

Flow field explorations and design improvements of a hybrid rocket motor LOx feed line

David John Rajendran^{a,*}, Mani Santhanakrishnan^a, Vassilios Pachidis^a, Jerome Messineo^b

^a Cranfield Air and Space Propulsion Institute, Centre for Propulsion and Thermal Power Engineering, Cranfield University, Cranfield, Bedfordshire, United Kingdom

^b HyImpulse Technologies GmbH, Wilhelm-Maybach-Straße 5, 74196, Neuenstadt am Kocher, Germany

ARTICLE INFO

Keywords:

Liquid oxygen
Flow shear stress rates
Vorticity
Total pressure loss
Stream tube
Design exploration

ABSTRACT

The oxidizer system in a hybrid rocket motor needs to deliver the flow from a pressurized storage tank to multiple combustor ports. Pressure losses in the oxidizer system directly impacts combustor pressure and consequently the vehicle performance. However, oxidizer feed line designs till date have been done using simple 1D tools. Higher fidelity flow analysis methods have not been reported in the literature to identify loss generating features. Therefore, a design improvement study was carried out to identify and alleviate the impact of undesirable flow features in a typical oxidizer system design. An experimentally calibrated 3D RANS approach is applied to a typical LOx feed system which includes steps, splitters, ports, and pipes with multiple bends. These design features result in varying degrees of flow separation, secondary flows and vortical flow features and result in total pressure losses of up to 7 %. This loss means that the storage tank needs to be pressurized further to accommodate such losses and ensure combustor performance. A targeted design improvement approach that features simple, alternative, implementable solutions in the loss-generating regions is discussed. The best of these design improvements can reduce the total pressure loss to 4 %, indicating a 43 % reduction in the losses and reduced impact on storage tank design and combustor performance. Therefore, this paper demonstrates that a higher fidelity design enhancement process of the oxidizer feed system, which is often neglected in such detailed studies, can result in overall vehicle level design improvements to ensure mission targets are met effectively.

1. Introduction

Hybrid rocket motors, with liquid oxidizer and solid fuel, are increasingly becoming attractive due to several reasons that combine the best features of liquid and solid rocket motors. Even though hybrid rocket motors exhibit lower performance metrics compared to liquid rockets and lack the simplicity of solid rocket architectures, they can be appropriate for specific mission choices. In such cases, hybrid rocket motors are often the solution where there is a clear trade-off between simplicity and performance. Hybrids are characterized by simpler, cheaper, safer, and more robust architectures for near comparable performance with liquid motors. This benefit is pronounced in the case of hybrid rocket motors that use cryogenic oxidizers with solid fuels for enhanced performance, because of the requirement to only manage the on-board storage and distribution of a single cryogenic component. Additionally, hybrid motors are throttleable, a feature that is not easily feasible in solid rockets. Moreover, the ability to use non-toxic propellants in hybrid rocket motors is an important feature that will ensure

sustainable, eco-friendly access to orbit. Therefore, the development of hybrid rocket motors is one of the potential growth strategies being considered across the world for enhancing accessibility to space in the future [1–5]. One of the key technological elements that unlock the potential benefits afforded by hybrid rocket motors is the efficient design and development of the cryogenic liquid oxidizer feed system. The delivery of the liquid oxidizer at the required flow rate, pressure, and temperature from the storage tank to the combustion chamber throughout the flight profile of the rocket is critical to achieve the mission targets. The oxidizer properties play a major role in various key phenomena in a hybrid motor such as the fuel regression rate, combustion stability and efficiency [6–9]. This fluid dynamic challenge is further accentuated by the accompanying mechanical and structural design challenges associated with the handling of a cryogenic liquid on-board the vehicle. Therefore, improvements in the elements of the liquid oxidizer storage and delivery system are of interest in optimizing the performance of hybrid rocket motors.

The oxidizer system in hybrid rocket motors is typically pressure fed

* Corresponding author.

E-mail address: d.rajendran@cranfield.ac.uk (D.J. Rajendran).

<https://doi.org/10.1016/j.actaastro.2024.09.023>

Received 1 February 2024; Received in revised form 21 August 2024; Accepted 8 September 2024

Available online 13 September 2024

0094-5765/© 2024 The Authors. Published by Elsevier Ltd on behalf of IAA. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

to retain the architectural simplicity of overall vehicle. The delivery system of the oxidizer from the pressurized tank involves feed line ducting from the storage tank. There is a single duct that is connected to the tank to retain the simplicity and robustness of the storage system. Thereafter, the flow from the single duct from the tank is divided into multiple feed lines that provide radially distributed flow at the combustion chamber entry. The duct from the tank is centrally positioned at the tank outlet, while the ports in the combustion chamber are arranged around the outer periphery to ensure uniform oxidizer distribution. Therefore, the feed lines to the ports are bent radially to distribute the flow from a central position to peripheral positions. Moreover, additional bends may be introduced in the feed lines to reduce the packing volume and accommodate other rocket vehicle systems to enable an optimized overall assembly configuration. The estimation of the flow field properties, which include total pressure loss and flux rates, after this ducting and feed line in the oxidizer system is important to ensure optimal rocket performance and can contribute directly to better design optimization of the vehicle system [10,11].

The flow property requirements from the oxidizer feed system are estimated from system level, low fidelity overall vehicle performance models. Thereafter, the configuration design of the oxidizer system is done by using 1D piping system models. These models utilize standard pipe flow engineering correlations like Darcy-Wiesbach coefficient maps as implemented in commercial system dynamics tools like EcoSim, Simcenter AMESIM etc. to estimate the total pressure losses and flux rates in the feed system. Several examples of such 1D modelling for system design in electric pump-based feed systems and control strategy development is reported in the literature, predominantly for liquid rocket motors and to a limited extent in hybrid motors. A few of these are highlighted in Refs. [12–15]. Subsequently, the ability of the oxidizer system design to deliver the required flow properties are experimentally verified, after which the vehicle performance model is updated to ascertain the vehicle's ability to meet the mission requirements [16–18]. Higher fidelity flow simulations that include 3D (U)RANS analysis are used for other elements of the storage system, injector spray modelling, combustion process and the like to aid in the detailed engineering of the respective components [19–21]. On the other hand, the feed lines of the distribution system are generally not explored in higher fidelity and reported in the literature due to perceived conceptions regarding the lower level of benefits achievable from such an investigation. The typical design elements used in such systems like steps, splitters and bends have been explored in canonical fluid dynamic studies – selected examples of which are available in Refs. [22–24]. These canonical flow fields are like those observed in liquid oxidizer systems and serve as a qualitative validation for focused high fidelity feed system design explorations as in the present study. Additionally, few studies have described design methods to improve the flow field past such design features, as highlighted in Refs. [25–27]. But these canonical and design studies have focused on a generalized flow field improvement with no geometric or flow property constraints to the scope of modifications. The implementation of such improvements is not feasible for realistic liquid oxidizer systems because of restrictions on volume packaging, and interaction with other rocket systems. Therefore, a research gap exists as to the identification of alternate, implementable design improvements with constraints regarding the delivery of a fixed flow rate and packaging for specific application to liquid oxidizer systems in hybrid rocket motors. Additionally, it is desirable that these design improvements are advised by the reported knowledge regarding flow field improvements from the previous canonical studies and does not indicate significant departure from present day practice to ensure optimal integration with the rest of the system and ease of adoption.

Therefore, in this study, the use of higher fidelity flow analyses to improve the total pressure loss characteristics for the required mass flow rates with packaging constraints are investigated using geometry elements that are associated with a realistic liquid oxidizer feed system. The paper demonstrates that higher fidelity flow field exploration studies for

the elements in a piping system and simple, implementable design modifications identified from such studies can significantly reduce the pressure losses and provide added value in the overall optimization of the hybrid rocket vehicle. Such a first-in-kind identification of the extent of benefits achievable in terms of reduction in total pressure loss in the oxidizer feed system, demonstrates the importance of utilizing high fidelity analysis for this system, the design of which has hitherto relied only on simplified 1D models.

2. Model description and methods

2.1. Geometry choice for exploration

Three geometry features that are typically found in oxidizer feed lines, as shown in Fig. 1, are chosen for the design improvement studies. These three features are considered because they completely represent design features that are necessary to feed fuel to multiple ports from a central supply. The features and the reason for inclusion in liquid oxidizer systems are explained below:

1. A radial step element in the central ducting from the storage tank. This appears due to concentric pipe arrangements that are usual in cryogenic systems for thermal management and robust mechanical design.
2. A conical flow splitter that transitions the duct flow into annular flow before division into the individual feed lines. Associated with the conical splitter are small radial steps in the flow path that are necessary to transition the whole annular flow into split flows for the several feed pipes. In addition to the radial steps, there are also circumferential steps due to the requirement of the annular flow to enter discrete feed pipes that are arranged around the circumference of the flow distribution plane.
3. Several individual feed pipes that feature multiple bends to accommodate distribution from a central pipe to peripheral ports and for minimizing package volumes. An exemplary feed pipe with two bends of representative radius of curvature and bend angles are considered in this study. These geometries are put together to represent a baseline configuration that is used in pressure fed oxidizer systems after an engineering correlation-based design process. The boundaries of the volume considered for flow modelling is the part of the oxidizer system that is after the tank and up to the feed pipes before for the combustion chamber.

2.2. Domain discretization

A hybrid computational domain discretization approach is adopted for the geometry elements considered in the analysis. Such hybrid discretization approaches have been used to resolve the flow field with high accuracy in cases where good resolution of boundary layers is required as in the present study [28,29]. A tetrahedral core region is combined with a prismatic near wall region to optimally resolve the flow development. This is done with an explicit aim of capturing the boundary layer development that is critical for the estimation of the total pressure loss. The near wall region inflation layer is chosen to resolve the flow into the inertial sub-layer with appropriate choice of grid y^+ along with ensuring that enough elements are placed throughout the boundary layer. y^+ values of less than 1 are used in high gradient change regions, while ensuring that the y^+ value never exceeds more than 3 in the entirety of the computational domain. These values are sufficient to appropriately resolve boundary layer development. Additional refinement of core nodes is intentionally done in high curvature and step regions to ensure solver robustness and accuracy at possible high flow gradient regions. Several levels of grid refinement are tested before second order of accuracy in the interpolated results based on Richardson Extrapolation method is ascertained [18]. The selected grid size is ensured to be in the asymptotic region of the total pressure loss value

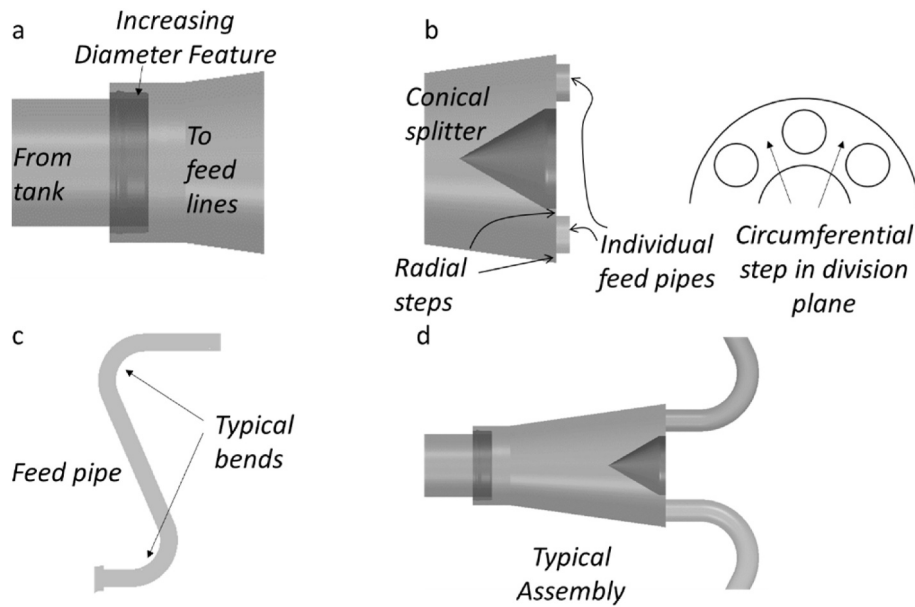


Fig. 1. Geometry elements considered in the analysis. a) Increasing diameter feature b) Conical splitter that transitions the duct into annular flow c) An exemplary bent feed pipe d) Assembly of these features.

with a significant margin away from regions of change. Additionally, the independence of the predicted flow properties of interest, viz. static pressures at discrete pipes, components of flow velocities into the splitter, and overall total pressure loss from the entry to the port exit for each pipe, to the degree of grid resolution is verified before finalization of the grid size. The total number of grid elements in the finalized grid used for the baseline geometry is around 25 million elements. The representative grid used for the analysis is shown in Fig. 2. Similar computational discretization strategies as in the baseline are followed for the subsequent design improvement studies as well.

2.3. Computational solution methods

The oxidizer fluid considered in this study is cryogenic Liquid Oxygen (LOx) because of its significant engineering usage experience in both liquid and hybrid rocket motors. Cryogenic LOx has higher performance

than other oxidizers because it is not diluted with inert elements and can be used with a wide variety of fuels. Moreover, even though other fluorine-based oxidizers have better performance, LOx is preferred because of its low toxicity. Therefore, it is a generally favored oxidizer due to its availability, and ability to deliver required performance when combined with paraffin based solid propellant fuels, and low toxicity [30].

In this study, the fluid behavior of LOx is generated from a full thermodynamic Equation of State (EoS) model. This is subsequently integrated into the solver setup for the total pressure loss estimation in both the baseline and the improved configurations. Fig. 3 shows the pressure-enthalpy diagram for oxygen from the EoS model. The operating region of pressure fed LOx systems are annotated within the Figure. By design, LOx is stored at pressures and temperatures away from the vapor dome as a cryogenic liquid to minimize the tank storage volume. LOx storage at these conditions results in fluid properties that

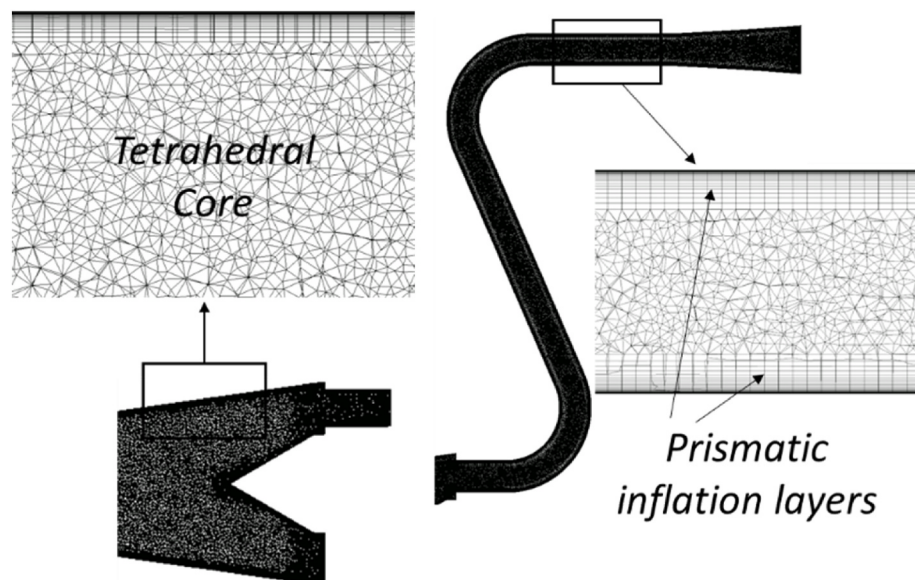


Fig. 2. Representative grid of the baseline geometry indicating hybrid discretization approach.

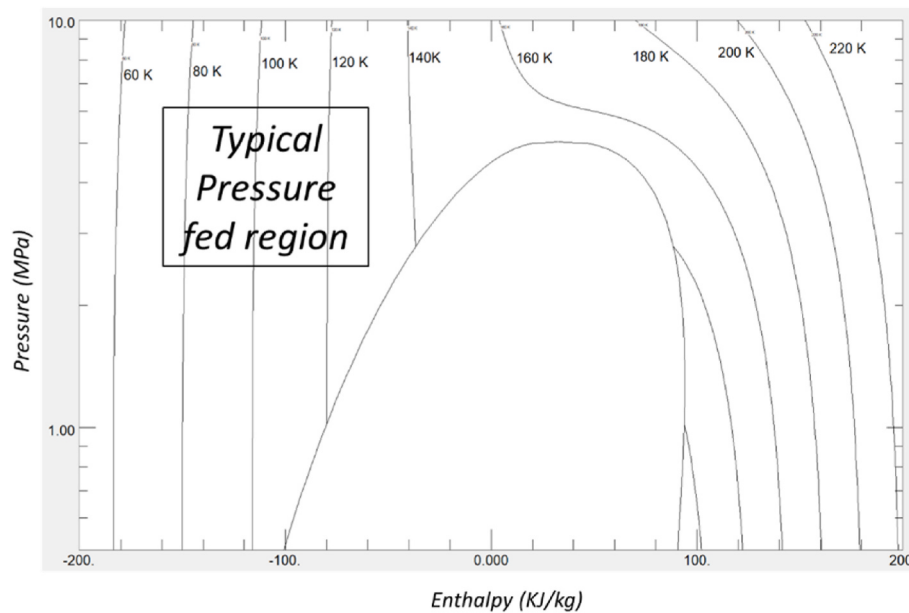


Fig. 3. Pressure-enthalpy diagram of oxygen showing LOX system operating region in pressure fed hybrids.

are within $\pm 10\%$ around a density of 1000 kg/m^3 , dynamic viscosity of $170 \times 10^{-6} \text{ Pa.s}$ and a feed pipe diameter based Reynolds number of 4 million. Absolute values of pressure and temperature corresponding to these storage conditions are within $\pm 10\%$ of 40 bar and 100 K [31]. The extents of total pressure loss in the feed system can amount to a worst-case limit of not more than 10 %. This would result in changes of LOx fluid properties of not more than 2 % in terms of density, specific heat capacity at constant pressure and thermal conductivity. Therefore, computed averages of thermodynamic and fluid properties in the operating regime of the LOx line are considered in the fluid model for the analysis. The near negligible changes in density across the LOx feed line warrants the adoption of a viscous incompressible algorithm to accurately predict the flow field features and properties.

The flow field solutions are carried out in an implicit finite volume implementation with Green-Gauss based interpolation method and higher order flux splitting schemes in ANSYS Fluent. The prescription of total pressure and total temperature at the inlet and chamber static pressure at the feed pipe outlet results in a well-posed numerical problem for the solver. Alternative implementations with mass flow rates specified in the feed pipe outlets are used to verify the static pressure in the chamber. This is then iteratively implemented in the baseline static pressure outlet boundary condition to match the mass flow rates. Turbulence closure in the RANS equations is implemented using the two-equation turbulence kinetic energy (k) – turbulence dissipation rate (ω) Shear Stress Transport (SST) model. At least second order accurate advection and turbulence numeric schemes are used in the solution of the discretized equations in the computational domain. Parametric and solver residual-based convergence criteria are used to determine convergence in the steady state flow field analyses when the iterative differences in the parameters of interest are less than 0.1 % and in a maximum equation residual tolerance of less than 1×10^{-4} . Second order backward Euler time-stepping schemes are used for exploratory unsteady analyses. These (U)RANS analyses were used to shed light on the possible transience of feed flow at the feed pipe outlets. The computational solutions are carried out in the High-Performance Computing (HPC) facility, 'DELTA 2', that has a peak processing power of 60 Tflops with 1888 cores. Using a block of 128 cores, and a Message Passing Interface (MPI) parallelization scheme, each steady state analyses require around 3 h to complete. Each iteration in the unsteady run with at least 10 inner loop iterations for ensuring spatial convergence before time marching requires 10 min. A full unsteady run with very fine

advection time scales to capture high vortex shedding frequencies requires around 48 h for resolving multiple flow passes of $\sim 3\text{s}$ real time.

The computational methods are initially applied to the baseline geometry to identify flow features and their associated loss generating mechanisms. Based on the flow field results, targeted design modifications to address individual design features that are responsible for increased total pressure loss are implemented. These design modifications deal with methods to suppress flow separations by eliminating adverse pressure gradients or geometric discontinuities [32], and for managing secondary flow components that result in a loss generating vortical flow field [33]. Each of these zonal design modifications are discretized and computationally solved using the same method as the baseline geometry to provide an indication of the extent of total pressure loss reduction possible. The individual design modifications are ensured to respect manufacturing, structural and assembly constraints and are therefore representative of achievable simple improvements in engineering practice.

2.4. Experimental calibration of computational approach

The key element in the application of higher fidelity computational methods for design improvement as attempted in this study is the extent of confidence in the generated results and the general applicability of the computational method chosen to resolve the flow field. Therefore, the study utilized an experimental campaign in a similar LOx feed line system to pre-calibrate and ascertain the ability of the approach to predict the flow field properties. The experimental setup model included a conical flow splitter arrangement with several feed pipes with bend parameters. However, the experiments did not include the radial step in the central duct as in the baseline geometry. This is because the radial step feature is typically used in the vehicle for thermal and structural considerations and are not necessary in the experimental arrangement. Even though the omission of the radial step would impact the overall pressure loss in the system, the test set up supports to 'calibrate' the flow analysis for investigation of pressure loss reducing features. Furthermore, the key challenging element of the investigation regarding the flow division into discrete pipes from the central duct is achieved through the experiments. Additionally, the individual feed pipe geometries also featured differences compared to the flight geometry at the combustor outlet port. In the typical flight geometry, the feed pipe diameter and the outlet port have the same diameter. However, in the

experiments, the outlet port diameter was only a third of the feed pipe diameter. This is shown in inset of Fig. 4. The remaining circumferential two-thirds were blocked using an annular piece of metal to match the mass flow rate available for the experiments. The inclusion of this feature introduces an adverse pressure gradient that is numerically more challenging to simulate than the increasing step feature. Therefore, calibrating the method with this feature increases the confidence in the subsequent results.

The experimental geometry was solved using the same computational approach that was considered for the design improvement study. The experiments measured static pressures and temperature upstream of the conical splitter – station 1 and downstream of the second bend in the feed pipe – before the diverging portion to the outlet – station 2, and the pressure at the outlet that is representative of the chamber pressure. The static pressure taps were located at a single outboard location on the pipe circumferential periphery and the temperatures were measured at a single point. The experimental conditions were specified in terms of the tank conditions of the liquid oxygen and the mass flow rates during the run. The non-dimensional static pressure contours at a mid-surface plane in the central duct and in one of the pipes from the experimental setup are shown in Fig. 4. Fig. 4 shows one of the several discrete pipes that branch off from the central duct. The analysis was carried out with all the pipes around annulus and the boundary conditions specified at all the pipe outlets. The static pressures at station 1 and 2 annotated in the figure as numerical results are averages computed at the outer annulus. These values show 0.3 % and 1.9 % difference at stations 1 and 2 respectively. The uncertainty in the static pressure measurements in the experiments are around 0.5 % and are single point measurements. The imposition of the chamber pressure measured in the experiments resulted in a computed mass flow value that has a variation of less than 0.1 % from the measurements. All the temperatures in the experiments were matched in the model within 0.1 %. Furthermore, in all the measurement locations, the computations under-predict the static pressure which would consequently lead to higher predicted flow velocities and losses than in the experiments. This makes the computational solutions conservative in their estimate of subsequent quanta of improvements achievable in the design modifications. Therefore, from this calibration study it is noted that the computational approach developed in this study is sufficient for the purposes of a design improvement study due to the ability of the numerical model to estimate the measured parameters within a narrow error band.

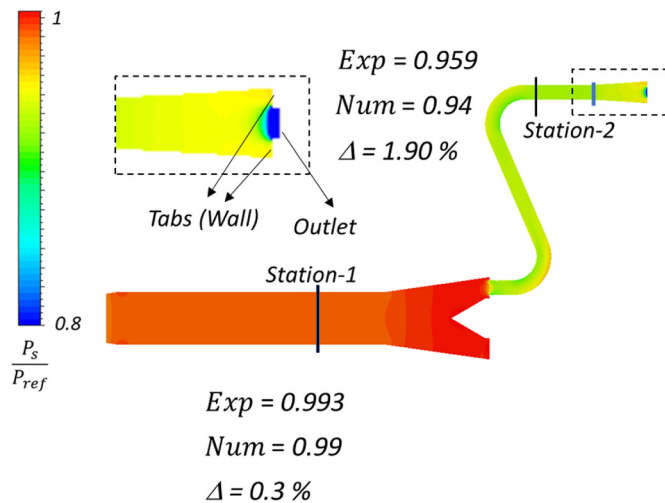


Fig. 4. Non-dimensional static pressure contours in rig setup used for calibration of computational approach.

3. Results and discussions

To highlight the specific loss generating mechanisms and associated exemplary design improvements, the discussion is organized as follows:

1. Initially, the flow field signatures associated with each of the three chosen design features in the baseline geometry is described for flight operating conditions. This discussion will quantify the extent of total pressure losses associated with each of these flow mechanisms due to the design elements.
2. Based on the results of the baseline flow field, a zonal design improvement approach that aims to ameliorate the undesirable loss generation mechanisms associated with each design feature is adopted. The improvements in the total pressure loss achievable by each of these modifications are explained.
3. Finally, the wider applicability and added value from the higher fidelity flow analyses vis-à-vis the traditional engineering correlation-based approach is qualitatively examined.

3.1. Baseline flowfield features

Central Duct Radial Step Feature. The presence of the discontinuous radial step increase in the LOx flowpath – Fig. 1a – would be expected to result in an increased total pressure loss from typical engineering correlations. The classical approach using correlations for sudden step expansion in pipes based on the Darcy friction factor. The resistance coefficient (K) for such expansion in high Reynolds number flows is computed by the equation below

$$K = (1 + 0.8f) \left[1 - \frac{D_{in}^2}{D_{out}^2} \right]^2$$

where f is the Darcy friction factor, D_{in} is the diameter of the smaller pipe and D_{out} is the diameter of the larger pipe [34]. The pressure loss is then given by the resistance coefficient multiplied by the dynamic head. The application of this formula for the increasing diameter feature results in a total pressure loss of 0.4 %. Such a value of total pressure loss from the correlation does not provide information on the flow distribution downstream of this sudden increase in diameter. The streamlines of the flow from the tank to the conical splitter is shown in Fig. 5 using non-dimensional total pressure contours on a mid-surface plane. The streamlines in the baseline case without step is shown for reference in Fig. 5.

The LOx flow stream tube from the tank ducting continues in the radial bounds of the upstream pipe before the radial step without any significant spread-out, as is expected in dense liquids. The regions beyond the stream tube bounds are occupied by regions of recirculation. The recirculation is primarily fed by the pressure gradient imposed by the stream tube outer bounds. The dynamics of flow and pressure evolution around the stream tube indicates that a negligible amount of flux rate is sufficient to saturate this outer periphery into a metastable recirculation region. The development of this recirculation region is further bolstered by the diverging portion in the higher diameter pipe. This diverging portion is used to ensure a smooth transition into the conical flow splitter before subsequent division into the feed pipes. The tightness of the LOx stream tube and the surrounding impact of the recirculation region causes a squeezing of the central LOx stream tube. This causes the flow that enters the conical splitter to be attached to the inner annulus around the cone. The outer annular regions around the cone are occupied by the recirculating flow. Moreover, there are minor signatures of circumferential asymmetry in the conical splitter annular exit plane before the division into the feed pipes due to the recirculation flow. The net effect of this flow field in terms of the total pressure loss, defined as the ratio of the drop in total pressure across the feature to inlet total pressure, amounts to ~3 %. This computation for the total

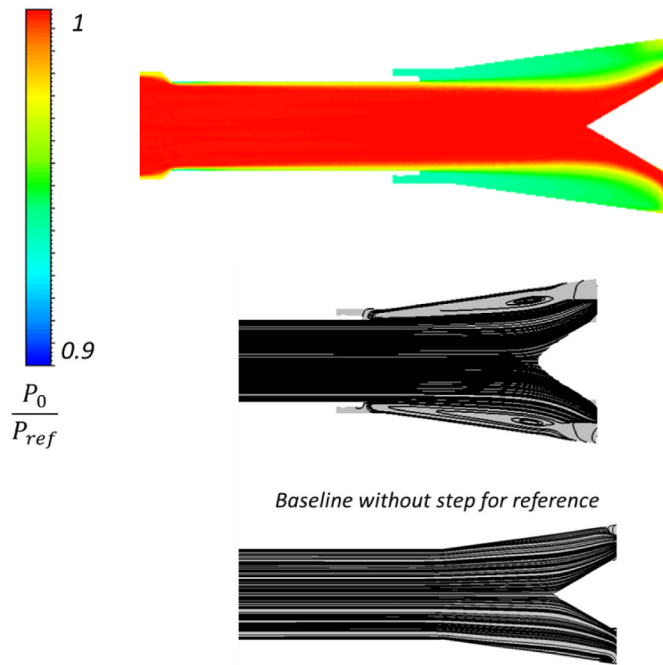


Fig. 5. Non-dimensional total pressure contours and streamlines due to the baseline increasing diameter feature.

pressure loss considers a plane $0.25D$ upstream and downstream of the design feature. The traditional estimate utilizing engineering correlations, as mentioned above, cannot provide accurate levels of pressure loss estimates or any additional insight into the stream tube concentration along the inner annulus of the larger diameter duct. Therefore, the use of such correlations can have significant impact on the total pressure loss estimate of any downstream components that would need to handle such a flow field.

Conical Splitter. The conical splitter – Fig. 1b – transitions the flow from a pipe flow into an annular flow and subsequently divides the annular flow into discrete feed flows. The conical splitter includes radial and circumferential steps due to the requirement to split the flow into individual feed pipes, which are manufactured and assembled separately. Conventionally, these steps are often considered to be significant

loss generating mechanisms in the splitters into the discrete pipes and are as such used as handles to access the total pressure loss correlations. However, these correlations cannot account for the flow property distribution before the splitter and is therefore not attempted for this feature. But instead, the differences caused by the centrally bound tube in the realistic case is highlighted by comparison with a notional uniform stream tube case. The non-dimensional total pressure contour in the notional uniform case and the realistic central stream tube case is shown in Fig. 6. The projected surface streamlines at planes upstream, P1, and downstream, P2, of the division into a single feed pipe are also shown in Fig. 6.

In the notional uniform case, the division of the annular flow into discrete feed pipes is characterized by the pre-divisional organization of the streamlines. Since the flow must develop through radial and circumferential steps before entering the individual pipes, the corresponding minor changes in the streamline are visible in plane P1. However, the flow field is predominantly uniform before the division into the individual pipes. Immediately after the flow enters the feed pipes for the notional case, the effect of the radial and circumferential step appears as a region of low-pressure separated flow around the circumference of the feed pipe. An outboard flow migration due to radial deflection of the stream tube in the conical splitter is observed. Thereafter, a swirling flow field with two counter rotating vortices is observed due to the circumferential pressure gradient. The net effect is that the inboard regions of the feed pipe are starved of the flow. This region occupies nearly a third of the flow area.

In the realistic case, the radial skew in flow distribution accentuates the non-uniformity in the flow field. In the plane P1, the pre-divisional streamline spread indicates a radial skewness with the outer annular region occupied by the recirculating region. This introduces residual swirl to the flow before division into the feed pipes. The signature of the flow occupying the inner annular region and spreading between the circumferential steps is also visible as regions of high total pressure. Two counter rotating vortices are observed in the plane P2 due to the swirl introduced by the radial and circumferential steps before the flow enters the individual pipes. Further, owing to the larger extent occupied by the low momentum flow in the feed pipe inboard regions, the separated flow region also indicates residual recirculation. The net effect of all these pressure gradients, flow migrations and vortical flow fields appear as a larger pressure drop than the notional uniform case. The total pressure loss due to the conical splitter, by considering the measurement planes $0.25D$ upstream and downstream of the feed pipe division, in the

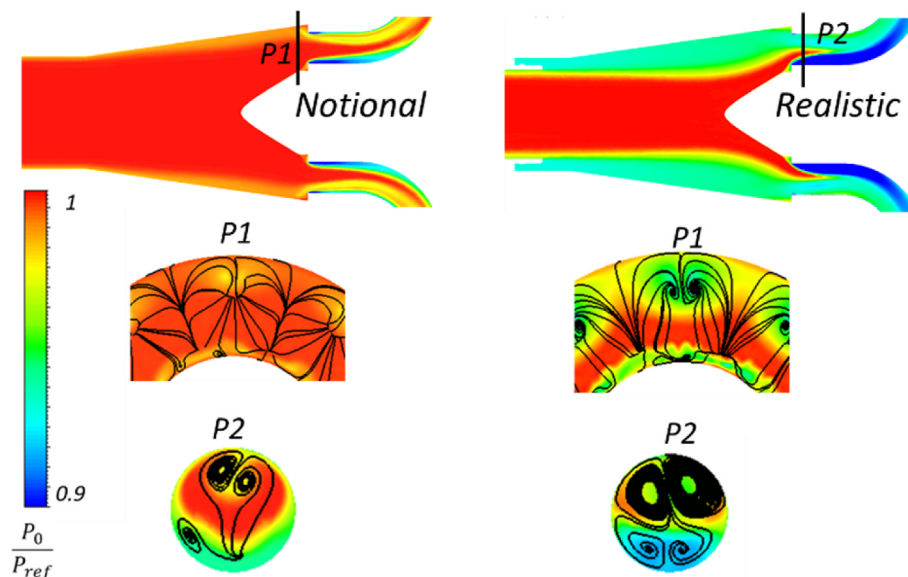


Fig. 6. Non-dimensional total pressure contours and projected surface streamlines due to the baseline conical splitter.

realistic case is $\sim 3\%$, while in the notional case it is $\sim 1.4\%$. The doubling of the pressure loss in the realistic case indicates the importance of resolving flow non-uniformity that is typically neglected in oxidizer feed line design process. Additionally, the effect of upstream flow features affecting the pressure loss of the component is demonstrated. Such upstream effects cannot be captured using typical Darcy-Weisbach correlations for estimating pressure loss.

Feed Pipe. The flow development in the individual feed pipes through the bends result in the appearance of vortical flow features and separations that contribute to the total pressure loss. The non-dimensional total pressure loss contours with the projected surface streamlines for one such feed pipe in a mid-surface plane and in cut planes along the flowpath are shown in Fig. 7. The surface streamlines on the cut planes are projected streamlines which indicate regions of vortical flow development perpendicular to the plane. The overall bulk flow as indicated in the mid-surface plane clearly indicates the convection of the

separated flow features and the subsequent response of the flow to the bends.

At the first bend at the lower radius, the inward turning of the flow helps in the redistribution of the outward migrated streamlines. This redistribution helps in reducing the extent of the separated flow region that would be expected in the inward region of the first bend. The signature of the canonical Dean vortices associated with flow turning with the bend are noticed in the 3 cut planes, 2 to 4, showing the flow evolution through the bend. As the flow traverses the bend, the counter rotating vortex pair develops in the inboard regions before they are suppressed by the bulk flow development beyond the inward separated region. Therefore, the vorticity of this turn vortices is greatly reduced in the cut plane 5 at the middle of the feed pipe between the bends.

At the second bend at the higher radius, the pressure gradient induced by the bend appears again as vortices that occupy the complete pipe diameter. This is because the second bend has a tighter radius of

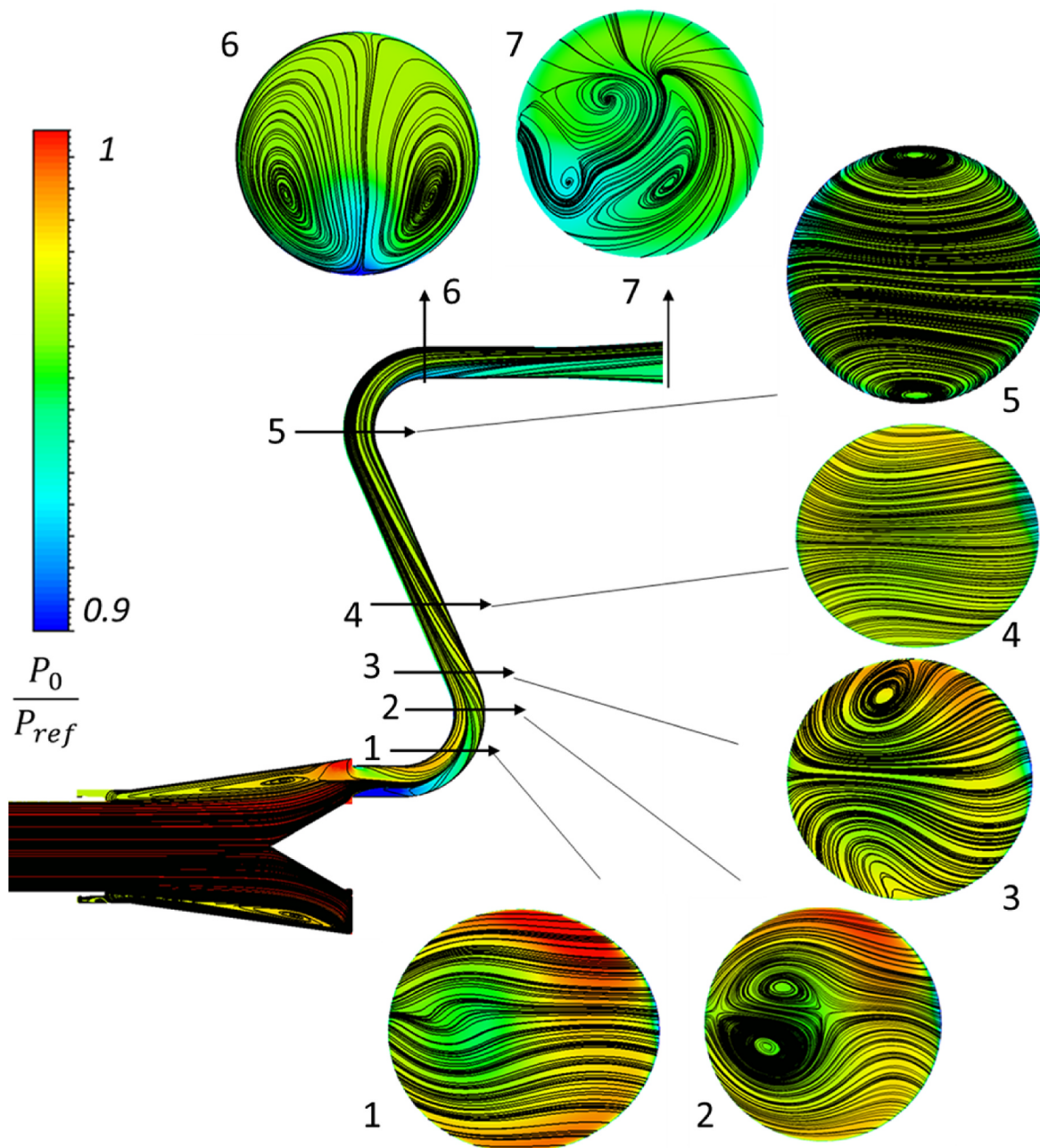


Fig. 7. Non-dimensional total pressure contours and projected surface streamlines on a feed pipe.

curvature which induces a stronger pressure gradient. The bulk flow in the second bend separates from the inboard side and is deflected radially outwards. It is in this region of deprived flow in the inboard side that the vortex core is located. Therefore, these vortices, though they appear elongated does not significantly affect the flux rates other than inducing additional minor secondary swirl components of velocity. Beyond the second bend, there are no relieving pressure gradients that can aid the flow reattachment in the inboard side. Therefore, the flow continues along the outboard side before it redistributes based on the potential field induced by the feed outlets. This flow distribution induces additional swirl and radial components that break the coherence of the turn vortices before it feeds the feed outlets. Even though the overall flow field in the feed pipes appear to be dominated by vortices, the separated flow regions in the bends are the dominant loss generation mechanisms. This is because vortical regions in liquid flow are associated with low shear values. The net effect of the feed pipe in terms of the pressure loss coefficient from the conical splitter outlet calculation plane and the feed outlet plane is $\sim 1\%$. The flow field development in the notional uniform case, without the radial step, is also similar with slightly different levels of vorticity and total pressure loss because the bend flows respond primarily to the upstream flux distributions which are not significantly different in both cases.

3.2. Zonal design adaptations

A zonal approach that targets alleviation of specific flow field features without compromising manufacturability, structural integrity and assembly requirements are explored. This is done to identify achievable improvements in the total pressure loss over the baseline geometry designed using the classical approach with engineering correlations.

Contoured Sleeve Insert with Ogival Splitter. This design modification is targeted to manage the tight stream tube that occupies the inner annulus in the central duct. This would reduce the consequent total pressure loss and the non-uniform flow property distribution introduced by the increasing diameter feature. Contoured sleeve inserts which could be placed within the baseline geometry is posited as a feasible solution to manage the sudden discontinuous change in the flow path diameter. The schematic of the design modification is shown in Fig. 8. The design modification is shown in a darker schematic to emphasize all changes are within the flowpath and geometry of the baseline case. Therefore, the concentric pipe arrangement that is required for managing thermostructural issues is retained externally. The sleeve could be either metallic or a polymer-based material with metal edges to enable welding or other alternative modes of assembly depending upon the mission flight time. The key feature of this design modification is that it does not involve any change in the LOx feed system external geometry because it

could be completely fitted within the existing geometries. The sleeve insert would match the inner radius of the central duct step and the conical splitter annular plane outer radius at its extremities through a contoured flowpath that controls the flow splitting and diffusion. Since the discontinuous step in the geometry is just upstream of the splitter and the flow field generated by it affects the annular transition of the flow, the sleeve design modification is combined with a change of the internal conical geometry with an ogival splitter geometry. This change to the ogival geometry can be done internally within the flow path following traditional manufacturing methods without affecting the overall assembly. The combination of the ogival geometry and the contoured sleeve provides a controlled flow splitting and area progression to the annular division plane. The inner radial step in the feed pipe division plane is removed by mating it with the end of the ogival splitter. This could be done by an additional insert from the annular plane or as part of the ogival splitter.

The flow field change with the contoured sleeve insert in terms of the non-dimensional total pressure contours is shown along with the baseline in Fig. 9. The contoured sleeve with the ogival splitter can effectively distribute the LOx stream tube and transition it smoothly into an annular flow. The outboard recirculation regions in the central duct are eliminated and the only total pressure loss generating mechanism is wall shear. Consequently, the total pressure loss in the region of the radial step in the baseline has reduced from 2.5 % to less than 1 % in the contoured region. The total pressure loss calculation in the improved design includes the ogival splitter because of its close integration with the sleeve. In addition to improving the total pressure loss in the central duct, the effective transition to annular flow has resulted in better flow characteristics down into the feed pipe as well. The overall total pressure loss in the complete geometry including the feed pipe outlets have reduced by 3 %.

‘Flower’ Circumferential Inserts. The improved flow field highlighted in the contoured sleeve-ogival splitter in the section above indicates separation in the flow splitting region that feeds the individual feed pipes. The reason for this separation is only the circumferential step around the discrete feed pipes that is fed by the combined annular flow. The radial steps are removed by the ogival splitter matching with the feed pipes. Therefore, to smoothly ease the combined annular flow into the feed pipes, simple 3D contoured inserts that can potentially be stuck or welded on to the transition plane is proposed. A schematic of the inserts is shown in Fig. 10. These inserts are placed in the circumferential steps on the transition plane in such a way that there is a smooth curvature onto the feed pipe entry. Since these inserts are individual small pieces, the arrangement around the circumference results in a distinctive petal like arrangement as in flowers and hence these are termed so. This design modification is implemented along with the contoured sleeve-ogival splitter design to isolate this affect over an already improved configuration. The change in the flow field with the ‘flower’ circumferential inserts over the contoured ogival reference is shown by using non-dimensional total pressure contours in the mid-surface plane and in two cut planes in the vicinity of the transition region in Fig. 11.

The smooth guidance of the flow into the feed pipes with the ‘flower’ inserts effectively eliminates the circumferential separation as observed in the cut planes P1 and P2. In the plane P1, the flow contour with the ‘flower’ insert is uniform except for minor radially inboard and outboard depressions associated with the upstream end wall boundary layer development. The plane P2 clearly indicates the smooth contoured transition into the feed pipe that eliminates the concomitant edge separation. The elimination of this circumferential separation is felt all the way up to the feed ports in the availability of a higher-pressure flow that effectively reduces the separation at the first bend and results in a reduction in the loss at the second bend. The net effect of this ‘flower’ insert appears as a $\sim 1\%$ improvement in the total pressure loss in the complete geometry over and above the already improved flow field of the contoured sleeve – ogival splitter modification.

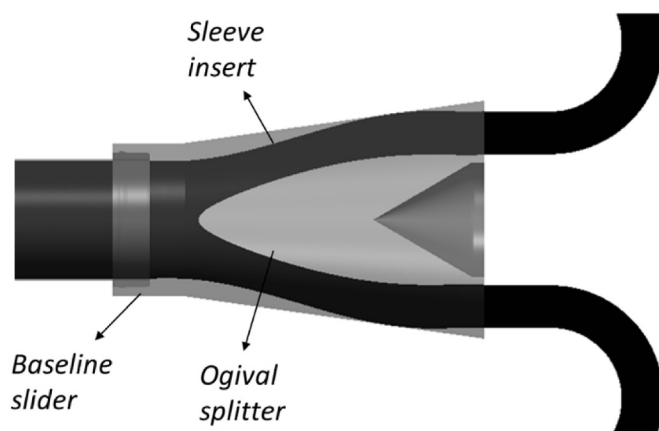


Fig. 8. Contoured sleeve insert with ogival splitter shown in dark colour. Baseline geometry is shown in grayscale.

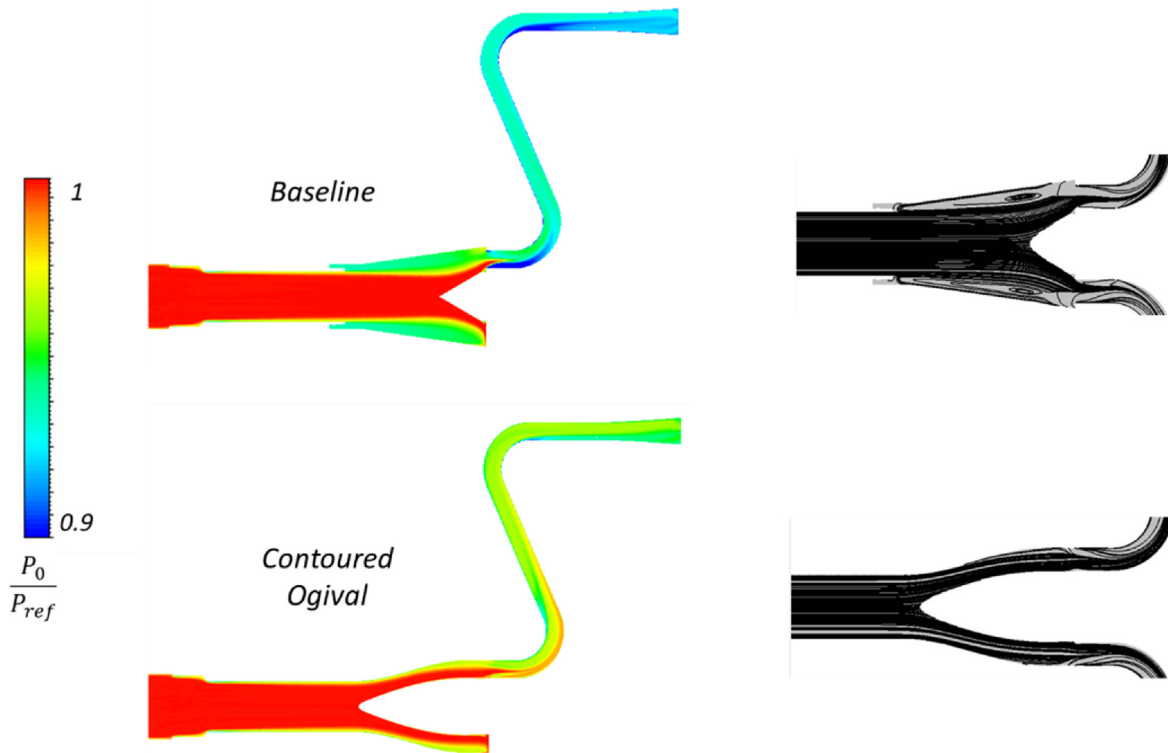


Fig. 9. Non-dimensional total pressure contours comparison between baseline and contoured sleeve with ogival splitter design modification.

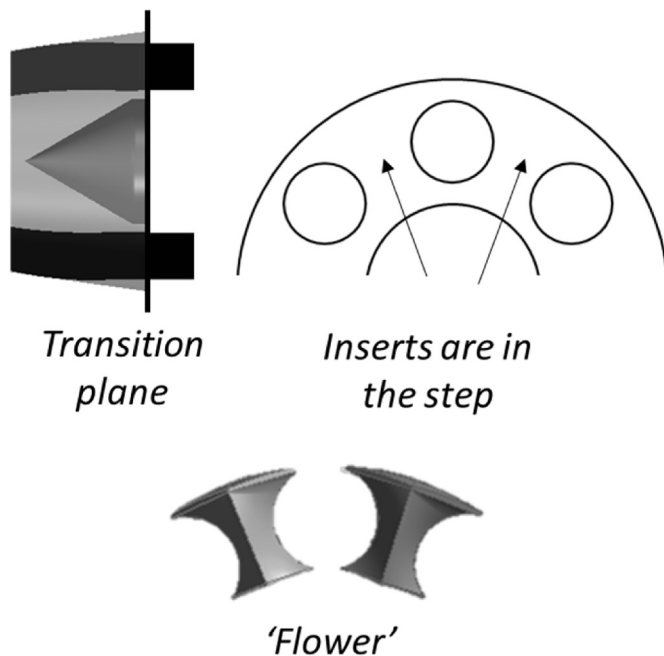


Fig. 10. 'Flower' circumferential insert to smooth the flow into the individual feed pipes.

Like the elimination of the circumferential step using these inserts, a concurrent modification in the baseline geometry that removes the radial step using a smooth ellipsoidal geometry was also explored. This resulted in an overall total pressure loss improvement of 0.6 %. As expected, this is lower than the circumferential insert because of the smaller step and the larger region of influence along the circumference.

Parametric Bend Manipulation in Feed Pipes. The baseline geometry

indicated that the higher radius second bend has a higher influence on the overall total pressure loss due to having a smaller radius of curvature. The effect of this total pressure loss is further accentuated due to the proximity of the second bend to the feed outlet. Moreover, the alleviating effect of the radially skewed flow due to the central duct radial step feature in comparison with a notional uniform flow was noted in the previous section. Therefore, to isolate the benefits due to the bend parameter manipulation in a more compromised flow field, the notional uniform flow geometry that sets up a higher loss generating flow in the feed pipe is considered for this design modification. Additionally, the manipulation is attempted only in the second bend to retain much of the original geometry without any change. The extent of parametric change explored was restricted by the assembly constraints imposed due to other rocket elements. An exemplary design modification and its impact on the non-dimensional total pressure contours at the mid-surface plane and in a cut plane after the second bend is shown in Fig. 12. The radius of curvature in the design modification is increased by a factor of 1.3 compared to the baseline. The increase in the radius of curvature is sufficient to significantly suppress the separation in the second bend. The change in the back pressure due to the reduction in separation also improves the flow throughout the feed pipe by reducing the separation in the first bend as well. The suppression of the separation also significantly reduces the flow vorticity in the plane P1 downstream of the bend leading to lower secondary components of velocity in the proximity of the feed pipe. The improvement in the overall flow throughout feed pipe also appears as more symmetric and lower levels of vorticity in the first bend as well. The net impact of the increase in the bend radius results in a 0.4 % reduction in the overall total pressure loss across the feed pipe. The higher radius of curvature in this study is selected in such a manner as to be able to ensure tangential continuity with the straight part of the pipe into the combustor port. Further increase in radii of curvature would compromise the flow development into the combustor port and are hence not explored.

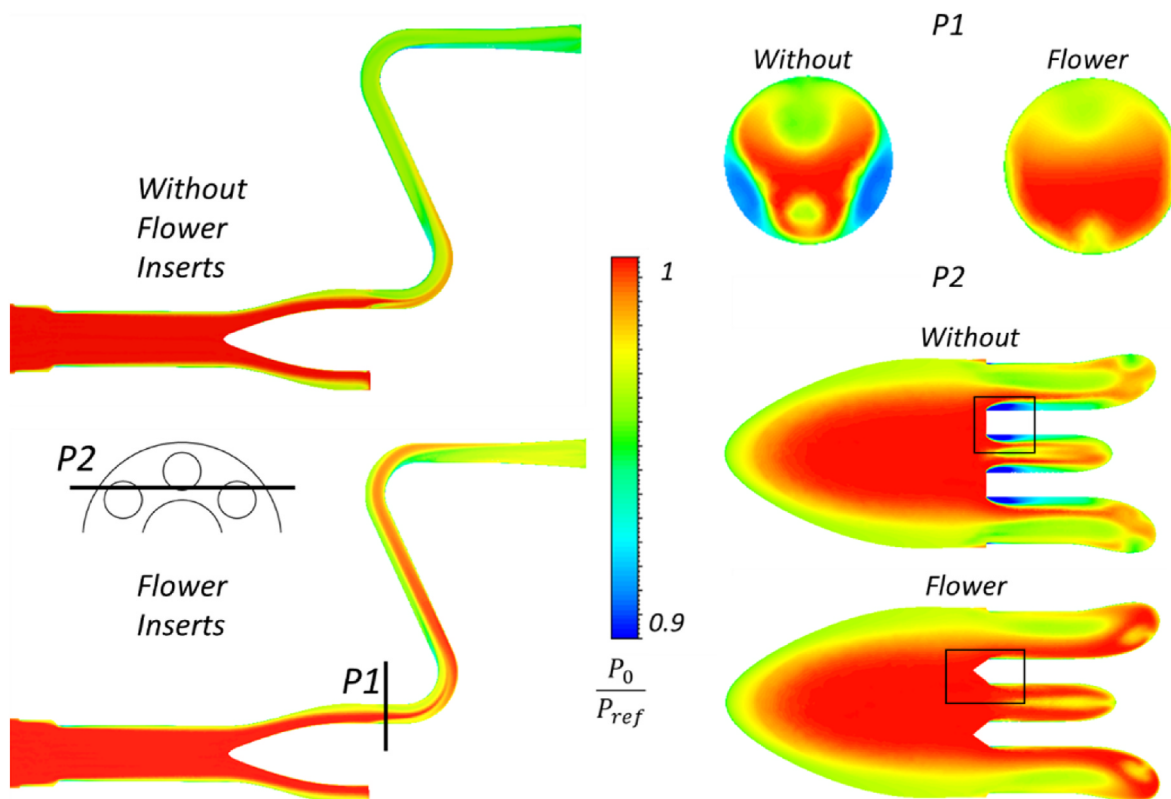


Fig. 11. Non-dimensional total pressure contours for the 'Flower' circumferential insert and the reference contoured sleeve-ogival splitter modification.

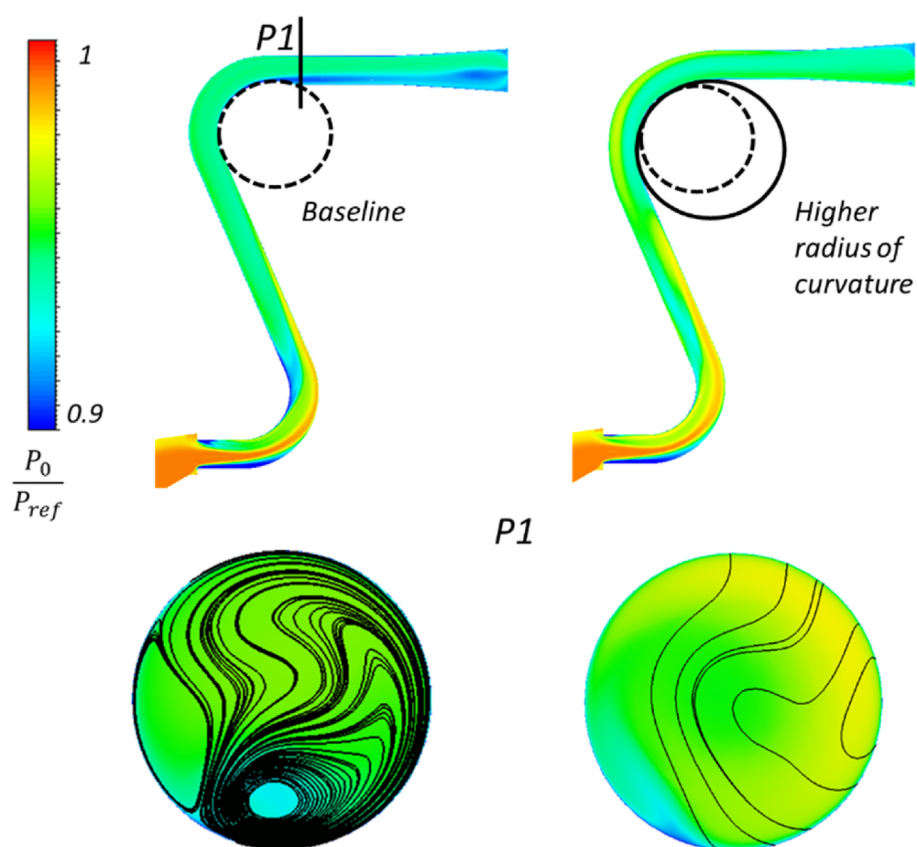


Fig. 12. Non-dimensional total pressure contours for different bend radius of curvatures in the feed pipe.

3.3. Applicability to general LOx feed system design

The computational approach as applied in this study utilizes design elements that are found in LOx feed systems of hybrid rocket motors. The total pressure loss and flow property distribution results observed in the study are dependent on the shape and form factors of the LOx line delivery elements rather than the specific dimensional forms. This includes general radius ratios introduced by radial step feature mechanisms, the geometric features of the conical flow splitter like axial positioning, annulus to central duct area ratio, radius ratios of the radial and circumferential steps, number of feed pipes, and their bend geometric parameters. While a generalized non-dimensional study of all possible combinations of these geometric parameters would provide a comprehensive map of the total pressure loss in the ducting, the results from this study provide value in operating within the most likely ranges of these parameters to identify critical flow features and their associated loss mechanisms. The identified loss mechanisms are ameliorated by using targeted design approaches that respect other vehicle level constraints and most often include easily implementable solutions that can provide significant improvements in the vehicle performance. All the computations are carried out by considering the fluid as cryogenic liquid oxygen in a representative thermodynamic property range that is likely to be found in flight vehicles. Such a choice, combined with an experimental calibration of the computational approach makes the study to provide insights into the extent of potential total pressure loss improvements possible in-flight geometry of any general LOx feed system.

4. Conclusions

This work has demonstrated the way short burst, quick, experimentally calibrated, higher fidelity flow analyses can aid in the design of lower total pressure loss LOx oxidizer feed systems for hybrid rockets by optimizing the geometries rather than an engineering correlation-based classical approach. This was explored by utilizing a notional baseline flight geometry of the LOx feed system that incorporates commonly used design features developed using a correlation-based approach. The key conclusions around the design elements considered in this study are:

1. The use of engineering pressure loss correlations based on Darcy friction factor for radial step features under predicts the pressure loss due to the discontinuity in the flow path. Contrastingly, a higher fidelity characterization can provide information on the stream tube development driven non-uniformity in the flux distributions which subsequently affect the performance of the entire LOx feed system downstream of this feature.
2. In the conical splitter, the presence of a non-uniform, skewed flow property distribution is likely to introduce differences up to 1.5 % in the estimate of the total pressure loss as compared to a notional uniform flow field assumed in typical design approaches. An inboard radially skewed flow field as in the baseline geometry with the splitter affects the subsequent division of flow in the feed pipes by introducing higher levels of non-uniformity. This was observed to result in unexpected benefits in managing separations in the bends due to the geometry and the pressure gradients developed from the skewed flow distribution as compared to a notional uniform flow.
3. A reasonable prediction of total pressure loss in the LOx feed system, as possible from a calibrated higher fidelity approach can help in the development of pre-experimental system models to verify vehicle capabilities.
4. A zonal design modification study indicated that a contoured sleeve with ogival splitter geometries can improve the total pressure loss by up to 3 %. The elimination of circumferential and radial steps can reduce the total pressure loss by up to 1 % and 0.6 % respectively. Additionally, increasing the radius of curvature at higher radius bends can reduce the total pressure loss by up to 0.4 %. Such an increase in the delivery pressure through such design improvements

will have first order impact on the vehicle performance and can feed back into the development of further optimized vehicle geometries.

A more generic optimization framework with assembly-based constraints imposed is planned to generate a notional geometry that can divide the central duct flow into several discrete feed pipes with minimum total pressure loss. While the experimental calibration of the computational approach provides reasonable confidence on the improvements predicted for the design modifications, a further validation campaign of few design elements will be helpful in establishing a multi-disciplinary optimization enabled ‘CFD-in-the-loop’ framework for future design optimizations.

Funding

The work was funded by UK Space Agency by grant P20491 of the LaunchUK Technology Investment program.

Nomenclature

D	diameter
EoS	Equation of State
f	Darcy friction factor
HPC	High Performance Computing
k	turbulence kinetic energy
K	resistance coefficient
LOx	Liquid Oxygen
MPI	Message Passing Interface
RANS	Reynolds Averaged Navier-Stokes
SST	Shear Stress Transport
(U)RANS	Unsteady RANS
y+	Non-dimensional distance
ω	turbulence dissipation rate

CRediT authorship contribution statement

David John Rajendran: Writing – review & editing, Writing – original draft, Visualization, Supervision, Software, Methodology, Investigation. **Mani Santhanakrishnan:** Software, Resources, Methodology, Investigation, Formal analysis, Data curation. **Vassilios Pachidis:** Supervision, Resources, Project administration, Funding acquisition, Conceptualization. **Jerome Messineo:** Writing – review & editing, Supervision, Methodology, Investigation, Conceptualization.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

- [1] N.A. Davydenko, R.G. Gollender, A.M. Gubertov, V.V. Mironov, N.N. Volkov, Hybrid rocket engines: the benefits and prospects, *Aero. Sci. Technol.* 11 (1) (2007) 55–60.
- [2] A. Okninski, On use of hybrid rocket propulsion for suborbital vehicles, *Acta Astronaut.* 145 (2018) 1–10.
- [3] A. Mazzetti, L. Merotto, G. Pinarello, Paraffin-based hybrid rocket engines applications: a review and a market perspective, *Acta Astronaut.* 126 (2016) 286–297.
- [4] A. Sibio, Hybrid space propulsion systems: development and main challenges. ESS2023-PaperNo:502-SFT-1. Proceedings of 58th Engine Systems Symposium 2023, March 15 – 16, 2023, Cranfield University, Cranfield, UK, 2023.
- [5] C. Schmierer, M. Kobald, K. Tomilin, U. Fischer, S. Schlechtriem, Low cost small-satellite access to space using hybrid rocket propulsion, *Acta Astronaut.* 159 (2019) 578–583.
- [6] G. Leccese, E. Cavallini, M. Pizzarelli, State of art and current challenges of the paraffin-based hybrid rocket technology, in: *AIAA Propulsion and Energy 2019 Forum*, 2019, p. 4010.

- [7] A. Karabeyoglu, B. Cantwell, J. Stevens, Evaluation of the homologous series of normal alkanes as hybrid rocket fuels, in: 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 2005, July, p. 3908.
- [8] A. Okninski, W. Kopacz, D. Kaniewski, K. Sobczak, Hybrid rocket propulsion technology for space transportation revisited-propellant solutions and challenges, *FirePhysChem* 1 (4) (2021) 260–271.
- [9] J. Lee, S. Rhee, J. Kim, H. Moon, Combustion instability mechanism in hybrid rocket motors with diaphragm, *J. Propul. Power* 38 (2) (2022) 309–318.
- [10] F. Barato, E. Toson, D. Pavarin, Variations and control of thrust and mixture ratio in hybrid rocket motors, *Advances in Astronautics Science and Technology* 4 (2021) 55–76.
- [11] S. Predoi, S. Grigorean, G. Dumitrascu, Case study regarding the use of three different fuels in a multi-stage hybrid rocket engine, in: *IOP Conference Series: Materials Science and Engineering*, vol. 997, IOP Publishing, 2020, December 012153, 1.
- [12] D. Spiller, A. Stabile, D. Lentini, Design and testing of a demonstrator electric—pump feed system for liquid propellant rocket engines, *Aerotecnica Missili & Spazio* 92 (2013) 123–130.
- [13] J. Li, N. Yu, P. Zeng, G. Cai, Design and integrated simulation of a pressurized feed system of the dual-thrust hybrid rocket motor, *Sci. China Technol. Sci.* 56 (2013) 989–1000.
- [14] C.P. Frank, O.J. Pinon-Fischer, D.N. Mavris, C.M. Tyl, Design methodology for the performance, weight, and economic assessment of chemical rocket engines, *J. Aero. Eng.* 30 (1) (2017) 04016071.
- [15] L. Casalino, D. Pastrone, Oxidizer control and optimal design of hybrid rockets for small satellites, *J. Propul. Power* 21 (2) (2005) 230–238.
- [16] M. Gieras, A. Gorgeri, Numerical modelling of the hybrid rocket engine performance, *Propulsion and Power Research* 10 (1) (2021) 15–22.
- [17] B. Genevieve, J.F. Pitot de la Beaujardiere, M.J. Brooks, A computational tool for predicting hybrid rocket motor performance, *R&D Journal* 33 (2017) 56–65.
- [18] A. Zavoli, P. Maria Zolla, L. Federici, M. Tindaro Migliorino, D. Bianchi, Surrogate neural network for rapid flight performance evaluation of hybrid rocket engines, *J. Spacecraft Rockets* 59 (6) (2022) 2003–2016.
- [19] J.E. Durand, F. Raynaud, J.Y. Lestrade, J. Anthoine, Turbulence modeling effects on fuel regression rate in hybrid rocket numerical simulations, *J. Propul. Power* 35 (6) (2019) 1127–1142.
- [20] T.I.A.N. Hui, D.U.A.N. Yu, Z.H.U. Hao, Three-dimensional numerical analysis on combustion performance and flow of hybrid rocket motor with multi-segmented grain, *Chin. J. Aeronaut.* 33 (4) (2020) 1181–1191.
- [21] H. Tian, X. Li, P. Zeng, N. Yu, G. Cai, Numerical and experimental studies of the hybrid rocket motor with multi-port fuel grain, *Acta Astronaut.* 96 (2014) 261–268.
- [22] H. Le, P. Moin, J. Kim, Direct numerical simulation of turbulent flow over a backward-facing step, *J. Fluid Mech.* 330 (1997) 349–374.
- [23] H. Liu, P. Li, J. Van Lew, CFD study on flow distribution uniformity in fuel distributors having multiple structural bifurcations of flow channels, *Int. J. Hydrogen Energy* 35 (17) (2010) 9186–9198.
- [24] P.M. Ligrani, R.D. Niver, Flow visualization of Dean vortices in a curved channel with 40 to 1 aspect ratio, *Phys. Fluids* 31 (12) (1988) 3605–3617.
- [25] W.A. El-Askary, M. Nasr, Performance of a bend–diffuser system: experimental and numerical studies, *Comput. Fluid* 38 (1) (2009) 160–170.
- [26] J. Zhang, D. Wang, W. Wang, Z. Zhu, Numerical investigation and optimization of the flow characteristics of bend pipe with different bending angles, *Processes* 10 (8) (2022) 1510.
- [27] J. Cao, M. Kraut, R. Dittmeyer, L. Zhang, H. Xu, Numerical analysis on the effect of bifurcation angle and inlet velocity on the distribution uniformity performance of consecutive bifurcating fluid flow distributors, *Int. Commun. Heat Mass Tran.* 93 (2018) 60–65.
- [28] S. Crippa, Application of novel hybrid mesh generation methodologies for improved unstructured CFD simulations, in: 28th AIAA Applied Aerodynamics Conference, 2010, p. 4672.
- [29] J. Marcon, M. Turner, J. Peiro, D. Moxey, C. Pollard, H. Bucklow, M. Gammon, High-order curvilinear hybrid mesh generation for CFD simulations, in: 2018 AIAA Aerospace Sciences Meeting, 2018, p. 1403.
- [30] David Altman, Rocket motors, hybrid, in: Robert A. Meyers (Ed.), *Encyclopedia of Physical Science and Technology*, third ed., Academic Press, 2003, pp. 303–321, <https://doi.org/10.1016/B0-12-227410-5/00835-8>. ISBN 9780122274107.
- [31] M.L. Huber, E.W. Lemmon, I.H. Bell, M.O. McLinden, The NIST REFPROP database for highly accurate properties of industrially important fluids, *Ind. Eng. Chem. Res.* 61 (42) (2022) 15449–15472.
- [32] Y. Nagano, M. Tagawa, T. Tsuji, Effects of adverse pressure gradients on mean flows and turbulence statistics in a boundary layer, in: *Turbulent Shear Flows 8: Selected Papers from the Eighth International Symposium on Turbulent Shear*

Flows, Munich, Germany, September 9–11, 1991, Springer Berlin Heidelberg, Berlin, Heidelberg, 1993, pp. 7–21.

- [33] S. Dey, Secondary boundary layer and wall shear for fully developed flow in curved pipes, *Proceedings of the Royal Society of London. Series A: Math. Phys. Eng. Sci.* 458 (2018) (2002) 283–298.
- [34] B. Silowash, *Piping Systems Manual*, McGraw Hill Professional, 2009.



Dr Rajendran is a Lecturer within the Rolls-Royce University Technology Centre for Aero Systems Design, Integration and Performance at Cranfield University. David's research expertise is in integrated system design space exploration with different levels of modelling fidelity. David's research interests include development of robust high fidelity methods for numerical flow solutions in complete integrated systems and system level interdependency explorations to develop optimum aerodynamic designs for future propulsion architectures. He plays an active role in several UK government and industry funded doctoral programmes and projects. He has received several prestigious awards in both the academia and the industry.



Dr Santhanakrishnan received the B.Tech. degree in Aeronautical Engineering from Madras Institute of Technology, Chennai and the M.Eng. degree in Aerospace Engineering from Indian Institute of Science, Bangalore, India. After working in aerospace industry for more than a decade, he embarked on PhD research in the field of Topology Optimization at the University of Greenwich, United Kingdom. He obtained his doctoral degree in 2018 and currently working as a research fellow in the Cranfield University, UK. He has authored 2 patents, 10 conference papers and 6 journal articles. His research interests are Aerodynamics, CFD, Turbomachinery, Engineering optimization and Heat transfer.



Prof. Pachidis is a Professor of Propulsion Integration Engineering and Head of the Centre for Propulsion and Thermal Power at Cranfield University in the UK. He is the Director of the Rolls-Royce University Technology Centre (UTC) in Aero Systems Design, Integration & Performance at Cranfield and the Director of the Cranfield Air & Space Propulsion Institute (CASPI). Prior to this, Vassilios acted as the Head of the Gas Turbine Engineering Group within the Centre for Propulsion. He has published widely (more than 80 peer-reviewed journal papers and more than 100 conference papers) and holds 19 joint patents with Rolls-Royce plc.



Dr. Messineo is the propulsion group leader at HyImpulse in Germany. His research work has been focused on hybrid rocket engines and he worked on this topic at ONERA in France and at JAXA in Japan before joining HyImpulse. He obtained several awards such as a best student paper at EUCASS conference, a thesis prize for his PhD, and obtained a H2020 Marie-Curie grant to support his work. At HyImpulse he leads the team that developed and qualified the largest hybrid rocket motor ever produced in Europe.



DSPACE

<https://dspace.org/>

Flow field explorations and design improvements of a hybrid rocket motor LOx feed line

Rajendran, David John; Santhanakrishnan, Mani Sekaran; Pachidis, Vassilios; Messineo, Jerome

2024-12

Elsevier

<https://doi.org/10.1016/j.actaastro.2024.09.023>

Rajendran DJ, Santhanakrishnan M, Pachidis V, Messineo J. (2024) Flow field explorations and design improvements of a hybrid rocket motor LOx feed line. Acta Astronautica, Volume 225, December 2024, pp. 273-284