torino, Turin, Italy
e-mail: erasmo.carrera@polito.it

[2] Mul2, Department of Mechanical and Aerospace Engineering, Politecnico di Torino, Turin, Italy
e-mail: {daniele.guarnera, alfonso.pagani}@polito.it

Keywords: Fluid dynamics in pipes, Carrera Unified Formulation, One-dimensional models.

Abstract. *Flow fields are described by the well-known Navier-Stokes equations, whose resolution is extremely challenging in almost every real situation. Nevertheless, simplifications of the governing equations and the formulation of 1D models for incompressible flows, for example, can be of interest for many applications in which fast resolution times are demanded, such as fluid-structure interaction of flows in compliant pipes and hemodynamics. This work proposes a higher-order 1D model for the Stokes flows in rigid pipes. This methodology is based on Carrera Unified Formulation (CUF), which was first employed in structural mechanics. Velocity and pressure are here expressed as arbitrary expansions of the generalized unknowns, which are functions of the 1D computational domain coordinate. As a consequence, the governing equations can be expressed in terms of fundamental nuclei, which are invariant of the theory approximation order. Thus, by utilizing the finite element method to solve the related Galerkin formulation, FE arrays of the problem at hand can be derived automatically and in a unified manner for different kinds of expansions and theory accuracies. The case of Poiseuille flow in circular section pipes is presented in this work to validate this novel methodology. The attention is mainly focussed on the use of hierarchical McLaurin polynomials as well as piece-wise non-local Lagrange expansions of the problem unknowns in the formulation of the 1D CUF models. The preliminary results show the great advantages in terms of computational costs of the proposed method when applied to Stokes flows. Also, they provide enough confidence for future extensions to more complex fluid-dynamics problems and fluid-structure interaction analysis.*

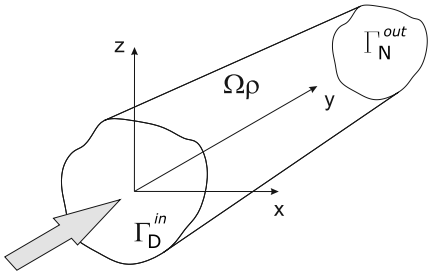


Figure 1: Computational domain Ω

1 Introduction

In nature, several kind of flows can be considered as mono-dimensional, and therefore can be approximate via 1D models, as suggested by Quarteroni et al. in [1] in the case of blood flow. In case of complex networks like the circulatory system, the utilization of 1D models can be convenient in terms of comprehension and computational costs. To this purpose, the employment of spectral method was proposed by Perotto in [2], whereas techniques based on proper orthogonal decomposition (POD) were presented by Salmoiraghi et al. in [3]. Formaggia et al. in [4], present non-linear 1D equations for blood pulsing propagation. A relevant contribute on reduced models for the circulatory system was proposed by Peiró and Veneziani in [5]. Another important aspect of the 1D approach is the capability to be coupled to more refined models (i.e 2D and 3D) in some parts of the domain, as suggested by Perotto et al. in [6].

A predominant direction allows the construction of a 1D FE model along the pipe longitudinal axis; the unknown field is then approximated trough the transversal cross-section by means of interpolation functions. This idea was exploited, initially, in structural mechanics by Carrera et al. in [7], by introducing Taylor (TE) polynomials as interpolation function. Subsequently, Lagrange (LE) and Legendre (HLE) functions were introduced, see Ref. [8] and [9]. The Carrera Unified Formulation (CUF), as presented in aforementioned works, allows to describe the unknown field as an arbitrary expansion of the generalized variables. 1D CUF has been employed in many other applications like composite beams and aerospace and civil engineering structures. Accounting for 1D CUF for fluid-dynamics problems, preliminary assess was obtained by Varello in [10] and by Pagani in [11] by means of TE polynomials. In the present work, one-dimensional flow models for incompressible and highly viscous fluids have been implemented employing the LE through the cross-section of the pipes.

2 One-dimensional CUF models for Stokes flow

The computational domain, is assumed in a Cartesian coordinate system as depicted in Fig. 1. Ω is the domain and $\partial\Omega$ represents the boundaries, whereas Γ is referred for the orthogonal cross-sections. Let consider the Galerkin approximation of the Stokes problem for incompressible flows with homogeneous boundary condition in Eq. 1:

$$\text{Find } \mathbf{u}_h \in V_h,\ p_h \in Q_h \text{ such that}$$

$$\begin{cases} \displaystyle\int_\Omega \nu\, \nabla \mathbf{u}_h : \nabla \mathbf{v}_h \,\mathrm{d}\Omega - \int_\Omega p_h\, \nabla \cdot \mathbf{v}_h \,\mathrm{d}\Omega = \int_\Omega \mathbf{f} \cdot \mathbf{v}_h \,\mathrm{d}\Omega & \forall\, \mathbf{v}_h \in V_h \\ \displaystyle -\int_\Omega q_h\, \nabla \cdot \mathbf{u}_h \,\mathrm{d}\Omega = 0 & \forall\, q_h \in Q_h \end{cases} \tag{1}$$

where ν is the kinematic viscosity in $[m^2/s]$ and $\mathbf{f}$ is the vector of body forces acting in Ω. $\mathbf{u_h}$ and p_h in Eqs. 1 are the discrete solutions of velocity in $[m/s]$, and pressure in $[m^2/s^2]$, of the Stokes problem in weak form. The Stokes formulation derives from the Navier-Stokes set of equations, by neglecting the non-linear convective term; this simplification is possible in presence of highly viscous fluids, as the case we are dealing with in this work.

The one-dimensional Carrera Unified Formulation (CUF) is here used along with FEM to approximate the Galerkin formulation of the Stokes equations. According to CUF, the velocity field $\mathbf{u}_h$ and the pressure field p_h are expressed, in a unified manner, as a generic expansion of the generalized unknowns through arbitrary functions of the cross-section domain coordinates:

$$\mathbf{u}_h(x,y,z) = F^U_\tau(x,z)\mathbf{u}_\tau(y), \qquad \tau = 1,2,\cdots,M^U \tag{2}$$

$$p_h(x,y,z) = F^P_m(x,z)p_m(y), \qquad m = 1,2,\cdots,M^P \tag{3}$$

where $\mathbf{u}_\tau(y)$ is the vector of velocity components and $p_m(y)$ is the scalar pressure, function of the pipe axis y. According to CUF, F^U_τ or F^P_m correspond to the expanding functions on the cross-section Γ, which is defined in the Cartesian plane xz, and M^U and M^P are the number of terms in the expansion, for velocity and pressure respectively. As briefly enunciated in the introduction, different kind of interpolation functions have been exploited in the framework of CUF models. Taylor expansion models (TE) employ hierarchical sets of 2D polynomials from Maclaurin series of the type $x^i z^j$ for the definition of $F_\tau(x,z)$. In this context, it is possible to increase the polynomial order with ease by adding higher-order terms to the unknown, enhancing the accuracy of the approximation. TE models have been extensively studied in many works, see for example [12, 13].

Lagrange expansion models (LE) are based on the use Lagrange-type polynomials as generic expansions on the pipe cross-section. The transversal physical surface is subdivided into a number of local expansion sub-domains, whose polynomial degree depends on the type of Lagrange expansion employed. Three-node linear L3, four-node bilinear L4, nine-node quadratic L9, and sixteen-node cubic L16 polynomials have been employed in the framework of CUF. The main feature of LE models is the possibility to make use of local expansions of pure unknowns variables, being these arbitrary placed over the cross-section surface. According to LE modelling, higher-order theories can be opportunely formulated by increasing the polynomial order (e.g., cubic L16) or by using a combination of polynomial sets on the conduit cross-section to have a piece-wise refined velocity (or pressure) field (see [8]).

2.1 Finite element formulation and CFD fundamental nuclei

As it concerns the FE approximation of the pipe axis, the generalized velocities $\mathbf{u}_\tau(y)$ and pressures $p_m(y)$ are described as a function of the unknown nodal vectors, $\mathbf{u}_{\tau i}$ and p_{mt} , and the 1D shape functions, N_i and N_t. Combining the FE classic procedure with CUF (Eqs. 3) the final expressions describing the unknown fields are

$$\mathbf{u}_h(x,\,y,\,z) \,=\, F^U_\tau(x,\,z)\,N^U_i(y)\,\mathbf{u}_{\tau i}, \qquad \tau = 1,\ldots,M^U \quad i = 1,\ldots,p^U+1 \tag{4}$$

$$p_h(x,\,y,\,z) \,=\, F^P_m(x,\,z)\,N^P_t(y)\,p_{mt}, \qquad m = 1,\ldots,M^P \quad t = 1,\ldots,p^P+1 \tag{5}$$

explained also in Fig. 2. In Eqs. 4 and 5, p^U and p^P represent the order of the FEM shape functions for velocity and pressure, respectively. The shape functions N^U_i and N^P_t can be arbitrary, and in general, different; in this work the classic lagrangian 1D shape function are considered, (see [14]). Approximating the generic discrete test function $\mathbf{v}_h \in V_h$ and $q_h \in$

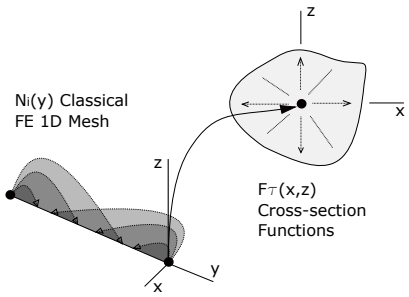


Figure 2: CUF technique

Q_h with a method equivalent to Eqs. 4 and 5, after extensive mathematical manipulations, the Galerkin approximation in Eq. 1 becomes the following system of algebraic equations:

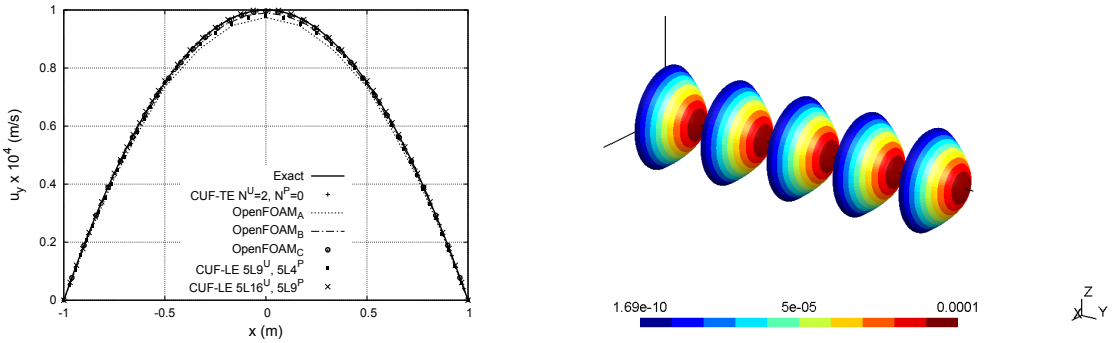
$$\begin{cases} \mathbf{A}^{\tau sij}\,\mathbf{q}_{sj} + \mathbf{B}^{\tau mit\,T}\,p_{mt} = \mathbf{F}^{\tau i} \\ \mathbf{B}^{mstj}\,\mathbf{q}_{sj} = \mathbf{0} \end{cases} \tag{6}$$

where $\mathbf{A}^{\tau sij}$, $\mathbf{B}^{\tau mit\,T}$, $\mathbf{F}^{\tau i}$ and $\mathbf{B}^{mstj}$ are the integrals in Eq. 1 written in terms of *fundamental nuclei*. The mathematical expressions of the nuclei are formally independent of the theory orders (N^U and N^P) and on the FEM shape functions (p^U and p^P) and this represents the core of the CUF technique. These nuclei have to be expanded against the indexes τ, s, m, i, j, and t and this leads to the construction of the elemental FE arrays associated to the Galerkin approximation of the Stokes problem. For further details on the expansions of the fundamental nuclei, interested readers are addressed to [12].

3 Numerical results: Poiseuille flow in circular-section pipe

The case study here presented, consists in the evaluation of fluid parameters in the case of Poiseuille flow in a cylindrical pipe. A number of 1D refined models are compared with analytical solutions and with finite volume results obtained with commercial software tool. The length of the pipe considered is $L = 6$ m and the radius is $r = 1$ m. No body forces are applied to the fluid and thus **f = 0** is taken into account in the Stokes equations. The fluid has a viscosity ν equal to $10^{-2}m^2/s$ to satisfy the condition of $Re \ll 1$. Moreover, the longitudinal axes of CUF models are discretized with 10 1D FEM elements, which ensure convergent results.

In the Poiseuille flow, the velocity **u** does not vary along the longitudinal axis y. In particular, the axial velocity component u_y describes a paraboloid in which the maximum value $u_{y_{max}}$ occurs at the centre of the section. To simulate this kind of flow, the model presents a homogeneous Dirichlet boundary condition on the lateral surface Γ^L (no-slip condition), and a homogeneous Neumann boundary condition on the outlet section Γ_N^{Out}. Conversely, a second-order non-homogeneous Dirichlet boundary condition is given at the inlet section Γ_D^{in}. According to the Poiseuille analytical solution [15], the axial velocity keeps a paraboloidal shape, as depicted in Fig. 3. As shown in Fig. 3(a), which gives the Poiseuille profiles for different 1D models and reference solution, a second-order for velocity and a zero-order for pressure are sufficient to detect the exact solution in the case of TE. On the contrary, in the case of LE, 5 cubic cross-section subdomains for velocity and 5 quadratic polynomials for the pressure are necessary to find the solution, as resumed in Tab. 1. On the other hand, 1D models are able to define the constant trend of the axial velocity, as depicted in Fig. 3(b), in agreement with Poiseuille analytical solution. The finite volume solutions are obtained by OpenFoam [16] and they come from three

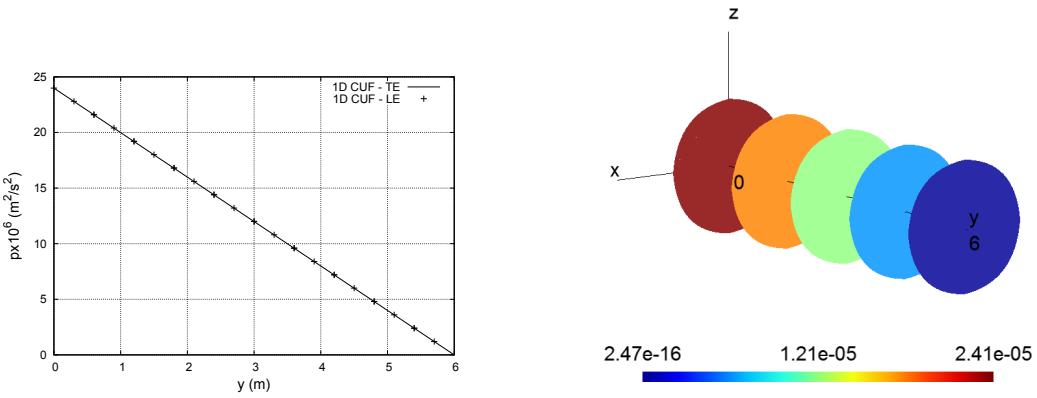


(a) Axial velocity profile at y = 3 m (b) 3D scheme of axial velocity along the pipe

Figure 3: Axial velocity u_y for Poiseuille Flow. Comparison of different models at midpspan section (a), 3D profile along the conduct (b). Results are in [m/s].

Model	$e_p(\%)$	$e_{u_y}(\%)$	$DOFs$
	OpenFOAM		
OpenFOAM$_A$	-0.72	-2.56	10560
OpenFOAM$_B$	-0.25	-0.97	54400
OpenFOAM$_C$	0.21	-0.39	435520
	CUF - TE		
$N^U 2, N^P 0$	-0.05	-0.05	389
	CUF - LE		
$5L9^U, 5L4^P$	+1.83	-2.19	2493
$5L16^U, 5L9^P$	-0.03	+0.24	5361

Table 1: Inlet pressure and maximum axial velocity in terms of percentage errors versus analytical solution for the Poiseuille flow. Comparison of OpenFOAM results with CUF results.



(a) Comparison between CUF-1D models

(b) 3D scheme of pressure trend in circular pipe

Figure 4: Pressure trend in Poiseuille flow. Comparison between 1D models TE and LE (a), constant value in cross-sections (b). Results are in $[m^2/s^2]$.

different discretizations. Namely, the model $OpenFOAM_A$ was constructed with 2640 finite volumes (132x20 mesh, where 132 stands for the number of volumes on the cross-section and 20 is the discretization along the y-axis), $OpenFOAM_B$ has 13600 (340 x 40) finite volumes, and $OpenFOAM_C$ has 108800 (1368 x 80) finite volumes. The value of pressure decreases along the pipe and does not have any dependency on the x and z coordinates, as in Fig. 4(b). In particular, it is important to underline that the outlet value is equal to zero, exactly as described by the Poiseuille analytical solution, see Fig. 4(a). Cubic and quadratic beam elements (i.e., B4 and B3) were used for the FE discretization along the y axis. The choice of the class of expansion is a key point of the analysis due to instability; in fact, finite elements of the same polynomial degree for both velocity and pressure are in general unstable.

4 Conclusions

In this work, the Carrera Unified Formulation (CUF) has been used for the analysis of incompressible, laminar and viscous fluids in rigid pipes. In particular, some higher-order 1D models for Stokes flows have been proposed. According to CUF, the primary variables of the flow (i.e. velocity and pressure) are expressed as arbitrary expansion of the generalized unknowns. By using these expanding functions on the cross-sectional plane, a unified finite element method has been developed straightforwardly. The case of Poiseuille flow has been taken into account as numerical assessment. In particular, Taylor expansion (TE) and Lagrange (LE) 1D models have been employed for the description of the flow. The results obtained from these 1D models have been compared with the analytical solution and with 3D solution obtained via finite volume commercial software. The analysis here conducted clearly underlines:

- LE and TE models based on CUF provide accurate results of velocity and pressure with respect to analytical solution.
- 1D models allow to obtain efficient results with lower computational costs compared with those obtained via 3D commercial software.

These promising preliminary results can be considered as a first step toward more advanced applications, like complex fluids and fluid-structure interaction.

REFERENCES

[1] A. Quarteroni L. Formaggia and A. Veneziani. *Cardiovascular Mathematics*, volume 1. 2009.

[2] S. Perotto, A. Reali, P. Rusconi, and A. Veneziani. Higamod: A hierarchical isogeometric approach for {MODel} reduction in curved pipes. *Computers and Fluids*, 142:21 – 29, 2017. Selected papers of 18th International Conference on Finite Elements in Flow Problems.

[3] F. Salmoiraghi, F. Ballarin, L. Heltai, and G. Rozza. Isogeometric analysis-based reduced order modelling for incompressible linear viscous flows in parametrized shapes. *Adv.Model. and Simul. in Eng. Sci.*, pages 3–21, 2016.

[4] L. Formaggia, D. Lamponi, and A. Quarteroni. One-dimensional models for blood flow in arteries. *Journal of Engineering Mathematics*, pages 251–279, 2003.

[5] J. Peiró and A. Veneziani. *Reduced models of the cardiovascular system*. Springer Milan, Milano, 2009.

[6] S. Perotto, A. Ern, and A. Veneziani. Hierarchical local model reduction for elliptic problems i: a domain decomposition approach. *A SIAM Interdisciplinary Journal*, 8:1102–1127, 1775.

[7] E. Carrera and G. Giunta. Refined beam theories based on a unified formulation. *Int. J. Appl. Mech.*, 2(1):117–143, 2010.

[8] E. Carrera and M. Petrolo. Refined beam elements with only displace- ment variables and plate/shell capabilities. *Meccanica*, 47(3):537–556, 2012.

[9] A. Pagani, A.G. de Miguel, M. Petrolo, and E. Carrera. Analysis of laminated beams via unified formulation and legendre polynomial expansions. *Composite Structures*, 156:78 – 92, 2016. 70th Anniversary of Professor J. N. Reddy.

[10] Alberto Varello. Advanced higher-order one-dimensional models for fluid-structure interaction analysis. *Ph.D Thesis*, 2013.

[11] Alfonso Pagani. Component-wise models for static,dynamic and aeroelastic analyses of metallic and composite aerospace structures. *Ph.D Thesis*, 2015.

[12] E. Carrera, M. Cinefra, M. Petrolo, and E. Zappino. *Finite element analysis of structures through Unified Formulation*. Wiley, 2014.

[13] G. Giunta, F. Biscani, S. Belouettar, A.J.M. Ferreira, and E. Carrera. Free vibration analysis of composite beams via refined theories. *Composites Part B: Engineering*, 44(1):540 – 552, 2013.

[14] Olgierd Cecil Zienkiewicz, Robert Leroy Taylor, and Robert Lee Taylor. *The finite element method*, volume 3. McGraw-hill London, 1977.

[15] S. Stera and R. Skalak. *The history of Poiseuille's Law*. 1993.

[16] OpenFOAM Fundation. OpenFOAM. www.openfoam.com, 2011–2014.